



Oasys GSA

Help Guide



YOUR IDEAS BROUGHT TO LIFE

13 Fitzroy Street
London
W1T 4BQ
Telephone: +44 (0) 20 7755 3302
Facsimile: +44 (0) 20 7755 3720

Central Square
Forth Street
Newcastle Upon Tyne
NE1 3PL
Telephone: +44 (0) 191 238 7559
Facsimile: +44 (0) 191 238 7555

e-mail: oasys@arup.com
Website: <http://www.oasys-software.com/>

Oasys GSA

Copyright © Oasys 1985–2017

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

Printed: June 2017

Table of Contents

Part I About GSA	23
1 Overview	23
2 GSA Analysis Features	23
3 GSA Design Features	24
4 GSA Program Features	25
5 Design Codes	26
6 Validation	32
7 Acknowledgements	32
Part II Step By Step Guide	34
1 Becoming familiar with GSA - An Example	34
Tabular Input	35
Graphical Input (Sculpt)	42
2 Constructing a GSA model	43
Generating models	44
Generating 2D element meshes	44
Sculpting models	48
Spreadsheets, CAD and Step files	48
3 Summary of analysis element types	48
4 Summary of analysis types	52
5 Requesting Analysis	54
Simple static analysis	54
Terminating an analysis	54
Post-analysis	54
Deleting results	55
Other types of analysis	55
Summary	55
6 Linear Static Analysis	56
7 Linear 2D Element Analysis	56
Modelling implications	56
Analysis	58
Results	58
Wall element limitations	58
8 P-delta Analysis	59
Modelling implications	59
Analysis	59
Results	60
9 Dynamic Analysis	60
Modal dynamic analysis	60
Modal P-delta analysis	62
Ritz dynamic analysis	63
Ritz P-delta analysis	63

10 Buckling analysis	64
Eigenvalue buckling analysis	64
Non-linear buckling analysis	65
11 Non-linear Analysis	66
Modelling implications of non-linear static analysis	67
Non-linear static analysis of ties and struts	67
Non-linear static analysis using dynamic relaxation	68
Analysis of fabric structures	69
Form-Finding Analysis	70
12 Seismic Analysis	71
Modelling implications	72
Equivalent Static Load	72
Response Spectrum Analysis	72
13 Harmonic Analysis	73
Modelling implications	74
Results	74
14 Linear Time History Analysis	74
Modelling implications	75
Results	75
15 Periodic Load Analysis	75
Modelling implications	75
Results	76
16 Footfall Induced Vibration Analysis	76
Modelling implications	77
Results	77
17 Raft & Piled-raft Analysis	78
Data requirements	78
Solution method	79
Raft analysis steps	79
Piled-raft analysis steps	80
Results	82
Notes on raft analysis	82
18 Bridge Analysis	83
Modelling implications	86
Delete Grid Loading Tool	87
Analysis of bridge structures	87
19 Analysis Stages	87
Modelling implications	88
Analysis and Design Layers	88
Analysis of stages	90
Results	90
Stages and Graphic Views	91
20 Analysis Envelopes	91
21 Environmental Impact	92
22 Steel Design	93
Modelling implications	93
Steel Restraint Properties	95
Member Restraints	96
Tools	97
Results	97

23 Concrete Design	97
RC Slab Reinforcement Design	98
24 Wave loading	98
25 Model Stability Analysis	102
26 Assemblies	103
Part III Working with the Program	106
1 Some Basic Concepts	106
Preferences	106
Toolbars	107
File formats	107
Using the New Model and Data Generation Wizards	108
Cases	108
2 Working with the Gateway	108
Right-click menus	109
Tables Tab	109
Output Tab	109
Views Tab	109
Copy and Paste	110
3 Working with the Object Viewer	110
Right-click menu	111
Properties Tab	111
Report Tab	112
4 Working with Table Views	112
Single and multi-page tables	112
Tabular data entry and editing	113
Basic operations in tables	114
Selecting blocks of cells in tables	115
Copying, cutting and pasting in tables	115
Delete, blank and insert in tables	116
Find, replace, go to and modify in tables	116
Copying to and from spreadsheets	117
Adjusting data display	117
Colour In tables	118
5 Working with Graphic Views	118
Graphic Settings	119
Basic orientation of the image	120
Scaling the image and zooming	121
Advanced orientation of the image	123
Identifying what is to be drawn	125
Current grid	127
Selection sets	128
Polylines in Graphic Views	131
Adornments	132
Shrinking elements	137
Colour In Graphic Views	137
Shading surfaces	138
Translucency	139
Unwrap Graphics	139
Highlighting element edges	140
Highlight coincident nodes	140
Highlight coincident elements	140

Resetting the display	140
Switch layer	141
Right-click menus	141
Graphic Fonts and Styles	141
Animation	142
Printing from Graphic Views	142
Output of the graphic image	142
6 Working with Chart Views	145
Chart Menus	145
Chart Styles	146
7 Working with Output Views	149
Output Settings	150
Output View table format	150
Selecting data to output	152
Case and entity lists	152
Outputting for a selection set of entities	152
Enveloping	152
Data extents	153
Output summary	153
Output By Case, By Property, By Group	153
Output units	153
Formatting of numeric output	154
Output axes	154
Printing from Output Views	154
Interacting with spreadsheets	154
8 Tools	155
Sculpting	155
9 Convert Orientation Node to Angle	173
10 Create Mass from Element/Member Mass	173
11 Undo and Redo	173
Undoing edits	173
Undoing views	174
12 Working with Saved Views and Preferred Views	175
Default View Settings	175
Preferred Views	176
Saved Views	177
Auto Views	177
Units and Numeric Format	177
View Lists	177
Batch Printing and Saving of Views	178
View Management	178
13 Working with Analysis Tasks and Cases	178
Task View	179
Task View Right-click Menu	179
Tasks, Cases and the Analysis Wizard	181
Copy and Paste Tasks and Cases	181
14 Working with Grid Planes and Grid Loading	182
Specifying Grid Planes	183
Specifying Grid Loading	183
Displaying Grid Planes and Grid Loading	184
15 Working with Geometric Entities	184

16 Data Management	184
Data Management Tools	185
Create Storey Grid Planes	185
Create Grid Planes from Storeys	185
Create Rigid Membranes from Storeys	186
Import GWA data	186
Comparing models	186
17 Miscellaneous	186
Unlock file	186
File backups	186
Delete results from files	187
Edit text file	187
User Modules	187
Evaluating expressions	188
Touch Gestures	189
Part IV Program Fundamentals	191
1 User Preferences	191
Advanced features	192
2 Units	192
Unit set	193
Preferred Units	193
Units in the model	194
Units in views	194
Units in Sections	194
3 Axes	194
Axis sets	195
Use of axis sets	196
Projected axes	197
Grid axes and the current grid	197
Constraint axes	198
Element and member axes	198
4 Grid Planes	198
Grid Plane Definition	198
Use of Grid Planes	199
5 Element types	199
Beam element	199
Bar type elements	200
Spring type elements	200
Mass elements	200
Link elements	201
Spacer elements	201
Cable elements	201
2D element types	201
6 Element axes	203
Beam element axes	203
Spring axes	204
Grounded spring and mass axes	204
Cable axes	204
Link axes	205
2D element axes	205
7 Beam Sections and Section Database	205

Section types	206
Naming convention for sections	207
Design section information	213
Section Database	213
8 Spring Supports and Ground Springs	213
9 Sets and Lists	213
Overview of sets and lists	213
Lists and embedded lists	214
List syntax	214
10 Use of Constraints	216
Restraints and generalised restraints	217
Settlements	218
Element offsets	218
Link elements and rigid constraints	218
Joints	219
Constraint equations	219
Tied Interfaces	219
Conflicting constraints	220
Automatic constraints	220
11 Applying Load	220
Node Loads	221
Applied Displacements	221
Beam Loading	222
2D Element Loads	225
Grid Loading	227
Gravity Loads	228
Selecting entities for loading	228
Patterned Load	228
12 Cases and Tasks	229
Load cases	229
Analysis cases	229
Analysis tasks	230
Analysis wizard	230
Combination cases	231
Enveloping in GSA	231
Syntax of combination case descriptions	232
13 Mass and Weight	236
14 Assemblies	237
15 Cursor modes in Graphic Views	237
16 Numeric formats	238
17 String IDs	238
Part V Program Data	240
1 Axes	240
2 Nodes	241
3 Elements	242
Elements - Definition	243
Elements - Releases	244
Elements - Offsets	244
4 Members	245

5 Grid Planes	247
6 2D Polylines	247
7 Material Properties	248
Standard materials	248
User-Defined Materials	248
8 Beam Sections	251
Use of section data in standard (GSS) analysis	251
Use of section data in GsRelax analysis	251
Section Properties	251
9 Properties	253
Spring Properties	253
Non-linear Spring Curves	254
Spring Matrices	254
Mass Properties	255
2D Element Properties	255
Link Properties	256
Cable Properties	257
Spacer Properties	257
10 Form-Finding Properties	258
Force Density 1D	259
Soap Film 1D	259
Force Density 2D	259
Soap Film 2D	259
11 Design Properties	260
Steel Beam Design Properties	260
Steel Restraint Properties	261
RC Member Design Properties	261
RC Slab Design Properties	262
RC Member Bar Layouts	263
12 Constraints	263
Generalised Restraints	263
Rigid Constraints	264
Joints	265
Constraint Equations	266
Tied Interfaces	266
13 Nodal Loading	267
Node Loads	267
Applied Displacements	268
Settlements	268
14 Beam Loading	269
Beam Loads	269
Pre-stress Loads	270
Distortion Loads	271
Thermal Loads	272
15 2D Element Loading	272
Face Loads	273
Edge Loads	274
Pre-stress Loads	274
Thermal Loads	275
16 Grid Loading	276
Grid Point Loads	276

Grid Line Loads	277
Grid Area Loads	278
17 Gravity	279
18 Dynamic Response	279
Response Spectra	280
Damping Table	280
Load Curve	281
Dynamic Load Factor	281
Frequency weighting curve	281
19 Raft	282
Pdisp	282
Raft Interaction	282
Pile Interaction	282
20 Bridge	283
Alignments	283
Paths	284
Vehicles	285
21 Bridge Loading	285
Bridge VUDL	285
Node Influence Effects	286
Beam Influence Effects	287
Path Loading	287
Static Bridge Loads	288
Moving Bridge Loads	289
Standard Bridge Loading	290
22 Analysis Stages	296
Stage Definition	296
Analysis Stage Properties	297
23 Cases and Tasks	297
Load Cases	297
Analysis Tasks and Cases	298
Combination Cases	298
24 General Data	299
Lists	299
Case Descriptions	299
Assemblies	299
Part VI Toolbars and Keyboard Accelerators	302
1 Toolbars	302
Assisted Input	302
Standard	303
GSA	303
Data Options	304
Lists	304
Orientation	305
Cursor Mode	305
Graphic Display	306
Sculpt toolbar	306
Command	307
Display Favourites	307
Recorder	308

2 Keyboard Accelerators	308
Part VII Data and Analysis Dialogs and Wizards	313
1 Welcome to GSA	313
2 New Model and Data Generation Wizards	313
New Model Wizard	313
Data Generation Wizard	314
3 Data Definition Dialogs and Wizards	321
Titles	323
General Specification	324
Units Specification	325
Currency Specification	326
Tolerances	326
Design Code Specification	327
Bridge Loading Analysis Specification	328
Environmental Impact Specification	329
Environmental Impact Wizard	330
Axis Definition	330
Modify Axis	330
Current Grid Definition	331
Grid Plane Definition	331
Grid Layout Definition	333
Grid Line Definition	333
Line Definition	334
Area Definition	335
Region Definition	336
2D Polyline Definition	339
Node Definition	339
Node Grid Settings Definition	340
Node Stiffness Definition	340
Node Mesh Attribute Definition	341
Element Wizard	341
Beam Element Releases	342
Element Releases	343
Member Wizard	343
Material Wizard	345
Section Wizard	347
Section Modifiers	352
Section - Multiple Sections	353
Spring Property Wizard	353
Spring Curve Definition	354
Spring Matrix Definition	354
Non-linear Spring Curve Definition	355
Mass Definition	355
2D Property Wizard	355
Link Property Definition	357
Cable Definition	357
Spacer Definition	358
Steel Beam Design Property Definition	358
Steel Restraint Property Definition	359
Member Restraint	360
RC Member Design Properties	360
RC Slab Design Properties	361
RC Member Bar Layouts	365

Generalised Restraint Definition	366
Rigid Constraint Definition	366
Joint Definition	367
Constraint Equation Definition	368
Tied Interfaces	368
Node Loading Definition	369
Beam Load Definition	370
Beam Pre-stress Definition	371
Beam Distortion Definition	371
Beam Thermal Load Definition	372
2D Element Face Load Definition	373
2D Element Edge Load Definition	374
2D Element Pre-stress Load Definition	375
2D Element Thermal Load Definition	375
Grid Loading Definition	376
Gravity Load Definition	377
Response Spectrum Wizard	378
Variable UDL Wizard	388
Path Loading	389
Load Case Titles	389
Combination Cases	390
Case Descriptions	390
Append Analysis Case, Append Combination Case, Append List Case	391
Append Envelope	391
Assemblies	391
4 Analysis Dialogs and Wizards	392
Analysis Case Definition	393
Analysis Wizard	393
Advanced Solver Settings	407
GsRaft Analysis Progress	413
GsRaft Damping Coefficient	414
GsRaft Displacement Residual	414
View List	414
Part VIII Other Dialogs	417
1 Graphic View Dialogs	417
Wizard: Graphic Settings	417
Labels and Display Methods	420
Deformation Settings	426
Contour Settings	427
Diagram Settings	430
Bridge Options	432
Further Options	432
Animation Settings	434
Orientation Settings	434
Graphic Fonts and Styles	435
2 Sculpt Dialogs	436
Create User Axes	436
Modify Nodes	437
Connect 1D Elements / Connect Lines	437
Split 1D Elements / Split Lines	438
Split Quad Elements	439
Refine Quad Elements	440
Split Tri Elements	441

Modify Elements	442
Modify Beam Element Releases	443
Modify Element Releases	443
Modify Element Offsets	443
Modify Members	444
Disconnect Elements	444
Move Entities / Copy Entities	445
Extrude	446
Flex	447
3 Output View Dialogs	449
Wizard: Output Settings	449
Output Wizard: Further Options	451
4 Chart View Dialogs	452
Non-linear Analysis Chart	453
Modal Analysis Details	453
Harmonic Analysis Chart	453
Periodic Load Analysis Chart	454
Linear Time-history Analysis Chart	454
Footfall Analysis Chart	454
Assembly Displacements and Forces	455
Storey Displacements and Forces Chart	456
Forces on 2D Element Cut Chart	456
Cut Section Forces	457
5 Tools	458
Create Members from Elements	459
Create Regions per Grid Plane	459
Create New Model from Deformed Geometry	460
Generate 3D Orthogonal Frame	461
Rationalize Lines	461
Split Warped 2D Elements	461
Check Entity Lists for Completeness	462
Expand Grid Loading	462
Expand Gravity Loading	462
Storey Displacements and Forces	463
Create a New Raft Model	464
Generate Static Vehicle Load	464
Optimise Path Loading	465
Expand Bridge Loading	466
Batch Bridge Analysis	466
Wheel Patch Definition	467
Create CEAP Modal Damping File	467
Effective Mass by Part	467
Equivalent Static / Accidental Torsion Load	468
Spectrum Scaling of Modes	469
Storey Masses	470
Footfall Response Data	470
Manage Data	470
Manage Nodal User Modules / Manage Element User Modules	472
Generate 2D Mesh for Polylines (legacy option)	473
Hollow Slab Definition	474
Create Load Panels on Surface	475
Create Facade List	475
Create Patterned Loads	475
Create Patterned Load Combinations	476

Create Mass from Loads	476
Create Mass from Grid Loads	477
Extract a Sub-model	477
6 Preference Dialogs	479
Initial Setup Wizard	479
Initial Setup: General Settings	479
Preferences	479
Saved Image Settings	486
Preferred Units	486
Preferred Section	487
Preferred Folders	487
Advanced Preferences	488
7 Miscellaneous Dialogs	489
Find	489
Replace	490
Modify	490
Go To	491
Modify Curve	491
Curve Data Selection	491
GWA Import Options	491
CAD Export Options	492
CAD Import Options	493
DXF Export Options (legacy option)	495
DXF Import Options (legacy option)	495
NASTRAN Export Options	496
OpenSees Export Options	496
ADC AdBeam Export	497
AdSec Export	497
Export Member Input Data to CSV	498
1D Element Results	498
Numeric Format	499
Axes	499
History	500
Part IX Output Options	502
1 Model Data	502
2 Load Data	506
3 Results	507
4 Bridge Data	512
5 Bridge Results	513
6 Analysis Stage Data	513
7 User Modules	513
8 Analysis Diagnostics	514
Error norm	514
9 Results	514
Static analysis results	515
Modal analysis results	515
Displacements	516
Reactions	516
Soil Contact Bearing Pressure	517
Beam element results	517

2D element results	524
Stress averaging in 2D elements	530
Part X Programming and Command Line Interface	533
1 COM Automation	533
COM Export Functions	534
COM VBA Example	557
COM C++ Example	557
Early and Late Binding	558
2 Command File	559
Command File Commands	559
Command File Example	562
Part XI Interaction with Other Software	564
1 Export options	564
Input data	564
GSA 6.x	565
LS-DYNA	565
SAP2000	566
NASTRAN	567
ANSYS ASAS	568
OpenSees	568
AutoCAD	568
CIMsteel	569
Pdisp	570
ADC	570
AdSec	570
Steel Member(s)	571
Footfall Response	571
Text output	571
2 Import options	571
Text files	572
AutoCAD	572
CIMsteel	573
Pdisp	573
3 Graphic View Save Options	573
4 Links With 3D Modelling Packages	574
Revit	574
Part XII Gss Theory	588
1 Active Degrees of Freedom	588
Degree of Freedom Ordering	589
Active Degrees of Freedom for Elements	589
Restraints	590
Slave Degrees of Freedom	590
Degrees of Freedom with no Local Stiffness	590
2 Analysis Options	591
Static & Static P-delta	591
Modal & Modal P-delta Dynamics	592
Ritz Analysis	592
Buckling	594
Long Term Analysis	594

Model Stability Analysis	594
3 Applied Displacements	595
4 Axes	596
5 Condition number	596
6 Constraints	597
Joints	598
Rigid constraints	598
Constraint equations	599
Tied Interface	599
7 Creep Analysis	600
8 Direction cosines	601
9 Dynamic and Mass Results	601
Dynamic Results	601
Axis transformations	602
Mass Details	603
10 Eigensolver	604
11 Elements	605
Topology	606
Local axes	606
Bar elements	607
Beam elements	608
Spring elements	610
Mass elements	611
2D elements	612
Link elements	619
Releases	620
Offsets	621
Transformations	622
Mass Options	622
12 Error Norms	622
13 Forces in 2D Elements	623
14 Ill Conditioning	624
15 Loads on 2D elements	625
Face and edge loads	626
Thermal Loads	626
In-plane loads	627
16 Loads on beams	627
Fully fixed beams	627
Gravity Loads	629
Pin ended beams	629
Projected loads	630
17 Mass distribution	631
18 Material models	631
19 Matrix Solver	634
20 Nodal Stresses, Forces and Moments	636
Direct extrapolation	636
Least squares extrapolation	637
21 Non-linear Analysis	638

22 Reactions	638
23 Shape functions	639
24 Stress and strain	639
Stress definitions	640
Stress in 2D elements	640
Strain definitions	641
Stress-strain relationships	642
Part XIII GsRelax Theory	648
1 Dynamic Relaxation: Solution method used by GsRelax	648
Damping	648
Solution Process	650
Fictitious masses and inertia used in dynamic relaxation analysis	650
Definition of Control Parameters: residuals	651
2 Element Behaviour	652
Beams	652
Fabric	656
Spacers and Sliding Cables	657
3 Non-linear behaviour and buckling	664
P-Delta effects, Eigenvalue buckling and GsRelax	664
Individual Member Buckling	665
Part XIV GsRelax Analysis	667
1 Static Non-linear Analysis Options	667
Single increment	667
Automatic load increment	668
Individual Member Buckling	669
Form-finding options	669
2 GsRelax analysis cases	671
3 GsRelax convergence and damping	671
Monitoring the iterative Process	672
Interrupting analysis	672
Optimising convergence	672
Termination of Analysis	674
Part XV Seismic Analysis	677
1 Equivalent Static Procedures	677
2 Response Spectrum Analysis	678
Part XVI Dynamic Response Analysis	683
1 Harmonic Analysis	683
2 Periodic Load Analysis	685
3 Linear Time History Analysis	686
4 Footfall Analysis	687
Part XVII GsRaft Analysis	693
Part XVIII GSBridge Theory	697

1 Automatic UK/HK Path Generation	697
2 Automatic EC1 Path Generation	697
3 EC1 - UK Carriageway Path load optimisation	697
4 Influence Analysis	697
Lane-by-lane Path load optimisation	698
UK/HK Carriageway Path load optimisation	698
Part XIX Steel Member Design	700
1 Steel Member Restraints	700
2 Steel Restraint Properties	702
3 Steel Member Section Axes	702
4 Member Results	702
Utilisation ratios	702
Flags	703
5 Steel Checks to BS5950-1:2000	704
Input Data Requirements	704
Local Checks	704
Buckling Checks	705
Input Data	706
Limitations	707
6 Steel Checks to EC3	707
Input Data Requirement	707
Local Checks	707
Buckling Checks	708
Input Data	708
National Annexes	708
Limitations	711
7 Steel Checks to AS4100:1998	711
Input Data Requirement	712
Effective Section Properties	712
Local Checks	712
Buckling Checks	713
8 Steel Checks to HKSUOS:05	713
Input Data Requirements	713
Effective Section Properties	713
Local Checks	714
Buckling Checks	714
9 Steel Checks to SANS 10162-1:2011	715
Input Data Requirements	715
Effective Section Properties	715
Local Checks	715
Buckling Checks	716
Limitations	716
10 Steel Checks to AISC360:05	716
11 Steel Checks to AISC LRFD	717
12 Steel Checks to IS800	717
Part XX RC Slab Theory	720

1 Introduction	720
2 Data requirements	720
3 Other symbols used in this theory	721
4 The RCSlab sign convention	723
5 RCSlab analysis procedure	723
Summary	723
Inclusion of moments resulting from minimum eccentricity	724
Division into layers	724
Applied stresses to each layer	724
Calculation of stress to be taken by reinforcement in a layer	725
Calculation of forces to be taken by reinforcement	727
Calculation of reinforcement strains	728
Determination of stress in reinforcement	734
Calculation of area of reinforcement	734
Non-iterative technique	734
6 Appendix 1	735
7 Appendix 2	738
8 Appendix 3	739
Part XXI Bibliography	745
Part XXII Technical Notes	748
1 Calculation of Torsion Constant	748
Introduction	748
Saint Venant's Approximation	749
Rectangular Sections	749
Other sections	750
Approximating the torsion constant	752
Factored Values in Concrete	753
2 Calculation of Shear Areas	753
Thin-Walled Sections	753
Solid Sections	755
Formulae	756
Stress Factors	757
Reference	757
3 Non-symmetric Beam Sections	757
4 Torce Lines	758
5 2D element shape checks	760
6 Hourglassing	761
7 Equivalent Static & Accidental Torsion Loading	761
8 Grid loading	763
One-way spanning	764
Two-way spanning	765
Multi-way spanning	765
9 Patterned Load Analysis	773
Part XXIII GSA Text (ASCII) File	776
1 Keywords	777

Part XXIV Miscellaneous Text Files	779
1 Curve File	779
Unit types	779
Example	781
2 Tab and Comma Separated Files	781
Part XXV Glossary	783

Part I

1 About GSA

[Overview](#)

[GSA Analysis Features](#)

[GSA Design Features](#)

[GSA Program Features](#)

[Validation](#)

1.1 Overview

GSA has developed from a program for the static analysis of three-dimensional structures composed of skeletal elements, to become a complete analysis package with connection to spreadsheet, CAD and design programs. GSA has been designed from the outset to be easy for the occasional user, yet powerful enough to cope with large, complex models. It incorporates options for graphical and tabular input of data and presentation of results. These allow input data to be entered and edited conveniently and accurately, errors in the data to be diagnosed, validity of the analysis to be established and performance of the structure to be assessed.

1.2 GSA Analysis Features

GSA 8 offers a wide range of structural analysis features:

- Static and dynamic* analysis
- Linear and non-linear* analysis
- Static analysis of vehicle loads*
- P-delta analysis*
- Eigenvalue and non-linear buckling analysis*
- Form-finding and lightweight structure analysis*
- 2D skeletal frame and grid analysis
- 3D skeletal frame and shell* structural analysis
- Seismic response analysis*
- Harmonic analysis
- Linear time history analysis
- Footfall induced vibration analysis
- Periodic load analysis
- Raft analysis and piled-raft analysis*
- Non-linear (tie and strut) elements
- Analysis in construction stages
- Two dimensional and axisymmetric analysis*
- Large capacity
- Efficient and stable solution algorithms
- Analysis of applied load, applied displacements and settlements

- Wizards to guide users in setting up data and analyses
- Export and import of data to and from other analysis programs

*(Program features marked * are optional and may not be available on all installations.)*

Additional features include:

- Sets of nodes may be constrained to act either identically in specified directions, or to act as a rigid body in specified planes.
- Loads may be applied on nodes, on elements or at locations in space. In addition to forces and moment applied to the structure, load types include gravity, pre-stress, thermal and distortion loads.
- Restraints and constraints may act in non-global, user defined directions.
- Displacements and forces are calculated at automatically determined intermediate points along beam and bar elements.
- Beam sections may be selected from a steel section database, which includes British, European, US and Australian sections. Other options are to specify the dimensions of a general section (rectangular, circular, I-section, etc) or to specify the properties required for analysis explicitly.

1.3 GSA Design Features

GSA 8 offers a range of design features:

- Design of steel members with I, H, CHS, RHS and EA sections. Tees and Channel sections are supported from GSA 8.6 onwards.
- Design of concrete members (beams and columns)
- Design of concrete slabs and walls
- Export of CIMsteel data

Steel member design features include:

- Results are available in various forms, including numeric utilization ratios and calculations as rich text.
- The different forms of result available mean that the user can avoid information overload, but can still audit the calculations at a very basic level when appropriate.
- Utilization ratios can be plotted (contoured) on the model just like any other numeric result, so enabling rapid understanding of which parts of a structure are overstressed or are overdesigned.
- Calculation results are marked up in colour to draw attention to problems and important results within the calculations. They are available in both brief and verbose formats.
- Restraints can be specified in detail so that the program can evaluate the effective buckling lengths of complex multi-span beams. Alternatively, the user can specify the effective lengths (as is common in other steelwork design packages). Assistance is available to set up restraints, which are necessarily more complex than restraints used in analysis models.

Concrete member design features include:

- Output of areas of steel
- Output of moment : ultimate moment ratios.
- Export of sections to [Oasys AdSec](#) for more details analysis

Concrete slab and wall design features include:

- Output of reinforcement areas

- All forces and moments are considered in calculating reinforcement areas, i.e. in-plane forces in both directions, in-plane shear force, out-of-plane bending moments in both direction and twisting moment.
- No limit for the shape of the slabs and walls, i.e. not limited the slabs/walls to be rectangle, they can be any shapes, e.g. triangle and polygon etc
- The reinforcements in the two directions are allowed to be in any angle, i.e. not limited to be right angle

1.4 GSA Program Features

Units

GSA [allows](#) the user to work in any units. Several standard sets of units are provided to cover most situations; these may be customised if required. Different units can be used in different views on the data and output is annotated accordingly.

Data Input

For regular structures a Data Generation wizard will allow the user to specify the structure in terms of few key parameters and automatically generate the model data. For more complex structures structural geometry and loading data may be input through either the graphical data generator ("Sculpt") or a more traditional spreadsheet format.

Sculpt enables all structural geometry to be rapidly generated and edited on the screen using a cursor controlled by a mouse. Dynamic viewing makes it possible to adjust the model as required. When the structure is complete, all loading may be similarly generated and displayed diagrammatically. Structures of complexity varying from simple to large skeletal frame models can be generated and complex 2D element meshes can be generated using the automatic mesh generation facilities.

The alternative spreadsheet method incorporates many useful features for the generation and copying of numerical data. Text input is normally specified by selection from a list of available items rather than by entering numeric codes.

The two data generation methods are linked so that they can be used side by side.

Analysis

When an analysis is requested the data are checked for inter-module consistency. If errors are detected, a report is prepared and the program returns to the data editing options; if not, the program proceeds with the analysis with progress being reported to the screen throughout.

Design

Steel Design results are not stored. Instead, they are calculated as required to produce the diagrams, contours or text output requested by the user, using the input data and analysis results to perform the calculations.

Concrete Design is handled in two ways in. For beam design the user can export a sub-frame model to AdBeam for design. For slab design the areas of steel are calculated as required to graphical or text output requested by the user, using the input data and analysis results to perform the calculations.

All types of Design results are only available if the corresponding analysis results are available.

Output

Output can be in the form of tables of results, or graphical views showing diagrams or contours.

The units used for output may be specified interactively as standard or non-standard and default to those used for input.

The axes directions in which results are given may be global, local (nodal or element) or user-

defined. User-defined axes may be Cartesian or cylindrical.

The format of numeric output may be specified in terms of either significant figures or decimal places.

Graphics

The graphics facility in GSA is designed to feel spontaneous in response to the commands of the user. Various single key commands have the effect of producing standard views, adjusting the orientation, scale and content of the drawn structure, windowing, zooming, labelling and annotating the display. Dynamic viewing and animation of results provide powerful tools to aid the engineer in understanding the behaviour of the structure.

Hard copy may be produced on a printer or plotter. Graphical output can be sent to a printer or in DXF format for exporting directly to CAD systems.

Tabular Output

A wide choice of results may be displayed on the screen or printed.

Output results available include displacements, forces, beam stresses, reactions, and analysis details. Tabular output can contain summaries of maximum and minimum values present in the selected output.

Links to Spreadsheets

Data from tables can be transferred to and from spreadsheets using the standard cut and paste facilities. The GWA file (ASCII data file) is written as a tab-separated file to facilitate input into spreadsheet programs.

1.5 Design Codes

Steel codes

The table below shows the current or most recent steel design codes that are available in GSA.

GSA Listing	Design Code Title	Publisher	Date of Publication	Notes
AISC360-10	Specification for Structural Steel Buildings	American Institute of Steel Construction	2010	<i>Design checks of I, H, RHS, CHS, Tee, Channel and Angle sections (strut and tie action only for angles) including local capacity and member buckling checks.</i>
AS4100-1998		Standards Australia	1998	<i>Design checks of I, H, RHS, CHS, Tee, Channel and Angle sections including local capacity and member buckling checks.</i>
CSA S16-09	Design of steel structures	CSA	2009	
BS5950-1:2000	Structural use of steelwork in building.	British Standards	2000	<i>Design checks of I, H, RHS, CHS, Tee,</i>

	Code of practice for design - Rolled and welded sections	Institution		<i>Channel and Angle sections (strut and tie action only for angles) including local capacity and member buckling checks.</i>
EN 1993-1-1 + 2005 Eurocode3	Eurocode 3: Design of steel structures	European Committee for Standardisation	2005	<i>Design checks of I, H, RHS, CHS, Tee, Channel and Angle sections (strut and tie action only for angles) including local capacity and member buckling checks.</i> <i>National Annexe included for FR, NL, UK, SS.</i>
EN 1993-1-2:2005 Eurocode 3	Eurocode 3: Design of steel structures - Fire	European Committee for Standardisation	2005	<i>National Annexe for UK</i>
HKSUOS-11	Code of Practice for the Structural Use of Steel	Building Authority Hong Kong	2011	<i>Design checks of I, H, RHS, CHS, Tee, Channel and Angle sections including local capacity and member buckling checks.</i>
IS800	General Construction in Steel — Code of Practice	Bureau of Indian Standards	2007	<i>Design checks of I, H, RHS, CHS, Tee, Channel and Angle sections (strut and tie action only for angles) including local capacity and member buckling checks</i>
SANS 10162-1:2011	The structural use of steel. Part 1: Limit-states design of hot-rolled steelwork	South African Bureau of Standards	2011	<i>Design checks of I, H, RHS, CHS, Tee, Channel and Angle sections including local capacity and member buckling checks.</i>
StressCheck -	-	-	-	<i>A simple check of axial stress and “web” shear stress in a member due to axial and bending stress. Consideration of the limitations of this approach should be made before using this checker: the standard</i>

of input data required is similar to that needed for other more robust checking methods such as the BS5950-1:2000 checker.

Legacy steel codes

The table below shows the legacy steel design codes that are available in GSA.

GSA Listing	Design Code Title	Publisher	Date of Publication	Notes
AISC360-05	Specification for Structural Steel Buildings	American Institute of Steel Construction	2005	<i>Design checks of I, H, RHS, CHS, Tee, Channel and Angle sections (strut and tie action only for angles) including local capacity and member buckling checks.</i>
AISC LRFD		American Institute of Steel Construction		<i>Design checks of I, H, RHS, Tee, Channel and CHS including local capacity and member buckling checks (non-sway frame only).</i>
AS4100-1998		Standards Australia	1998	<i>Design checks of I, H, RHS, CHS, Tee, Channel and Angle sections including local capacity and member buckling checks.</i>
BS5950-1:2000	Structural use of steelwork in building. Code of practice for design - Rolled and welded sections	British Standards Institution	2000	<i>Design checks of I, H, RHS, CHS, Tee, Channel and Angle sections (strut and tie action only for angles) including local capacity and member buckling checks.</i>
HKSUOS-05	Code of Practice for the Structural Use of Steel	Building Authority Hong Kong	2005	<i>Design checks of I, H, RHS, CHS, Tee, Channel and Angle sections including local capacity and member buckling checks.</i>
IS800	General Construction	Bureau of	1984	<i>Design checks of I, H,</i>

in Steel — Code of
Practice

Indian
Standards

*RHS, CHS, Tee,
Channel and Angle
sections (strut and tie
action only for angles)
including local capacity
and member buckling
checks*

Concrete

The table below shows the current or most recent concrete design codes that are available in GSA.

GSA Listing	Design Code Title	Publisher	Date of Publication
ACI318-02	Building Code Requirements for Structural Concrete (ACI318-02)	American Concrete Institute	2002
ACI318-05	Building Code Requirements for Structural Concrete (ACI318-05)	American Concrete Institute	2005
ACI318-08	Building Code Requirements for Structural Concrete (ACI318-08)	American Concrete Institute	2008
ACI318-11	Building Code Requirements for Structural Concrete (ACI318-08)	American Concrete Institute	2011
ACI318M-02	Building Code Requirements for Structural Concrete (ACI318M-02) (metric version)	American Concrete Institute	2002
ACI318M-05	Building Code Requirements for Structural Concrete (ACI318M-05) (metric version)	American Concrete Institute	2005
ACI318M-08	Building Code Requirements for Structural Concrete (ACI318M-08) (metric version)	American Concrete Institute	2008
ACI318M-11	Building Code Requirements for Structural Concrete (ACI318M-11) (metric version)	American Concrete Institute	2011

AS3600-2009	Australian Standard Concrete Structures 2009	Standards Australia	2001
BS5400-4 1990	Steel, concrete and composite bridges – Code of practice for design of concrete bridges	British Standards Institution	1990
BS8110-1:1997	Structural Use of Concrete Part 1. Code of practice for design and construction	British Standards Institution	1997 (Incorporating Amendments Nos. 1, 2 and 3)
CSA A23.3-04	Design of Concrete Structures	CSA	2004/07
EN1992-1-1: 2004 Eurocode 2	Eurocode 2: Design of concrete structures – Part 1-1: General rules and rules for buildings	European Committee for Standardisation	2004
Hong Kong Code of Practice – 2013	Code of Practice for the Structural Use of Concrete	Building Authority Hong Kong	2103
IRS Concrete Bridge Code	Code of Practice for Plain, Reinforced & Prestressed Concrete for General Bridge Construction	Indian Railway Standard	1997
IS 456:2000	Plain and reinforced concrete – Code of Practice	Bureau of Indian Standard	2000

Legacy concrete codes

The table below shows the legacy concrete design codes that are available in GSA.

GSA Listing	Design Code Title	Publisher	Date of Publication
ACI318-02	Building Code Requirements for Structural Concrete (ACI318-02)	American Concrete Institute	2002
ACI318-05	Building Code Requirements for Structural Concrete (ACI318-05)	American Concrete Institute	2005
ACI318-08	Building Code Requirements for Structural Concrete	American Concrete Institute	2008

(ACI318-08)			
ACI318M-02	Building Code Requirements for Structural Concrete (ACI318M-02) (metric version)	American Concrete Institute	2002
ACI318M-05	Building Code Requirements for Structural Concrete (ACI318M-05) (metric version)	American Concrete Institute	2005
ACI318M-08	Building Code Requirements for Structural Concrete (ACI318M-08) (metric version)	American Concrete Institute	2008
AS3600-2001	Australian Standard Concrete Structures 2001	Standards Australia	2001
BS5400-4 1990	Steel, concrete and composite bridges – Code of practice for design of concrete bridges	British Standards Institution	1990
BS8110-1:1985	Structural Use of Concrete Part 1. Code of practice for design and construction	British Standards Institution	1985
BS8110-1:1997	Structural Use of Concrete Part 1. Code of practice for design and construction	British Standards Institution	1997 (Incorporating Amendments Nos. 1 and 2)
Hong Kong Code of Practice – 1987	Code of Practice for the Structural Use of Concrete	Building Authority Hong Kong	1987
Hong Kong Code of Practice – 2004	Code of Practice for the Structural Use of Concrete	Building Authority Hong Kong	2004
Hong Kong Code of Practice – 2007	Code of Practice for the Structural Use of Concrete	Building Authority Hong Kong	2007

1.6 Validation

Before embarking on any analysis, the user should be aware of its limitations. The user should check the validity of the results. Computer models should be as straightforward as possible to minimise the risk of error, and designed to model the effects required by the user, e.g. a model to assess the stiffness of a structure would not necessarily be the same as one to check the strength. It is often good practice to start with a simple model and increase the complexity if the need arises.

Oasys Ltd implements a Quality Management System (QMS) that is certified to ISO 9001 and TickIT. The certificate of approval for the Oasys QMS is published on the Oasys web site. Oasys products are designed, developed and tested in accordance with procedures set out in this QMS. This process results in the completion of a Test Certificate for each version of a program. The Test Certificate for the current version of GSA is published on the GSA Downloads page on the Oasys web site. This provides evidence that the program has been tested in accordance with the Oasys QMS and therefore with ISO 9001.

1.7 Acknowledgements

The GSA Sparse solver option uses the Approximate Minimum Degree algorithm and LDL Sparse used by permission from the University of Florida, Department of Computers, Information Science and Engineering.

AMD V1.1:

AMD Version 1.1 (Jan. 21, 2004), Copyright (c) 2004 by Timothy A. Davis, Patrick R. Amestoy, and Iain S. Duff. All Rights Reserved.

Your use or distribution of AMD or any modified version of AMD implies that you agree to this License.

THIS MATERIAL IS PROVIDED AS IS, WITH ABSOLUTELY NO WARRANTY EXPRESSED OR IMPLIED. ANY USE IS AT YOUR OWN RISK.

Permission is hereby granted to use or copy this program, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses AMD or any modified version of AMD code must cite the Copyright, this License, the Availability note, and "Used by permission". Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. This software was developed with support from the National Science Foundation, and is provided to you free of charge.

<http://www.cise.ufl.edu/research/sparse/amd/>

LDL V1.1:

LDL Version 1.1 (Apr. 22, 2005), Copyright (c) 2003-2005 by Timothy A. Davis. All Rights Reserved.

Your use or distribution of LDL or any modified version of LDL implies that you agree to this License.

THIS MATERIAL IS PROVIDED AS IS, WITH ABSOLUTELY NO WARRANTY EXPRESSED OR IMPLIED. ANY USE IS AT YOUR OWN RISK.

Permission is hereby granted to use or copy this program, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses LDL or any modified version of LDL code must cite the Copyright, this License, the Availability note, and "Used by permission". Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. This software was developed with support from the National Science Foundation, and is provided to you free of charge.

<http://www.cise.ufl.edu/research/sparse/ldl/>

Part II

2 Step By Step Guide

This chapter gives step-by-step instructions on how to use GSA for its various applications. The [first section](#) gives a very basic introduction to the program by example of a static skeletal analysis. “[Constructing a model](#)” then gives an overview of the various methods that may be used for setting up a model for any application in GSA and “[Requesting Analysis](#)” covers the production and management of results.

The remaining sections deal with particular applications and assume an understanding of how to go about doing static skeletal analysis in GSA. They describe the extra steps that are required to achieve results in the particular application.

This chapter does not deal with the details of the data specification or program operation but does make frequent reference to following chapters that do.

[Becoming familiar with GSA — An Example](#)

[Constructing a GSA model](#)

[Summary of analysis element types](#)

[Summary of analysis types](#)

[Requesting Analysis](#)

[Linear Static Analysis](#)

[Linear 2D Element Analysis](#)

[P-delta Analysis](#)

[Dynamic Analysis](#)

[Buckling Analysis](#)

[Non-linear Analysis](#)

[Seismic Analysis](#)

[Harmonic Analysis](#)

[Linear Time History Analysis](#)

[Periodic Load Analysis](#)

[Footfall Induced Vibration Analysis](#)

[Raft & Piled-raft Analysis](#)

[Bridge Analysis](#)

[Analysis Stages](#)

[Analysis Envelopes](#)

[Steel Design](#)

[RC Slab Reinforcement Design](#)

[Wave loading](#)

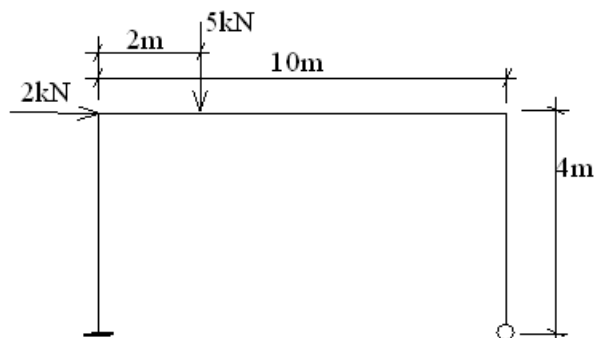
[Model Stability Analysis](#)

2.1 Becoming familiar with GSA - An Example

The purpose of this section is to enable new users to become quickly familiar with the main features, commands and operation of GSA. For the static analysis of a simple portal frame, the data is input and analysed; the results are output in both tabular and graphical form. Users should take time to understand the significance of each operation rather than merely input the commands by rote.

This help topic should be printed out before undertaking the example. The printer should be set to the best graphics quality available e.g. "Fine" rather than "Coarse" for LaserJets 4 and 5. It is inconvenient to keep this help box on the screen while the example is completed.

Consider a simple portal frame as an example of the process of constructing a model. The columns are from UC 203x203x60 and the beam a UB 406x178x54.



The data can be input either in entirely tabular form, or alternatively by a combination of graphical and tabular means. The method chosen for a particular problem is often one of user preference; the resulting data files are identical and can be edited subsequently by either means. For this simple example we will demonstrate both methods.

More:

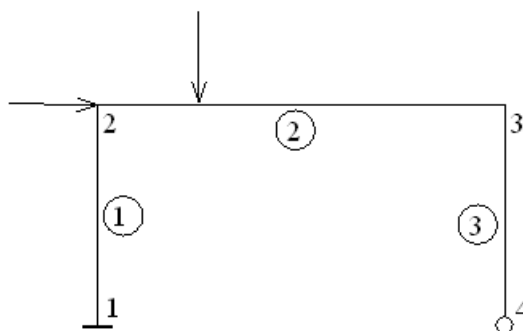
[Tabular Input](#)

[Graphical Input \(Sculpt\)](#)

2.1.1 Tabular Input

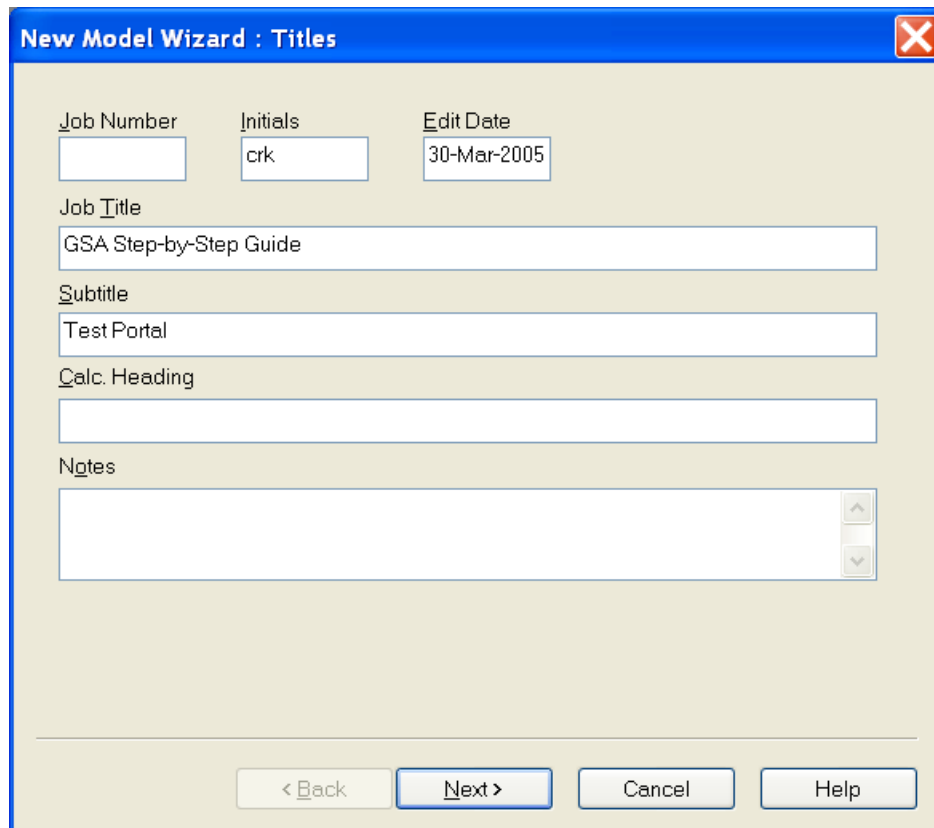
This help topic should be printed out before undertaking the example. The printer should be set to the best graphics quality available e.g. "Fine" rather than "Coarse" for LaserJets 4 and 5. It is inconvenient to keep this help box on the screen while the example is completed.

First, sketch out the GSA model, attributing numbers to the nodes and elements:



When GSA has been installed on your PC as described in the GSA 8 Installation Guide, a blue GSA

icon  will appear in your Start menu. Click on this icon and the first GSA screen will appear; click on the "New file" button, , at the left of the [Standard toolbar](#) (located under "File"). This opens the [New Model Wizard](#) at the [Titles page](#) to allow the job details to be entered.



New Model Wizard : Titles

Job Number:

Initials:

Edit Date:

Job Title:

Subtitle:

Calc. Heading:

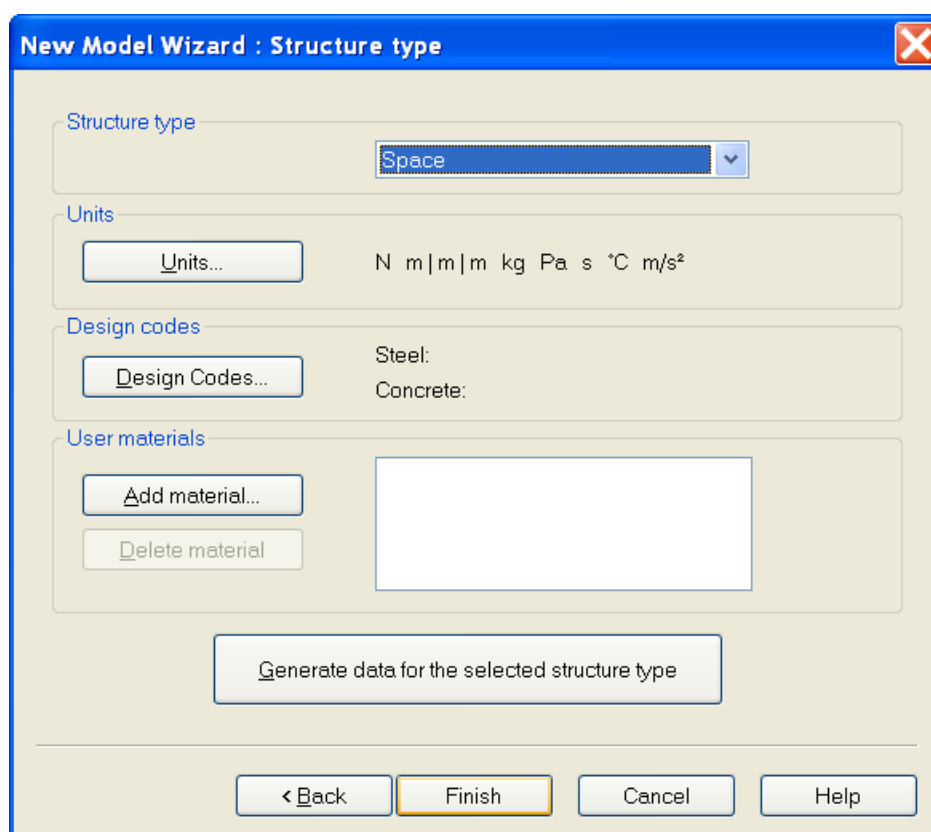
Notes:

< Back Next > Cancel Help

Context-sensitive help is available in most dialog boxes in GSA. If other information is required it can be obtained from the “Help | Contents and Index” menu command for individual topics. Usually the menu commands are not available when a dialog box is open.

Moving from cell to cell may be done by clicking on the cell or Tab and Shift+Tab.

Click on the “Next” button to change to the Structure type page of the New Model Wizard. Clicking “Back” would then move back to the Titles page if subsequent changes to that page were necessary.



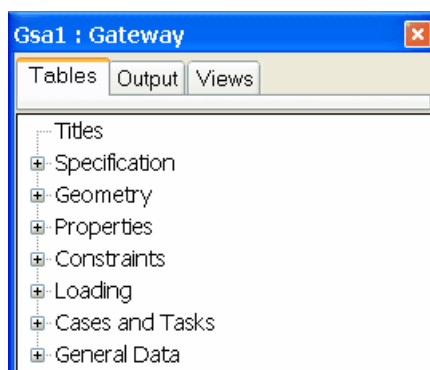
The portal frame is a vertically planar model so set the [Structure Type](#) to Plane.

Click on the “Units” button and change the force units to kN and the small length units to mm; confirm that the large length units are m before clicking OK. The default units are a modifiable [preference](#).

It is from this page that the model could be generated automatically. This time we are entering the data by tabular input so click “Finish”. At this stage the new model is set up and a Graphic View is opened.

The data that has been entered so far may be viewed and edited either from the Data menu commands or from the [Gateway](#) or from the [Assisted input toolbar](#). It is quite typical of Windows programs that there are several ways of doing the same thing. The menu commands are the most comprehensive set of commands but, in practice, they are the least often used. This is because the other methods are provided to give more convenient access. In this instance the Tables page on the Gateway gives quick access to all the data modules via an expandable tree control. The Assisted Input toolbar gives even quicker access to the most frequently used data modules.

Double-click on “Titles” on the Tables page of the Gateway.

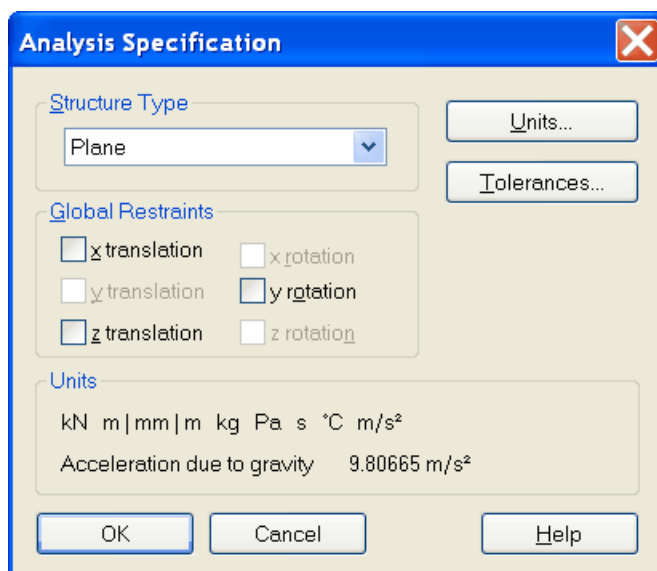


This opens the Titles dialog box displaying the data you entered in the New Model Wizard. Note that in this dialog box it is also possible to paste a graphic bitmap. This may be a useful contribution to the description of the model. Bitmaps may be copied from [Graphic Views](#). Close Titles by clicking the Close window button  at the top right of the Titles dialog box.

The [Assisted Input toolbar](#) is displayed along the bottom of the GSA window. If it is not displayed it may be switched on by giving the “View | Toolbars | Assisted Input” (Alt+F7) menu command.

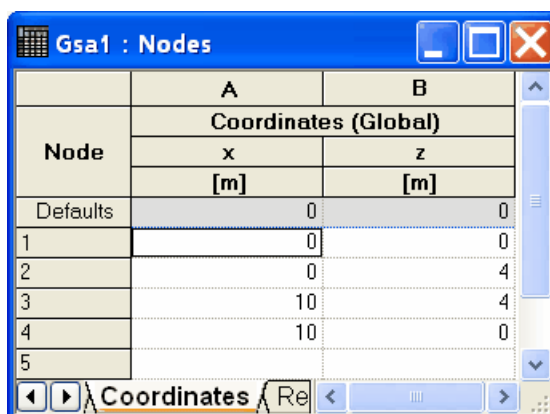


Click the Specification button on the Assisted Input toolbar to open the General Specification dialog box.



The plane structure type is defined in the XZ plane so the Y translational, X and Z rotational degrees of freedom are automatically greyed out. If edits have been made then click “OK”. Otherwise it is a good habit to click “Cancel” to ensure that no unintended changes are made to the model.

Click the **Nodes** button on the Assisted Input toolbar to open the Nodes table; the [Table View](#) in which nodes and nodal attributes are defined. The Nodes table has tabs along the bottom. On entry to the table the “Coordinates” tab will be active, so that the coordinates of four nodes can be defined (e.g. node 1 at (0,0), node 2 at (0,4), node 3 at (10,4) and node 4 at (10,0).



Node	A	B
	Coordinates (Global)	
	x	z
	[m]	[m]
Defaults	0	0
1	0	0
2	0	4
3	10	4
4	10	0
5		

Now click on the Restraints tab to apply the restraints. The restraint condition for node 1 can now be modified to “Encastré” by setting all three directions to “Yes”; similarly for Node 4, the “Pin” cell should be set to “Yes” and confirmed with return. The “Direction” fields will update to reflect these restraint conditions.

	A	B	C	D
Node	Constraint Axis	Restraint Direction		
		x	z	yy
Defaults	Global	~	~	~
1	Global	Yes	Yes	Yes
2	Global	~	~	~
3	Global	~	~	~
4	Global	Yes	Yes	~

Click **Elements** to open the Elements table where details of the elements are defined. On entry to the table the Definition tab should be active. Click on the Type cell for Element 1; it will change to "Beam". The three elements can be defined as Beams, with Property 1 for the verticals and Property 2 for the horizontal member. The Group column can be left as "1" and the Topology (the nodes between which the beam spans) should now be input i.e. element 1 from node 1 to 2, element 2 from 2 to 3 and element 3 from 3 to 4.

	A	B	C	D	E	F
Element	Type	Property	Group	Orientation Angle	Topology	
					1	2
Defaults	Beam	1	1	0	0	0
1	Beam	1	1	0	1	2
2	Beam	2	1	0	2	3
3	Beam	1	1	0	4	3
4						

Click the **Materials** button to give access to the Materials table. For a steel portal we can use the standard material properties for steel, so no action is required.

	A	B	C	D	E	F	G	H	I
Material	Name	Material Model	Material Type	wizard	Young's Modulus [Pa]	Poisson's Ratio	Shear Modulus [Pa]	Density [kg/m³]	Temperature Coeff. [°C]
Defaults	Material #	Elastic Isotropic	N/A		2.05e+011	0.3 E / 2 (1 + nu)		7850	1.2e-005
1	Steel	Elastic Isotropic	Steel	more...	2.05e+011	0.3 E / 2 (1 + nu)		7850	1.2e-005
2	Concrete short term	Elastic Isotropic	Concrete	more...	2.8e+010	0.2 E / 2 (1 + nu)		2400	1e-005
3	Concrete long term	Elastic Isotropic	Concrete	more...	1.4e+010	0.2 E / 2 (1 + nu)		2400	1e-005
4	Aluminium	Elastic Isotropic	Aluminium	more...	7e+010	0.34 E / 2 (1 + nu)		2710	2.3e-005
5	Glass	Elastic Isotropic	Glass	more...	7.03e+010	0.22 E / 2 (1 + nu)		2470	8e-006

Click the **Sections** button to give access to the Beam Sections table where the section properties are defined. In this example we have two section types but both are standard British sections. In the first record enter "Columns" in the "Name" field. Now click the Wizard button, , on the "Data Options" toolbar at the top of the screen to open the [Section Wizard](#). Wizards are available for many of the Table Views to facilitate the entry of a record of data. As demonstrated above it is not always essential to enter the data via the wizard. The wizard is recommended for Beam Sections data entry.

In the Section Wizard the material defaults to "Steel"; set the "Definition method" to "Catalogue". The "Next" button gives a screen, which allows selection of the section. The catalogue is "British", the type is Universal Column and the appropriate section should be selected. The next page allows modification of the standard properties; this can be ignored for this example. "Finish" then returns the user to the Beam Sections table that will have been updated with the relevant section details. This

is then repeated for the beam.

Gsa1 : Beam Sections					
	A	B	C	D	E
Property	Name	Material	Description	Type	Cost
					[GBP/kg]
Defaults	Section #	Steel	Explicit	N/A	
1	Section 1	Steel	CAT UC UC203x203x60	N/A	
2	Section 2	Steel	CAT UB UB406x178x54	N/A	
3					

This now completes the definition of the geometry and properties of the model. The loads can now be applied.

Double-click on “Loading | Nodal Loading | Node Loads” on the Tables page of the Gateway to open the Node Loads table where loads applied directly to the nodes are specified. (The **Gateway** Assisted Input toolbar button brings up the Gateway, if it is not already visible.) In this example, there is only one load case i.e. “1”; at node 2 apply a load in the global X direction of 2kN.

Gsa1 : Nodal Loading					
	A	B	C	D	E
Record	Nodes	Load Case	Axis	Direction	Value
					[kN]/[kNm]
Defaults	none	1	Global	z	0
1	2	1	Global	x	2
2					

Double-click on “Loading | Beam Loading | Beam Loads” on the Tables page of the Gateway to open the Beam Loads table where loads applied to beam and bar elements are specified. On element 2 there is a “Point” load in the “Global” Z direction at a position 2m from end 1 of -5kN. This can also be applied in load case 1. Positions are always measured from “end 1”. Note that “end 1” is as defined in the “Elements” table above. (It is not the “left”, “right”, “top” or “bottom”).

Gsa1 : Beam Loading					
	A	B	C		
Record	Beam List	Load Case	Type	Direction	Position 1
					[m]. [%]
Defaults	none	1	Uniform	Global z	0
1	2	1	Point	Global z	2

The **Graphics** Assisted Input toolbar button opens a [Graphic View](#). If a Graphic View has already been opened then the top-most Graphic View is brought to the top. Several of the commands on graphic related toolbars (typically down the sides of the GSA Window) also serve to open a Graphic View if one is not already open.

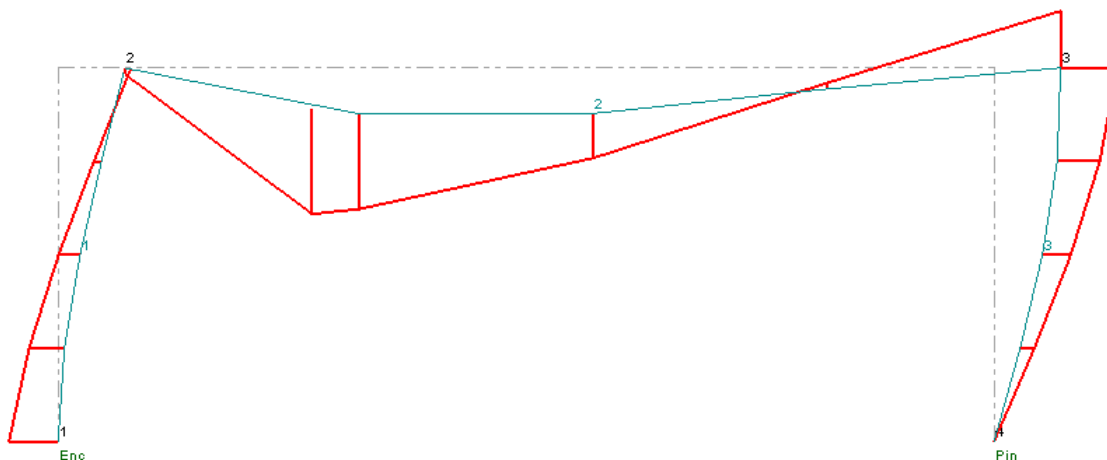
Use the graphic image to visually check the model.

Click the Y-elevation Orientation toolbar button, , to produce a view of the model in the global XZ plane, and the Shrink Graphic Display toolbar button, , to switch shrink off.

Click the Labels and Display Methods Graphic Display toolbar button  to open the [Labels and Display Methods dialog box](#) and enable the node and element number display on the appropriate tabs; confirm with OK. Note that the node number and element number labels can be toggled directly from the [Display Favourites toolbar](#).

The model is now ready for analysis. The simplest method of analysing the model is to use the Analyse GSA toolbar button . Click this and the model is first checked and then a static analysis is performed using the GSS solver. Messages from these processes are sent to the Report View; this may be closed upon completion of the analysis.

To inspect the deformed shape in the Graphic View, click the Deformed image Graphic Display toolbar button . Select the Diagrams settings button . The [Diagram Settings dialog box](#) will appear. Select "Beam and Spring Element Results", then click "Beam and Spring Forces and Moments", "Moment, Myy", "OK". The bending moment diagram will then be drawn on the deflected form. Note that the Myy moments can also be toggled on the Display Favourites toolbar.



The **Output** Assisted Input toolbar button opens an [Output View](#). If one has already been opened then the top-most Output View is brought to the top.

Output Views display tables of input data and results. Clicking the Wizard button  on the "[Data Options](#)" toolbar gives access to the full range of Output View options. Select "Beam and Spring Element Results", then click "Beam and Spring Forces and Moments", "OK" to display these in the Output View.

New Output Views may also be opened from the Output page on the Gateway.

Selective output can be obtained if required by specifying the Cases and Entities (Nodes and Elements) in the Lists toolbar.

The Print [Standard toolbar](#) button  will produce a formatted hard copy of the displayed information.

ARUP Ove Arup & Partners Program Manual Example Getting Started		Job No.	Sheet No.	Rev.
		5279/53		
		Proj. Ref.		
		Model by	Date	Date
		ROB	04-Mar-1988	global.gsa
		Checked		

Beam and Spring Element Forces and Moments

The force in an element at any point is the force required to maintain equilibrium if the element is cut at that point and the end 2 part of the element is discarded. Thus: $\rightarrow$ ve axial forces are tensile
 Element axes for springs are as defined by the spring property axis no.
 Output axes: local

Element	Cuse	Pos'n	Px	Py	Pz	Mxx	Myy	Mzz
		[kN]	[kN]	[kN]	[kNm]	[kNm]	[kNm]	[kNm]
1	A1	0.00	-3.679	0	-873.8E-2	0	4.794	0
		1.000	-3.679	0	-873.8E-2	0	1.299	0
2	A1	0.00	-1.126	0	-3.679	0	0	0
		0.300	-1.126	0	-3.679	0	-6.059	0
		0.300	-1.126	0	1.321	0	-6.059	0
		1.000	-1.126	0	1.321	0	4.505	0
3	A1	0.00	-1.321	0	-1.126	0	4.505	0
		1.000	-1.321	0	-1.126	0	4.505	0

Mocima

2	A1	0.00	-1.126	0	-3.679	0	1.299	0
1	A1	0.00	-3.679	0	-873.8E-2	0	4.794	0
2	A1	0.300	-1.126	0	1.321	0	-6.059	0
1	A1	0.00	-3.679	0	-873.8E-2	0	4.794	0
1	A1	0.00	-3.679	0	-873.8E-2	0	4.794	0
1	A1	0.00	-3.679	0	-873.8E-2	0	4.794	0

Mxvima

1	A1	0.00	-3.679	0	-873.8E-2	0	4.794	0
1	A1	0.00	-3.679	0	-873.8E-2	0	4.794	0
2	A1	0.00	-1.126	0	-3.679	0	1.299	0
1	A1	0.00	-3.679	0	-873.8E-2	0	4.794	0
2	A1	0.300	-1.126	0	1.321	0	-6.059	0
1	A1	0.00	-3.679	0	-873.8E-2	0	4.794	0

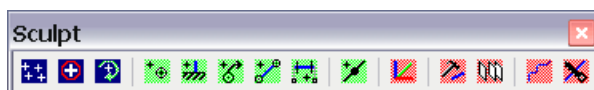
To close the data file, but remain within GSA, click on the Close button  on the [Standard toolbar](#), or select the "File | Close" menu command.

2.1.2 Graphical Input (Sculpt)

This help topic should be printed out before undertaking the example. The printer should be set to the best graphics quality available e.g. "Fine" rather than "Coarse" for LaserJets 4 and 5. It is inconvenient to keep this help box on the screen while the example is completed.

The same portal frame will now be entered graphically using Sculpt commands.

Select  to open a new data file and complete the Titles and general specification menus as before. Close both forms and maximise the graphics area. Click on the Y-elevation button  to produce a view in the global XZ plane.



Move this to a convenient location at the edge of the screen.

Click on the dark blue "Draw grid" toggle button  on left of the array to switch the construction grid, on the Shrink toggle button  to switch shrink off. Click on the label button  and enable the node, restraint and element number display on the appropriate tabs; confirm with OK. Note that these labels can also be switched via the [Display Favourites toolbar](#).

In the [Sculpt toolbar](#), click on the "Snap to grid points" button  at the left and on the green "Sculpt Geometry" icon  to its right; then position the cursor cross [+] on a grid point (node 1: 0,0) in the lower left of the graphics window and click. Holding the Control key down, (which allows a contiguous string of elements to be input), move the cursor four dots upwards and click on that grid point (node 2: 0,4). Note global (X, Y, Z) coordinates displayed in the bottom left of the screen. Repeat this operation for node 3 (10,4) and, releasing the Control key, for node 4 (10,0).

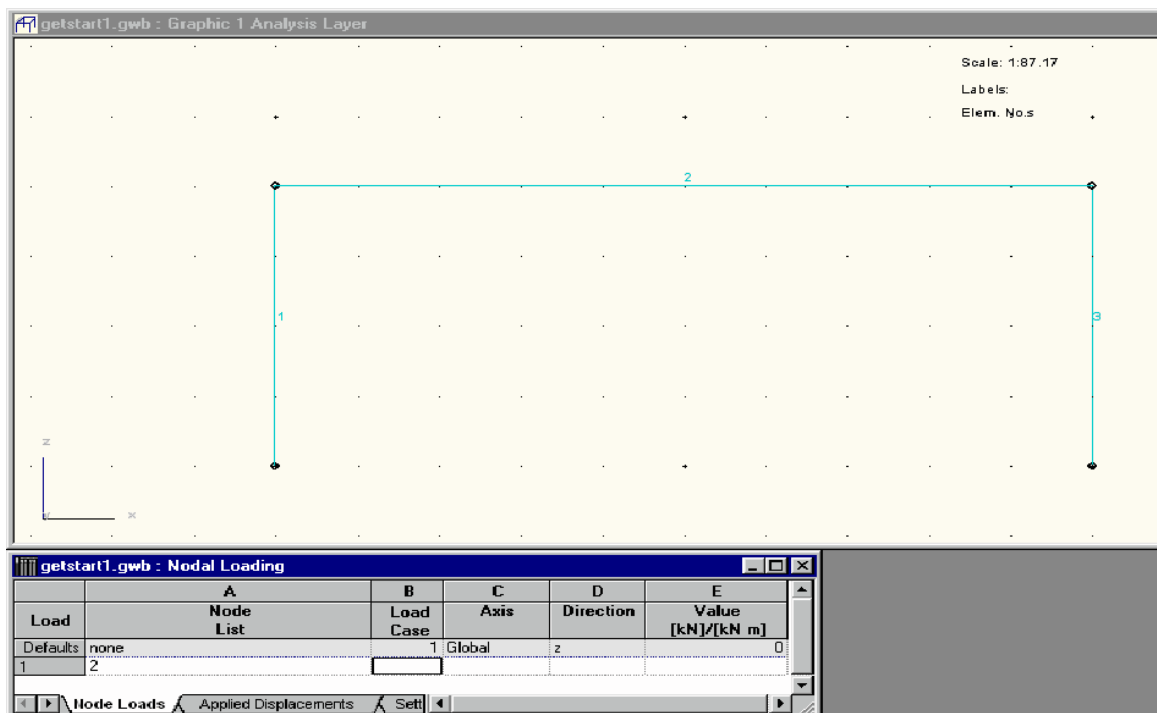
The portal frame should now be drawn. To add the encastré **support** at node 1, click on the green Select Nodes button  to the left of the window, node 1 on the screen, then the Modify Selection button on the Sculpt toolbar . The Modify Nodes menu will appear; click on "Restrain" for x, z, and yy, and confirm OK. Similarly set node 4 to a pin by setting x and z directions to "Restrain" and yy

to “Free”.

There is no explicit Sculpt facility for **loading** input; however nodes or elements can be picked or windowed graphically, and subsequently copied into the node or beam list in the appropriate loading module.

Click on the **Node Loads** button and scale the table with the cursor to fit at the lower edge of the screen, with only the first two lines showing. Click on the title bar of the Graphic View and then on the Select Nodes icon .

Click on node 2 and then copy this node number on to the clipboard by clicking on the Copy icon  at the top of the screen. In the Loads module click on the List cell and then on the Paste icon ; “2” will appear in the cell. Note that lists of several nodes or elements can be copied and pasted using this technique.



The remainder of the line is input manually, i.e. in load case 1, in the global X direction, a load of 2 kN. Close the table.

Return to the Graphics View and similarly input the beam load. Click on the **Beam Loads** button and scale the table to display only the first two data lines only.

Click on the title bar of the Graphic View. Click on the Select Elements  button, then near the middle of the horizontal beam (on which a confirmatory cross will appear), the Copy icon , the Beam List cell, and the Paste icon . A “2” will appear in the cell.

In load case 1, in the global Z direction, input at position 1 of 2m, a point load of –5 kN. Close the module.

The Material and Section data, analysis and post-processing now proceed as for the entirely tabular input data method above.

2.2 Constructing a GSA model

Data in GSA is held in modules that can (in most cases) be thought of as tables. So, for example, a simple model may have a nodes module, an elements module, a beam sections module and some load modules. The data for these tables can be entered directly via Table Views. However it is usually better to use some more convenient (and less error prone) method of entering the data.

This section is concerned with the construction of the geometrical model (nodes, elements and properties) and does not consider the loading.

There is no one correct way of building a model. The most appropriate method will depend on the shape and complexity of the model. Familiarity with other programs, such as spreadsheets and CAD packages will affect the approach, as will the availability of the geometric data in other formats.

More:

[Generating models](#)

[Sculpting models](#)

[Spreadsheets, CAD and Step files](#)

2.2.1 Generating models

The simplest way of constructing a model is to use the “[Data Generation](#)” wizard. Templates are offered for typical structural forms. The structure is specified by selecting a template and defining a few key parameters.

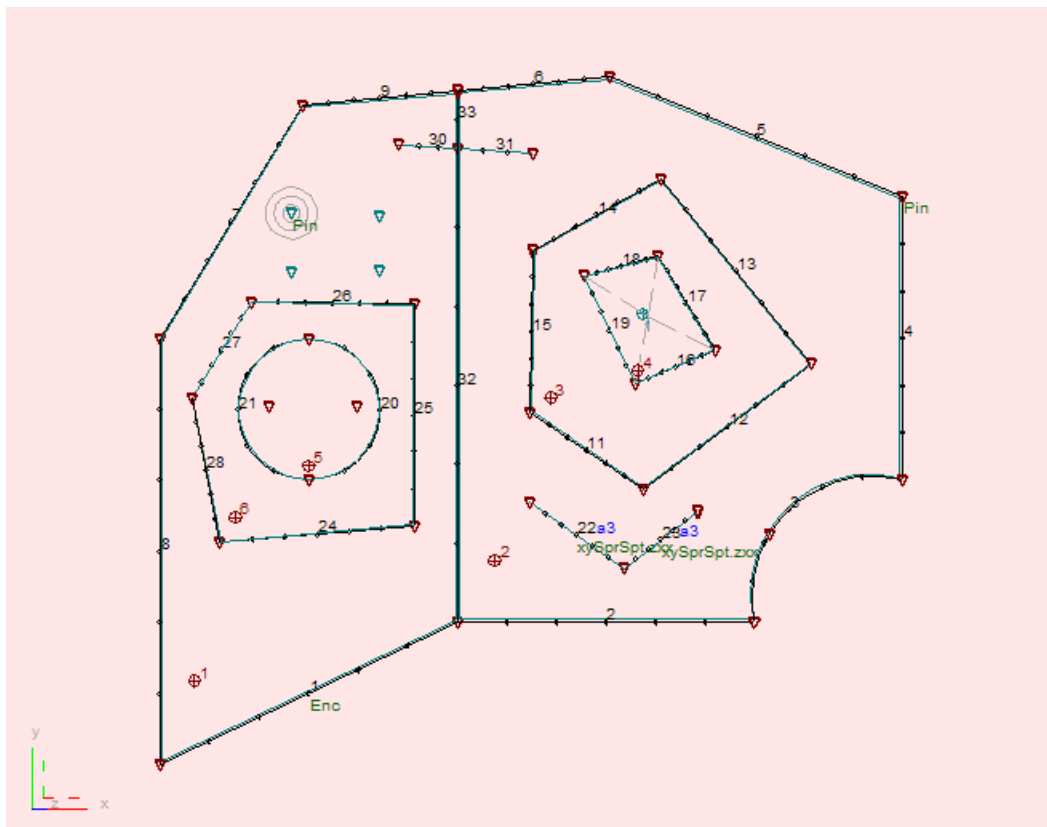
Since this method is so quick and easy it is sometimes useful to generate from a template even if it is known that the resulting geometry will have to be edited to arrive at the desired geometry.

2.2.2 Generating 2D element meshes

2D element mesh generation is based on the region entity. This section describes how to set up a region for mesh generation.

Note that mesh generation may also be carried out using the [Generate 2D Mesh for Polylines](#) legacy option.

Here is an example of a [region](#) that defines a mesh:



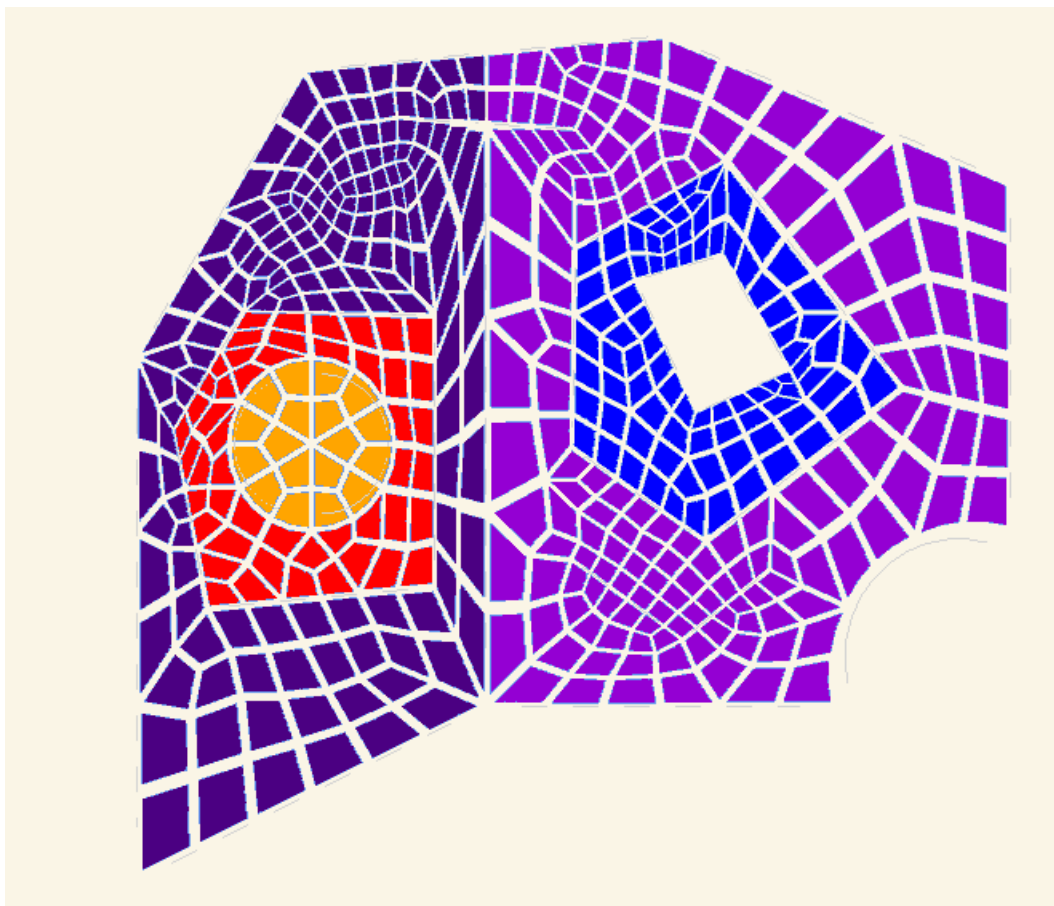
Using this example to illustrate the scope of the mesh generation feature:

- The region is made up of two adjacent areas, [areas](#) 1 and 2. (The area selection points are represented by maroon crossed circles inside the first corner of the area.)
- These areas each have different 2D property references. Generated elements will be assigned the property reference associated with the related area.
- The [lines](#) bounding the areas each have a parameter that defines the number of elements along the line, represented here by dots along the line. The mesh generation engine will be guided by this in determining the number of elements to generate along the line.
- The areas share a common boundary along lines 32 and 33.
- Within area 1 is area 6 and within that, area 5. These areas also have different 2D properties.
- Within area 2 is area 3, with yet another 2D property reference and area 4, which is a void area.
- In area 2 there are also two internal lines, 22 and 23, each with restraints and support stiffnesses assigned. Element edges will be formed along the internal lines and nodes generated along the lines will be assigned the restraint and support stiffness of the related line.
- A line crosses the common boundary but has had to be split at the boundary of areas 1 and 2, as 'internal' lines may attach to an area boundary but may not cross the boundary.
- All nodes used for mesh generation are [grid nodes](#), represented by 'inverted triangle' symbols.
- In area 1 there are four internal grid nodes. The generated mesh will align with and attach to these internal nodes.
- One of these internal grid nodes has an element edge length specified for a specified radius of influence. The mesh generation engine will be guided by these parameters to result in a mesh concentration around the node.
- The complete [region](#) assembly is as follows:
 - Areas: 1 to 6
 - Lines: 22 23 30 31
 - Nodes: 27 to 30
- Only internal nodes and lines need be explicitly referenced by the region. Nodes attached to included lines and lines attached to included areas are implicitly included and, therefore, need not be explicitly referenced.
- (The region selection point is represented by a teal crossed circle at the centre of the largest referenced area.)
- The colours of grid nodes, lines and areas that are included in regions are changed from their native colour to teal to indicate their association with the region.

The region may be checked by giving the 'Tools | Region Mesh Generation | Check All Regions for Mesh Generation' command or the 'Check Region for Mesh Generation' command on the Graphic View right-click menu for the region.

The mesh may be generated for the region by giving the 'Tools | Region Mesh Generation | Generate 2D Mesh for Regions' command or the 'Generate 2D Mesh for Region' command on the Graphic View right-click menu for the region.

Here is the resulting mesh:



Geometry checks are carried out on the generated elements as a post-generation step. If the generation has been requested from a Graphic View then the option is given to select any problem elements.

Elements are attached at common boundaries. The mesh has been generated with nodes at the internal lines and internal node positions.

The generated mesh, i.e. the new nodes and 2D elements created during the mesh generation, are internally associated with the region. Deleting the region results in the associated mesh also being deleting (with prior warning).

The mesh generated for a region may be deleted by giving the 'Tools | Region Mesh Generation | Delete 2D Mesh for Regions' command or the 'Delete 2D Mesh for Region' command on the Graphic View right-click menu for the region.

Note: There is currently no guard against the editing of geometric entities that form the basis of a generated mesh. Care should be taken to avoid doing this.

More:

[Modelling regions for mesh generation](#)

[Modelling tips](#)

2.2.2.1 Modelling regions for mesh generation

A [region](#) is an assembly of [areas](#), [lines](#) and [grid nodes](#). Together these are termed [geometric entities](#).

The mesh generator generates a mesh for a region, so a region may be thought of as a mesh generation 'task'. The mesh generator operates in 2D; GSA prepares the region data for meshing by

first projecting it on to a plane defined by the region grid plane. The pre-generation region data checks warn if the data is not in the region grid plane. A model may contain several regions.

Geometric entities may be created and edited graphically on the Graphic View design layer. The following sculpt commands are relevant:

- [Add lines sculpt tool](#) – adds grid nodes, as necessary, and grid lines. If the added grid line closes one or more line loops then the option is given to create area(s) made of these lines.
- Add Area command – adds a new area made of the currently selected lines.
- Add Region command – adds a new region made of the currently selected areas, lines and nodes.
- [Split Lines](#) command – splits currently selected lines
- [Connect Lines](#) command – connects selected lines at intersection points
- [Create Lines from 1D Elements](#) command – creates lines based on the topologies of either 1D elements or members

Various tools are available for manipulating geometric entities, as follows:

- [Create Regions per Grid Plane](#) – creates lines, areas and regions per grid plane
- [Rationalize Lines](#) – rationalizes all co-linear lines into a string of lines

Generally, new entities are created with [default parameters](#).

Geometric entities may also be imported from [CAD](#) and [3D](#) packages.

2.2.2.2 Modelling tips

Here are some tips on modelling regions for mesh generation:

- Model the region to achieve the desired mesh; this is not necessarily the same as a physical representation.
- Try to avoid placing internal nodes or the ends of internal elements close to area boundaries.
- Model adjacent areas by having the areas reference the same common line(s) rather than different but coincident lines.
- The actual element edge lengths generated along a line will always be such that there is an even number of elements along a line (minimum 2). The specified number will be increased, or the element edge length reduced, so as to achieve this.
- The number of elements generated along a line will be increased automatically as necessary, in even increments, to achieve a feasible mesh and to reduce large discrepancies in the mesh. Note that this *will not happen* if any line has the 'number of segments' defined (as opposed to 'step size') and the region 'steps' parameter is set to 'constant'.
- Consider adding internal 'construction' lines so as to simplify the task presented to the mesh generator. E.g. a square slab with a square void can be simplified by adding lines between each corner of slab and corresponding corner of void.
- If the size of a region is causing difficulties for the mesh generator (in terms of numbers of areas, lines and nodes) then consider splitting the region into several adjoining regions. Connectivity between the regions can be achieved either by setting 'steps' to 'constant' for the regions, or by setting the 'Tie meshes in adjoining regions along this line' line attribute on the common lines.

CM2 MeshTools:

- New lines will have the default number of elements (6) assigned, regardless of line length. You can specify a desired number of elements or element edge length at both the ends of line.
- The region advanced parameter 'Internal element target size' can be used to reduce or increase

the total number of elements generated.

- Do not change the default advanced regional mesh attributes unless you are sure about them.

Quad-Build (legacy option):

- New lines will have the default number of segments (6) assigned, regardless of line length. You can specify a desired number of segment or step size (element edge length).
- The 'ratio between last and first step sizes' parameter is ignored (i.e. assumed to be 1.0) when the region 'steps' parameter is set to 'linear'.

2.2.3 Sculpting models

Sculpting is the term used for graphical editing of the model via a Graphic View. The clear visual feedback makes this method popular for all shapes, sizes and complexities of structure.

The mesh generation option in GSA makes sculpting a powerful tool for the meshing of irregular areas with 2D elements.

Sculpting is covered in more detail in the "Working with the Program" section under "[Working with Graphic Views](#)".

2.2.4 Spreadsheets, CAD and Step files

For some structures there is a regularity which lends itself to definition of the structure in a spreadsheet. In other cases the structure can be defined in terms of a number of parameters that are subject to change, so spreadsheets give the flexibility to quickly generate variations on complex models. Data that is generated in a spreadsheet can either be read as a [text file](#), or included directly in tables using cut and paste.

Structural models that exist in CAD packages may be transferred to GSA via the DXF and DWG file formats (provided the information can be saved as a [DXF file](#) in the CAD package). This is useful both when the geometric form has been defined by others and supplied in a CAD format and when it is felt that CAD is the most appropriate tool for defining the geometry.

Structural models that exist in packages that support the [CIMsteel Step file](#) format may be transferred to GSA via this file format.

2.3 Summary of analysis element types

The table below may aid decisions on which kinds of elements should be used in a model.

Element Type	No of Nodes	Description	Limitations	See also
0-D Element Types				
Grounded Spring	1	Similar to a 2 noded Spring (see below) but one of the nodes is attached to the "ground" and so does not need to be specified.		
Mass	1	A mass element provides the facility to simulate any rigid body's dynamic behaviour	Mass can also be present in a model from the "create mass from loading option" in the	

by the specification analysis wizard, of a mass scalar, and from other elements' self moments of inertia masses. of the body about its three principal axes (whose orientation is specified using any user axis or the global axes).

1-D Element Types

[Spring](#)

2

A spring can have stiffness in one, or three directions specified by one, two or three stiffness parameters depending on the type selected: for example: only one parameter is relevant for an axial spring, whereas three stiffness parameters are required for a general spring. Springs may be translational or rotational. The spring's stiffness direction can refer to any user defined axes or the global axes.

Releases not allowed.

[Grounded spring element](#)

[Bar](#)

2

Models a bar or beam with axial, stiffness only – no bending stiffness.

Releases are not relevant

[Strut element](#), [Tie element](#)

[Beam](#)

2

Models a beam with axial, bending, shear and torsional stiffness: Releases allow the ends of beam elements to be pinned in any of the local three rotation axes.

Warping degrees of freedom are not considered, so lateral torsional buckling modes cannot be considered in the way that Euler strut buckling modes are.

[Bar element](#), [Strut element](#), [Tie element](#)

[Strut](#)

2

Compression-only element. Stiffness depends on section area, Young's

No moment stiffness. Analysis cases involving models containing

[Bar element](#), [Beam element](#)

		modulus and length. these elements cannot be combined, because they are non-linear. Treated as bar element for modal analysis	
Tie	2	Tension-only element. Stiffness depends on section area, Young's modulus and length. these elements cannot be combined, because they are non-linear. Treated as bar element for modal analysis	Bar element , Beam element
Link	2	Use links instead of very stiff elements. They simulate elements with infinite stiffness without causing numerical solver problems. Various types are possible, that are stiff/free in different axes and directions of movement.	Every link has a slave node Rigid constraint
Cable	2	Tension only elements. Connect individual cable elements together into a cable by giving them the same property number.	Tie element , Spacer element
Spacer	2	Spacers don't affect the stiffness of a model: they only control where the nodes end up on the form-found surface. Connect individual spacer elements together into a cable by giving them the same property number.	Only for use with Soap-film form-finding with GsRelax solver. Cable element

2-D Element Types (see [2-D element property types](#) table below)

Quad4	4	Linear shape functions.	2-D element property types

Quad8	8	Parabolic shape functions		2-D element property types
Tri3	3	Linear shape functions	Cannot calculate shear strains accurately due to insufficient degrees of freedom.	2-D element property types
Tri6	6	Parabolic shape functions		2-D element property types

This table shows the available material models for 2-D elements.

2-D Element Property Types	Stiff degrees of freedom per node	Limitations	Comments
Plane stress	2: In-plane translation only. (x,y)	Disregards in-plane moments.	Assumes zero stress normal to element
Plane Strain	2: In-plane translation only. (x,y)	Only allowed in Plane strain structures.	Assumes zero strain normal to element
Axisymmetric	2: In-plane translation only. (x,y)	Only allowed in Axisymmetric structures.	Strain normal to elements is "hoop" strain (proportional to radial movement and distance from centre)
Fabric	2: In-plane translation only. (x,y)	Disregards all moments and out of plane forces.	Warp and weft stiffness as well as in-plane Poisson's ratio can be specified. Normally tension only (non-linear). A non-linear solver (such as GsRelax) is needed to get "out of plane" stiffness effects due to fabric displacement.
Flat plate	3: Out-of-plane translation and rotations only. (z,xx,yy)	Disregards in-plane moments and forces.	Analogous to a grillage model of a floor: you're not interested in axial forces and minor axis bending of the beams.
Flat shell	5: All except rotation about local z axis. (x,y,z,xx,yy)	Disregards in-plane moments.	Element is flat
Curved shell	6: All (x,y,z,xx,yy,zz)	Not currently supported by GSS or GsRelax, but available for export.	Element can be curved out of plane.
Wall element	6. All (x,y,z,xx,yy,zz)	Forces and moments within the element are not available in this version of GSA	Element is flat and must be quad4.
Load panel	none	Linear elements only	For application of load to adjacent beam elements

2.4 Summary of analysis types

<u>Solver</u>	<u>Solver option</u>	<u>Description</u>	<u>Limitations</u>	<u>See also</u>
<u>Gss</u>	<u>Static</u>	Linear analysis (except where tie, strut or cable elements are used). Result cases can be combined post-analysis	Ignores non-linear effects except for strut and ties.	<u>Gss Static P-delta</u> , <u>GsRelax Non-linear static</u>
	<u>Modal dynamic</u>	Calculates dynamic natural frequencies and mode shapes using eigen-analysis. You choose how many modes are calculated.	Linear behaviour only. Struts and ties are modelled as bars.	<u>Gss Modal dynamic P-delta</u> and <u>Gss Ritz analysis</u>
	<u>Ritz analysis</u>	Calculates approximate natural frequencies and mode shapes using force-dependent Ritz vectors.	Linear behaviour only. Struts and ties are treated as bars.	<u>Gss Modal dynamic</u>
	<u>Buckling</u>	Calculates buckling mode shapes and load factors using eigen analysis.	Struts, ties and cable are treated as bars. The internal geometric stiffness effects on wall elements is not included. Does not consider buckling modes involving warping.	<u>GsRelax Member Buckling</u>
	<u>Static P-delta</u>	As static option, but stiffness is modified using geometric stiffness from the same or another analysis case.	Geometric stiffness limited to beam and bar elements.	<u>Gss Static</u>
	<u>Modal dynamic P-delta</u>	As modal dynamic, but stiffness is modified using geometric stiffness from a defined loading.	Linear behaviour only. Struts and ties are treated as bars.	<u>Gss Modal dynamic</u>
	<u>Ritz P-delta</u>	As Ritz, but stiffness is modified using geometric stiffness from a defined loading.	Linear behaviour only. Struts and ties are treated as bars.	<u>Gss Modal dynamic</u>
	<u>Mass</u>	Adds up total structure mass and mass by node.		

GsRelax	Static Non-linear static	Applies load and iterates until nodal forces are in an equilibrium state.	Won't find all possible results (e.g. snap through buckling may lead to more than one valid solution). Slower than GSS. Doesn't always converge.	Gss Static , Gss Static P-delta
	Form finding	For use with fabric and cable element models. Analogous to hanging chain model.		
	Individual member buckling analysis	Allows determination of member(s) effective buckling lengths, taking into consideration the restraint afforded by the rest of the structure.		Gss Buckling
GsSpec	Response spectrum analysis. UBC 1994.	Earthquake response calculation based on statistical characterisation of seismic event.		Gss Modal dynamic
	Harmonic analysis	Structure responses at steady state under harmonic loadings		Gss Modal dynamic
	Linear time history analysis	Time history responses of structures subjected to dynamic loads or base acceleration	Linear responses only, i.e. large deformation and material plasticity are ignored	Gss Modal dynamic
	Footfall induced vibration analysis	Dynamic response analysis of structures subjected to the actions of human footfalls		Gss Modal dynamic
GsRaft	Raft and piled-raft analysis	Soil-structure (raft or piled-raft) interaction analysis		Pdisp
GsBridge	Bridge load optimisation.	Automatic path generation. Influence analysis. Lane by lane path optimisation. Carriageway		

[GsEnvelope](#)

Batch envelope generation.

optimisation.

Generates and stores envelope results. An alternative to the traditional “on-the-fly” envelopes.

2.5 Requesting Analysis

Having set up the data the model must be analysed. This section deals with requesting an analysis, deleting results and the part that analysis cases play in this.

See also [Working With Analysis Tasks and Cases](#).

More:[Simple static analysis](#)[Terminating an analysis](#)[Post-analysis](#)[Deleting results](#)[Other types of analysis](#)[Summary](#)

2.5.1 Simple static analysis

In the simplest case the analysis will be a static analysis of each load case. This may be requested by using the “Analysis | Analyse All” menu command. The “Analyse All” command is also available as a button  on the [GSA toolbar](#). If any [analysis tasks](#) have been set up in the [Analysis Wizard](#), they will be analysed instead.

Giving the “Analyse All” command in this way automatically sets up analysis cases for each load case. (A discussion on this subject is given in [Program Fundamentals — Cases and Tasks](#).) A Report View is opened for reporting progress during the analysis. A check is done on the data and then, provided that there are no errors a GSS static analysis is performed.

2.5.2 Terminating an analysis

The analysis may be aborted at any stage during the analysis by using the “Stop” button  on the [GSA toolbar](#) or by closing the Report View.

2.5.3 Post-analysis

After a successful analysis the “Analyse All” command will be disabled and results will be available for viewing in Graphic Views and Output Views. The input data is locked against editing, since any editing of data upon which results depend would invalidate the results.

After a simple static analysis the results for load case 1 are stored in analysis case 1 etc. Since there is this unambiguous, one to one relationship between load and analysis cases in this circumstance GSA does not insist on specific reference to load cases or analysis cases. E.g. “1” may be used to refer to “L1” and “A1”.

2.5.4 Deleting results

The results may be deleted using the “Analysis | Delete All Results” menu command. The “Delete All Results” command is also available as a button  on the [GSA toolbar](#). When the analysis cases have been set up automatically, as described in “Simple static analysis” above, the analysis cases are deleted with the results.

2.5.5 Other types of analysis

Analysis cases will be set up automatically only for simple static analysis. If it is required to analyse compound load cases or analyse using a different method or solver it is necessary to set up analysis cases to describe the required analysis. This is done in the [Analysis Wizard](#) which may be opened using the “Analysis | New Analysis Task” menu command. The “Wizard” button  on the [Data Options toolbar](#) opens the same wizard when the current view is the Analysis Cases Table View.

When analysis cases have been set up (in the Wizard) but not analysed yet, the “Analyse” command is enabled and will analyse all analysis cases currently at “pre-analysis” status.

Using the “Delete All Results” command when analysis cases exist that have been set up via the Analysis Wizard does not automatically delete the analysis cases with the results; however the option is given to delete the analysis cases.

2.5.6 Summary

Analysis cases associate results with analysis method and load cases or mode number etc.

A simple static analysis is a GSS static analysis of each load case.

Analysis cases are set up automatically for a simple static analysis and may also be set up via the [Analysis Wizard](#).

The “Analyse All” and “Delete All Results” commands behave as follows:

Analyse All

If no results exist, no analysis cases exist and some loads exist the “Analyse All” command sets up analysis cases for and invokes a simple static analysis.

If some analysis cases exist that are at pre-analysis status the “Analyse All” command invokes the analysis of these analysis case.

Otherwise the “Analyse” command is disabled.

Delete All Results

The “Delete All Results” command deletes all results.

If the only analysis cases that exist were automatically created by a simple static analysis the “Delete All Results” command will delete all the analysis cases.

If any analysis cases exist that were not automatically created then the “Delete All Results” command prompts whether analysis cases are to be deleted.

There is always an opportunity to cancel the operation before any deleting is actually done.

The “Delete All Results” command is enabled whenever analysis cases exist.

It is evident that the “Analyse” and “Delete All Results” commands may both be enabled in some circumstances.

The Analysis Wizard offers facilities for management of analysis cases.

The “Stop” command or closing the Report View aborts an analysis.

2.6 Linear Static Analysis

For most users the most common use of GSA is for linear static analysis. The [Becoming familiar with GSA example](#) guides a new user through the stages of building and carrying out a linear static analysis. In all cases a knowledge of linear static analysis is a pre-requisite for the more advanced analysis options.

Analysis

A linear static analysis can be carried out using the “Analysis | Analyse All” menu command. (See [Requesting Analysis](#) above.)

Results

The results of a linear static analysis break down into three categories:

- **Global results**—total loads and reactions
- **Nodal results**—e.g. displacements at all nodes and reactions at restrained nodes
- **Element results**—e.g. forces and moments in elements

Details of results available and how they can be viewed are in the [Output Options section](#).

2.7 Linear 2D Element Analysis

Some structures can only be adequately modelled using 2D elements. Examples of these structures are cores (where deformations in the plane of the element dominate) or floor slabs (where deflections normal to the plane of the element dominate). This section gives an introduction to 2D elements for linear analysis. Non-linear analysis using 2D elements is discussed later.

See also the Step By Step Guide section on [Generating 2D element meshes](#).

More:

[Modelling implications of linear 2D element analysis](#)

[Analysis](#)

[Results of linear 2D element analysis](#)

2.7.1 Modelling implications

Modelling using 2D elements is less intuitive than modelling with skeletal elements, and very different results can be obtained for the same problem, by changing mesh size for example.

Structure types

There are a number of different types of 2D analysis available in GSA depending on the structure type set in the “[General Specification](#)”:

- **Space**—allowing 2D elements to be combined with other element types in 3D space.
- **Plane stress**—where a condition of plane stress (in-plane stresses only) is required, for example a shear wall.
- **Plane strain**—where a condition of plane strain (in-plane strains only) is required, for example a tunnel section.
- **Axisymmetric**—where a radial slice of an axisymmetric structure is required, for example a cylindrical tank.

Plane stress, plane strain and axisymmetric analyses are restricted to 2 dimensions (x and y only).

Elements

There are two aspects of 2D elements that need to be considered: the geometrical aspects (element shapes and mesh) and the appropriate properties.

In skeletal analysis the element mesh is defined by the structure, while in 2D analysis the user needs to provide a mesh of 2D elements which will give an adequate representation of the structural behaviour in the region. This will usually be done using the option for [generating 2D element meshes](#).

The type of element chosen will depend on the type of structure and the analysis method that will be used. 2D elements can be either quadrilateral or triangular and linear (nodes at the corners) or parabolic (nodes at corners and at mid-sides). The recommended elements for linear analysis are parabolic quad (Quad8) elements. However it is often more convenient to generate a mesh of Quad4 elements and use the “[Modify elements](#)” command on the [Sculpt toolbar](#) to convert the elements from linear to parabolic before analysis. Internally the elements are mapped onto simpler shapes, so for example, a Quad8 element regardless of its shape is mapped to a square. The further the shape of the element departs from a square the less accurate the elements are likely to be.

The properties of 2D elements are defined in the “[2D Element Properties](#)” table. For plane stress, plane strain and axisymmetric structures no type is required as this is implied by the structure type. For space elements there is a choice of:

- **Plane stress**—an element which has only in-plane stiffness
- **Fabric**—not available for linear analysis
- **Flat plate**—an element which has only out-of-plane stiffness
- **Shell**—an element which has both in-plane and out-of-plane stiffness
- **Curved shell**—not available at present in the solver
- **Wall**—a superelement which has both in-plane and out-of-plane stiffness. Wall element is internally meshed by 4-by-4 Quad8 shell elements, so a single wall element can be used to model a wall between two floors. Note the [wall element limitations](#) when using this element type.

If out-of-plane bending stiffness is to be ignored (e.g. shear wall), plane stress 2D elements can be used, while for a slab the flat plate may be the best choice if the slab is to take no in-plane forces. If both in-plane and out-of-plane actions are to be considered then shell elements should be used.

In principle, wall elements are the same as shell elements, but larger size can be used for wall elements without losing accuracy of analysis results. In this version of GSA, wall elements only have edge force and moment results expressed by equivalent beam elements in primary and secondary (local x and y) directions. Note that [wall element equivalent beam results](#) are reported with respect to element axes so care should be taken to align wall element axes consistently.

It is useful to be able to assign a set of [axes to 2D elements](#) that applies to a whole group of elements irrespective of the individual orientations determined by the topology. This can be done by setting the axis in the property table.

The added mass allows for an extra non-structural mass, such as a screed, to be included when gravity loads are being considered.

Element formulation for linear elements

GSA provides two element formulations for linear elements; Mindlin and MITC. These are the methods GSA uses to construct the internal stiffness of an element. The Mindlin formulation is the original formulation and is kept in GSA for compatibility with previous models. For new models the MITC formulation is recommended and provides better predictive capabilities and stability. It is possible to change the formulation used in [Advanced Solver Settings](#) of each analysis task or set a global preference on the miscellaneous tab in the GSA [Preferences](#).

Loading

Loading can be applied to 2D elements in a similar way to loading on beam elements, on individual or lists of elements. The 2D element loading modules are accessible from the “[2D Element Loading](#)” table. The loading types are:

- **Face Loads**
- **Edge Loads**
- **Pre-stress**
- **Thermal**

As with beam elements gravity loads can be applied to lists of 2D elements.

The loading that can be applied will depend on the type of element, so for example face loads cannot be applied to plane stress elements and edge loads cannot be applied to flat plate elements. In general it is preferable to apply loads to elements than to the nodes around the elements as the nodal forces resulting from 2D loads are far from intuitive.

2.7.2 Analysis

Linear analysis of 2D elements is carried out in the same way as linear analysis of skeletal structures.

2.7.3 Results

The same set of results is available from a 2D element analysis with a number of additional results for forces and moments, and stresses in 2D elements.

Results for 2D analysis are often most easily understood graphically. This can be done by selecting the “Graphics | Display | Setting | Contour Settings” or “Graphics | Display | Settings | Diagram Settings” menu commands or the “Contour settings” or “Diagram settings” buttons on the [Graphic display toolbar](#). Details of results available and how they can be viewed are in the [Output Options section](#).

Unlike analysis with beam elements there is no requirement for continuity of stress and force results across 2D elements so contour discontinuities are to be expected. The user may choose in the contour settings to average the results at the nodes giving smooth contours, but this should not be done until the user is satisfied that the discontinuities are minor, so as not to mask poor results. Large discontinuities indicate that the mesh is too coarse. 2D elements results are discussed in more details in [Interpreting Data and Results — 2D element results](#).

Wall elements have a different result set from other 2D elements. Wall element results are discussed in [Interpreting Data and Results — 2D element results — 2D wall element results](#).

If all results for 2D elements are stored the model can become very large, so the user may select which force and stress results to store. This can be done from the “Tools | Preferences” (Ctrl+F7) menu command; select “Results” and select the “2D Element Results” that are required. This has no effect on existing results and only affects subsequent analyses.

2.7.4 Wall element limitations

Wall elements have the following limitations in the current version of GSA:

- Wall elements must be 4 node element, i.e. Quad4.
- Non-global nodal constraint axes cannot be assigned to nodes connected to wall elements.
- Settlement and applied displacement loads cannot be applied to nodes connected to wall elements.
- 'Forces on 2D element Cut...' is not available for wall elements, but 'Cut Section Forces' and 'assembly forces & moments' are available.
- If a beam element is connected to the two nodes along an edge of a wall element, the displacements at the middle points along this edge are not compatible between beam element and

wall element.

- Only equivalent beam results in the two orthogonal directions of the wall elements are available. (See [2D wall element results](#) for details.)
- RC slab design cannot be done for wall elements as element results within the elements are not available.
- If both nodes of an edge are constrained by the same constraint equation or tied interface, the whole edge should be also constrained by the same constraint equation & tied interface, but this is not the case at this version of GSA, i.e. even though both nodes of an edge are constrained, but the hidden intermediate nodes along the edge are not constrained.
- If wall elements have face, edge, pre-stress or thermal loads, only the equivalent nodal loads at the 4 corner nodes of the wall elements are considered in the analysis.
- Wall elements cannot have nodal releases

2.8 P-delta Analysis

For some structures it is important to be able to take account of the changes in the stiffness of a structure due to the load. When a column is subjected to an increasing compressive axial load its ability to carry transverse load is reduced until the Euler load is reached when it can no longer carry any transverse load.

The aim of a P-delta analysis is to take account of the changes in the stiffness due to the load.

To be able to access the P-delta options go to the “Tools | Preferences” (Ctrl+F7) menu command, select “Advanced Features” and ensure that the “P-delta and buckling analysis” option is enabled.

More:

[Modelling implications of P-delta analysis](#)

[Analysis](#)

[Results of P-delta analysis](#)

2.8.1 Modelling implications

In most cases a P-delta analysis follows on from a linear static analysis, so a model set up for skeletal static analysis is the normal starting point. As P-delta effects are three dimensional, the structure type in the “General Specification” should be set to space.

2.8.2 Analysis

In a P-delta analysis there is a geometric or differential stiffness in addition to the normal structure stiffness. The geometric stiffness is derived from the forces in the structure, so the solution requires two passes. The first pass establishes the forces in the structure allowing the geometric stiffness to be established for the second pass. The Gss solver handles both passes in a single solution procedure.

A P-delta analysis is set up using the Analysis Wizard from the “Analysis | New Analysis Task” menu command. Select the option to create new analysis cases, then select the Gss and Static P-delta options. The next choice is

P-delta analysis with each analysis case defining its own differential stiffness

- P-delta effects for all analysis cases defined by case...

For a true P-delta analysis the loads used to establish the geometric stiffness are then used in the solution in the second pass (the first option above). However there are circumstances where one

load dominates, for example the dead loads in a bridge, so it is possible to base the geometric stiffness on one set of loads and analyse for another set of loads.

Once this is selected the analysis cases are defined in the same way as for a static analysis and finally the user is given a choice to analyse immediately or later.

In linear static analysis, provided the model is properly restrained, the structure should always be stable, so a solution is always possible. In a P-delta analysis this is not necessarily the case. If the axial forces in the element are too high the elements may be unstable so that a solution cannot be found.

If there are wall elements in the model, the p-delta effects within the wall elements is not considered and only the overall p-delta effects of the wall elements to the whole model is considered.

2.8.3 Results

The same set of results is available from a P-delta analysis as for a linear static analysis. Details of results available and how they can be viewed are in the [Output Options section](#).

The effect of the geometric stiffness is to either stiffen the structure, reducing displacements, if the internal forces are predominantly in tension or make the structure more flexible, increasing displacements, if the internal forces are predominantly in compression.

For a true P-delta analysis the results for each analysis case are based on a different stiffness matrix so superposition of results in combination cases should be avoided. Even scaling of analysis case results will be incorrect, as the stiffness depends on the loading.

2.9 Dynamic Analysis

A dynamic analysis is to establish either the dynamic characteristics of a structure or the dynamic response of a structure to a specified excitation.

To be able to access the Dynamic options go to the “Tools | Preferences” (Ctrl+F7) menu command, select “Advanced Features” and ensure that the “Modal dynamic analysis” option is enabled. In addition for Modal P-delta analysis the “P-delta and buckling analysis” option should be enabled and for Response Spectrum analysis the “Dynamic response analysis” option should be enabled.

More:

[Modal dynamic analysis](#)

[Modal P-delta analysis](#)

[Ritz dynamic analysis](#)

[Ritz P-delta analysis](#)

2.9.1 Modal dynamic analysis

A modal analysis is used to determine the dynamic characteristic of a structure. The results of a modal analysis are a set of natural frequencies and the accompanying mode shapes, which represent free vibration of the structure without reference to any loads.

More:

[Modelling implications of modal analysis](#)

[Modal dynamic analysis](#)

[Results of modal dynamic analysis](#)

2.9.1.1 Modelling implications

In most cases a dynamic analysis follows on from a linear static analysis, so a model set up for static analysis is the normal starting point. The dynamic response should normally be considered in three dimensions so generally the structure type in the “General Specification” should be set to space.

Mass definition

In a modal analysis there is a structure mass matrix in addition to structure stiffness matrix. This is assembled from the element mass matrices in a similar way to the stiffness matrix. This requires some additional specification information, which is set when setting up a modal analysis. The user must decide how the mass is to be distributed around the structure and can derive additional mass from loading.

The three mass options allow for the mass of the structure to be accounted for in different ways.

- **Mass lumped at nodes**—the mass of the elements (including mass elements) is lumped at the nodes (inertias are ignored). For a beam element half of the mass is assigned to each node. The resulting mass matrix is the lumped mass matrix.
- **Masses calculated from the element shape function**—the mass and inertia of the element (including the mass element) are accounted for. The resulting mass matrix is the consistent mass matrix.
- **Mass of mass elements only**—the mass of the structural elements is ignored and the user has to specify mass elements to distribute mass around the structure.

In general the first option will be the most useful. If the mass calculated from the element shape function is used it may lead to modes which involve vibration of individual elements rather than the structure as a whole.

The element mass is derived from the element properties and geometry. For beam and bar elements the mass is not the catalogue mass per unit length but is derived from the unmodified section area, element length and material density.

In most cases not all the mass of the structure is accounted for by the mass of the structural elements, but some is represented in the model as loading on the structure. The option to derive mass from loads allows for some loading to be converted to mass for the modal analysis. Loads are vectors, whereas mass is a scalar so the direction allows the appropriate component of the loading to be selected. The converted loads (load component divided by g) are then considered like lumped masses at the nodes.

Non-linear elements

Modal analysis is by definition only applicable to a linear model, so if the model contains non-linear elements (ties, struts and cables) these need to be linearized. This is done by treating these elements as bars (able to take both compression and tension).

2.9.1.2 Modal dynamic analysis

A modal dynamic analysis is set up using the Analysis Wizard from the “Analysis | New Analysis Task” menu command. Select the “Modal dynamic” option. The next page allow the user to specify the task name (a name to associate with this modal analysis), the number of modes and start mode, and the maximum number of iterations. In most cases only the number of modes needs to be changed here. The number of modes that are required will depend on the structure and what is to be done with the modal results. For a simple structure or where all that is required is the frequency of the fundamental mode only a few modes are required, while a large, complex structure may require up to or in excess of 100 modes.

2.9.1.3 Results

The results for a modal analysis are similar to those for a static analysis, but the interpretation of the results is different.

The displacements represent the mode shape, rather than an actual deflected form, and are arbitrarily scaled or normalised. The default normalisation gives a maximum displacement of 1m. The basis of normalisation can be changed in the [GSS Advanced Settings | Eigensolution](#) dialog, both in terms of units and whether based on mode shape or modal mass.

In addition to the modal displacements, forces and reactions there are the dynamic details results with information such as frequency, modal mass and stiffness, participation factors and effective masses. All of these quantities will depend on the displacement units in that they depend on the modal displacements.

Details of results available and how they can be viewed are in the [Output Options section](#).

2.9.2 Modal P-delta analysis

A modal P-delta analysis is similar to a [modal analysis](#), only the deflected shape is the mode shape. The same model (with a few changes) can be used for both. The difference in the case of P-delta analyses is that the stiffness is modified to include geometric stiffness effects.

If non-linear elements (ties, struts and cables) are included in the model

More:

[Modelling implications](#)

[Results](#)

2.9.2.1 Modelling implications

A modal P-delta dynamic analysis is set up using the Analysis Wizard from the “Analysis | New Analysis Task” menu command. Select the “Modal P-delta” option. The next page is similar to the page for a modal analysis without P-delta effects with the difference in this case that a P-delta case should also be defined.

Non-linear elements

Modal analysis is by definition only applicable to a linear model, so if the model contains non-linear elements (ties, struts and cables) these need to be linearized. For a straightforward modal analysis this is done by treating these elements as bars (able to take both compression and tension). In the case of a P-delta analysis the stiffness matrix is modified by the P-delta effects and this modified stiffness is used in the eigensolver. If the non-linear elements are inactive following the P-delta pass they are then excluded from the stiffness matrix for the modal analysis, if they are included they are treated as bars.

2.9.2.2 Results

The same set of results is available from a modal P-delta analysis as for a [linear modal analysis](#).

If the geometric stiffness acts to stiffen the structure the result will be that the natural frequencies are increased, while if the geometric stiffness reduces the stiffness of the structure the result will be that the natural frequencies are lowered.

2.9.3 Ritz dynamic analysis

A Ritz analysis is a way of calculating approximations to the dynamic modes of a structure. In that respect a Ritz analysis is very similar to a modal dynamic analysis.

More:

[Modelling implications](#)

[Results of Ritz analysis](#)

2.9.3.1 Modelling implications

In a Ritz analysis there is a structure mass matrix and structure stiffness matrix, just as with the [modal analysis](#).

A Ritz analysis is set up using the Analysis Wizard from the “Analysis | New Analysis Task” menu command. Select the “Ritz analysis” option. The next page allows the user to specify the task name (a name to associate with this modal analysis), the number of Ritz vectors and the “load” direction, and the maximum number of iterations. The number of Ritz vectors determines the number of approximate modes. The number that are required will depend on the structure and what is to be done with the modal results.

2.9.3.2 Results of Ritz analysis

The results for a Ritz analysis are almost identical to those for a [modal analysis](#). However the mode shapes, represented by the displacements, are in this case only approximations to the true mode shapes. The reliability of the mode shape can be checked by examining the error norm which for an exact calculation of the true mode shape would be zero.

Details of results available and how they can be viewed are in the [Output Options section](#).

2.9.4 Ritz P-delta analysis

A Ritz P-delta analysis is similar to a [Ritz analysis](#), only the deflected shape is the mode shape. The same model (with a few changes) can be used for both. The difference in the case of P-delta analyses is that the stiffness is modified to include geometric stiffness effects.

More:

[Modelling implications](#)

[Results](#)

2.9.4.1 Modelling implications

A Ritz P-delta analysis is set up using the Analysis Wizard from the “Analysis | New Analysis Task” menu command. Select the “Ritz P-delta” option. The next page is similar to the page for a Ritz analysis without P-delta effects with the difference in this case that a P-delta case should also be defined.

2.9.4.2 Results

The same set of results is available from a modal P-delta analysis as for a [Ritz analysis](#).

If the geometric stiffness acts to stiffen the structure the result will be that the natural frequencies are increased, while if the geometric stiffness reduces the stiffness of the structure the result will be that the natural frequencies are lowered.

2.10 Buckling analysis

A buckling analysis is required where it is important to investigate the potential effects of buckling on the structure. These effects can be significant well before a structure actually buckles.

There are two approaches to buckling — eigenvalue buckling and non-linear buckling. Eigenvalue buckling analysis looks at buckling load factors and the corresponding buckling mode shapes, for a structure with a particular load. Non-linear buckling analysis looks at the response of the structure to a particular load as the load is increased and the effects of buckling begin to show. Non-linear buckling cannot, in general, reveal higher modes.

To be able to access the Buckling options go to the “Tools | Preferences” (Ctrl+F7) menu command, select “Advanced Features” and ensure that the “P-delta and buckling analysis” option is enabled for eigenvalue buckling and that both the “P-delta and buckling analysis” and “Non-linear analysis” options are enabled for non-linear buckling.

More:

[Eigenvalue buckling analysis](#)

[Non-linear buckling analysis](#)

2.10.1 Eigenvalue buckling analysis

An eigenvalue buckling analysis looks at the potential for buckling on a structure with a given load. Unlike a [modal dynamic analysis](#), which characterises the structure, a buckling analysis characterises the structure for a given load.

When a column is subjected to an increasing compressive axial load its ability to carry transverse load is reduced until the Euler load is reached when it can no longer carry any transverse load. For a given axial load the ratio of the Euler load to the actual load is the load factor for that mode. If the column is supported at its midpoint then the first mode will be suppressed and higher modes with corresponding higher loads factors are possible. The buckling analysis should reveal all the modes of concern.

More:

[Modelling implications of buckling analysis](#)

[Results for buckling analysis](#)

2.10.1.1 Modelling implications

In most cases a buckling analysis follows on from a [linear static analysis](#), so a model set up for static analysis is the normal starting point. As buckling effects are three dimensional, the structure type in the “General Specification” should be set to space.

Buckling analysis

In a P-delta analysis there is a geometric or differential stiffness in addition to the normal structure stiffness. The geometric stiffness is derived from the forces in the structure, so the solution requires two passes. The first pass establishes the forces in the structure allowing the geometric stiffness to be established for the second pass, which solves for the modes. The Gss solver handles both passes in a single solution procedure.

A buckling analysis is set up using the [Analysis Wizard](#) from the “Analysis | New Analysis Task” menu command. Select the option to create new analysis cases, then select the Gss Buckling options. The next page allows the user to specify the task name (a name to associate with this modal analysis), the number of modes and start mode, and the maximum number of iterations. In most cases only the number of modes needs to be changed here. The number of modes that are required will depend on the structure and what is to be done with the modal results. For a simple structure or where all that is required is the frequency of the fundamental mode only a few modes

are required, while a large, complex structure may require up to or in excess of 100 modes.

Limitations

Eigenvalue buckling only works for linear elements. Tie and strut elements are treated as bars for buckling analysis. In the Gss solver cable elements in general act like tie elements. For a buckling analysis cables are assumed to act in a similar way to bar elements. So for a buckling analysis the geometric stiffness of the tie, strut and cable elements are identical to those for a bar.

If there are wall elements in a model, the buckling within the wall elements is ignored. Only overall buckling of the whole model is considered for wall element model.

Generating an imperfect geometry

It is useful, particularly for non-linear buckling to be able to specify an imperfect geometry. The results of an eigenvalue buckling can be scaled to give an appropriate imperfection. Using the "Tools | Manipulate Model | Create New Model from Deformed Geometry" menu command allows the user to select an analysis case and specify either a scale factor or maximum imperfection to apply to the displacements to be used to update the model geometry.

2.10.1.2 Results

The results for a buckling analysis are similar to those for a [static analysis](#), but the interpretation of the results is different. The displacements represent the mode shape, rather than an actual deflection form, and are arbitrarily scaled to give a maximum displacement of 1m. In addition to the modal displacements, forces and reactions there are the buckling details results with information such as load factor, modal stiffness and geometric stiffness.

Details of results available and how they can be viewed are in the [Output Options section](#).

In general negative eigenvalues mean that buckling cannot occur under the loading as applied. However if the loading is reversed so that tension and compression are reversed in the structure then the load factors would become positive. If the magnitude of the load factors is greater than 10 then the effects of buckling can generally be ignored. But for values between 1 and 10 further checks are required. The load factor should not be considered as a factor of safety against buckling. It is better to think of the effects of buckling being ever present, and it being necessary to make specific allowance for then if the load factor is less than 10.

2.10.2 Non-linear buckling analysis

Non-linear analysis is performed by the GsRelax solver. Note that for some structures buckling is not a sudden phenomenon and can be preceded by a gradual reduction in stiffness as forces increase. The GsRelax solver recalculates element forces and overall model stiffness based on the deformed geometry, therefore "p-delta" effects and the reduction in stiffness associated with the onset of buckling is considered automatically for all GsRelax analyses. In addition, the stiffness of individual beam elements is reduced when they are in compression.

There are a number of GsRelax analysis options that can be used to investigate the behaviour of a structure as it approaches its first mode of buckling. It can be helpful to run the model through the GSS buckling analysis first to get an estimate of the buckling load factor.

The automatic load increment option in non-linear static analysis can be used to search for the collapse / buckling load of a model, starting from a given percentage of imposed loads. Note that: (1) the loads are increased monotonically and the results are valid only for that load pattern; (2) the internal loads, e.g. prestress and thermal loads etc, are applied at the first increment and then kept constant during the analysis.

Individual member buckling analysis can be used to estimate the maximum axial load capacity of one or a chain of beam elements in a model while the total applied loads to the model are kept constant. This option can be used to estimate the effective restraint to the elements offered by the surrounding structure under the given load combination, i.e. the effective length of the elements can be evaluated from the maximum axial load capacity from this analysis.

More:[Modelling implications for non-linear buckling analysis](#)[Results for non-linear buckling analysis](#)**2.10.2.1 Modelling implications**

Beam elements should be used for non-linear buckling investigations. Bars and struts will not buckle and will exhibit no reduction in stiffness as the compression increases.

2.10.2.2 Results

The results for non-linear buckling analyses will include the real deflections and forces from the buckling load. This differs from the [GSS buckling analysis](#), which gives displacements that represent a scaled mode shape.

The individual member buckling analysis will also give load deflection/rotation results for the member under consideration for increasing values of member compression. These are reported in the “Axial Force / Displacement Relationship” Output View option.

The automatic increment analysis will give load factor deflection/rotation results per load increment for the node under consideration. These are reported in the “Load Factor / Displacement Relationship” Output View option.

2.11 Non-linear Analysis

There are several different types of non-linear analysis, but there are two different effects that need to be considered.

- **Geometric non-linearity**—where the loading causes changes in the shape of the structure which must be taken into account in order to get an accurate solution.
- **Material non-linearity**—where the loading causes material to behave in a non-linear manner, typically through yielding.

Different analysis options in GSA allow these effects to be accounted for in different ways.

The simplest geometrically non-linear problem is a [static P-delta analysis](#). This takes account of the fact that displacements in the structure can be either beneficial or detrimental to its performance.

The simplest material non-linearity problem is static analysis with ties and struts. In this case the material can take either no compression or no tension.

More general non-linear static analysis will model both geometrical and material non-linear effects. To be able to access these non-linear options go to the “Tools | Preferences” (Ctrl+F7) menu command, select “Advanced Features” and then choose a non-linear option. This will ensure that additional non-linear data types are made available in the data tables. The options given are the basic [Non-linear static analysis](#), [Fabric analysis](#) and [Form-finding analysis](#). Once a non-linear analysis preference has been set and the model defined it can be analysed with the non-linear analysis solver [GsRelax](#).

The analysis solver for all the general non-linear analysis options is called GsRelax and is based on [Dynamic Relaxation](#). The GsRelax solver takes a completely different approach to the GSS solver. Because of this some data items function differently with the GSS and GsRelax solvers. Other data items are only appropriate for use with a specific solver option. The data affected is mentioned briefly below and is covered in detail in the relevant data definition sections.

More:[Modelling implications of non-linear static analysis](#)[Non-linear static analysis of ties and struts](#)

[Non-linear static analysis using dynamic relaxation](#)

[Analysis of fabric structures](#)

[Form-Finding Analysis](#)

2.11.1 Modelling implications of non-linear static analysis

The principle of superposition is not valid for non-linear behaviour so unlike linear static analysis, results from different analysis cases **must not** be added together. Combined load cases (e.g. 1.4L1 +1.6L6) should be analysed if analysis results are required for this load combination. Nonlinear analysis cases may be used in envelope case, e.g. A1 or A2.

2.11.2 Non-linear static analysis of ties and struts

Analysis of structures with ties and struts is the simplest non-linear option. No special action is required by the user to activate this option as it depends only on the types of elements in the model.

The effect of the geometric stiffness can be included by selecting a static P-delta analysis rather than a static analysis.

More:

[Modelling implications of ties and struts](#)

[Non-linear static analysis](#)

[Results of non-linear static analysis](#)

2.11.2.1 Modelling implications

The following elements cause a solution to be non-linear:

- [Tie](#)—tension only elements that otherwise resemble bars.
- [Strut](#)—compression only elements that otherwise resemble bars.
- [Cables](#)—similar to tie element but the properties are specified differently.

Non-linear behaviour is only appropriate for a static analysis and cannot be incorporated into a modal analysis (dynamics or buckling). In these cases these elements behave as bars.

2.11.2.2 Non-linear static analysis

The analysis of ties and struts uses an iterative solution technique. This means that each load case has to be considered separately. The report view shows the number of iterations required for convergence. If the solution fails to converge the number of iterations and the convergence tolerance can be modified in the “[Advanced](#)” dialog in the [Analysis Wizard](#).

2.11.2.3 Results

The same set of results is available from a non-linear static analysis as for a linear static analysis. However in this case superposition of results is not valid. Details of results available and how they can be viewed are in the [Output Options section](#).

2.11.3 Non-linear static analysis using dynamic relaxation

This is the basic non-linear option in the GsRelax solver. Selecting this option means that the majority of non-linear data and analysis features will be offered.

More:

[Modelling implications of non-linear static analysis](#)

[Non-linear static analysis](#)

[Results for non-linear static analysis](#)

2.11.3.1 Modelling implications

Elements

The following elements function differently with the non-linear analysis dynamic relaxation solver:

- 1D Elements — [Cable](#) elements behave as sliding cable elements.
- 2D Elements — [Quad 4](#) and [Tri 3](#) are the only 2D elements allowed with the non-linear analysis solver and their [properties](#) must be set to Plane Stress, Plane strain, Fabric or shell.

Properties

Materials — To take account of material non-linearity, User Defined Materials with a yield stress can be defined using the [materials wizard](#). The user defined materials are assigned to the section in the selection/numeric material field of the Section table.

Cases

As the principle of superposition is not valid for non-linear behaviour results from different analyses **must not** be added together. Combined load cases (e.g. 1.4L1+1.6L6) should be analysed if analysis results are required for this load combination.

Dynamic data

Although the solver uses a method called dynamic relaxation, the non-linear options cannot be used to study the dynamic behaviour of a structure. Any data (e.g. mass elements) relating to dynamic analysis included in the model will be ignored during GsRelax analysis.

2.11.3.2 Non-linear static analysis

Once the structural model is defined using the data tables and graphics, the analysis is set-up using the [Analysis Wizard](#). Note that the “Analysis | Check data” menu command checks the data without knowing which solution method is intended. Error messages may be generated which are not relevant to GsRelax. Also note that setting an advanced feature preference does not set the analysis. It merely makes a suitable analysis available. The “Non-linear static” solver option should be selected in the Analysis Wizard. (The solver will be reported as GsRelax and the solution option as Static.) The Wizard then prompts for the information to define an analysis. The data is fully checked for GsRelax at the start of the analysis.

Within the wizard an [Analysis Case](#) can be defined or selected, the loading can be applied in one or many automatic steps and the particular non-linear analysis selected. The basic analysis option is “single increment”, i.e. ‘Automatic load increment’ option is not checked.. This will set up an analysis which most closely follows a linear static analysis in that the results are the forces and deflections in the model for imposed loading defined as an analysis case. The loading is all imposed on the model at the start of the analysis. For details of the other analysis options, look in the [GsRelax Analysis section](#).

The GsRelax solver uses an iterative process similar to transient dynamic analysis with time increment. As GsRelax analysis is a static analysis, it does not use real time increment, but unit time increment and dummy masses rather than real masses. As GsRelax uses dynamic analysis to

simulate static analysis, [damping](#) is also used to enable the vibration to come to rest. The results and progress of the analysis can be viewed and [adjusted](#) while the analysis is in progress. The iterative process can be tuned in the analysis wizard, and the format and frequency of reporting analysis progress adjusted.

2.11.3.3 Results

Once the analysis has [converged](#) to a solution, the results are available for display in [Output Views](#) and [Graphic Views](#). During the analysis a report window displays the intermediate results and progress information. This report window also displays the results summary of axial force and load factor for individual member buckling and automatic increment options respectively.

2.11.4 Analysis of fabric structures

Fabric structures rely on deflection and prestress to achieve equilibrium. Also fabric material is orthotropic with distinct warp and weft properties. The non-linear solver [GsRelax](#) offers features to deal with both of these issues.

The first step is to enable 'Fabric Analysis' as an "[Advanced Feature Preference](#)". This will ensure that the additional non-linear data types needed are made available in the data table views. The advanced feature of non-linear analysis will be automatically enabled.

More:

[Modelling implications of analysing fabric structures](#)

[Analysis and results](#)

2.11.4.1 Modelling implications

Proceed in the same way as for a non-linear static analysis using Quad 4 and Tri 3 element types to define fabric elements. The initial geometry of the fabric surface is a function of the stress in the fabric and the support conditions. To define the geometry prior to analysing the fabric structure, a [Form-Finding Analysis](#) can be performed. This is the most likely method of defining a fabric mode, i.e. define a primary fabric model (e.g. within a plane) and then do a form-finding analysis to find the shape of the fabric model prior to the analysis with imposed loads.

Define the [Fabric Material properties](#) in the materials table. The [warp direction](#) will be aligned with the [element x axis](#) of the 2D element. It is worth considering this before starting to input fabric elements.

Then open the [2D element properties](#) table. Set the property type to Fabric. Type the Material property record number to assign the Fabric Material.

To analyse a fabric model, it normally requires that the fabric, bar/tie elements have been pre-stressed, the pre-stress can be defined as prestress loads for 2D and 1D elements. Alternatively the element pre-stress can be generated from a Form-Finding analysis.

Spacer elements are needed for soap-film form-finding analysis to control nodal spacing a certain way. Spacer elements are only active in soap-film form-finding analysis and they will be ignored in any other analyses, so it is not necessary to delete spacer elements before doing any analysis on the model.

2.11.4.2 Analysis and results

Once the structural model is defined, the analysis is set-up using the [Analysis Wizard](#) as described for the basic Non-Linear static analysis. The "Non-linear static" solver option should be selected in the Analysis Wizard. The GSS solver cannot analyse 2D elements with Fabric Properties and is not appropriate for the large deflection behaviour of these structures. The Wizard then prompts for the data to define the analysis.

Fabric analysis is the same as Non-linear analysis except that fabric elements are enabled. Therefore the same analysis options apply.

2.11.5 Form-Finding Analysis

This can find the geometry of an artificial structure that is in equilibrium with user defined internal and external forces and constraints. There are two types of artificial structures available: soap-film structures to model membranes, and force-density structures to model cable nets.

When a form-finding analysis converges, the nodal co-ordinate data is overwritten with the deformed shape, and load cases are created from the equilibrium element distortions. Therefore the form-finding analysis method is suitable for investigating construction sequences of non-linear structures using the option “form-finding analysis – ignoring form-finding properties”.

Before embarking on a form-finding exercise consider the approximate shape and the boundary conditions required. This should include estimating the average radii of curvature of membranes and boundary cables, and identifying fixed high and low points. Use approximate hand calculations to estimate the state of equilibrium pre-stress that will result.

The first step is to select [Form-Finding Analysis](#) as an [Advanced Feature Preference](#). This will ensure that the additional Form-Finding data types are made available in the data tables. The advanced feature “Non-linear Analysis”, will be automatically enabled. If the structure is a membrane select Fabric Analysis as well.

More:

[Modelling Implications of form-finding analysis](#)

[Soap film form-finding](#)

[Force density form-finding](#)

[Form-finding analysis ignoring form-finding properties](#)

2.11.5.1 Modelling Implications

Proceed in the same way as for a non-linear static analysis except that the elements can have [Form-finding Properties](#) assigned as well as, or instead of, their true physical properties. The solver looks for the appropriate form-finding property for each element, and if none is defined it uses the normal property. Therefore the model can include a mixture of stiff “real” elements and form-finding elements.

Spacer elements are used to control the mesh (nodal spacing) in soap-film form-finding analysis. These elements are ignored in all the other analysis options.

2.11.5.2 Soap film form-finding

Membrane geometries are defined using soap-film form-finding. In this method, all or part of the structure is modelled as having zero stiffness by assigning Soap Film Form-Finding Properties. It is useful to visualise this as making 1D elements into elastic bands, and 2D elements a soap film.

Use Quad4 and Tri3 to define fabric elements. The initial geometry of the fabric surface only needs to be an approximation of the final expected geometry. Therefore generating simplified flat grids of 2D Quad4 or Tri3 elements can be a fast method of constructing a model. The edges of these simplified grids can be modified to form the boundaries and supports as required.

During soap film form-finding, the nodes within soap film elements tend to wander and coalesce. [Spacer elements](#) and [Spacer Properties](#) should be input to control the final node positions generated during the form-finding process.

A state of pre-stress needs to be defined in fabric and boundary elements using [Soap Film Properties](#) to give the desired geometry and boundary conditions.

Once the model is defined store the data to a new file name as the results will overwrite the original data. Set up the analysis using the [Analysis Wizard](#) and selecting the “Form finding” solver option. An analysis case is prompted for but external loading is not always needed for form-finding. If no external loading is to be considered a blank entry is acceptable.

Once converged, the deflected shape represents a form that is in equilibrium with the specified soap film forces. The resulting forces in Elements with “real” properties will be in equilibrium with their deflected shape. These forces will differ from any prestress input for “real” elements. The option is given to overwrite the data file with the deformed geometry and to store resulting forces as a specific prestress load case.

2.11.5.3 Force density form-finding

Force density form-finding is a method of form-finding cable networks. The tension in each bar or tie elements in the network is made proportional to its length. The form that is found is that giving the minimum strain energy i.e. the form that minimises the sum of the squares of the lengths of the elements multiplied by the force-density of the elements.

The procedure is the same as for soap film form finding, except that force density properties are used instead of soap-film or real properties. When force density form finding is selected in the Analysis Wizard, the program uses force density properties where these exist.

2.11.5.4 Form-finding analysis ignoring form-finding properties

This simply performs a non-linear analysis with the GsRelax solver using the materials and section properties defined in the data tables (i.e. ignoring soap film or force density properties and using real physical element properties).

The analysis is therefore identical to the static non-linear analysis. The results, however, are reported very differently. The displaced nodal co-ordinates overwrite the original geometry and element distortions and tensile forces are stored in the additional load case. The structure in the deformed shape plus the additional load case is in equilibrium with the total loading applied in the form-finding analysis. So if the deformed model is analysed with the same applied loading as before plus the new load case, negligible further movement should occur. An exception to this occasionally arises where distributed element loading has been used, as the equivalent nodal loads of the element loads will be generated based on the deformed geometry. They are different from the equivalent nodal loads used in form-finding analysis that were generated based on original undeformed geometry.

No “analysis results” will be produced from the form-finding analysis.

2.12 Seismic Analysis

A number of approaches are possible for seismic analysis and the particular approach that is used will depend on the anticipated levels of seismicity and the type of structure. In some cases a relatively simple static analysis may suffice – at the other extreme full non-linear time-history analysis may be required. The seismic options in GSA can be split into two categories

- Equivalent static analysis
- Response spectrum analysis

A number of seismic codes require calculations by storey. GSA can output a breakdown of [mass and inertia by storey](#).

More:

[Modelling implications in Seismic Analysis](#)

[Equivalent Static Load](#)

[Response Spectrum Analysis](#)

2.12.1 Modelling implications

The basic model can be built as any other GSA model but the following points should be considered for seismic analysis.

The vertical direction should be global Z.

Define storeys for the structure. Most codes make reference to storeys at some point and for simple structures they can be defined automatically by the “Tools | Create Storeys” menu command. A storey is defined by a height in the Z direction along with appropriate storey tolerances.

Codes give the situations where floors are allowed to be treated as rigid diaphragms. Where this is the case create rigid constraints (xy-plane type) either graphically or if you have defined storeys using the “Tools | Create Rigid Membranes from Storeys” menu command.

In most cases seismic design will be carried out with reference to a particular design code, GSA supports to a lesser or greater extent ASCE, Eurocode, FEMA, GB (Chinese), IBC, IS (Indian), Ordinanza PCM (Italian) and UBC codes. The seismic data is stored along with the response spectrum data as the two are closely interrelated.

In all seismic analysis the first stage is to determine the levels of ground motion that are to be considered. This can be expressed in different ways and varies from one code to another and will typically vary depending on the type of building. When working to a particular code, the various parameters required will be defined in the code. Values are either supplied by the code or have to be found by reference to external sources. For example the US Geological Survey gives detailed information of S_S and S_1 values for all of the USA.

2.12.2 Equivalent Static Load

This is the simplest approach to seismic analysis and avoids the need for dynamic analysis. However, as earthquakes are dynamic events the equivalent static procedure may not always be appropriate.

At the simplest level an earthquake can be considered as a lateral gravity load which the structure must be able to withstand. In practice the codes specify the type of loading that must be used. Typically this is a load distribution that increases with elevation above ground level. The total applied load is calculated to give a particular base shear as defined in the code.

In addition to the lateral load the codes require consideration of an accidental torsion load. This considers the effect of a torque applied to the structure equal to the equivalent static load being offset by a proportion of the floor width.

More details of equivalent static loads are in the [Seismic Calculation](#) section.

The equivalent static and accidental torsion loads defined in the seismic code can be generated in GSA from the “Tools | Seismic Analysis | Equivalent Static / Accidental Torsion Load” menu command.

2.12.3 Response Spectrum Analysis

A [modal analysis](#) allows the user to investigate the dynamic characteristics of a structure. The next stage is often to determine the response to a particular dynamic excitation. Two approaches are possible, one is to solve for the response of the structure to a particular excitation time history. The other is to solve for the response of the structure to an excitation characterised by a response spectrum.

Only the response spectrum option is available in GSA. This approach is generally useful for seismic analysis. More details of response spectrum analysis are in the [Seismic Calculation](#) section.

A Response Spectra is used to characterise the earthquake. The user can either import a response spectrum or use a code defined spectrum (e.g. UBC, IBC, Eurocode 8).

The basic responses define how the response spectra are to be used in the response in the global directions. The response spectral accelerations, the modes included in the response (normally all), and a combination method (normally complete quadratic combination — CQC).

Seismic events produce excitation in all three directions and different methods can be used to combine the responses in three orthogonal directions. Combination cases can be set up to combine the basic responses using either a square root sum of the squares (SRSS) method or linear (“30% rule”) combination cases.

One aspect of concern for a seismic analysis is the storey drift. This is defined by a list of nodes in geometric order.

The set of results available from a response spectrum analysis is similar to that for a static analysis, however, the user has the option of requesting nodal velocities and accelerations. The displacements in this case are the maximum displacements that might be anticipated for the given response spectrum.

Unlike modal analysis, which gives information about the dynamic response of the structure in isolation, the response spectrum analysis gives information about the dynamic response of the structure to a particular dynamic loading, and consequently, these results can validly be combined with those from a static analysis.

The results of a response spectrum analysis are not a set of results in equilibrium but the maximum values that can be expected. This means that it is not possible to calculate intermediate forces and moments along individual elements post-analysis. Any analysis cases or combination cases that include response spectrum results will only use linear interpolation of results along the element.

Details of results available and how they can be viewed are in the [Output Options section](#).

Warning: any derived results should from a response spectrum case should be used with caution as these will tend to over-estimate the actual effect. So for example in RC Slab in-plane forces and bending moment will both be made positive while design may require bending moments of the opposite sign to the in-plane forces.

See also:

the [bibliography](#)

2.13 Harmonic Analysis

Harmonic analysis of structures is to calculate the maximum elastic response of a structure subjected to harmonic loading at steady state. The structure responses include nodal displacements, velocities and accelerations as well as element forces and moments etc. The harmonic loading is the load that varies sinusoidally along with time. Harmonic analysis is based on modal dynamic analysis results (frequencies, mode shapes & modal masses etc). It calculates the maximum structure responses for the given harmonic loads using modal superposition method.

This analysis utilizes modal dynamic analysis results, so modal dynamic analysis must be conducted before doing this analysis.

More:

[Modelling implications](#)

[Results](#)

2.13.1 Modelling implications

Harmonic analysis utilizes modal dynamic analysis results and the construction of GSA model for harmonic analysis is the same as that for modal dynamic analysis. See [Modeling implications](#) of [Modal dynamic analysis](#) for the requirements of building up a model for harmonic analysis.

The magnitudes and locations of harmonic loads can be defined in the same way of defining a static load. Load combination (e.g. 1.4L1 + 1.6L2) can also be used in harmonic analysis. Harmonic loads vary sinusoidally and its frequency is defined in the 'Harmonic Analysis' page of the analysis wizard.

The results of modal dynamic analysis are essential for harmonic analysis, before doing harmonic analysis, at least one modal dynamic analysis task with analysis results must exist.

See also:

Analysis Wizard : [GsSpec Harmonic Analysis](#)

2.13.2 Results

The results of a harmonic analysis are the same as that from Response Spectrum Analysis, they include nodal displacements, velocities and accelerations and element forces, moments and stresses etc. The results are not a set of results in equilibrium but the maximum values that can be expected in the whole history of the vibration response. This means that it is not possible to calculate meaningful intermediate forces and moments along individual elements. Any analysis cases or combination cases that include harmonic analysis results will only use linear interpolation of results along the element.

The results can be viewed in the same way as those from other analyses. See section [Output Options section](#) for viewing the results.

In addition to the normal node and element results mentioned above, chart views that show the nodal responses (e.g. displacement, velocity and acceleration etc) versus harmonic load frequencies are also available.

See also:

Analysis Wizard : [GsSpec Harmonic Analysis](#)

2.14 Linear Time History Analysis

Linear time history analysis is to calculate the linear responses of structures that are subjected to dynamic loads (force excitation) or base accelerations defined by the combination of the applied loads or base acceleration and [load curve](#). The structure responses include nodal displacements, velocities and accelerations as well as element forces and moments etc at the chosen time intervals. If it is force excitation, the magnitude and locations of the dynamic loads are defined in the same way as that for static analysis and the variation of the dynamic loads along with time is defined by [load curve](#). If it is base acceleration, the maximum base acceleration is defined in the analysis wizard and the variation of the base acceleration is also defined by [load curve](#).

The initial displacements and initial velocities are assumed to be zero in GSA linear time history analysis.

Linear time history analysis utilizes modal dynamic analysis results (frequencies, mode shapes & modal masses etc), so at least one modal dynamic analysis must exist with results before doing linear time history analysis.

In addition to the nodal and element results at the given time intervals, the nodal displacements, velocities and accelerations can also be plotted against time from Chart view output.

More:

[Modelling implications](#)

[Results](#)

2.14.1 Modelling implications

Linear time history analysis utilizes modal dynamic analysis results, so the construction of a GSA model for linear time history analysis is the same as that for modal dynamic analysis. See [Modeling implications](#) of [Modal dynamic analysis](#) for the requirements of building up a model for modal dynamic analysis which also applies to linear time history analysis.

The spatial distribution and magnitude of the dynamic loads are defined in the same way as for defining loads for static analysis. Load combinations (e.g. 1.4L1 + 1.6L2) can also be used in linear time history analysis. The variation of the dynamic loads along with time is defined by load curve in the load curve table.

As linear time history analysis utilizes the results of modal dynamic analysis, at least one modal dynamic analysis task must be analysed before doing linear time history analysis.

See also:

Analysis Wizard : [GsSpec Linear Time history Analysis](#)

2.14.2 Results

The results of a linear time history analysis include nodal displacements, velocities and accelerations and element forces and moments at the chosen time intervals. In addition, the nodal displacements, velocities and accelerations can also be plotted against time on the output of chart views.

See also:

Analysis Wizard : [GsSpec Linear Time History Analysis](#)

2.15 Periodic Load Analysis

Periodic load analysis is to calculate the maximum responses of structures that are subjected to periodic loads. The structure responses include the maximum nodal displacements, velocities and accelerations. The periodic loads are generic and the dynamic load factors of each of the harmonic components of the periodic loads at a given frequency are defined by the users through dynamic load factor table.

Periodic load analysis utilizes modal dynamic analysis results (frequencies, mode shapes & modal masses etc), so at least one modal dynamic analysis must exist with results before doing periodic load analysis.

In addition to the nodal displacements, velocities and accelerations, the nodal displacements, velocities and accelerations can also be plotted against the frequencies of the periodic loads.

More:

[Modelling implications](#)

[Results](#)

2.15.1 Modelling implications

Periodic load analysis utilizes modal dynamic analysis results, so the construction of a GSA model for linear time history analysis is the same as that for modal dynamic analysis. See [Modeling implications](#) of [Modal dynamic analysis](#) for the requirements of building up a model for modal dynamic analysis which also applies to periodic load analysis.

The spatial distribution and magnitude of the periodic loads are defined in the same way as for defining loads for static analysis. Load combinations (e.g. 1.4L1 + 1.6L2) can also be used in periodic load analysis. The dynamic load factors of each of the harmonic components of the periodic loads at a given frequency are defined in the dynamic load factor table.

As periodic load analysis utilizes the results of modal dynamic analysis, at least one modal dynamic analysis task must be analysed before doing periodic load analysis.

See also:

Analysis Wizard : [GsSpec Periodic Load Analysis](#)

2.15.2 Results

The results of a periodic load analysis include the nodal displacements, velocities and accelerations. In addition, the nodal displacements, velocities and accelerations can also be plotted against frequencies of the periodic loads from chart view output.

See also:

Analysis Wizard : [GsSpec Periodic Load Analysis](#)

2.16 Footfall Induced Vibration Analysis

Footfall induced vibration analysis is to evaluate the responses of structures subjected to the actions of human footfalls. The structural responses include nodal accelerations, velocities and response factors etc. The human footfall loads are considered as periodical loads which are represented by a number of harmonic loads according to Fourier series theory. The detailed descriptions of human footfall loads can be found from references 1, 29, 35 and 39 in the [Bibliography](#) which are also listed below. Footfall induced vibration analysis utilizes dynamic analysis results (frequencies, mode shapes & modal masses etc) to calculate the structure responses. The outputs of footfall analysis are the maximum responses of the structure for the given ranges of walking/running frequencies etc. The following four design guides of footfall analysis can be considered:

1. AISC Steel Design Guide Series, Floor Vibration Due to Human Activity, October 2003.
2. Willford, M.R. & Young, P. (2006) A Design Guide for Footfall Induced Vibration of Structures, The Concrete Centre, CCIP-016.
3. Smith, A.L., Hicks, S.J., & Devine, P.J. (2007) Design of Floors for Vibration: A New Approach, The Steel Construction Institute, P354.
4. AISC Steel Design Guide 11, second edition, Vibrations of Steel-Framed Structural Systems Due to Human Activity, May 2016

In addition to use the four design guides, user defined DLF (Dynamic Load Factor) curve can also be used in footfall analysis if you know the Fourier coefficients (DLFs) of each of the harmonic components of the footfall loads.

As footfall analysis utilizes modal dynamic analysis results, modal dynamic analysis results must be available before doing footfall analysis.

Response factors from footfall analysis are calculated using frequency weighting curves (FWC), i.e. the calculated RMS accelerations times the weighting factors from the chosen frequency weighting curve, then divided by 0.005 m/s^2 to get the response factors. Frequency weighting curve can be standard or user defined, there are three standard frequency weighting curves (W_b , W_d and W_g) from BS6841 that can be used directly by footfall analysis. User defined frequency weighting curves can also be used in footfall analysis if it has been defined in Frequency weighting curve table, this makes it possible for GSA footfall analysis to cope with any type of frequency weighting options.

More:

[Modelling implications](#)

[Results](#)

2.16.1 Modelling implications

Footfall induced vibration analysis utilizes modal dynamic analysis results and the construction of a GSA model for footfall analysis is the same as that for modal dynamic analysis. See [Modeling implications](#) of [Modal dynamic analysis](#) for the requirements of setting up a model for Footfall analysis.

The results of modal dynamic analysis are essential for footfall induced vibration analysis, before doing footfall induced vibration analysis, at least one modal dynamic analysis task with analysis results must exist.

See also:

Analysis Wizard : [GsSpec Footfall Induced Vibration Analysis](#)

2.16.2 Results

The results of a footfall induced vibration analysis are nodal results and they include:

- Resonant analysis
 1. Maximum acceleration based response factors
 2. Peak velocity
 3. RMS velocity
 4. Peak acceleration
 5. RMS acceleration
 6. Critical nodes - the excitation node (where walking is taking place) that give maximum response factor for the response node
 7. Critical walking frequencies - the frequency of walking that give the maximum response factor for the response node
- Transient analysis
 1. Maximum velocity based response factors
 2. Peak velocity
 3. RMS velocity
 4. Peak acceleration
 5. RMS acceleration
 6. RMQ Critical nodes
 7. RMQ Critical walking frequencies

These results can be viewed from contours, diagrams and output tables.

After a footfall induced vibration analysis, results can also be shown on chart views, chart views show the plot of Response factor versus Walking frequency, Velocity versus Time etc. Results from chart view are for individual excitation and response nodes.

See also:

Analysis Wizard : [GsSpec Footfall Induced Vibration Analysis](#)

2.17 Raft & Piled-raft Analysis

The raft analysis option analyses vertical soil-structure interaction for raft and both vertical and horizontal soil-structure interaction for piles. It provides a means of linking the GSA static analysis of a structural model (typically a raft or piled-raft) with [Pdisp](#) soil settlement analysis. Pdisp is a program which calculates the soil settlements (and stresses if Boussinesq analysis method is used) under the normal and/or shear pressure loads on vertical and/or horizontal planes within the soil. Pdisp has been embedded in GSA as the soil settlement analysis engine, so standalone Pdisp program is not needed in GSA raft analysis. The data modules in GSA relating to soil properties and analysis can be imported from or exported to the Pdisp program.

To be able to access the raft analysis option first go to the “Tools | Preferences” (Ctrl+F7) menu command, select “Advanced Features” and ensure that the “Raft analysis” option is enabled.

More:

[Data requirements](#)

[Raft & piled-raft solution method](#)

[Raft analysis steps](#)

[Piled-raft analysis steps](#)

[Results of raft & piled-raft analysis](#)

[Notes on raft analysis](#)

2.17.1 Data requirements

The input data for a raft and piled-raft analysis can be divided into three categories:

Raft structure data

The structural model of the raft or piled-raft is modelled in GSA in the usual way as other structure models except that the nodes that will interact with soil should be kept free in vertical direction for the raft nodes and kept free in all three directions for pile nodes.

Vertical support stiffness for the soil interaction nodes may be specified in which case the specified stiffness are used as the initial stiffness for the interaction nodes, if the specified initial support spring stiffness are closer to the real soil stiffness, the analysis speed can be enhanced, but it has no effect on the analysis results. If nodal support stiffness are not defined, default initial nodal support stiffness will be used, the default initial nodal support stiffness can be modified on the Analysis Wizard when setting up the raft & piled-raft analysis.

Raft can be modelled by grillage using beam elements or modelled by 2D elements. For piled-raft model, the piles need to be modelled by a number of beam elements and the beams must be vertical (in global Z direction).

Soil property and soil zone data

These data are the same as those used in the standalone Pdisp program. Refer to Pdisp program user manual which is in GSA installation folder for guidance. The input tables for these data can be opened either from “Data | Raft | Pdisp Data” menu command or from the Gateway using the “Raft | Pdisp Data” option.

The loading to the soil from raft and/or piles is generated automatically during raft or piled-raft analysis and they should not be included in the Pdisp load data table. Only additional loads, i.e. extra to the loads from raft, can be specified in Pdisp load data table.

Soil-Raft interaction data

The Raft Interaction table defines the nodes on the raft to interact with soil. This table is opened from the “Data | Raft | Raft Interaction” menu command or from the Gateway using the “Data | Raft | Raft Interaction” option. The interaction area of the raft interaction nodes can be defined on this table or

choose Automatic to allow GSA to calculate the interaction areas automatically, the automatic interaction area calculation can only be done for the rafts that are modelled by grillage. The elevation of the interaction can also be defined or choose Automatic to take the nodal z coordinate as the interaction elevation. Minimum and maximum soil pressure can also be defined in this table. Zero minimum soil pressure means that soil will not take any tensile stresses. If a negative minimum soil pressure is defined, it will be used as the soil tensile strength. The maximum soil pressure is the compressive strength of soil. If this pressure is reached during the analysis, the soil-raft contact pressure will not increase any more to allow soil local yield effect to be considered.

The Pile Interaction table defines the nodes on the piles to interact with soil. This table is opened from the "Data | Raft | Pile Interaction" menu command or from the Gateway using the "Data | Raft | Pile Interaction" option. The pile-soil interaction areas (dimensions of X, Y, Z & perimeter) of the pile interaction nodes can be defined on this table or choose Automatic to allow GSA to calculate the interaction areas automatically. The same as raft interaction nodes, the elevation of the interaction can also be defined or choose Automatic to take the nodal z coordinate as the interaction elevation.

The graphical display of raft & pile interaction nodes and/or raft & pile interaction areas are available from Graphic View [label](#) options.

2.17.2 Solution method

Raft & piled-raft analysis is an iterative process linking the linear static analysis solver for the structure analysis with the Pdisp analysis solver for the soil settlement analysis. The process iterates until the difference of the nodal displacements and soil settlements for all the interaction nodes are smaller than the specified tolerance.

The first stage in a raft analysis is a linear static analysis. The soil is represented by support springs of the interaction nodes. For raft nodes, only vertical interaction is considered and for pile nodes, the interaction in all three directions will be considered.

After linear static analysis, the support spring forces are known, so the pressures on soil can be calculated using the nodal interaction areas. Then, a Pdisp soil settlement analysis is carried out to obtain soil settlements. From these settlements and the spring forces, new support stiffness can be calculated for the interaction nodes. For pile interaction nodes, the reaction forces from soil will be calculated based on the differences of pile displacements and soil settlements using the data defined in Pile-soil interaction table and PSIC (Pile-Soil Interaction Coefficient) curves. The detailed description on how to calculate soil reactions on pile interaction nodes is given in [GsRaft Analysis](#) section. This iteration continues until the raft and pile displacements and soil settlements are compatible, i.e. differences of soil settlements and raft displacements are smaller than the predefined tolerance.

If maximum pressure of soil has been specified for the raft interaction nodes and the maximum pressure has been reached, the check of the compatibility of raft displacements and soil settlements will be ignored, i.e. the raft displacements and the soil settlements are allowed to be different.

By default, the temporary support springs used during the analysis will be deleted after the analysis is converged or cancelled. However, if "Save soil stiffness" box is checked in the analysis wizard, the support spring stiffness at the last iteration will be kept, therefore the Raft model will be modified.

2.17.3 Raft analysis steps

The following are the main steps required for doing a raft analysis.

1. Build up a raft model, the raft can be modelled by a grillage using beam elements or by a horizontal plate using 2D elements.
2. Define loads on the raft in the same way as for doing other GSA analysis.
3. Open "Raft Analysis Specification" dialog box from Gateway or from menu "Data | Specification | Raft analysis specification..." to define rigid boundary level etc soil settlement analysis parameters

4. Open "Soil Profiles" table from Gateway or from menu "Data | Raft | Pdisp Data | Soil Profile" to define soil properties. One soil profile defines the soil properties in a vertical direction from the top surface of the soil to the rigid boundary level below it. One soil profile may include soil properties of many layers depending on the real soil conditions. There is no limit for the number of soil profiles to be defined and the number of soil profiles depends on the actual site requirements. Soil profiles are then assigned to rectangular areas called soil zones.
5. Open "Soil Zones" table from Gateway or from menu "Data | Raft | Pdisp Data | Soil Zones" to assign soil profiles (properties) to relevant rectangular areas called soil zones. If soil zones are overlapped, the overlapped areas will use the soil-profile defined later in the soil zone table.
6. Open "Raft Interaction" table from Gateway or from menu "Data | Raft | Raft Interaction" to define the nodes on raft to interact with soil. The interaction areas of the raft interaction nodes can be defined on this table or choose Automatic to allow GSA to calculate the interaction areas automatically, the automatic interaction area calculation can only be done for the rafts that are modelled by grillage. The elevation of the interaction can also be defined here or choose Automatic to take the nodal z coordinate as the interaction elevation. Minimum and maximum soil pressure can also be defined in this table. Zero minimum soil pressure means that soil will not take any tensile stresses. If a negative minimum soil pressure is defined, it will be used as the soil tensile strength. The maximum soil pressure is the compressive strength of soil. If this pressure is reached during the analysis, the soil-raft contact pressure will not increase any more to allow soil local yield effect to be considered.
7. Now we are ready to do raft analysis by going through the analysis wizard start from menu "Analysis | New Analysis Task..." in the same way as doing other GSA analysis.

Note:

The data described above are the data always required for doing raft analysis. There are also some other Pdisp data which are not required and necessary in most of the raft analysis. If they are required, they can be defined as explained below:

1. "Non-linear Curve" - If soil non-linearity need to be considered and the soil strain - Young's modulus (or Young's modulus reduction factor) relationship is known, the non-linear curve defining the strain - Young's modulus reduction factor can be defined and then referred to by the relevant soil profile. Once this has been done, the soil Young's modulus will be changing according to the current soil strain so non-linear properties will be considered.
2. "Load Data" - The loads defined in the "Load Data" table are the loads acting on directly on the soil in addition to the loads come from the raft. Normally, these load data are not required as the soil loads from raft are defined as raft loads. The loads on raft will be transferred to soil through the interaction nodes during the analysis. If there are really some extra loads directly on the soil, e.g. construction loads acting on directly to soil, they can be defined here to increase the flexibility of GSA raft analysis.
3. "Displacement Data" - By default, the soil settlements will be given at the soil-raft interaction points. If the soil settlements at other points, lines or grids are interested, they can be defined in this table. If this has been defined, there will be more soil settlement results.
4. "Results" - This is an output view showing soil analysis results and no data input for this view.

See also:

[Notes on raft analysis](#)

Iteration scheme in section [GsRaft Analysis](#)

2.17.4 Piled-raft analysis steps

The following are the main steps required for doing a piled-raft analysis.

1. Build up a piled-raft model, the raft can be modelled by a grillage using beam elements or by a horizontal plate using 2D elements. A pile should be model by a number of vertical beam elements, since pile-soil interaction is defined by the nodes on the pile, the number of beam

elements used to model a pile depends on the length and section sizes of the pile. Normally, 10 beam elements may be required to model a single pile.

2. Define loads on the raft in the same way as for doing other GSA analysis. If there are loads acting on pile directly, they can also be defined.
3. Open "Raft Analysis Specification" dialog box from Gateway or from menu "Data | Specification | Raft analysis specification..." to define rigid boundary level etc soil settlement analysis parameters.
4. Open "Soil Profiles" table from Gateway or from menu "Data | Raft | Pdisp Data | Soil Profile" to define soil properties. One soil profile defines the soil properties in a vertical direction from the top surface of the soil to the rigid boundary level below it. One soil profile may include soil properties of many layers depending on the real soil conditions. There is no limit for the number of soil profiles to be defined and the number of soil profiles depends on the actual site requirements. Soil profiles are then assigned to rectangular areas called soil zones.
5. Open "Soil Zones" table from Gateway or from menu "Data | Raft | Pdisp Data | Soil Zones" to assign soil profiles (properties) to relevant rectangular areas called soil zones. If soil zones are overlapped, the overlapped areas will use the soil-profile defined later in the soil zone table.
6. Open "Pile-soil interaction properties" table from Gateway or from menu "Data | Raft | Pdisp Data | Pile-soil interaction properties" to define the pile-soil interaction properties. The properties include the maximum soil stresses (strengths) at the top and bottom of the soil layer in X, Y & Z directions as well as at the pile base. The interaction in X, Y & base are normal contact and the interaction in Z is shear contact between pile surface and soil in vertical direction. A "Pile-soil interaction property" is referred to by one or more soil layers defined in soil profile table. The pile-soil interactions are non-linear and the non-linear relationships are defined by "Pile-soil interaction coefficients" which is curve defining the relationship between normalized relative soil-pile displacements and the reduction factor of the soil strength. Which "Pile-soil interaction coefficients (curves)" to be used for this pile-soil interaction property in X, Y, Z & base need to be defined in this table as a reference to the "Pile-soil interaction coefficients" table.
7. Open "Pile-soil interaction coefficients" table from Gateway or from menu "Data | Raft | Pdisp Data | Pile-soil interaction coefficients" to define the pile-soil interaction coefficients (curves). These coefficients (curves) define the relationship between the normalized relative soil-pile displacements and the reduction factor of the soil strength. It is used (referred to) by the Pile-soil interaction properties.
8. Open "Raft Interaction" table from Gateway or from menu "Data | Raft | Raft Interaction" to define the nodes on raft to interact with soil. The interaction areas of the raft interaction nodes can be defined on this table or choose Automatic to allow GSA to calculate the interaction areas automatically, the automatic interaction area calculation can only be done for the rafts that are modelled by grillage. The elevation of the interaction can also be defined here or choose Automatic to take the nodal z coordinate as the interaction elevation. Minimum and maximum soil pressure can also be defined in this table. Zero minimum soil pressure means that soil will not take any tensile stresses. If a negative minimum soil pressure is defined, it will be used as the soil tensile strength. The maximum soil pressure is the compressive strength of soil. If this pressure is reached during the analysis, the soil-raft contact pressure will not increase any more to allow soil local yield effect to be considered.
9. Open "Pile Interaction" table from Gateway or from menu "Data | Raft | Pile Interaction" to define the nodes on piles to interact with soil. The interaction areas of the pile interaction nodes can be defined on this table or choose Automatic to allow GSA to calculate the interaction areas automatically. The elevation of the interaction can also be defined here or choose Automatic to take the nodal z coordinate as the interaction elevation.
10. Now we are ready to do piled-raft analysis by going through the analysis wizard start from menu "Analysis | New Analysis Task..." in the same way as doing other GSA analysis.

Note:

The data described above are the data always required for doing piled-raft analysis. There are also some other Pdisp data which are not normally required and necessary in most of the raft analysis. If

they are required, they can be defined as explained below:

1. "Non-linear Curve" - If soil non-linearity need to be considered and the soil strain - Young's modulus (or Young's modulus reduction factor) relationship is known, the non-linear curve defining the strain - Young's modulus reduction factor can be defined and then referred to by the relevant soil profile. Once this has been done, the soil Young's modulus will be changing according to the current soil strain so non-linear properties will be considered.
2. "Load Data" - The loads defined in the "Load Data" table are the loads acting on directly on the soil in addition to the loads come from the raft. Normally, these load data are not required as the soil loads from raft are defined as raft loads. The loads on raft will be transferred to soil through the interaction nodes during the analysis. If there are really some extra loads directly on the soil, e.g. construction loads acting on directly to soil, they can be defined here to increase the flexibility of GSA raft analysis.
3. "Displacement Data" - By default, the soil settlements will be given at the soil-raft interaction points. If the soil settlements at other points, lines or grids are interested, they can be defined in this table. If this has been defined, there will be more soil settlement results.
4. "Results" - This is a output view showing soil analysis results and no data input for this view.

See also:

[Notes on raft analysis](#)

Iteration scheme in section [GsRaft Analysis](#)

2.17.5 Results

Raft & piled-raft analysis results can also be divided into three categories.

Raft & piled-raft results

They are the same as from other GSA analysis, e.g. the nodal displacements and element forces etc.

Soil results

They are the same as from Pdisp analysis (standalone Pdisp program), e.g. soil settlements and stresses. Soil results can be viewed by selecting the "Data | Raft | Pdisp Data | Results" menu command.

Interaction results

The interaction results include the contact pressures between raft interaction nodes and soil and the contact and shear pressures between pile interaction nodes and soil. These results can be contoured or viewed from tabular outputs.

The total reaction at all soil supports can be viewed from "Total Loads & Reactions output".

See also "[The Generate New Structural Model Tool](#)".

2.17.6 Notes on raft analysis

Multiple GsRaft Analyses

More than one raft analysis can be run on a model and multiple sets of GSA results can be stored. However only one set of Pdisp results are held which is the last case analysed.

Combination of Raft analysis results

In general, raft analysis results cannot be combined as it is non-linear analysis if not all raft interaction nodes are in contact with soil or piles are included. However, if it is a pure raft analysis and all the interaction nodes are in contact with soil (not separated from soil) in the raft analysis cases to be combined, the combination of those raft analysis cases can be used since in this case the results are linear.

Export and Import of Pdisp data

Pdisp data files (*.pdd) can be imported into GSA using the "File | Import | Pdisp" menu command. The analysis results in Pdisp data will be ignored in the import since they are meaningless in GSA.

The Pdisp data created in GSA can also be exported to Pdisp data file. The export can be done from the "File | Export | Pdisp" menu command. If Raft results exist when exporting Pdisp data the equivalent soil loads will be calculated and they will be exported along with the other Pdisp data.

The Create New Raft Model Tool

After the completion of a converged raft analysis it is possible to generate a new raft model in which support stiffness equivalent to the soil reactions are added to the current raft model.

This is done using the "Tools | Raft Analysis | Create New Raft Model" menu command.

Creating a new model in this way will modify the current model so it is recommended that the current model be saved before running this command.

The support stiffness are derived from the soil reaction forces and the nodal displacements. If raft analysis results exist for several analysis cases then an analysis case must be selected for the calculation.

Continuation runs

The "Create New Raft Model" tool can be used to restart a raft analysis since the preserved support stiffnesses will be used as initial stiffnesses in subsequent raft analyses.

It is sometimes useful to do an initial analysis with coarse convergence criteria to examine the potential for convergence. The raft analysis can then be continued by using the "Create New Raft Model" tool to generate a model with revised initial support stiffnesses and then analysing with more demanding convergence criteria.

Soil settlement (Pdisp) analysis methods (Mindlin or Boussinesq) to be used

If soil stresses are not interested from raft analysis, it is recommended to use Mindlin analysis method since it is generally faster and less sensitive to "number of intermediate displacement levels" defined for soil layers compared with Boussinesq analysis method. However, Mindlin analysis method is not able to consider soil non-linearity, therefore if soil non-linearity (defined by non-linear curves) need to be considered, Boussinesq analysis method should be used as the non-linear curves will be ignored in Mindlin analysis. For piled-raft analysis, only Mindlin analysis method can be used since Boussinesq method is not able to consider horizontal loads and horizontal displacements in the soil that are required by pile-soil interactions. In piled-raft analysis, the soil non-linearity is considered by the "Pile-soil interaction coefficient" through the non-linear springs used to connect piles and soil. The followings are a summary of the recommended soil settlement analysis methods:

Pure raft analysis:

If soil stresses are not interested and soil non-linearity is not considered, use Mindlin method.

If soil stresses are required or soil non-linearity need to be considered, use Boussinesq method.

Piled-raft analysis:

Only Mindlin analysis method can be used.

2.18 Bridge Analysis

For bridge analysis the live load requirements are very different to those for ordinary structural analysis. A number of tools are available in GSA to simplify the generation of these load cases. Three different ways of using the tools are available for different design codes and different situations:

- A) The most powerful tools are available for specific design codes and constant width carriageways.
- B) For other design codes or more complex carriageway geometry, lane by lane optimisation is possible.
- C) When an influence analysis is not appropriate, loads can be placed directly on a structure.

The steps to be carried out are described below and should normally be carried out in sequence:

Method A – For UK or HK design loading (HA and HB), assessment loading (HA and SV), EC1-UK loading (LM1 and LM3) Australian (AS), or US (AASHTO or LRFD) only on constant width carriageways

1. Enable Bridge Analysis – as method B.
2. Use the [Bridge Specification dialog](#) to select the design code, limit state(s) to be considered and loading type.
3. Define a User Axis – as method B
4. Define a Grid Plane – as method B
5. Define an Alignment – as method B
6. Define a series of [Paths](#) – carriageways/footways only (otherwise as method B)
7. Define influence effects – as method B
8. Carry out [GSBridge Analysis](#) which will produce analysed Grid Loading load cases and Combination Cases.

The Bridge Analysis generates data for a number of input modules. These are generally deleted automatically when the Influence Analysis is deleted.

For large files with many Influence Effects the GSBridge Analysis may take a long time.

The process may be split into two stages by carrying out an Influence Analysis (which includes Optimisation in this method) and reviewing the Influence Lines and generated Static Bridge Loads before Expanding Bridge Loading and carrying out the final static analysis. Note that the user will then need to create Combination Cases manually, so this is only recommended for large models where the user wants to make manual changes to Static Bridge Loading (for example to delete cases which will not govern design by inspection, such as sagging at supports). If the user wants to make minor alterations to the generated loading, without deleting cases, the recommended approach is to carry out a full Bridge Analysis first, then delete the static Analysis Cases (using the [Analysis Wizard](#), not Delete All Results which will delete the Carriageway Paths and Static Bridge Loads as well) and Grid Loading (using the [Delete Grid Loads Tool](#)), then modify the Static Bridge Loading, and follow steps 11 and 12 of method B.

Method B – to optimise on a lane by lane basis for any design code

1. Enable Bridge Analysis (go to the “Tools | Preferences” (Ctrl+F7) menu command, select “Advanced Features” and ensure that the “Bridge analysis” option is enabled).
2. Use the [Bridge Specification dialog](#) to set the design code to “Other”.
3. Define a [User Axis](#) with origin at the start of the alignment, chainage 0 and the x axis in direction of increasing chainage (or arrange the model so that the alignment starts at global (0,0,0) in the Global X direction).
4. Define a [Grid Plane](#) on which the load is to be applied. Note that all bridge loading is applied in the Grid Plane, and that the elements selected in this plane need to form a simple grillage where elements only meet at nodes. Further details of these restrictions are given under “[Grid Point Loads](#)”.
5. Define an [Alignment](#), defined by a series of curvatures at chainages and the Grid Plane.
6. Define a series of [Paths](#) – lanes/footways/tracks/vehicle path (defined by offsets from the Alignment, note that positive offsets are to the right of the alignment, looking in the direction of increasing chainage, not in the direction of positive local y) along which bridge loading is to be

moved. Paths need to be within [closed panels of the structure](#), except at the ends of the structure.

7. Define a series of [Node Influence Effects](#) or [Beam Influence Effects](#) that are to be maximised/minimised by the Optimiser.
8. Define a series of [Path Loads](#) that are to be applied to Paths.
9. Carry out an [Influence Analysis](#) using the GSBridge option in the Analysis Wizard. Tip: It is often useful to save the results at this stage to allow a quick return to this point.
10. Use the [Optimize Path Loading Tool](#) to derive Static Bridge Loads for each Path Load and each Influence Effect. Tip: increase the default group number by one to allow the group number and load case number to be made identical to the analysis case number, see step 12.
11. Use the [Expand Bridge Loading Tool](#) to convert Static Bridge Loads (or Moving Bridge Loads) to [Grid Loads](#). Note that only the Grid Loads form part of the input data to a static analysis, and changes to the other data described above will not have any effect on subsequent analysis unless the old Grid Point Loads are deleted and new ones generated using this tool.
12. Carry out a Static Analysis of the Grid Load cases using the [analysis wizard](#). Tip: create default analysis cases then delete the dummy load case created at step 10 to make analysis case numbers identical to load case numbers.
13. Using the Total Effect information in the [Static Vehicle Loads](#) module as a guide, set up [Combination Cases](#) to give the worst effect due to simultaneous loading on several Paths.

Having gone through these steps once for a model it is sometimes useful to use the GSBridge dialog box to carry out steps 9 to 12 in one operation. Expanding Bridge Loads generates Grid Loading and 2D Polyines. The [Delete Grid Loading Tool](#) allows these to be deleted quickly before carrying out another analysis.

Note that generated Static Bridge Loads are not deleted automatically with the Influence Analysis Results in this method, and should normally be deleted before carrying out another Influence Analysis.

Method C – Placing load directly on the structure

1. Enable Bridge Analysis – as method B.
2. Use the [Bridge Specification dialog](#) to set the design code to “Other”.
3. Define a User Axis – as method B
4. Define a Grid Plane – as method B
5. The [Generate Static Vehicle Loading tool](#) can be used to place a vehicle directly on the Grid Plane, otherwise:
6. Define an Alignment – as method B
7. Define a series of [Paths](#) – as method B
8. The Generate Static Vehicle Loading tool can also be used to place a vehicle directly on a path, or offset from an alignment, otherwise:
9. Vehicles can be moved along Paths at regular intervals as [Moving Bridge Loads](#) and [Static Bridge Loads](#) can be placed directly on paths.
10. Not used
11. Expand bridge loading – as method B.

More:

[Modelling implications for bridge analysis](#)

[Delete Grid Loading Tool](#)

[Analysis of bridge structures](#)

2.18.1 Modelling implications

Defining the structure

Different models are usually needed to determine global effects on the main structural members, and local effects (local bending of slabs/plates). The main GsBridge features are designed for determining global effects. Grillage models of complete bridges are unlikely to be sufficiently refined to give a reasonable estimate of local effects, which also depend on wheel contact areas etc. These effects on local parts of bridges can be investigated using fine grillage meshes for short lengths of bridge, and using the [Generate Static Vehicle tool](#) to model critical wheel loads as patches.

Bridge loading is applied to Grid Planes in GSA, and the results from a GsBridge analysis will depend on the type of Grid Plane chosen. The [Technical Note on Grid Loading](#) describes in detail how the loads are distributed to elements and defines some of the terms used below. Some knowledge of how the load is transferred from space on to the structure will avoid some of the potential problems associated with bridge analysis.

Structure edges

When checking the paths a pass is made along the path checking where the path first crosses and where it last leaves the structure and looks for any positions between these where the path “falls off” the edge of the structure. The method of doing this relies on identifying the edge of the structure. If no edge can be found the checks will fail.

Note: Where a bridge deck is constructed from longitudinal beams with transverse beams that cantilever out from the main beams, the resulting comb like sides cannot be interpreted as edges of the structure. This problem can be avoided by adding dummy elements between the tips of the cantilever elements. These elements are ignored in the analysis but they allow the identification of the structure edge and panels for loading.

Tolerances

When checking which elements lie in the grid plane and thus those forming the deck the “Grid plane tolerance” is used. If the deck is not flat the tolerance may be adjusted to include all deck elements, but if panels are significantly out of the grid plane a solution is unlikely to be found.

When checking for the start and end chainages for bridge analysis the “Grid loading edge tolerance” is used. This may require adjustment if the path is reported as crossing the edge of the structure.

Grid plane span direction

There is no difference between two-way spanning and multi-way spanning unless a Grid Area Load completely covers a lot of simple rectangular panels, in which case two-way spanning will give quicker results. One way spanning, because the span direction is related to the grid plane definition axes, will only give meaningful results for straight bridges. Normally it is recommended to use multiway spanning onto a list of elements (defined with the grid plane) which only includes the longitudinal beams and end diaphragms. This will model the spreading effect of the slab without confusing global and local effects.

Influence line interval

During an Influence Analysis, values of effects are calculated at a number of regularly spaced points along the length of the structure (provided the alignment extends over the full length). By default the program will try to provide approximately 100 points. Where the structure is very long and an effect is very local (eg bending of a transverse element) there may not be sufficient calculation points in this local area to accurately position the load to pick up the maximum effect. If this is a concern then the number of influence calculation points can be increased in the Bridge Specification dialog.

Note: Increasing the number of points will increase the analysis time, so the number of points should be chosen to be sufficient but not excessive.

2.18.2 Delete Grid Loading Tool

Bridge Analysis generates Grid Point Loads, Grid Line Loads and Grid Area Loads. These loads will be deleted automatically when deleting bridge analysis results, but if these loads are used by other analysis tasks, they will not be deleted along with other bridge analysis results. As there may be circumstances that all those loads need to be deleted, the Delete Grid Loading Tool does this in a single operation.

2.18.3 Analysis of bridge structures

[GSBridge analysis](#) is selected in the Analysis Wizard.

Analysis of bridge structures consists of a number of stages:

- Influence analysis
- Optimization
- Expansion
- Static analysis

For method A as described in the [Step-by-Step guide to Bridge Loading](#) all these stages are usually carried out in a single operation, while for method B it is more likely that only the influence analysis will be carried out initially and the other operations will be done using the tools described.

The influence analysis is carried out by applying a unit load across the width of the path at a series of positions along each path, the result for each influence effect being extracted for each load position. As this involves solving for many sets of loads the influence analysis can be quite slow.

The influence analysis is selected in the “[Analysis Wizard](#)”. The influence analysis is carried out by applying a unit load across the width of the path at a series of positions along each path, the result for each influence effect being extracted for each load position. As this involves solving for many sets of loads the influence analysis can be quite slow.

The optimization uses [the path loading](#) and the influence results to determine the most onerous load positions (those that maximise the response at the positions of the influence effects) and creates a set of static bridge loads.

The third stage in the bridge analysis is to carry out a static analysis of the structure subject to the [static bridge loads](#) determined by the optimizer. The first stage in this analysis is to expand the bridge loads “[Tools | Bridge Analysis | Expand Bridge Loads](#)” menu command. This converts the bridge loads defined in the “Bridge Loads” table into loads on the structure. The static analysis is then selected in the “Analysis Wizard”.

2.19 Analysis Stages

The analysis stage facility allows the elements, element properties, restraints and constraint conditions to be specified for particular analysis cases. It is provided to allow the results from models differing in these ways to be combined (e.g. for the analysis of construction stages).

Analysis stages may be analysed by either the GSS or GsRelax solver. They may not be analysed by GsSpec or GsRaft.

To use these features go to the “Tools | Preferences” (Ctrl+F7) menu command, select “Advanced Features” and ensure that the “Analysis stages” option is enabled.

More:

[Modelling implications of analysis stages](#)

[Analysis of stages](#)

[Results for analysis stages](#)

[Stages and Graphic Views](#)

2.19.1 Modelling implications

The model is set up in the usual way. In the context of staged analysis the model that is set up without reference to an analysis stage is referred to as the whole model. The following should be considered when setting up the whole model.

- The elements that constitute an analysis stage will be a list of elements from the whole model so the whole model must be the superset of all analysis stage elements.
- The element properties used in an analysis stage default to the whole model properties. The properties for a given property number may be replaced for specific stages (see below) so whole model element property numbers should be organised accordingly.
- Nodal restraints and spring supports are applied in all stages in which the nodes to which they are applied are active. If it is required that a node be restrained in some stages but not in others then this must be done in the Generalised Restraints module (see below), not in the Nodes module. If it is required that a node has a spring support in some stages but not in others then this must be done using a grounded spring element, not in the Nodes module.

Analysis stages are defined in the [Stage Definition](#) module. A stage has a name and a list of elements that are to be active for the stage. Other data that contributes to the definition of the stage is as follows.

- If the properties associated with an element property number are to be different for a specific analysis stage the required property record may be mapped on to the element property number for the stage in the [Analysis Stage Properties](#) module. This module does distinguish between different property types so mapping Beam Sections record 10 on to element property number 5 will only affect elements that refer to Beam Section property 5. Note that the element local axes may not vary from stage to stage so, for example, changing the 2D Element Property axis from that specified for the whole model will result in an error.
- [Generalised restraints](#) may be associated with a particular stage. (This field is hidden when the analysis stages feature is not enabled.) When the stage is set to zero the restraint condition is applied to the nodes in the whole model, irrespective of stage. Otherwise it is applied just in the specified stage. If it is required to apply a restraint to a stage made up of all the elements in the whole model but not to all stages then a stage must be set up containing all of the elements.
- [Rigid constraints](#) may be associated with a particular stage, as for Generalised Restraints. Note that it is valid for the slave node list to identify nodes that are not active for the stage; a warning is given in this circumstance. The master node is implicitly active for the stage.
- [Joints](#) may be associated with a particular stage, as for Generalised Restraints. The joint is ignored if either the slave or master node is not active for the stage.
- [Constraint equations](#) may be associated with a particular stage, as for Generalised Restraints. The equation is ignored if either the slave or any master node is not active for the stage.

Note that when specifying element lists references to element properties in the list (to identify elements with a specified property) are always interpreted as whole model element properties. For example, beam loads applied to elements PB5 are applied to beam and bar elements that have element property number 5 in the whole model even if the analysis of that load case is of a stage for which the beam section property has been mapped to a different record.

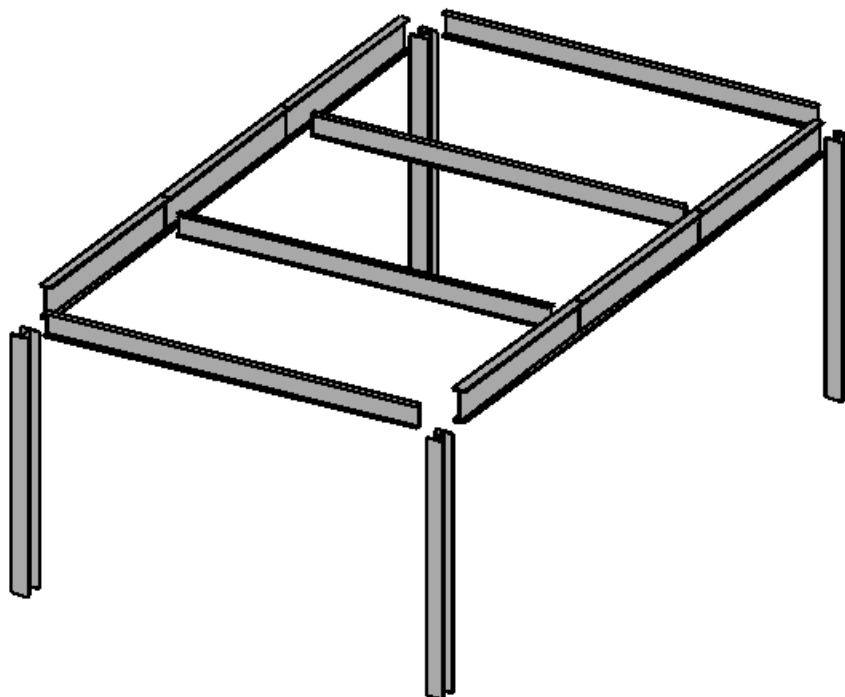
2.19.2 Analysis and Design Layers

GSA can hold two versions of a model — these are best thought of as two “layers”, an analysis layer and a design layer. The analysis layer is composed of elements and provides the topological connectivity between the different parts of the structure. The design layer is composed of members and provides a description of the members (individual pieces of steel, etc.). The two layers are

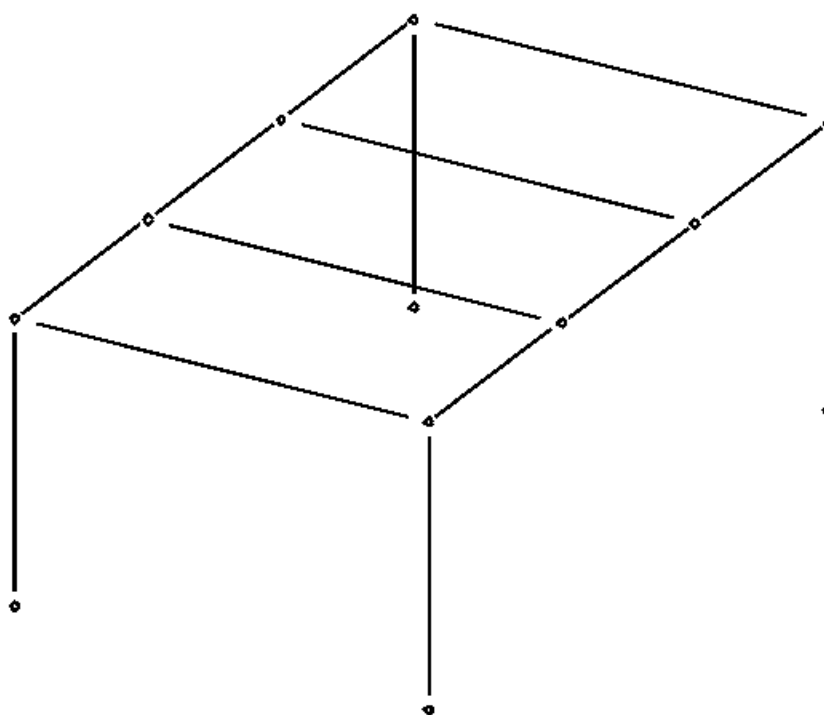
connected through the nodes that are common to both elements and members.

Note that there is no need to consider members and the design layer if the design options are not being used.

For a simple frame the main beams spanning between the columns are represented by a single member, but secondary beams frame into these members. The nodes at the ends of the secondary beams lie along the length of the primary beam members but are not part of the topology of the member, which is defined only by the end nodes.



The design layer is not appropriate for analysis as it does not represent the physical connectivity between the primary and secondary beams. This connectivity is essential for the analysis to perform correctly. The analysis layer consists of elements and in this case the primary beams (which in the design layer are represented by a single member) are represented by three elements.



Member definition is similar to element definition but only beam and bar member types are available. In addition to the section properties of an element, the members have also a design property that gives details of design specific data.

Tools provide a means of generating the analysis layer from the design layer and vice versa. The mapping of members to elements will work for situations such as those above but at present only a one-for-one mapping of elements to members is provided.

2.19.3 Analysis of stages

The analysis of a stage is set up using the [Analysis Wizard](#) from the “Analysis | New Analysis Task” menu command. Select the option to create new analysis cases, then, on the [Solver Option page](#), select the analysis code, whether GSS or GsRelax, and the analysis stage. The remainder of the data in the wizard is entered in the usual way.

2.19.4 Results

Results are produced for analysis stages in the same way as for whole model analysis. Values of zero are stored for nodes and elements that are inactive for the stage.

Results are not output for inactive nodes and elements where this is reasonable, though records representing that data are reported if, say, output is requested for several cases and the node or element is active in some of the cases.

When combining results from different analysis stages there are circumstances in which it is inappropriate to output the combined values. The following restrictions apply.

- Combined element stress and force results (and results derived from these) are not output when the geometric properties of the element differ for any of the analysis cases being combined. For this purpose geometric properties are deemed to differ:

- If any Beam Section description differs or if any modification factor, other than a shear factor modification factors, differs.
- If any 2D Element Property thickness differs.
- If any Link Property linkage differs.

respectively. Properties for other element types are not checked for consistency and are deemed not to differ. Note that the material is not checked for consistency so these results will be combined even if any material differs.

- Combined reactions are not output when the restraint condition differs for any of the analysis cases being combined.
- Combined rigid constraint, joint and constraint equation forces are always output regardless of whether the constraint condition for the node differs for the analysis cases being combined.

Result envelopes are not output when the result for any contributing permutation is not calculated for any of the reasons stated above.

In staged analysis combined 2D element stresses and forces are not averaged across elements when the geometric property (i.e. thickness) or material differs for either element being considered for any of the analysis cases being combined.

2.19.5 Stages and Graphic Views

Graphic Views can be set to display a specific stage. The stage is specified on the “[Lists](#)” toolbar for the Graphic View by first setting the Display Option to “Stages” and then selecting the required stage in the Display List. The “Stages” Display Option is not offered when no stages are present in the model. The stage can also be set in the in the [Graphic Settings dialog box](#).

Setting the current stage to other than whole model has the following affect.

- Elements not in the current stage are either not drawn or drawn in wire-frame style, depending on whether “Draw inactive elements” is set on the [Display Methods page](#) of the Labels and Display methods dialog box.
- Element properties, restraints, rigid constraints, joints and constraint equations displayed are those relating to the current stage.
- When edges are highlighted the edge calculation is based on the elements that are active for the current stage.

The display of results is independent of the current stage. Results are produced by the analysis of a stage so it is the stage that relates to the analysis case that determines whether or not results are available for a node or element and the analysis stage properties are used in calculating beam stresses, etc.

The analysis stage in which an element first appears can be labelled by setting the “Initial stage” option on the [On Elements page](#) of the Labels and Display methods dialog box. Elements can be coloured according to the analysis stage in which they first appear by setting “Colour Elements | By initial stage” on the [Display Methods page](#) of the Labels and Display methods dialog box.

References to element properties in the element list on the [Lists toolbar](#) (to identify elements with a specified property) are always interpreted as whole model element properties regardless of the current stage.

2.20 Analysis Envelopes

An analysis envelope is a *stored* envelope of results produced in way similar to other analysis options. I.e Analysis envelopes are requested by setting up an appropriate analysis task via the [Analysis Wizard](#) and then analysing this. The envelope is produced by comparing results from

cases described in the syntax of a standard [enveloping combination case](#). Since it is stored it is very much quicker to display than the alternative combination envelope that is produced by examining all effected analysis case results at the time of display. The disadvantage of analysis envelopes is that derivation of other results from those stored is not possible.

The specification of an analysis envelope includes a case description, that describes which cases are to be compared, and the result types for which envelopes are required. In the case of element results it is also necessary to specify whether an envelope is required at each node on the element or just at the worst position. Setting up an analysis envelope task produces several analysis cases: a case each for the maximum and minimum values for each position for each data type selected. Performing the analysis determines which permutation causes the maximum or minimum value for the data type at the position at each node or element.

Having produced the analysis envelope the results to which the envelope permutation refers plus coexistent values can be output in Graphic Views, as contours or diagrams, and in Output Views as tables of results.

Analysis envelope cases may be referred to by combination cases.

2.21 Environmental Impact

The Environmental Impact feature enables environmental impact parameters to be assigned to the structure, and for the environmental impact of the structure to be reported both as a summary and to be represented graphically.

Environmental Impact Parameters

The following environmental impact parameters can be assigned:

- Embodied energy (eE)
- Embodied carbon (eCO₂)
- Recycled content (%RC)
- a user defined parameter

A set of environmental impact parameters can be assigned to the structure for each material type: Steel, Concrete, Rebar, Aluminium, Glass, and Wood. These global values are assigned in the [Environmental Impact Specification](#) dialog.

The global environmental impact parameters can be overridden by parameters assigned to user defined materials via the [Material Wizard](#). Standard materials always assume the global parameters.

Note that the parameters are based on material type. Elements and members must be assigned to materials of a defined type to accrue environmental impact.

Where an element refers to a concrete material type, the parameters assigned to the element are both the concrete and rebar parameters in the specified proportion.

Environmental Impact Wizard

Environmental impact parameters are assigned via the [Environmental Impact Wizard](#). The wizard suggests parameters for the material type, country, material variant, and material grade, though these 'typical values' can be overridden. The material variants and material grades offered are adjusted depending on the material type and country settings. For example, when specifying parameters for concrete, the variant is expressed in terms of the level of cementitious content, and the grade, with notation respective of the country/region selected.

Note that the typical values are based on information available, and for countries for which information is available, at the time of publication. The data will become out-dated rapidly as environmental circumstances change and as the understanding of the industry evolves. The 'date of reference data' is reported in the wizard to give a measure of confidence.

Output of Environmental Impact

Environmental impact can be output as follows:

In Output Views: the 'Environmental Impact Summary' output reports environmental impact *totals* per property, per material property and for the whole structure. Recycled content is proportioned by the quantity of total material, resulting in total recycled content for the structure. The user should judge whether the summation of the user defined impact is valid.

In Graphic Views: environmental impact parameters can be contoured on elements, for the current stage, and on members.

2.22 Steel Design

The analysis capabilities allow the user to determine forces and stresses in elements. The steel design option checks these against the capacity of the “members”. Steel design is integrated within GSA so that the design model coexists with the analysis model. Steel design is carried out on the design model using analysis results to establish the steel utilisation to the selected [design code](#).

To use these features go to the “Tools | Preferences” (Ctrl+F7) menu command, select “Advanced Features” and ensure that the “Steel Design” option is enabled.

More:

[Modelling implications of steel design](#)

[Steel Restraint Properties](#)

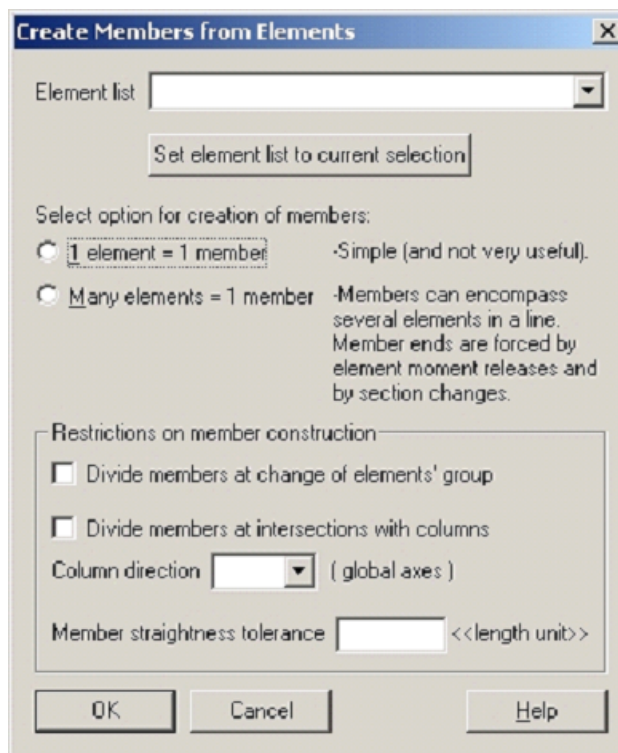
[Member Restraints](#)

[Tools](#)

[Results for steel design](#)

2.22.1 Modelling implications

The main difference between analysis and design is that analysis is carried out on elements while design is carried out on members. If the starting point is an analysis model then the “[Tools | Manipulate Model | Create Members from Elements](#)” menu command can be used to create members from elements. This tool automates the creation of members from elements, although members can be defined manually as well. A steel member can consist of any number of elements arranged in a continuous straight line. For example, a beam spanning between two columns with a secondary beam attached at mid span requires two elements for analysis but can be represented by a single member. The tool ignores elements that are already covered by members.



Members can be displayed in a Graphic View by switching to the design layer. The layer is specified on the “[Lists](#)” toolbar for the Graphic View by first setting the Display Option to “Layer” and then selecting “Analysis” or “Design” in the Display List. The layer can also be set in the in the [Graphic Settings dialog box](#) or from the graphics right-click menu. Sculpt commands that operate on elements also operate on members. The design layer is selected only when the graphic view background is pink, instead of “parchment” which would indicate the analysis layer is shown.

Members are listed in separate, but parallel tables to elements.

Steel members have three properties:

- Section property (as analysis elements)
- [Steel Member Design Property](#) (addresses design specific issues like effective buckling lengths)
- [Steel Restraint Property](#).

Initially the sections used by the member should be the same as those of the elements it represents, but some extra properties are required to fully define the member. The two extra properties required are the design property and the steel restraint property. The design property allows the specification of design specific attributes including the net-to gross area ratio for the entire section, the grade of steel, the maximum allowable plastic:elastic moment capacity ratio and the shear lag factor. In addition, it allows the option of specifying the effective length of the member either using an absolute length, or as a multiple of the member’s true length. If the effective lengths are not specified by the user program uses the Restraint Property to calculate the effective lengths. Restraint properties are necessarily more complicated than restraints used in analysis models, and are covered in the following section.

Member forces are required for the design calculations. Since analysis results are calculated for elements, not members, it is necessary for there to be a relationship between elements and members. This relationship is formed automatically by GSA based on the element and member topologies.

2.22.2 Steel Restraint Properties

Steel Restraint properties consist of a list of Steel Member Restraints (see below), each assigned to a node in the member or smeared over a constituent element. A member can be restrained at its constituent nodes, and over the “spans” (elements) between its nodes. The span restraints are assumed to be distributed over the span: an example of this is the restraint afforded to a window mullion by the glazing. The nodes and spans are referred to in the restraint properties table by use of their index and the terms “end1”, “end2” and “all”. For example, the fourth (4th) node in a member could be referred to using its index: 4. The keyword “all” can be used to specify that all nodes (or spans) are restrained using the specified restraint. Nodes and spans referred to specifically and by the “all” keyword will be restrained by the specifically assigned restraints. This allows convenient specification of restraint properties: for example a simply supported node with lateral restraints at every intermediate node could be restrained with just three entries in its restraint property: simple supports at nodes at ends 1&2, and lateral restraint to all nodes: the end restraints override the “all” restraints for those individually restrained nodes.

Steel Member Restraints are assigned to members via restraint properties because many members in a structure are likely to have the same pattern of restraints, and because the specification of design restraints is relatively slow, assembling them into a reusable property will save the user effort.

Select a row to be edited by clicking on the row's entry in the first column. The rows details will be copied down to the Edit box below. Click on the Delete button to remove a row from the list.

Compose or edit restraint details in this box. Press the Update List button to copy the new details up to the list above. Press the Edit Restraint... button to get assistance in specifying the individual restraint.

These shortcut buttons allow definition of the entire Restraint Property in one hit. Existing settings will be lost.

Restraint property 1

Name: main stringers

Node/Span	Ref.	Load ht.	Description
node	end1	0m	enc
node	2	0m	TR
node	3	0m	TR
node	4	0m	TR
node	5	0m	TR
node	end2	0m	TR

Delete

Edit

☐ Node ☐ Span

Reference:

Description:

Destabilizing load: m

Update list

Edit restraint...

Common restraint properties

Composite (top flange restrained by deck)

Simply supported

Cantilever (end 1 enc.)

Cantilever (end 2 enc.)

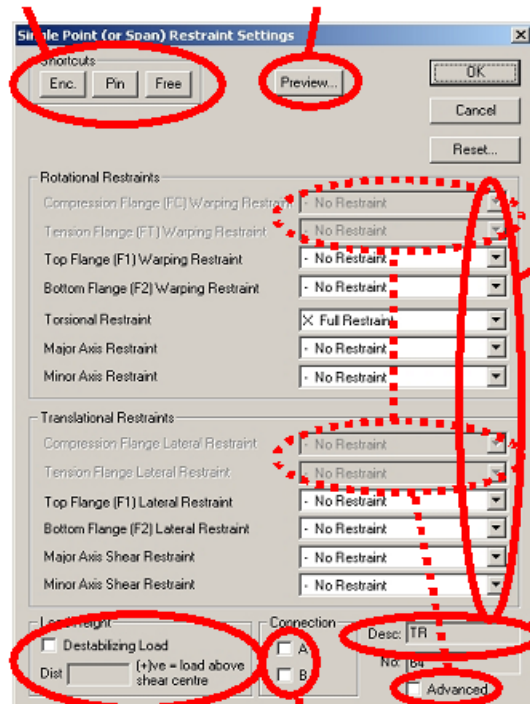
OK Cancel Help

Steel Restraint Properties can be specified in the restraint properties table, or by using the [Steel Restraint Property Definition dialog](#). The dialog can be accessed by right clicking on a member in a graphic view and selecting “edit restraint property”, or by clicking on the wizard button (the magic wand on the data options toolbar) when the restraint property table is open. The wizard allows quick setting-up of common restraint properties by the provision of shortcut buttons.

2.22.3 Member Restraints

These shortcut buttons can be used to set common restraints quickly. Use them to speed up setting custom restraints: you can modify the initial settings to your exact needs.

Use this button to see a text description of the restraint you have selected. Remember: Some restraint axes can become restrained by implication... for example: If you set lateral restraint on both flanges, that implies torsional restraint as well.



Use these combo box pulldowns to select the level of restraint in each "axis".

Advanced Users: Watch this mnemonic change as you adjust your restraint settings. You can avoid using this dialog box entirely if you know how to type these mnemonics into the Restraint Property Wizard or the Restraint Property Table. If you make a mistake, the correct syntax is displayed in a message box.

Allows specification of load height: this may affect the selection of effective buckling lengths. Clicking in the check box enables this option.

Additional information about the connection that may be required for some section types and design codes. For example, These check boxes can be used to specify which legs of an angle section are connected.

Flanges can be referred to by their position (top/bottom) or by whether their bending stress is compressive or tensile. This option could lead to confusion, and so is disabled by default. Click in the advanced check box to enable this feature.

The individual member restraints that make up a restraint property are analogous to nodal restraints, but are more detailed, and are specific to 1-dimensional beam type elements. They can be specified directly by typing their shorthand code into the restraint property table or by using the Member Restraint dialog box. This dialog box is accessible from the member restraint wizard via its "edit restraint..." button. If the specification is typed directly, any errors will be notified, and an explanation of the syntax will be displayed in a message box. The [Member Restraint dialog box](#) shows the code for the current restraint, and updates the code as the restraint is edited. It is recommended that the dialogue box is used until a reasonable level of understanding is obtained. Use of the direct text method affords considerable time savings thereafter.

2.22.4 Tools

Various menu items are available to assist you in the construction and maintenance of design models. These are accessible from the Tools and Design menus:

Menu	Menu Item	Description
Tools	Create Members from Elements	Automates the creation of members, given an existing analysis model.
	Create Elements from Members	Automates the construction of an analysis model, given an existing design model.
Design	Reset Member Sections	Changes member section property numbers so that they match those of the corresponding elements.
	Update Member Sections	Change element section property numbers so that they match those of the corresponding members.
	Check Member-Element Relationships	Do elements and members match up spatially?
	Check Steel Design Data	Does every member have valid properties such as a valid steel restraint property?

2.22.5 Results

Results are available for any valid analysis case or combination. There is no stage at which the user requests a design to be processed. Instead, when the data is correctly specified to enable the design calculations to be carried out the design results options become available. The design results are calculated only at the time they are needed for display.

The “Design | Check Steel Design Data” menu command may be used at any stage to establish whether the data has been correctly specified for steel design. The details of any problems appear in a message box that is displayed after this command is given.

A convenient way of inspecting results for individual members in rapid succession is to use the report tab of the [Object Viewer](#). Once the Object Viewer window is set to display member results (for example: brief results), the member to be inspected is selected by clicking on the relevant member in any graphic view that is showing any members. The view could usefully be displaying a contour of member utilization, so that members of particular interest could be picked.

Steel utilisation factors can be output in the same way as stress results, i.e. they may be shown in Graphic Views as contours or diagrams, in Output Views as tables of steel utilisation, or in the Object Viewer.

Steel design summaries, brief calculations, verbose calculations and effective lengths are also available in Output Views and the Object Viewer.

2.23 Concrete Design

All RC calculations are code dependent. Select the appropriate concrete [design code](#) via “Data | Specification | Design Code Specification”.

2.23.1 RC Slab Reinforcement Design

The RC slab reinforcement design process designs reinforcement areas for RC slabs modelled using 2D elements of uniform thickness, subject to any combination of in-plane axial or shear force and out-of-plane bending moment and torsion. The process does not design for through thickness shear.

Two directions of reinforcement are specified; these need not be orthogonal. Reinforcement areas are calculated for each direction at the top and bottom of the slab.

To use these features go to the “Tools | Preferences” (Ctrl+F7) menu command, select “Advanced Features” and ensure that the “Concrete Design” option is enabled. Select the appropriate RC Slab design code via “Data | Specification | Design Code Specification”.

See also:

[RC Slab Reinforcement Design Theory](#)

More:

[Modelling implications of RC slab design](#)

[Results for RC slab design](#)

2.23.1.1 Modelling implications

RC slab reinforcement design is carried out directly on a 2D element model that represents a slab. The only data required additional to the analysis model data are the RC Slab Design Properties and the selection of an RC Slab design code. These may be specified before or after analysis.

The element property number in the Elements table identifies the RC Slab Design Property record associated with the element, — just as it does the 2D Element Property record that is used for analysis. Often there will be the same number of RC Slab Design Properties specified as 2D Element Properties. Reinforcement is not calculated for elements that do not have RC Slab Design Properties. Refer to [Program Data: RC Slab Design Properties](#) for details.

2.23.1.2 Results

There is no stage at which the user requests a design to be processed. Instead, when the data is correctly specified to enable the design calculations to be carried out the design results options become available. The design results are calculated only at the time they are needed for display. The “Design | Check RC Slab Design Data” menu command may be used at any stage to establish whether the data has been correctly specified for RC slab design.

Reinforcement areas can be shown in Graphic Views as contours and diagrams, and in Output Views as tables of RC Slab Reinforcement. Areas shown with positive values represent tension reinforcement. Areas shown with negative values represent compression reinforcement.

Occasionally the process fails to produce a result for a particular point in a slab. The reason for such a failure is given in the Output View RC Slab Reinforcement table. Such failures are displayed in contoured output as out-of-range values, typically displayed black.

2.24 Wave loading

The wave loading tool calculates wave loads on 1D elements using Morison's equation. Wave loading is an option on the Tools menu. The wave profile is supplied by ASCII input file (wave profile data file). The calculated wave loads are represented by beam loads. The wave loads can be calculated according to either Airy linear theory or Stokes's Fifth Order theory that can be set in the wave profile data file.

Wave loading has the following capabilities:

1. A choice of two wave theories:
 - Linear Airy theory.
 - Fifth Order Stokes's theory.
2. A series of phase angles can be specified.
3. A wave profile can be specified.
4. A drag and inertia coefficients profile can be specified.
5. A marine growth profile can be specified.
6. Skip level (whole elements below this level are not considered in the wave load generation).
7. Skipped elements list (elements in this list are not considered in the wave load generation).
8. Option to define the number of points along elements for the generated poly-loads.
9. A summary file (*.sum) will be generated after running Wave Loading which summarizes the wave profile data and the total wave loads.

Running Wave Loading

The generation of wave loads can be run from 'Tools | Offshore | Wave Loading' menu command. First a file dialog box is displayed in which the wave profile data file (*.wvp) must be selected. The format of wave profile data file is explained below. Then the wave loading dialog box prompts for load case information. Pressing OK in the wave loading dialog box commences the wave load generation process. Errors and warnings encountered during the generation process are displayed in the GSA report view. A successful wave load generation produces beam loads on the GSA model.

The format of Wave Profile Data File (*.wvp file)

Each line in the wave profile data file contains a key word followed by data pertaining to that key word. Keywords and data can be separated by spaces, tabs or spaces and tabs. The key words and data are summarized in the following table:

Wave profile data file (key words and data)

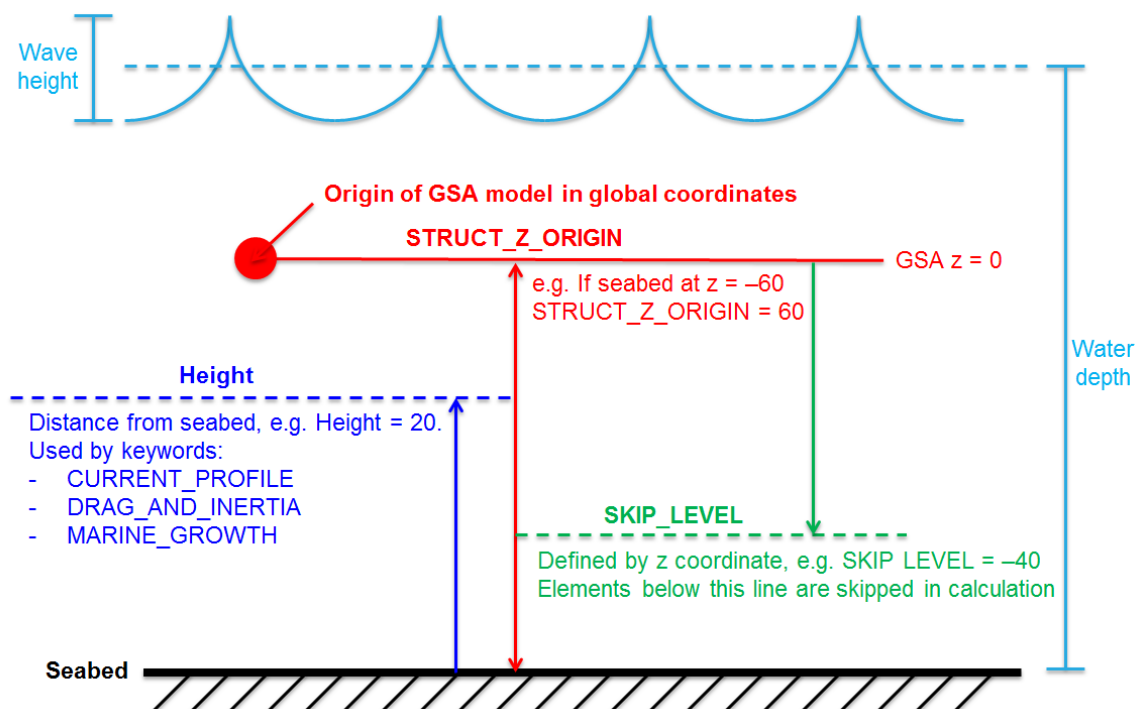
Key words	1st data	2nd data	3rd data	4th data
WAVE	<i>Wave height</i>	<i>Wave period</i>	<i>Water depth</i>	<i>Wave direction</i>
PHASE	<i>Start phase</i>	<i>Finish phase</i>	<i>Phase increment</i>	
STRUCT_Z_ORIGIN	<i>Distance from seabed to GSA structure origin</i>			
WAVE_THEORY	<i>STOKES or AIRY</i>			
CURRENT_PROFILE	<i>Height</i>	<i>Current velocity</i>		
DRAG_AND_INERTIA	<i>Height</i>	<i>CD</i>	<i>CM</i>	
MARINE_GROWTH	<i>Height</i>	<i>Thickness</i>		
SKIP_LEVEL	<i>Water level</i>			

SKIPPED_ELEMENTS	<i>Element list</i>			
POLYLOAD_POINT	<i>Number of points</i>			

Unit: metre, kN and degree

Note: (also see the diagram below for illustrations of some of the parameters)

WAVE	Key word for wave profile (one entry only and compulsory)
Wave height	wave height, the vertical distance from valley to peak. If wave height is zero, the wave loads generated will be from current only and phase angle will be ignored
Wave period	wave period (second)
Water depth	Water depth – distance from seabed to surface
Wave direction	0 is in X axis direction and the angle is measured in degree on anticlockwise direction
PHASE	Key word for phases (one entry only compulsory) in unit of degree
Start phase	first phase to be calculated
Finish phase	last phase to be calculated
Phase increment	phase angle increment
STRUCT_Z_ORIGIN	Key word for GSA structure origin (one entry only, if omitted zero is assumed)
Distance from seabed to GSA structure origin (i.e. GSA global Z axis origin), e.g. if sea-bed is at – 60 on GSA global coordinate system, STRUCT_Z_ORIGIN should be 60.	
WAVE_THEORY	Key word for wave theory (one entry only and compulsory)
Either STOKES or AIRY	
CURRENT_PROFILE	Key word for current profile (as many entries as needed and optional)
Height	distance from seabed
Current velocity	the current velocity at above water 'Height' level
DRAG_AND_INERTIA	Key word for drag and inertia coefficients (as many entries as needed and compulsory)
Height	distance from seabed
CD	Drag coefficient at above water 'Height' level
CM	Inertia coefficient at above water 'Height' level
MARINE_GROWTH	Key word for marine growth, radium growth of the tubular members (as many entries as needed and optional)
Height	distance from seabed
Thickness	growth thickness at above water 'Height' level
SKIP_LEVEL	Key word for elements skip level (one entry only and optional)
Water level	GSA Z coordinate, elements below this level will be skipped in wave load calculation
SKIPPED_ELEMENTS	Key word for skipped elements (as many entries as needed and optional)
Element list	list of the element number that to be skipped in wave load calculation (refer to the GSA documentation for details of valid element list syntax)
POLYLOAD_POINT	Key word for the number of points of the poly-loads (one entry only and optional, default is 8)
Number of points	the number of the generated poly-load points



Example Wave Profile file

```
!Lines starting with - Lines start with ! are comment lines
!
!Wave      wave_height  wave_period  water_depth  direction_angle
WAVE       17.5        13.4        60          90
!
!Phase     start_phase  end_phase   phase_step
PHASE      0          180         45
!
!Struct_Z_origin  distance_from_sea_bed_to_GSA_structure_origin
STRUCT_Z_ORIGIN  60
!
!Wave_theory  STOKES  or  AIRY
WAVE_THEORY  STOKES
!
! Current profile  height  current_velocity
CURRENT_PROFILE  0      0.55
CURRENT_PROFILE  100    0.55
!
! Drag and Inertia  Height  CD  and CM
DRAG_AND_INERTIA  0      1    2
DRAG_AND_INERTIA  60     1    2
DRAG_AND_INERTIA  60.1   0.7  2
```

```

DRAG_AND_INERTIA      100      0.7      2
!
! Marine_Growth Height Thickness
MARINE_GROWTH  0      0.05
MARINE_GROWTH  60      0.05
MARINE_GROWTH  60.1    0
MARINE_GROWTH  100     0
!
SKIP_LEVEL          -30
SKIPPED_ELEMENTS    1 to 5
POLYLOAD_POINT 15

```

2.25 Model Stability Analysis

Model stability analysis can be helpful in detecting the causes of [ill conditioning](#) in GSA models. As mentioned in [model stability analysis](#) theory, if a model is reported as being ill-conditioned in a static analysis, a model stability task can be setup and run. It is best to start with the default values of low and high stiffness modes requested. The task returns two sets of results

- Element virtual energies that can be contoured or viewed as an output table.
- Eigenvalues of the stiffness matrix, which can be viewed from 'Global Results | Model Stability Summary'.

Assuming the number of low stiffness modes requested was w and high stiffness modes was h , then virtual energies for modes 1 to w represent element virtual kinetic energies whereas modes $(N-h)$ to N represent element virtual strain energies. Here N stands for the number of degrees of freedom in the model.

Once the contours are generated, it can be seen that a handful of elements have relatively large energies (compared with other elements) in either low stiffness or high stiffness modes. These are to be viewed in conjunction with the eigenvalues reported in the Model Stability Summary output. The eigenvalues corresponding to low stiffness modes (the first w eigenvalues) will often have a 'gap' in their magnitude i.e. there is an $i < w$ such that

$$\frac{\lambda_i}{\lambda_{i+1}} \gg \frac{\lambda_{i-1}}{\lambda_i}.$$

If such a gap is not found in the w smallest eigenvalues computed, the task must be modified by increasing w (say by doubling it) and executed again.

Elements with large relative virtual kinetic energies (low stiffness modes) are badly connected with the rest of the model. This can happen, for example, when beam elements are free to spin about their X axis due to insufficient restraints when moments are released or when they are connected to bars. Another common reason is when a beam is normal to the plane of a connected QUAD8 element, it does not have restraint against rotation about its axis.

Elements with large relative virtual strain energies (high stiffness modes) are ones that have disproportionately large stiffnesses compared with other elements in the model. The most common reason for this are beams with very short lengths but a large section size, which can result when a model is imported from CAD.

In both cases, the model definition in the vicinity of the elements identified needs to be examined for the presence of modeling errors. Once such errors are found and fixed, the task must be run again to ensure the condition number is below the warning threshold. Thus the analysis is an iterative procedure.

2.26 Assemblies

There are many cases where the modelling requires a single component to be modelled using many elements, but results are required for the component as a whole. Examples of this are items such as building cores or trusses where many element are used to get the required analysis accuracy but results such as axial force and bending moment are required for the component as a whole.

An [assembly](#) allow a component to considered as a stick and be defined as a collection of elements (a list), orientation information and result positions. An assembly can either be defined by topology or using an axis and position information.

When defined by topology an assembly can be either straight or curved. A curved assembly can either be defined with a third node on the arc or by a curve using Lagrange interpolation through a set of nodes. The line or curve joining the topology items defines the x axis of assembly. An orientation node defines the xy plane of the assembly, and the local z axis is normal to the x axis and xy plane.

When defined by an axis the local axes of the assembly are aligned with this axis, so a core, for example, is likely to use the Vertical axis. The extents of the assembly are then specified either directly or calculated from the elements in the assembly. The transverse offset located the x axis of the assembly relative to the x axis of the reference axis. This can be specified directly or calculated by GSA from the stiffness (EA) centroid of the elements in the assembly.

As well as defining the basic assembly it is important to specify where results are required. GSA offers a number of options for this:

- Number of points – the internal result positions are spaced evenly along the assembly.
- Spacing of points – the internal result positions are set at the specified spacing.
- By storey – the internal result positions correspond to the storeys (for Vertical axis only)
- Explicit – the internal result positions are specified directly along the x axis of the assembly.

The results for an assembly are broadly in two categories: displacements and forces. In both cases the results are calculated for each of the internal positions. At each of these positions a cutting plane is specified which is normal to the x axis of the assembly. A check is made to see which elements are cut by this plane. For displacement results the displacements of each of these elements are calculated and the assembly displacement is the average of these displacements.

$$\mathbf{u} = \frac{1}{n} \sum_{i=1}^n \mathbf{u}_i$$

The rotations are based on the average of the displacement components causing a rotation at position (y,z) on the cutting plane

$$\begin{Bmatrix} \theta_x \\ \theta_y \\ \theta_z \end{Bmatrix} = \begin{bmatrix} 0 & -z/r & y/r \\ 1/z & 0 & 0 \\ -1/y & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_x \\ u_y \\ u_z \end{Bmatrix}$$

where

$$r^2 = x^2 + y^2$$

gives a rotation of

$$\boldsymbol{\theta} = \frac{1}{n} \sum_{i=1}^n \boldsymbol{\theta}_i$$

For the force calculation the forces on each element at the cutting plane summed

$$\mathbf{f} = \sum_{i=1}^n \mathbf{f}_i$$

and for the moment calculation

$$\begin{Bmatrix} m_x \\ m_y \\ m_z \end{Bmatrix} = \begin{Bmatrix} m_{x,0} \\ m_{y,0} \\ m_{z,0} \end{Bmatrix} + \begin{bmatrix} 0 & -z/r & y/r \\ 1/z & 0 & 0 \\ -1/y & 0 & 0 \end{bmatrix} \begin{Bmatrix} f_{x,0} \\ f_{y,0} \\ f_{z,0} \end{Bmatrix}$$

where subscript 0 refers to the element cut position on the cutting plane, gives a moment of

$$\mathbf{m} = \frac{1}{n} \sum_{i=1}^n \mathbf{m}_i$$

These results are then reported or displayed in the graphics for the assembly as a whole.

Part III

3 Working with the Program

This chapter describes how to drive GSA. It is written for the user who knows what he wants to achieve but does not know how to achieve it. Other chapters explain the details of particular GSA commands and frequent reference is made to these from this chapter.

The initial section introduces [some basic concepts](#) upon which the design of GSA is based.

Following sections deal with how to drive the main types of view in GSA, namely:

[The Gateway](#)—access to Table Views, Output Views, Saved Views and Preferred Views.

[The Object Viewer](#)—display properties, display results for individual nodes, elements and members and access data defaults.

[Table Views](#)—allow tables of input data to be edited.

[Graphic Views](#)—display a graphic image of the model offering graphical editing and graphical representation of input data and results.

[Output Views](#)—display input data and results in tabular format.

Particular operations warrant their own sections:

[Sculpting](#)—graphical editing of data.

[Undoing Edits](#)—the Undo and Redo commands.

[Saved Views and Preferred Views](#)—saving Graphic View and Output View settings for re-use either with the particular model or as preferences.

3.1 Some Basic Concepts

[Preferences](#)

[Toolbars](#)

[File formats](#)

[Using the New Model and Data Generation Wizards](#)

[Cases](#)

[Analysis and Design Layers](#)

3.1.1 Preferences

In many cases the user will want to be able to have preferred settings. These settings or “Preferences” are stored between one session of GSA and the next.

Many preferences are set in the [Preferences Dialog](#) available from the “Tools | Preferences” (Ctrl +F7) menu command. For example if the user wants by default a preferred set of units other than SI. This can be changed within the “Preferred Units” dialog accessed from the “Units” button on the Miscellaneous page.

Some preferences, such as the toolbar selection and positioning and window sizes are automatically set each time the program is closed.

Default settings for new Graphic Views and Output Views are saved as preferences. Several preferred Graphic and Output Views may also be saved as preferences. Details are given in the [Working with Saved Views and Preferred Views](#) documentation.

3.1.2 Toolbars

Many of the commonly used commands are available on toolbars. With the exception of the [Assisted Input toolbar](#) all toolbars can be either docked (attached to the main frame) or floating. The toolbars can be switched on and off from the “View | Toolbars” menu command. Hovering the mouse over a toolbar button will display a small window (tooltip) with the name of the command.

Toolbars are covered in detail in the [Toolbars and Keyboard Accelerators](#) section.

3.1.3 File formats

GSA can read and write a number of file formats, however, the default file type for GSA is a GWB file.

In the Open dialog, Files of type offers options as follows:

GSA Files (*.gwb) — the default option.

GSA Files (Input data only) (*.gwb) — any results in the GWB file are ignored.

GSA Files (Selected results only) (*.gwb) — only selected results in the GWB file are read. See 'Selected Results Only', below.

Text (Tab delimited) (*.gwa, *.txt) — GSA text file, which can be edited.

Text (Comma delimited) (*.csv) — GSA text file, which can be edited.

There are also options to interact with other software through Import and Export functions. These are described in detail in the [Interaction with other software section](#).

Selected Results Only

The option to open a model with 'Selected Results Only' allows a subset of results to be opened with the model, offering faster post-processing of huge models.

Having requested to open a model with 'Selected Results Only' GSA prompts for a list of elements and cases for which results are to be read. Results are read only for the specified elements, and for nodes attached to those elements, and for analysis cases referenced by the analysis and combination cases in the specified case list. Loading is also reduced to the loads upon which the selected results depend.

Having opened a model with selected results only:

- A flag is set in the model to indicate that the model has selected results.
- The default filename for the model has '_[SELECTED_RESULTS_n]' appended.
- Lists of nodes, elements and cases are saved, named "[RESULTS_ELEMENTS]", etc. as an indication of the results that are available.
- The saving of a model with selected results is available. The model retains the 'selected results' flag.
- The results in a file that has selected results can be reduced further by, again, choosing to open with 'Selected Results Only'.
- The deletion of all results is available; you are warned that the loading is reduced from what was previously analysed.
- (The partial deletion of results is not permitted.)

3.1.4 Using the New Model and Data Generation Wizards

For users unfamiliar with GSA the [New Model Wizard](#) and [Data Generation Wizard](#) provide a quick way of generating simple structural forms.

When a new document is opened the New Model Wizard is used to set up preliminary details about the structure such as [job details](#), [structure type](#) and [units](#). The user may finish at this stage with an empty model or proceed via “Generate data for the selected structure type” to select a structural form and then generate a model based on a set of parameters defining this structure. At the end of this process a complete model is generated ready for loading to be applied.

These wizards are covered in more detail in the [Dialog Boxes and Wizards](#) section.

3.1.5 Cases

Loading and results are grouped into cases. Loads are grouped into load cases, primary results (those calculated directly) are grouped into analysis case and secondary results (combinations of primary results) are grouped into combination cases. For design there are design cases which associate a set of results with a design code.

[Cases](#) are covered in more detail in the [Program Fundamentals](#) section.

3.2 Working with the Gateway

The Gateway gives access to all the data modules that are available for setting up a GSA model and to all the output tables available for the model.

The Gateway behaves like a toolbar in that it can be displayed or hidden and, when displayed, can be docked onto an edge of the GSA window or floating anywhere on the screen. When floating it is displayed on top of any other view. When docked the part of the screen that is available for other views excludes that occupied by the Gateway. The content of the Gateway always relates to the current model. There is never more than one Gateway displayed even if there is more than one model open. (The Object Viewer behaves similarly.)

The Gateway can be opened and closed using the “View | Gateway” (Alt+O) menu command. “Gateway” is also available on the [GSA toolbar](#).

The Gateway has three tabs: “Tables” which gives access to the data editing, “Output” for viewing tabulated data and results and “Views” for opening preferred and saved Graphic and Output Views.

Top level categories can be expanded by clicking on the “+” symbol beside the name or by double clicking on the name. Clicking on the “-” symbol or double clicking on the name when expanded will close up the item. A branch in the view is fully expanded when the items have no symbol beside them.

Double clicking on an item will open the appropriate view.

More:

[Right-click menus](#)

[Tables Tab](#)

[Output Tab](#)

[Views Tab](#)

[Copy and Paste](#)

3.2.1 Right-click menus

Clicking the right mouse button when the cursor is pointing at an item in the Gateway displays a floating menu that relates to that item.

3.2.2 Tables Tab

The Tables tab in the Gateway organises the data into several categories. Not all of these categories may be visible as the list is updated dynamically, depending on the “Advanced Features” set in the “Tools | Preferences” menu command.

Against each item is reported the number of records currently specified for the data module.

The right-click menu includes the following options:

- **Delete**—delete the contents of the selected module or a branch of modules. (Only enabled when the clicked item is editable.)
- **Copy**—copy the contents of the selected module to the clipboard in GWA format.
- **Copy All**—copy the whole model to the clipboard in GWA format.
- **Paste**—paste the GWA formatted clipboard contents, overwriting the current data.
- **Paste Special**—specify, via the [GWA Import Options dialog box](#), how to paste the GWA formatted clipboard data, including the option to overwrite or append and the option to select the modules that are to be pasted.

3.2.3 Output Tab

The Output tab in the Gateway organises the output data and results into several categories. Not all of these categories may be visible as the list is updated dynamically; an item is offered only if there are data or results to display and a category is only visible if there are items offered within that category.

Some results are available in more than one Output View, for example displacements are available under “Nodal Results” as displacement at nodes or under “Beam and Spring Element Results” and “2D Element Results” as displacements at positions on elements. Results that apply to the model as a whole are reported as “Global Results”.

The right-click menu includes options to “Specify Output” and to “Open Summary”. “Specify Output” displays the [Output Settings dialog](#) with the clicked item selected, giving the opportunity to further specify the Output View settings before opening the view. “Open Summary” opens an Output View of just the summary (Maxima and Minima) of the clicked item. In the “User Modules” branch the right-click menu also gives access to the user module management dialogs.

3.2.4 Views Tab

The Views tab in the Gateway gives access to:

- New Graphic Views
- Preferred Graphic Views
- Saved Graphic Views
- New Output Views
- Preferred Output Views
- Saved Output Views
- View Lists

The right-click menus also offer commands for the management of these.

Opening a New Graphic View or New Output View simply opens a Graphic View or Output View with the default settings. (These have the same affect as giving the “View | New Graphic View” or “View | New Output View” menu commands, or the equivalent toolbar commands.)

Preferred views and saved views are offered only if these have previously been saved. An asterisk, “*”, is appended to “Views” on the tab label when some saved Graphic Views or Output Views are present in the current model.

Different right-click menus are displayed depending on whether the clicked item is a new view, preferred view, saved view or view list.

The right-click menu for new views includes the option to reset the settings to GSA default. Note that the “Tools | Reset Preferences” menu command also gives the option to reset the default Graphic View and Output View settings.

The right-click menu for preferred views includes options to rename and delete the preferred view and to “Delete All Preferred Graphic Views” or “Delete All Preferred Output Views”, respectively. Deleting all preferred views does not reset the default view settings.

The right-click menu for saved views includes options to edit, copy, rename and delete the saved view and to “Delete All Saved Graphic Views” or “Delete All Saved Output Views”, respectively.

Options to create new view lists, to edit, copy and otherwise manage existing view lists are available in the right-click menu for view lists.

Preferred views, saved views and view lists can be printed directly from the right-click menu. Refer to [Batch Printing of Views](#) for details.

Details of how to produce preferred and saved views and view lists are given in the [Working with Saved Views and Preferred Views](#) documentation.

3.2.5 Copy and Paste

The following copy and paste commands are available from the Gateway:

- **Copy** – Copy the currently selected module or the modules beneath the currently selected branch to the clipboard. (Tables and Views tabs only.)
- **Copy All** – Copy all input data modules to the clipboard. (Tables tab only.)
- **Paste** – Paste the clipboard data into the model. Data records that are identified by a record number (and often referred to by that number) such as nodes, elements and properties, overwrite existing data. Other data append to existing data.
- **Paste Special** – The [GWA Import Options dialog box](#) is displayed to allow the specification of the paste.

All of these commands operate with data in text format and GWA syntax.

3.3 Working with the Object Viewer

The Object Viewer displays properties and results for selected items. It also gives access to data defaults for the model.

The Object Viewer behaves like a toolbar in that it can be displayed or hidden and, when displayed, can be docked onto an edge of the GSA window or floating anywhere on the screen. When floating it is displayed on top of any other view. When docked the part of the screen that is available for other views excludes that occupied by the Object Viewer. The content of the Object Viewer always relates to the current model. There is never more than one Object Viewer displayed even if there is more than one model open. (The Gateway behaves similarly.)

The Object Viewer can be opened and closed using the “View | Object Viewer” (Alt+3) menu

command. “Object Viewer” is also available on the [GSA toolbar](#).

The Object Viewer has two tabs: “Properties” which displays the properties for a selected item and “Report” which displays output for a selected item.

More:

[Right-click menu](#)

[Properties Tab](#)

[Report Tab](#)

3.3.1 Right-click menu

Clicking the right mouse button when the cursor is over the Object Viewer displays a floating menu offering the following commands:

- Close Object Viewer
- Copy
- Select All
- Print

3.3.2 Properties Tab

The Properties tab displays properties for a selected item or a set of selected items.

In Graphic Views, clicking on a node, element or member when the Object Viewer is open on the Properties tab causes the properties for that item to be displayed, regardless of the current cursor mode.

Also in Graphic Views the “Graphics | Selection Properties” command displays properties in the Object Viewer relating to the current selection of nodes, elements, members or current polyline. For Beam, Bar, Tie and Strut elements, selection properties include:

- total length,
- total mass and
- total surface area.

For 2D elements, selection properties include:

- total area,
- total element mass and
- total additional mass.

For nodes, if two or three nodes are selected, selection properties include:

- distance between nodes,

and, if three nodes are selected:

- angle at nodes.

In some Table Views properties for the current record can be displayed by giving the “View | Properties” menu command. The “Properties” command is also accessible from the Table View right-click menu.

3.3.3 Report Tab

The Report tab displays results for a selected item. This is done as follows:

1. Select the required output option on the Report tab of the Object Viewer. Only those options currently available for the model are displayed.
2. Select the required case or cases in the Cases list at the top of a Graphic View.
3. In the Graphic View, click on a node, element or member.

The output is reported in the Object Viewer.

The default output option is "<displayed result>". In this setting clicking on a node, element or member in the Graphic View causes the currently displayed result in that Graphic View to be reported in the Object Viewer. The currently displayed result in the Graphic View is determined by the following precedence:

1. The contoured data, if contouring is displayed.
2. The last specified diagram, if any diagrams are displayed.
3. Node or element displacements (depending on whether a node or element is clicked), if the deformed image is displayed.

3.4 Working with Table Views

Most input data required by GSA can be entered in tables. The details of the data entered in each table are covered in the [Program Data](#) section.

Tables in GSA are similar to spreadsheets, but there are a number of significant differences. These are highlighted in detail below.

More:

[Single and multi-page tables](#)

[Tabular data entry and editing](#)

[Basic operations in tables](#)

[Selecting blocks of cells in Tables](#)

[Copying, cutting and pasting in tables](#)

[Delete, blank and insert in tables](#)

[Find, replace, go to and modify in tables](#)

[Copying to and from spreadsheets](#)

[Adjusting data display](#)

[Colour in tables](#)

3.4.1 Single and multi-page tables

The simplest type of table in GSA contains data on a single page, which scrolls horizontally as required to give a view on the complete module.

In other cases the data to be displayed is more complex or several data modules are related so it is convenient to display the data on more than one page of a table. This leads to the following options:

For more complex data it is useful to be able to have more than one view on the data in the table, with different pages providing a view on a different subset of the data. An example of this type of table is the "Nodes" table – here it is convenient to separate the node Coordinates, Constraints and Supports on to separate pages. These pages give a view of different parts of the same node record. The complete record can be viewed by selecting the All page. In tables like this there is an "All"

page to provide a view of all the data.

Another situation where multiple pages are convenient is where each record is in itself a table. In these situations each record forms a page. In these tables the user can right click on the tab to insert or delete pages or change the name of that page. An example of this type of table is the “Alignments” table, where each alignment is defined by a table of curvature as a function of chainage.

Sometimes it is convenient to be able to group related modules into a single table. An example of this is the “Nodal Loading” table. Here the “Node Loads”, “Applied Displacements” and “Settlements” are grouped together on a single table: each page of the table relates to a different module.

For the multi-page tables the pages are changed by clicking on the appropriate tab.

3.4.2 Tabular data entry and editing

The details of the data entered in each table is covered in the [Program Data](#) section. This section covers the basic navigation and use of the data tables by keyboard and mouse, and the Edit menu functions specific to tables.

Changes made in a table are immediately reflected in other tables, Output Views and Graphic Views, and vice versa.

Note that in general, data cannot be entered or edited if results exist.

More:

[Wizards](#)

[Field Types](#)

[Defaults in tables](#)

3.4.2.1 Wizards

For many of the tables there is the option of defining the data in a Wizard. The [data wizards](#) are available from the Wizard button in the [Data Options toolbar](#). The use of wizards is necessary for some of the modules where it is not otherwise possible to select or set up the required data.

3.4.2.2 Field Types

All the fields in GSA belong to one of the six following types:

- **Text**—for names and labels (e.g. [property name](#)).
- **Numeric**—for item numbers and numeric values (e.g. [topology item](#) or [node coordinate](#)).
- **Numeric/Percentage**—for entering values such as locations that can be entered as an actual value or as a percentage (e.g. position of [load on a beam](#)).
- **Selection fields**—for selecting from a list of items (e.g. [element type](#)). Depending on the “Show drop-down list in selection fields” [preference setting](#) the required item may either be selected from a drop-down list or typed in directly. In the latter case the program auto-completes with the first option that matches what has been typed.
- **Selection/Numeric fields**—for selecting from standard items or entering a value for a user defined property (e.g. an axis Local or Global from list or 1, 2, etc for user defined).
- **List**—for entry of lists (e.g. nodes, elements, etc).

3.4.2.3 Defaults in tables

The shaded cells across the top of a table contain default values, which can be changed by the user. These are placed in the current cell when the cell contents are entered as blank.

Note that some [Cell operators](#) operate with default values.

3.4.2.4 Cell operators

The following operators may be entered in cells instead of typing in values.

Copy operators

=	to copy the value from the same cell in the preceding record.
=n	to copy the value from the same cell in record n.
==	to copy the values from the same cell and all following cells in the preceding record.
==n	to copy the values from the same cell and all following cells in record n.

Increment operators

The increment operators copy the value from the same cell in the preceding record and perform an operation on that. They apply to numeric fields only.

+=f	to add f to the copied value.
-=f	to subtract f from the copied value.
*=f	to multiply the copied value by f.
/=f	to divide the copied value by f.

If the increment, f, is omitted then the default value is used as the increment.

3.4.3 Basic operations in tables

The current cell in the table is indicated by depression of the grey cells at the top and left of the table. When navigating around the table the current cell is also indicated by a bold border around the cell. When editing a cell the cursor flashes at the current position in the cell and the cell is said to be in edit mode. Basic navigation and entry of data in tables is as follows.

Navigation

Moving around the table is done by using the arrow keys, **Tab**, **Return**, mouse clicks or the “Edit | Go To” (Ctrl+G) menu command.

Ctrl+Home and **Ctrl+End** move the current cell to the first and last cell in the table, respectively.

Page Up and **Page Down** move the current cell a window-full of records up and down, respectively.

Simply navigating to a cell does not put the cell into edit mode.

Editing

The following actions change a cell to edit mode:

- Typing in the cell, to cause the existing data to be replaced by what is typed.
- Press **F2**, to highlight the existing data. Subsequent typing will replace any highlighted data.
- Press **Home** or **End**, to place the cursor at the beginning or end of the existing data, respectively.
- Clicking in the current cell, to place the cursor at the clicked position.

Having edited the contents of a cell the contents of the cell must be registered by doing one of the following:

- Press **Return** or **Tab**, moves to the next cell.
- Press the **up** or **down**, moves row.
- Press the **left** or **right** when the cursor is at the left-most or right-most positions, moves to the adjacent cell.
- **Click** on another cell.

In all cases the contents of the cell are validated and if invalid input is detected it must be corrected before moving.

The **Esc** key can be used to undo an edit.

3.4.4 Selecting blocks of cells in tables

To select all the cells containing data, use the “Edit | Select All” (Ctrl+A) menu command or click on the grey box at the very top left of the table. The selection is highlighted in inverted colours (as with any block selection)

Select records or fields (rows or columns) of cells by clicking on their headers in the grey area.

To select any particular set of cells there are three options:

- Use the “Edit | Select” menu command, which displays a dialogue box where a set of cells can be defined in terms of records and fields.
- Drag a box around them using the mouse.
- Click on the start cell of the proposed block, then click on the final cell while holding down shift.

To cancel a selection, use “Edit | Select None” menu command or click anywhere in the table.

3.4.5 Copying, cutting and pasting in tables

Any [selection of table data](#) can be cut or copied to the clipboard, and the contents of the clipboard can be pasted into a selected location in a table. The clipboard can be used to transfer data between table views, graphic views, spreadsheets and other packages.

When cutting, copying or pasting complete records the clip-board data is in [GWA format](#) and includes data for the complete object represented by the record, not just the data shown in the table.

Cut

Cut records from the document and put them on the clipboard. Complete records must be selected before cutting data. Cutting data to the clipboard replaces the contents previously stored there.

Many data records (e.g. loads) are deleted from the Table View after cutting the data. This avoids empty records and causes subsequent records to be renumbered.

Records that are not renumbered are those whose record numbers are cross-referenced to other data, and alteration would affect that other data. For example Node record numbers are not renumbered as they are used to define element topology.

Copy

Copy works in two ways depending on the selection. If partial records are selected then copy copies blocks of selected data to the clipboard as tab delimited text. If complete records are selected then the data is copied to the clipboard in GWA format, as described above.

Copying data to the clipboard replaces the contents previously stored there. Use the [Export command](#) to **save** selected data as tab delimited text.

Paste

Use the Paste command to insert a copy of the clipboard contents at the currently selected location. Blocks of data are inserted if acceptable, see below.

If a complete record is pasted and the clip-board data is in GWA format the whole object will be overwritten, as described above. Note that item numbers in the GWA data are ignored when pasting to tables; the records are written into the table consecutively starting from the first selected record in the table. For example, if GWA records for nodes 1, 3, and 5 are on the clip-board then pasting into an empty Nodes table will result in nodes 1, 2, and 3 in the table. (Whereas [pasting into the Gateway](#) is sensitive of the item number. So, pasting nodes 1, 3, and 5 into the Gateway will paste nodes 1, 3 and 5 into the model, overwriting these if they previously existed.)

If you highlight a destination block to paste data to, everything in the destination block will be overwritten on pasting. The size and form of the destination block must be the same as that for data being pasted from.

Pasting a block of data when no cells are selected results in the block being pasted starting at the current cell. In this case:

- If pasting a block to an existing record, GSA gives the option to overwrite the existing data or to insert the new data. If insert is chosen, data is inserted before the record containing the current cell.
- If pasting a block to a cell beyond the existing data, it is pasted appended.

Note that for any of the paste operations, if partial record data is pasted resulting in blank cells, the blank cells will be set to default values.

Other methods

Note that [Cell operators](#) may also be used to copy values from existing cells.

3.4.6 Delete, blank and insert in tables

Complete records can be inserted, deleted (with renumbering of subsequent records) or blanked (without renumbering).

To **delete** a selected record(s), use “Edit | Delete” (Del) menu command or the Delete key. The action of the delete key will depend on the type of table. In tables in which no blank records are permitted, deleted records are removed and the table is closed up. In tables in which blank records are permitted, delete will clear or blank the selected records leaving subsequent records unchanged. If all the selected records are already blank, the records are deleted to result in subsequent records being renumbered.

To **insert** a blank record(s) above the currently selected cell or record, use the “Edit | Insert” (Ins) menu command. This inserts a record at a time and causes subsequent records to be renumbered.

3.4.7 Find, replace, go to and modify in tables

Find

Use the “Edit | Find” (Ctrl+F) menu command or the “Find” button on the [Data Options toolbar](#) to find specified text or numbers in a Table. The Find Dialog Box appears:

- Specify the exact entry you wish to find. This can be made case sensitive by checking the “Match case” box.
- Choose to search the “whole” table or a just a “selection”. Note — if a selection is required, it must be highlighted before choosing the find command.
- Choose to search “up” or “down” from the current cell. Once the end of the table is reached, the search reverts back to the beginning of the table.

- Click the “Find Next” button to find the first matching entry, and again for each subsequent matching entry. GSA displays a warning message if the specified text is not found.

Go to

To find a specific **record** or **page**, use the “Edit | Go To” (Ctrl+G) menu command or the “Go To” button on the [Data Options toolbar](#). The record/page can be specified in several ways:

n	go to record/page n
$+n$	go forward by n records
$-n$	go back by n records
$+$	go to next non-blank record
$-$	go to previous non-blank record
$*$	go to highest non-blank record/page

Replace

Use the “Edit | Replace” (Ctrl+H) menu command or the “Go To” button on the [Data Options toolbar](#) to perform a search, as for Find, and also replace the specified text or numbers. The [Replace Dialog Box](#) appears. This operates as for Find dialog with additions:

- Specify the new entry you wish to replace the existing entry.
- Upon finding a matching entry, click the “replace” button, or you can choose to “replace all” matching entries.

Modify

Use the “Edit | Modify” (Ctrl+M) menu command or the Modify button on the [Data Options toolbar](#) to modify numerical cell entries. Cells containing text remain unmodified. The [Table View Modify dialog box](#) appears. The modifications specified here are applied to every cell value in the “selection” or the “whole” table view. Specify the “modify by” value to be used in one of the following ways:

- **Add**—add the value in the Modify By box.
- **Factor**—factor by the value in the box.
- **Power**—raise to the power of the value in the box.
- **Absolute**—modify the selection to the absolute value (the modify by is ignored in this case).

3.4.8 Copying to and from spreadsheets

For many purposes the preparation of some part of the model or loading, or post-processing of results will require the use of spreadsheets.

To **save** data for use in spreadsheets, use the [Export command](#).

Information can be cut/copied from tables in GSA and pasted into spreadsheets and vice versa. Also results can be copied from Output Views and pasted into spreadsheets. Data copied from GSA or spreadsheets is held as Tab delimited text.

If the user intends to create data in a spreadsheet, it is recommended that a dummy record be created in GSA and copied to form a “template” in the spreadsheet.

3.4.9 Adjusting data display

There are a number of options for adjusting the display of data in tables. These are available from the “Window | Settings” menu and from the [Data Options toolbar](#).

Font

The font that is used in the table can be selected. This gives a standard Font selection dialog.

Units

This is available in tables in which units are used. Some of the data may be entered in one unit, the units changed and subsequent data entered in a different set of units. All data in GSA is stored in SI units and converted to the requested units upon display.

Axes

This option is only available for the nodes table and allows the coordinates to be entered or viewed in a different sets of coordinates. This allows, for example, some of the nodes can be defined in a Cartesian coordinate system and other in a cylindrical system.

Size Columns to Fit, Size Rows to Fit

The widths or heights of table columns and rows on the screen can be reduced to fit the displayed text.

3.4.10 Colour In tables

The default colours in which entities are displayed in [Graphic Views](#) can be overridden by explicitly defined colour. Explicit colour can be assigned to nodes, elements, members, materials, properties, and analysis stages.

In tables, the rows for objects that have explicit colour definition are highlighted with a colour-wash representation of the assigned colour.

Assignment of explicit colours in tables is by use of the 'Edit | **Modify Colour**' menu command, having first selected the records that are to be modified. 'Modify Colour' is also available on the right-click menu. This opens the Select Colour dialog. The selected colour is indicated by a colour swatch, and the derived colour-wash, by the partially hidden colour swatch. Setting the colour to 'undefined' unsets the explicit colour assignment.

Note that explicit colour assignment can also be applied via [Graphic Views](#).

In the element and member tables there is an option to check for and highlight incomplete elements. These are entities that refer to non-existent nodes or properties, or where topology items are repeated. Non-existent items are highlighted in red and repeated items are highlighted in blue.

3.5 Working with Graphic Views

Graphic Views may be used to view a graphical representation of the structure. Labels may be applied to identify the entities that make up the model. Diagrams and contouring may be applied to gain an understanding of the model and its behaviour.

A number of sculpt tools are provided to enable the model to be edited graphically. [Sculpting](#) is documented in a separate section.

Several Graphic Views of a model may be open at a time.

A new Graphic View may be opened using the "View | New Graphic View" (Alt+1) menu command. "New Graphic View" is also available on the [GSA toolbar](#).

The "Graphics" button on the [Assisted Input toolbar](#) and, indeed, any of the enabled graphics toolbar buttons may be used to open a Graphic View when one is not already open. When at least one Graphic View is already open, giving these commands causes the most recently accessed Graphic View to be made the current view.

By default the rendering of the graphics for a Graphic View is carried out in a separate thread, to provide better performance and to enable the rendering of the graphics to be interrupted. It is also possible to open a Graphic View for which the rendering is processed in the same thread as the main GSA application, by using the "View | Misc. View Options | New Graphic View (main thread)" menu command.

Default view settings, preferred views and saved views are available for Graphic Views. Refer to [Working with Saved Views and Preferred Views](#) for details and to [Working with the Gateway](#) for accessing these.

More:

[Graphic Settings](#)

[Basic orientation of the image](#)

[Scaling the image and zooming](#)

[Advanced orientation of the image](#)

[Identifying what is to be drawn](#)

[Current grid](#)

[Selection sets](#)

[Polylines in Graphic Views](#)

[Adornments](#)

[Shrinking elements](#)

[Highlighting element edges](#)

[Highlight coincident nodes](#)

[Highlight coincident elements](#)

[Resetting the display](#)

[Switch layer](#)

[Right-click menus](#)

[Graphic Fonts and Styles](#)

[Animation](#)

[Printing from Graphic Views](#)

[Output of the graphic image](#)

3.5.1 Graphic Settings

All of the settings that specify the content of a Graphic View can be set either in the [Graphic Settings dialog box](#) or in dialog boxes accessible from this. The Graphic Settings dialog box can be accessed by giving the “Graphics | Graphic Settings” (Ctrl+W) menu command or the “Wizard” command on the [Data Options toolbar](#). Double-clicking on the background of a Graphic View opens the Graphic Settings dialog for that view (whereas double-clicking on an entity opens the definition dialog for that entity).

When a new Graphic View is opened the [default view settings](#) are adopted. This is true except when there exists a [saved Graphic View](#) named “startup” (case insensitive) in which case this is the view that is opened when the model is opened.

Changes to view settings may be undone and redone using the “View | Undo View” (Ctrl+Alt+Z) and “View | Redo View” (Ctrl+Alt+Y) menu commands. These commands are also available on the [Standard toolbar](#).

View settings for the current view may be applied to another Graphic View by clicking the “View settings painter” on the [Standard toolbar](#) to grab the current settings and then clicking in the other Graphic View. Opening a new Graphic View when the settings have been grabbed applies the grabbed settings to the new view. Click the “View settings painter” again or press <Esc> to switch off the option without applying grabbed settings.

View settings may be copied to the clipboard from within the Graphic Settings dialog box. View

settings on the clipboard may be pasted into the view (i.e. applied to the view) either using “Edit | Paste” (Ctrl+V) or from within the Graphic Settings dialog box. Note that “Edit | Copy” copies either the actual content of the view (see [Copying the graphic image to the clipboard](#)) or the current selection (see [Copying sets to the clipboard](#)), not the view settings.

There are often other ways of editing the settings, as described below.

3.5.2 Basic orientation of the image

The initial view of the model depends on the dimensions of the model. Typically, vertically planar models are displayed in elevation and other models are displayed in plan.

More:

[Standard views](#)

[Changing the orientation of the image](#)

[Regenerating the image](#)

3.5.2.1 Standard views

The following commands (and their associated accelerator keys) are available from the “Graphics | Orientation” menu to provide standard views of the image. The commands are also available from the “Orientation” toolbar.

- **Plan (P).**
- **X Elevation (X).**
- **Y Elevation (Y).**
- **Isometric (I).**
- **Skew View (K).** This is a perspective projection.
- **Ground View (G).** This is a perspective projection.
- **Reverse Plan (Alt+P).**
- **Reverse X Elevation (Alt+X).**
- **Reverse Y Elevation (Alt+Y).**

In all cases a [scale to fit](#) is also carried out.

3.5.2.2 Changing the orientation of the image

The following methods may be used to rotate the image.

Dragging—Set the [cursor mode](#) to Rotate and drag the mouse. The sensitivity of the mouse movement may be adjusted in the [Graphics Preferences](#).

Shift+Drag with the mouse wheel (or middle button) held down (in any cursor mode) operates as dragging in the Rotate cursor mode.

Arrow Keys—Each keystroke rotates the image about the object point by the amount currently set in the [Graphics Preferences](#). These commands may also be found in the “Graphics | Orientation | Rotate” menu, for the sake of completeness.

Explicitly—The orientation may be specified in terms of longitude and latitude in the [Orientation Settings dialog box](#).

In all cases the rotations occur about the [object point](#).

3.5.2.3 Regenerating the image

The “Graphics | Orientation | Regenerate” (F5) menu command (also on the [Orientation toolbar](#)) regenerates and redraws the image without changing any parameters that specify the image.

Any operation that causes information to be added or removed from the image will also cause the image to be regenerated. The image is always regenerated prior to printing.

Impact of zooming on symbols, diagrams and contours

Many of the operations dealing with the orientation, scaling and projection of the image do so without regenerating the display. This makes the display of the image much quicker and enables these operations to happen “live” on the full image. A consequence of this is that if, say, dots are drawn at nodes when the image is zoomed the dots are zoomed by the same amount as the structure. The same occurs to diagrams and contours, invalidating reported data scales. In this circumstance a message is reported in the legend to indicate that data scales are unreliable.

Regenerating causes the image to be re-established from the data and view settings to result in all items being drawn at the intended scale.

3.5.3 Scaling the image and zooming

The image of the model is drawn in a Graphic View in a specified orientation, at a specified scale, centred at a specified position. This section deals with scaling and positioning of the image, including zooming and panning.

Note that many operations that change the structure scale reproduce the display without regenerating the image. This can invalidate the reported scale for diagrams and contours. See [Regenerating the image](#) for more details.

More:

[Scaling to fit](#)

[Zooming and panning](#)

[Setting the scale explicitly](#)

[Scaling for printed output \(and changed window sizes\)](#)

3.5.3.1 Scaling to fit

A scale to fit involves an automatic adjustment of the scale, object point, mid-point and eye distance to result in the image fitting in the current view.

Scaling to fit occurs when any [standard view](#) is selected, when the “Graphics | Orientation | Scale to fit” (Ctrl+Home) menu command is given and when “Do scale to fit” is selected in the [Graphic Settings dialog box](#). “Scale to fit” is also available on the [Orientation toolbar](#).

First the extents of the image are determined. These are based on the undeformed state of the part of the model currently being drawn.

The object point is set to the centre of the extents.

The mid-point is set to 0,0,0 to result in the object point being drawn at the centre of the view.

A scale is calculated that will result in the extents, in their current orientation, fitting within the Auto-scale margin as set in the [Graphics Preferences](#). The image is [regenerated](#) if the ‘Regenerate graphics on standard view’ preference is set in the [Graphics Preferences](#).

Scaling may be set to be to an engineering scale by toggling the “Graphics | Orientation | Engineering Scale” menu command and in the [Graphic Settings dialog box](#). When set, the calculated scale is rounded up to the next engineering scale. If the view is an isometric view then it is the isometric scale that is set at an engineering scale.

An eye distance is arbitrarily set. This is five times a dimension of the image, where the dimension

is the near extent to the far extent or, if that is zero, the left to right or the top to bottom.

3.5.3.2 Zooming and panning

In terms of the parameters that define a view, zooming and panning affect the structure scale and the mid-point. These parameters may be adjusted explicitly via the [Orientation Settings dialog box](#). More commonly, though, higher level operations may be used to zoom or pan.

Zoom Cursor Mode—When in the Zoom [cursor mode](#):

- **Dragging a rectangle** in the window causes the image to be zoomed and panned to result in that rectangle filling the Window. Only when the aspect ratios of the rectangle and window are identical will this be exact; otherwise the scale is such that *at least* all of the rectangle remains visible.
- **Click** pans and zooms in. The image is panned to result in the point clicked on being at the mid-point of the view. The structure scale is multiplied by (zoom factor) where the zoom factor is a [Graphics Preferences](#).
- **Ctrl+Click** pans and zooms out. The image is panned to result in the point clicked on being at the mid-point of the view. The structure scale is multiplied by (1 / zoom factor) where the zoom factor is a [Graphics Preferences](#).
- **Shift+Click** pans. The image is panned to result in the point clicked on being at the mid-point of the view.

Dragging—When in the Rotate [cursor mode](#):

- **Ctrl+Drag** up and down zooms in and out respectively. The sensitivity is governed by the zoom increment mouse sensitivity [Graphics Preferences](#).
- **Shift+Drag** pans the image. The sensitivity is governed by the pan increment mouse sensitivity [Graphics Preferences](#).

Intellimouse wheel—In any [cursor mode](#):

- **Rolling the mouse wheel** forwards and backwards zooms in and out respectively. Each notch on the mouse wheel is equivalent to one Zoom In or Zoom Out command. If when rolling the mouse wheel the cursor is over the Graphic View then the zoom occurs pinned at the cursor position.
- **Drag with the mouse wheel** (or middle button) held down pans the image. The sensitivity is as for Shift+Drag, under Dragging above.
- **Ctrl+Drag with the mouse wheel** (or middle button) held down operates as Ctrl+Drag in the Rotate cursor mode.

Menu commands—The following commands (and their associated accelerator keys) are available from the “Graphics | Orientation” menu to allow zooming and panning.

- **Zoom In** (Ctrl+Up). Multiplies the structure scale by (zoom factor) where the zoom factor is a [Graphics Preferences](#).
- **Zoom Out** (Ctrl+Dn). Multiplies the structure scale by (1 / zoom factor).
- **Reset Pan** (Shift+Home). Sets the mid-point to (0, 0, 0).
- **Pan Right** (Shift+Rt). Adjusts the mid-point to result in the picture shifting left by $1/16$ of the picture width.
- **Pan Left** (Shift+Lft). Adjusts the mid-point to result in the picture shifting right by $1/16$ of the picture width.
- **Pan Up** (Shift+Up). Adjusts the mid-point to result in the picture shifting down by $1/16$ of the picture height.

- **Pan Down** (Shft+Down). Adjusts the mid-point to result in the picture shifting up by $\frac{1}{16}$ of the picture height.

Note that [scaling to fit](#) effectively resets any zooming and panning.

Note also that these operations do not affect the object point so, for example, rotations that occur after a zoom or pan may not have the desired affect unless the [object point](#) is adjusted.

3.5.3.3 Setting the scale explicitly

The scale may be explicitly defined in the [Graphic Settings dialog box](#). Note that when “Lock scale” is checked in this dialog box the specified scale is preserved when the window is resized and when the image is printed.

3.5.3.4 Scaling for printed output (and changed window sizes)

There are various ways of specifying the structure scale, as discussed above. Broadly these fall into three categories, namely by scaling to fit, by zooming and by specifying the scale explicitly. What happens to the scale when producing printed graphical output or whenever the picture size changes depends on how the current scale was specified, as follows.

Was scaled to fit—The image is scaled to fit the new picture size.

Was by zoom (or the model was rotated since a scale to fit)—The image is scaled to result in at least the extents previously shown being visible in the new picture. More will be visible in one direction if the aspect ratio of the new and old picture sizes are not the same.

Was explicitly defined and locked—The scale is not changed.

This default behaviour may be over-ridden by specifying the structure scale for the printed image in the [Graphic Settings dialog box](#).

3.5.4 Advanced orientation of the image

[Adjusting the object point](#)

[Orthographic and perspective projections](#)

[Adjusting the eye distance](#)

3.5.4.1 Adjusting the object point

The object point is the point in global space about which rotations occur.

The object point may be adjusted as follows.

Clicking—Set the [cursor mode](#) to Rotate and Ctrl+Click on the node or [grid point](#) at which you want the object point to be set.

Node Selection—[Select](#) one node and give the 'Graphics | Orientation | Rotate | Rotate About Selected Node' command. 'Rotate About Selected Node' is also available on the [right-click menu](#) when in the Select Node cursor mode.

Explicitly—Specify the coordinates of the object point in the [Orientation Settings dialog box](#).

Note that [scaling to fit](#) resets the object point to the centre of the cuboid surrounding the extents of the currently drawn part of the model.

3.5.4.2 Orthographic and perspective projections

The view may be toggled between orthographic and perspective projections using the “Graphics | Orientation | Perspective Proj.” (Ctrl+Alt+P) menu command.

If the eye distance has not been set when a perspective projection is requested then a default eye distance is calculated as when the image is [scaled to fit](#).

Note that some [standard views](#) are in perspective projection.

3.5.4.3 Adjusting the eye distance

The eye distance may be adjusted as follows.

Dragging—In the Rotate [cursor mode](#) Ctrl+Drag+Lft and +Rt to move the eye distance out and in respectively. The sensitivity is governed by the eye distance increment mouse sensitivity [Graphics Preferences](#).

Ctrl+Drag with the mouse wheel (or middle button) held down (in any cursor mode) operates as Ctrl+Drag in the Rotate cursor mode.

Menu commands—The following commands (and their associated accelerator keys) are available from the “Graphics | Orientation” menu to adjust the eye distance.

- **Eye Distance In** (Ctrl+Rt). Multiplies the eye distance by (eye distance increment) where the eye distance increment is a [Graphics Preferences](#).
- **Eye Distance Out** (Ctrl+Lft). Multiplies the eye distance by (1 / eye distance increment).
- **Explicitly**—Specify the eye distance in the [Orientation Settings dialog box](#).

The program imposes a minimum eye distance of 1mm.

Note that [scaling to fit](#) resets the eye distance.

3.5.4.4 Orientation about non-global axes

By default, orientation of the model in Graphic Views is with respect to the global axis set. So, for example, a plan is a view along the negative global Z, viewed with the global Y up the “page” and the global X to the right, and any rotation caused by sideways movement of the mouse (or adjustment to the [Orientation Settings](#) longitude) results in rotation of the model about the global Z axis. Giving the “Graphics | Orientation | Orient About Grid Axes” menu command (also available on the [Sculpt toolbar](#)) causes orientation operations to be carried out with respect to the axes of the [current grid](#) instead of the global axes. The axes about which orientation occurs are referred to as the orientation axes.

When non-global orientation axes are set the orientation axes are drawn dashed alongside the global axes in the image.

Having given the “Orient About Grid Axes” command this remains set until the command is given again to switch it off. While set, the orientation axes are changed each time the [current grid is redefined](#).

Changing the orientation axes does not affect the current longitude and latitude settings. Therefore if the view is a plan before changing the orientation axes it will remain a plan after, but the plan will be with respect to the new orientation axes.

[Saving a view](#) with non-global orientation axes set causes the axes to be saved with the view; – not a reference to the current grid. Thus opening the saved view will result in the model being oriented as saved and not adjusted to account for any change that may have occurred to the current grid.

3.5.5 Identifying what is to be drawn

The part of the model that is drawn in a Graphic View may be specified in two ways: by volumes and by lists. Both of these methods are extremely powerful ways of breaking down a large model into more manageable or more presentable chunks. The volume method is better for situations where the part or parts of the model that are to be included or excluded can conveniently be identified graphically; the list method makes use of the [list syntax](#) to identify the drawn part of the model.

The two methods may be used independently or in combination.

Note that the [scale to fit](#) operation sets the extents of the drawn image on the basis of the current volumes and lists. However, simply specifying a volume or list does not cause the extents or, indeed, object point to be adjusted.

Note that “Reset to All Entities” (or “All” in the “[Lists](#)” toolbar) resets both the volumes and entities lists to result in the whole model being drawn.

The “Graphics | Display | Hide | Hide Elements” command temporarily switches the display of elements off. Elements may then be un-hidden by giving the “Hide Elements” command again. “Hide Elements” is also available on the right-click menu. “Hide” commands are also available for members, lines areas and regions.

The “Graphics | Display | Display Excluded Entities Faint (Ctrl+Alt+F)” command causes entities excluded from display, whether by exclusion by volume or display list, are drawn faint, rather than not at all. (Auto-scaling ignores excluded entities regardless of this setting.)

More:

[Volume clipping](#)

[Entity lists](#)

3.5.5.1 Volume clipping

When identifying the drawn entities by the volume method the mouse is dragged to form a rectangle which specifies the part of the model that is to be drawn. Since a Graphic View is a 2D representation of 3D space a volume should be considered as a rectangular prism in space.

A volume is defined graphically by dragging the mouse when the [cursor mode](#) is set to “Volume”. The view may be in any orientation at the time the volume is specified since part of the definition of the volume is an axis set which is constructed internally, based on the orientation at the time of forming the volume.

When forming a volume by dragging the mouse, elements within the volume are **included**. When Ctrl-dragging the mouse, elements within the volume are **excluded**. Whether an element is within a volume or not is determined by the current “Reference point of selection items” setting in the [Sculpting Preferences](#).

Several volumes may be defined at once, — at different orientations, — some inclusive, some exclusive. The included entities are the sub-set of entities included in all the volumes.

A volume is “undone” by clicking in the Graphic View when the cursor mode is set to Volume. Each click removes the most recently specified volume. Shift+Click removes all specified volumes for the current Graphic View.

3.5.5.2 Entity lists

When identifying the drawn entities by the list method a list of entities is specified to identify the part of the model that is to be included. The list is specified on the “[Lists](#)” toolbar for the view by first setting the Display Option to the required entity type, whether Nodes, Elements or Members. The list of entities is then specified in the Display List. The full [GSA list syntax](#) is available when specifying the list.

When the list is a list of elements and the Analysis Layer is being displayed or when the list is a list of members and the Design Layer is being displayed, the list simply identifies the elements, or members, that are to be included.

When the list is a list of elements and the Design Layer is being displayed, those members that are made up of an element in the list are included.

When the list is a list of members and the Analysis Layer is being displayed, those elements that are part of a member in the list are included.

When the list is a list of nodes then those elements or members attached to nodes in the list are included.

When an entity list is set to "<current selection>" that entity list is set to the current [selection set](#) for that entity type. If the current selection is empty then "All" is assumed. (Unlike in Output Views, once acted upon the "<current selection>" setting is forgotten about and the list is set to the contents of the model set.)

When **Pin Display** is checked, referencing a grid plane in the list of elements on the "Lists" toolbar also changes the current grid to that grid plane (and vice versa).

Draw Selection command

Another way of specifying the list of entities to be drawn is by using the "Edit | Draw Selection" menu command. "Draw Selection" is also available on the [right-click menu](#) that is displayed when the [cursor mode](#) is set to Select Nodes or Select Elements. The procedure is as follows.

1. Set the [cursor mode](#) to Select Nodes or Select Elements.
2. Select the nodes or elements to be drawn.
3. Give the "Draw Selection" command.

The entity type and list are then set according to the current selection.

Omit From Display cursor mode

The Omit From Display [cursor mode](#) allows entities to be removed from display by selection. The behaviour of the Omit cursor mode is as follows.

- **Click** on element or member omits that entity and all related entities, depending on the current [Colour Elements](#) method.
 - **Colour by type** - if the selected entity has been assigned a colour then all entities that have that colour assigned are omitted, otherwise all entities that are of the same type and do not have a colour assigned are omitted.
 - **Colour by property** - all entities that reference the same property as the selected entity are omitted.
 - **Colour by group** - all entities that are in the same group as the selected entity are omitted.
 - **Colour by material** - all entities that reference the same material as the selected entity are omitted.
 - **Colour by initial stage** - all entities that first appear in the same analysis stage as the selected entity are omitted.

(Note that [keyboard accelerators](#) may be used to switch the Colour Elements method.)

- **Shft-click** or **Shft-drag** omits the selected entities, regardless of the current Colour Elements method.
- **Ctrl-click** undoes the last omission.*
- **Ctrl-Shft-click** undoes the all omissions.*

* Entities are removed from the display by appending a 'not' clause to the entity display list. Omissions are undone by removing trailing 'not' clauses that seem to have been appended by the

omit cursor mode.

3.5.6 Current grid

The current grid is a plane of grid points or grid lines that may be displayed in a Graphic View; it does not form part of the model but it may be used in conjunction with various [sculpt commands](#) to edit the model. Coordinates are reported with respect to current grid axes.

The current grid is defined by reference to a [grid plane](#), a [grid layout](#) and a set of [grid lines](#).

When displayed in a Graphic View the current grid is represented as grid points or grid lines on the current grid plane. Grid points are displayed on the current grid plane at the spacing specified by the current grid layout. Grid lines, which, by definition, are in the global XY plane, are displayed projected onto the current grid plane, but not at all if the current grid plane is vertical.

More:

[Defining and adjusting the current grid](#)

[How to use the grid](#)

[Grid coordinates and snapping to grid points](#)

3.5.6.1 Defining and adjusting the current grid

The current grid is a reference to a grid plane. The current grid can be defined in the [Current Grid Definition](#) dialog box, accessible from the “Define Current Grid” (Ctrl+Alt+W) menu command on the “Edit” and “Sculpt” menus and on the Graphic View right-click menu.

The current grid plane may also be set on the “[Lists](#)” toolbar for the view by first setting the Display Option to “Cur. Grid”. The current grid is then displayed in the Display List and the available grid planes are offered in the drop-down list. When **Pin Display** is checked, setting the current grid on the “Lists” toolbar also sets the list of elements to those in the grid plane (and vice versa).

A quick way of switching the current grid to one at the elevation of a particular node is to right-click on the node and give the “Set Current Grid to This” command on the right-click menu. The current grid is set to a grid on which the node lies, first looking for one parallel to the existing current grid, then for any. If none exists then the program offers to create one parallel to the existing current grid.

3.5.6.2 How to use the grid

The grid can be switched on and off by giving the “Sculpt | Draw Grid” (Ctrl+Alt+G) menu command (also available on the [Sculpt toolbar](#)).

The “Draw Grid” command toggles between displaying grid lines, grid points and no grid, but grid lines are only included in the toggle if grid lines have been specified and grid points are not included if the current grid layout has been disabled in the [Current Grid Definition](#) dialog box.

Various [sculpt operations](#) make use of the grid.

3.5.6.3 Grid coordinates and snapping to grid points

The coordinates reported in the status bar are the coordinates of the cursor position on the current grid. These are referred to as the grid coordinates. The grid coordinates are reported in the [current grid axes](#) (and global axes, if these differ).

Snap to grid may be set by giving the “Sculpt | Snap to Grid Points” (Ctrl+Alt+S) menu command (also available on the [Sculpt toolbar](#)). When this is set on, the grid coordinates are the coordinates of the nearest grid point or intersection of grid lines on the current grid. Otherwise, the accuracy of the grid coordinates is based on the screen resolution and the structure scale.

Note that when the construction grid is normal to the picture plane then no coordinates are reported

in the status bar.

3.5.7 Selection sets

A set is a collection of nodes, elements, members or grid points. Sets are only displayed graphically and may only be specified graphically. There is no facility to save sets. However there are facilities to translate between sets and [lists](#). Many [sculpt operations](#) operate on sets. Sets may also be used to identify the part of the model that is to be [adorned](#) with labels and diagrams etc.

More:

[The model sets](#)

[Use of selection sets](#)

[Forming selection sets](#)

[Interaction of sets with lists](#)

[Finding entities](#)

[Copying sets to the clipboard](#)

3.5.7.1 The model sets

There are four selection sets established for a model at the time a model is opened in GSA, referred to as the model sets. These are sets of nodes, elements, members and grid points. The model sets are initially empty.

The contents of the model sets may be edited from any Graphic View when the related selection [cursor mode](#) is current. The selected items in a model set are highlighted on the image; when not in the respective selection cursor mode the highlighting is greyed to indicate that the selection is disabled. Reverting to that selection mode enables the selection.

Since the model sets are “owned” by the model and not the particular Graphic View and since the cursor mode is set for all Graphic Views and not just the current one, the highlighting of the model sets occurs on all open Graphic Views simultaneously.

3.5.7.2 Use of selection sets

Many [sculpt commands](#) operate on model sets.

It is possible to [adorn](#) just items in a model set with labels, diagrams and contouring.

[Annotation](#) of diagrams and contouring can be applied to a set. In this case it is not a model set that is used but a set associated with the particular diagram or contour display. The set is automatically configured to be of the type (node or element etc.) appropriate for the data being annotated.

3.5.7.3 Forming selection sets

Sets are formed using the mouse in an appropriate Graphic View [cursor mode](#).

The functionality of the mouse is as follows.

- **Clicking** or **dragging** empties the set, unless the [Append to selection when click is unmodified](#) preference is set, in which case the previous selection is not emptied. Items identified by the click or drag are selected.
- **Ctrl+Clicking** or **Ctrl+Dragging** toggles the selection state of the identified items, unless the [Append to selection when click is unmodified](#) preference is set, in which case the identified items are de-selected. (The previous selection is not emptied.)
- **Shft+Clicking** or **Shft+Dragging** selects the identified items regardless of whether they were

previously selected. (The previous selection is not emptied.)

Right-clicking on an entity displays a [right-click menu](#) that includes the option to select that entity. For elements, members and areas the right-click menu also includes options to select all entities that reference the same property, the same material, and that are in the same group. The previous selection is replaced unless the [Append to selection when click is unmodified](#) preference is set.

The following menu commands are also available.

- “Edit | Select All” (Ctrl+A)
- “Edit | Select None” (Esc) – This clears the selection for the current selection mode, but if there are no items selected for the current selection mode then it clears all other selections.
- “Edit | Invert Selection”
- “Edit | Select String” – When in the Select Elements cursor mode this selects a string of elements attached to and aligned with any currently selected elements. When in the Select Nodes cursor mode nodes along the string of elements are selected. The [straightness tolerance](#) preference is used to determine whether an element is aligned, allowing this command to be used to select curved strings of elements and nodes.
- “Edit | Select Highest Coincident” – When in the Select Nodes or Select Elements or Select Lines cursor modes this deselects all but the highest numbered coincident items in the current selection. E.g. If nodes 1 to 30 are selected and nodes 1 and 11 are coincident and nodes 2, 12 and 22 are coincident then giving this command will result in only nodes 11 and 22 being selected. This command is useful for deleting coincident nodes and elements (though note that nodes may not be deleted graphically when they are referred to by elements). A check for common topological connectivity is performed for elements and lines. (See also “Select Overlapping” below, which performs a rigorous geometric check for overlapping entities.)
- “Edit | Select Overlapping” – When in the Select Elements or Select Lines cursor modes this selects all entities that completely or partially overlap one another. A rigorous geometric check is performed, which may take a while for large models. (See also “Select Highest Coincident” above, which simply checks the topological connectivity of elements and lines.)
- “Edit | Select Close To Vertical 1D Elements” – When in the Select Elements cursor mode this selects all 1D elements that are within 1° of vertical but are not sufficiently close to vertical to be deemed vertical for the local element axes definition.
- “Edit | Select Out-Of-Range” – Enabled only when contours are displayed, this command selects all entities for which the contoured data is out of the range of the current contour settings. E.g. If nodal Uz displacements are being contoured and the lowest contour value is set to -10mm and the highest to 0mm then “Select Out-Of-Range” will select all nodes for which the Uz displacement is less than -10mm or greater than 0mm.

Entity selection can be by **selection within a polyline**. The procedure is as follows:

- In the Polyline [cursor](#) mode, [form a Polyline](#).
- Switch to the required selection cursor mode ('Select Nodes' or 'Select Elements', etc.) cursor mode. All entities within the polyline (as viewed) are appended to the current selection set for that entity type.
- When the [reference point of selection items](#) is set to **Drag to right — all; drag to left — any part**, a polyline specified in the clockwise direction (as viewed) selects all entities wholly within the polyline; whereas an anti-clockwise polyline selects entities within or partly within the polyline.

In a graphic view the Edit menu offers a number of options for selecting entities. The particular options offered depend on the currently selected entity type.

- “Edit | Select By List” – This allows a list to be selected and copied to the selection. The list can

be a saved list, referred to by name, or explicitly defined. This latter option is a useful method for showing the location of particular items. If entities are already selected the dialog offers the option of appending the list data to the current selection.

- “Edit | Select By Criteria” – This allows a set of criteria to be selected, so for example it is possible to select all the vertical elements that are blue. The rule applied can be either any of the criteria should be met or that all of the criteria should be met. If entities are already selected the dialog offers the option of appending to the current selection.
- “Edit | Select By Name” – This allows the construction of a selection set based on entity names. The match can either be a simple match or for more complex selections regular expressions can be used. For example “^roof.*” will match all names that begin with roof, or “^roof[0-9]” will match all names that are of the form roof followed by a digit. More details of regular expressions can be found online, for example www.regular-expressions.info. If entities are already selected the dialog offers the option of appending to the current selection.

Note also the scope for interaction with [lists](#) and the [clipboard](#).

3.5.7.4 Interaction of sets with lists

When in a selection mode the “Edit | Save Selection As List” menu command saves the current selection set to the Lists module as a list of the appropriate type. The list formed is a simple list made up of item numbers and ranges of numbers; no attempt is made to recognise, for example, patterns in the element properties or groups of selected elements.

Also, when in a selection mode the “Edit | Select By List” menu command allows a list to be appended to the current selection set. The list can be a saved list, referred to by name, or explicitly defined. This latter option is a useful method for showing the location of particular items.

Both “Save Selection As List” and “Select By List” are also available on the Graphic View right-click menu when in a selection mode.

3.5.7.5 Finding entities

The [interaction of sets with lists](#) offers a method for finding entities that are described in list syntax to result in those entities being selected in the Graphic View. The “Edit | Find” menu command combines the various steps into one. Find is also available on the [Data Options toolbar](#).

3.5.7.6 Copying sets to the clipboard

When the current [cursor mode](#) is a selection mode the “Edit | Copy” (Ctrl+C) menu command copies the current selection set to the clipboard in [GSA list format](#). This list may be pasted anywhere where a GSA list is expected. The list formed is a simple list made up of item numbers and ranges of numbers; no attempt is made to recognise, for example, patterns in the element properties or groups of selected elements. When the ‘[Copy lists expanded](#)’ preference is set then ranges are expanded to separate items.

Also, when in a selection mode the “Edit | Paste” (Ctrl+V) menu command pastes the list on the clipboard to the view by appending the items to the current selection set. In this circumstance items in the list may be separated by either spaces, tabs or may be on separate lines. An error message is given if the content of the clipboard is not a valid list.

Both “Copy” and “Paste” are also available on the [Graphic View right-click menu](#) when in a selection mode.

Nothing in a list held on the clipboard identifies the type of list so, for example, it is possible to copy a set of nodes and paste the resulting list to somewhere where an element list is expected. This is unlikely to be useful.

3.5.8 Polylines in Graphic Views

In GSA a polyline is a sequence of points in a 2D plane. There is a [2D Polylines module](#) in which polylines can be saved. There is also a current polyline which is displayed in Graphic Views when in the Polyline [cursor mode](#).

The current polyline is drawn in the plane of the current grid and its points are positioned with respect to the current grid axes. Polylines are represented in Graphic Views by lines connecting the points and a faint line closing the polyline. Whether a polyline is considered as closed depends on the context in which it is used. For example, a polyline used in the definition of a grid area load is assumed to be closed whereas one used for a grid line load is not.

The current polyline may be edited from any Graphic View when in the Polyline cursor mode. The current polyline is displayed on the image when in the Polyline cursor mode and greyed when the cursor mode is changed.

Since the current polyline is “owned” by the model and not the particular Graphic View and since the cursor mode is set for all Graphic Views and not just the current one, the display of the current polyline occurs on all open Graphic Views simultaneously.

More:

[Use of polylines](#)

[Forming polylines](#)

[Interaction of graphic polylines with the 2D Polylines module](#)

3.5.8.1 Use of polylines

Selection sets may be [formed](#) by selecting entities within the current polyline.

When in the Polyline cursor mode the properties of the polyline can be obtained using the “Graphics | Selection Properties” menu command. “Selection Properties” is also available on the Graphic View right-click menu when in the Polyline mode. The properties include the length, perimeter, area and accumulated length.

Several [sculpt commands](#) require a polyline as input, including [Create Grid Loading](#) and the (legacy) [Generate 2D Elements](#) commands.

3.5.8.2 Forming polylines

Polylines are formed using the mouse in the Polyline [cursor mode](#).

In the Polyline cursor mode the functionality of the mouse is as follows.

- **Clicking** when the grid is drawn adds a point to the polyline at that position on the grid.
- **Clicking** on a node adds a point to the polyline at the position of the node. If the node is not in the plane of the current grid the option is given to change the current grid to one at the elevation of the node. If the current grid is changed then the polyline is displayed in the new current grid plane; otherwise the new point is positioned at the projected position of the node on the current grid plane.
- **Dragging** an existing point changes the position of that point to the grid position dragged to.

The following commands are also available on the right-click menu when in the Polyline cursor mode.

- “Delete All Polyline” (Esc)
- “Delete Polyline Point” (Del)
- “Select Polyline”

And on the main menu:

- “Edit | Select String” – This extends the polyline forward from the last polyline segment, adding vertices at the positions of closest aligned displayed nodes or grid points. The [straightness tolerance](#) preference is used to determine whether a node or grid point is aligned, allowing this command to be used to select curved polylines.

3.5.8.3 Interaction of graphic polylines with the 2D Polyline module

When in the Polyline cursor mode the “Edit | Save Selection As 2D Polyline” menu command saves the currently displayed polyline to the 2D Polyline module, giving the option to overwrite an existing 2D Polyline or create a new one.

Also, when in the Polyline cursor mode the “Edit | Select 2D Polyline” menu command allows an existing 2D Polyline to be displayed as the current polyline.

On giving the “Save Selection As 2D Polyline” or “Select 2D Polyline” command an association is formed between the current polyline and the saved polyline. This saved polyline is then offered as the default when calling these commands. The association is cleared when the current polyline is deleted.

Both “Save Selection As 2D Polyline” and “Select 2D Polyline” are also available on the Graphic View right-click menu when in the Polyline cursor mode.

3.5.9 Adornments

The image of the model can be adorned in various ways to add information to the image.

More:

[Labels](#)

[Display methods](#)

[Diagrams](#)

[Contours](#)

[Deformed image](#)

[Scaling of diagrams, contours and deformations](#)

[Annotating diagrams and contours](#)

[Units, Numeric Format and Axes](#)

3.5.9.1 Labels

Nodes and elements can have labels applied to them. Generally the node and element labels identify attributes of these such as node numbers, restraint conditions, element numbers, element axes etc.

Axes can also be labelled. An axis set is labelled by displaying the axis triad correctly positioned and orientated on the image.

Labels are set using the “Graphics | Display | Labels and Display Methods” menu command.

“Labels and Display Methods” is also available on the [Graphic Display toolbar](#). Giving this command opens the [Labels and Display Methods dialog box](#).

Labels can be applied just to a selection of nodes or elements. If this is required then the items should be selected prior to giving the “Labels and Display Methods” command. Refer to the [Applying adornments to a selection](#) section for details.

Some of the more frequently used labelling options are accessible directly from the [Display Favourites toolbar](#).

Note that [resetting the display](#) reverts to the labels set in the [default view settings](#).

Refer to the [Output Options chapter](#) for a full list of available options.

The “Graphics | Display | Hide | Hide Labels” command temporarily switches the labels off. Labels may then be un-hidden by giving the “Hide Labels” command again. Adjusting any label settings while the labels are hidden automatically un-hides the labels. “Hide Labels” is also available on the right-click menu.

3.5.9.2 Display methods

Display methods are the methods by which the various element types are displayed. For example, beam elements can be drawn as lines or solid sections, spring elements can be lines or coils, 2D elements can be outlines, filled, with or without thickness.

The display methods include various options that affect the colour in which elements are displayed. The colour may be according to the element type, element property, element group, element material, or by the initial analysis stage in which the element is represented. Colouring by type helps distinguish between, say, beam and bar elements. When colouring by property each property number used by the model is automatically assigned a unique colour and the elements are coloured on that basis. Note that in models referring to a large number of properties the colours of closely numbered properties become less distinct. Colouring by element group or material is handled similarly.

The display methods also offer the option to set whether the various type of unattached nodes are drawn.

Display methods are set using the “Graphics | Display | Labels and Display Methods” menu command. “Labels and Display Methods” is also available on the [Graphic Display toolbar](#). Giving this command opens the [Labels and Display Methods dialog box](#).

A common use of display methods is to display elements as solid entities. Switching on the Section Display button on the [Graphic Display toolbar](#) has the effect of setting the display method for beams and bars to “Outlined & Filled” and the display method for 2D elements to “Filled & Thickness”. Switching this off reverts to “Line” and “Outline”, respectively.

Note that [resetting the display](#) resets all display methods to the [default view settings](#).

Refer to the [Labels and Display Methods dialog box](#) documentation for a full list of available options.

3.5.9.3 Diagrams

Diagrams can be drawn on the image to represent various input data and results. Typically diagrams are associated with nodes or elements; for example, reactions or bending moments. These diagrams are displayed only if the related node or element is drawn.

Some diagrams are not associated with nodes or elements; for example, grid point loads. These diagrams are displayed if their position lies within the extents of the currently drawn structure.

Diagrams are specified using the “Graphics | Display | Settings | Diagram Settings” menu command. “Diagram Settings” is also available on the [Graphic Display toolbar](#). Giving this command opens the [Diagram Settings dialog box](#).

Diagrams can be applied just to a selection of nodes or elements. If this is required then the items should be selected prior to giving the “Diagram Settings” command. Refer to the [Applying adornments to a selection](#) section for details.

Ranges of interest, units, numeric format, axes and scaling can be specified. Refer to the Diagram Settings dialog box documentation for a full list of available options.

Some of the more frequently used diagram options are accessible directly from the [Display Favourites toolbar](#). Selecting diagrams from the toolbar has the same affect as specifying the data type in the Diagram Settings dialog box and leaving all other settings as default.

Refer to the [Output Options chapter](#) for a full list of available diagrams.

Diagrams of several data types can be displayed together in a Graphic View. This can be done either from the [Diagram Settings dialog box](#), by using the New Diagram tab when a diagram is already being displayed, or by using the Display Favourites toolbar commands.

Where the diagram is of data that is stored by case, the case displayed is as specified in the Cases field in the [Lists toolbar](#). Diagrams are drawn for each case referred to in the field; diagrams for each case are superimposed. Diagram envelopes are displayed for enveloping cases.

Note that [resetting the display](#) switches off all diagrams.

The “Graphics | Display | Hide | Hide Diagrams” command temporarily switches any diagrams off. Diagrams may then be un-hidden by giving the “Hide Diagrams” command again. Adjusting any diagram settings or rescaling the data while the diagrams are hidden automatically un-hides the diagrams. “Hide Diagrams” is also available on the right-click menu.

See also:

[Scaling of Diagrams, Contours and Deformations](#)

[Annotating diagrams and contours](#)

3.5.9.4 Contours

Algorithms exist in GSA to enable any scalar value to be contoured for 0D, 1D, and 2D elements and nodes. Contours on nodes and 0D elements are displayed as balloons of varying diameters and colours. Contours on 1D elements are displayed as barrels of varying diameters and colours.

Contours are displayed only if the related node or element is drawn.

Contours are specified using the “Graphics | Display | Settings | Contour Settings” menu command. “Contour Settings” is also available on the [Graphic Display toolbar](#). Giving this command opens the [Contour Settings dialog box](#).

Contours can be applied just to a selection of nodes or elements. If this is required then the items should be selected prior to giving the “Contour Settings” command. Refer to the [Applying adornments to a selection](#) section for details.

By default the number of contours displayed is eight; the lowest and highest contour values are set at the minimum and maximum values of data being contoured and the intervening contours are equally dispersed.

The values and colours of contours or a constant contour interval can be specified.

The number of contours can be specified. No limit is imposed on the number that may be requested; the more contours drawn, the less the distinction between colours of adjacent contours (though the colours may be adjusted manually).

Contours may be drawn as lines or filled.

Contouring on elements and members when 'section display' is switched on displays the contouring on the surface of the section for beams and on the top and bottom surfaces of 2D elements. Note that in this mode it is not the surface values that are being contoured; the same values are being contoured as in other contour representations but displayed on the surface of the elements.

Units, numeric format, axes and scaling can be specified. Refer to the Contour Settings dialog box documentation for a full list of available options.

Refer to the [Output Options chapter](#) for a full list of data available for contouring.

Where the contouring is of data that is stored by case, the case displayed is as specified in the Cases field in the [Lists toolbar](#). Contours are drawn for only the first case referred to in the field. Either the maximum or minimum value is contoured for enveloping cases, as specified in the Contour Settings dialog box.

Note that [resetting the display](#) switches off all contours.

The “Graphics | Display | Hide | Hide Contours” command temporarily switches any contours off.

Contours may then be un-hidden by giving the “Hide Contours” command again. Adjusting any contour settings or rescaling the data while the contours are hidden automatically un-hides the contours. “Hide Contours” is also available on the right-click menu.

See also:

[Scaling of Diagrams, Contours and Deformations](#)

[Annotating diagrams and contours](#)

3.5.9.5 Deformed image

Translational and rotational displacements can be represented as both diagrams and contours. A further option is to display the image in its deformed state. In the deformed image the model is deformed at nodes and along elements by the translational displacement, magnified by a factor to result in a visible deformation.

The deformed image can be switched on using the “Graphics | Display | Deformed Image” menu command. “Deformed Image” is also available on the [Graphic Display toolbar](#).

When the deformed image is displayed the undeformed or initial state image can be superimposed as a series of dashed lines. The initial state image is displayed with the deformed image when the “Initial state” option is checked in the [Deformation Settings dialog box](#).

The deformation settings can be specified using the “Graphics | Display | Settings | Deformation Settings” menu command. Giving this command opens the [Deformation Settings dialog box](#).

When the deformed image is switched on [animating](#) the image results in the deformation being animated.

Note that [resetting the display](#) switches off the deformed image.

See also:

[Scaling of diagrams, contours and deformations](#)

3.5.9.6 Case list

The case list for a Graphic View can be specified both in the [Graphic Settings dialog box](#) and via the [Lists toolbar](#). The full [GSA list syntax](#) is available when specifying the list.

Diagrams and deformed images are drawn for each case specified in the case list. Contouring is displayed for only the first case in the case list, referred to as the 'contouring case' in the legend.

When **Condense Cases** is checked, the case list pull-down on the Lists toolbar is automatically trimmed to offer only those cases in which current diagram and contouring data exists for the currently displayed entities.

3.5.9.7 Scaling of diagrams, contours and deformations

The scales at which diagrams, contour balloons and contour barrels are drawn and the magnification factor applied to displacements to arrive at a deformed image may be referred to as data scales.

Data scales can be explicitly specified in the [Further Options dialog box](#), which is accessed via the respective “settings” dialog boxes. If they have not been specified or if the “Do auto-scale” option has been checked then the scale is automatically calculated to result in a maximum size of diagram etc. of 15mm. If “Lock scale” is checked the scale for that diagram or contour is not affected by any automatic rescaling.

Scaling may be set to be to an engineering scale by checking “Auto-scale to engineering scale” in the [Further Options dialog box](#). When set, the calculated scale is rounded up to the next engineering scale.

The auto-scaling process includes a calculation of the extents of the data being displayed. The data extents are used when calculating contour values and when interpreting extents of interest in diagrams when expressed in terms of percentages.

Whether data scales or data extents are recalculated when the case is changed depends on the “Recalculate displayed data scales on change of case” and “Recalculate extents of displayed data on change of case” settings in the [Graphics Preferences](#).

Giving the “Graphics | Display | Rescale Data” (Alt+Home) recalculates the scales for all current diagrams, contours and the deformed image. “Rescale Data” is also available on the [Graphic Display toolbar](#).

A scale factor is applied to the calculated or specified data scale to result in a scale used for the display. The scale factor is provided to offer a convenient way of increasing or decreasing the displayed scale of diagrams, contours and deformed image. While the scale factors may be specified via the respective “settings” dialog box usually they are adjusted via the “Double size of diagrams” and “Halve size of diagrams” commands on the [Graphic Display toolbar](#). “Rescale data” resets the scale factor to unity. The scale factor is applied to ‘constant size’ contouring.

3.5.9.8 Applying adornments to a selection

Labels, diagrams and contours may be applied to all currently displayed entities (i.e. nodes or elements) or they may be applied to a [selection set](#). Once applied to a set that set is preserved with the adornment. Changing the selection set has no effect on the entities to which currently displayed adornments are applied. This allows, for example, bending moment diagrams to be displayed on one set of elements and shear force diagrams to be displayed on a different set.

The easy way of applying labels, diagrams or contours to a selection set is by using the “Graphics | Display | Apply Adornments to Selection” command. “Apply Adornments to Selection” is also available on the [Display Favourites toolbar](#). While the “Apply Adornments to Selection” command is set, any labels, diagrams or contours are applied to the current selection set. If there is no selection of the relevant type (e.g. no elements selected when bending moment diagrams are requested) then the diagrams are applied to all entities.

The “Apply Adornments to Selection” command is a quick way of specifying what can otherwise be set in the [Labels and Display Methods dialog box](#) or ‘Diagram Settings | [Further Options dialog box](#)’ or ‘Contour Settings | [Further Options dialog box](#)’, respectively.

3.5.9.9 Annotating diagrams and contours

Diagrams and contours can be annotated with the values represented by the diagram or contour. The units and numeric format of the annotation is as specified via the [Diagram Settings](#) or [Contour Settings](#) dialog box, respectively.

The annotation of 1D element diagrams can either be at diagram peaks or at all points along the element. Diagram peaks are at element ends and where the diagram gradient changes sign or, when the diagram is constant along the element, just at the element centre. Annotation is at all points when “Full annotation” is checked in the [Further Options dialog box](#) which is accessed via the Diagram Settings dialog box.

In the case of contouring, the annotation replaces the contouring; the annotation is displayed in the colour associated with the interval in which the value being annotated falls. Only the centre value is annotated for 2D elements unless there is found to be space to annotate at each nodal position.

Annotation may be applied to all entities, to a selection of entities or to none. This annotation setting is set in the [Further Options dialog box](#), which is accessible from both the Diagram Settings and Contour Settings dialog boxes. Annotation “By Selection” is used in conjunction with the Select for Annotation [cursor mode](#) as follows.

1. Specify the diagrams and / or contours. Set the annotation to “By Selection” (the default).
2. Set the [cursor mode](#) to Select for Annotation.

3. Select the nodes or elements to be annotated. For the annotation of grid plane related data (e.g. grid loading) select any element that lies in the grid plane.
4. Note that the “Edit | Select All” (Ctrl+A), “Edit | Select None” and “Edit | Invert Selection” menu commands are available in this cursor mode.
5. Only diagrams and contours set at the time of selecting the nodes or elements are annotated.

Another way of annotating a selection of entities is by using the “Edit | Annotate Selection” menu command. “Annotate Selection” is also available on the [right-click menu](#) that is displayed when the [cursor mode](#) is set to Select Nodes or Select Elements. The procedure is as follows.

1. Set the [cursor mode](#) to Select Nodes or Select Elements.
2. Select the nodes or elements to be annotated.
3. Give the “Annotate Selection” command.

The Select for Annotation cursor mode and the “Annotate Selection” command have no effect on diagrams and contours that have the annotation setting set to “All” or “None”.

3.5.9.10 Units, Numeric Format and Axes

The units, numeric format and axes may each be set for individual diagrams and contours via the [Diagram Settings](#) or [Contour Settings](#) dialog box, respectively.

Some axes options are disabled for particular data options. Refer to the [Output Options chapter](#) for details.

The “Window | Settings | Units”, “Window | Settings | Numeric Format”, “Window | Settings | Increase Precision”, “Window | Settings | Decrease Precision” and “Window | Settings | Axes” menu commands apply the settings to all existing diagrams and contours in the current Graphic View and also set defaults for new diagrams and contours in the current Graphic View. The “Increase Precision” and “Decrease Precision” commands increase or decrease the number of significant figures or decimal places, depending on the current ‘Numeric Format’ setting. All these commands are also available on the [Data Options toolbar](#).

3.5.10 Shrinking elements

Elements may be drawn shrunk by using the “Graphics | Display | Shrink” menu command. “Shrink” is also available on the [Graphic Display toolbar](#).

Shrinking the elements has the effect of pulling the nodal positions of elements towards the element centre by the current Shrink factor. The Shrink factor is a [Graphics Preferences](#).

3.5.11 Colour In Graphic Views

The overall colour of Graphic Views, such as whether colour or monochrome, or whether light or dark background, can be adjusted via the [Graphic Fonts and Styles](#) settings.

The colour in which elements and members are displayed may be by element type, element property, element group, element material, or by the initial analysis stage in which the element is represented. This can be adjusted via the [Display methods](#) ‘colour elements’ setting.

The default colours assigned to entities (i.e. elements, members, lines, areas, regions) can be overridden by explicitly defined colour. Explicit colour can be assigned to elements, members, materials, properties, and analysis stages. Depending on the display methods ‘colour elements’ setting:

- when colours are ‘by type’ then the colour assigned to the entity is used.
- when colours are ‘by property’ then the colour assigned to the respective property is used.

- when colours are 'by group' explicit colours are ignored.
- when colours are 'by material' then the colour assigned to the material is used.
- when colours are 'by initial stage' then the colour assigned to the analysis stage is used.

Assignment of explicit colours is either by use of the 'Graphics | **Modify Colour of Selection**' menu command, having first selected the entities that are to be modified, or by right-clicking on an entity and giving the '**Modify Colour of This**' menu command.

- These commands open the Select Colour dialog.
- The selected colour is indicated by a colour swatch, and the derived colour-wash, by the partially hidden colour swatch.
- Whether it is the explicit colour of the entity, or property, or material, or analysis stage that is modified is determined by the current display methods 'colour elements' setting. E.g. if the 'colour elements' setting is 'by property' then it is the colour of the property that is modified.
- Setting the colour to 'undefined' unsets the explicit colour assignment.

The last colour assigned is 'sticky', in as much as that the Select Colour dialog remembers the last colour specified. The 'right-click | **Pick Colour of This**' command sets that remembered colour to the colour of the entity that was right-clicked on, as displayed; e.g. that would be the colour of the property if the 'colour elements' setting is 'by property'.

All explicit colour assignment can be removed by using the 'Graphics | **Clear Colour Modifications**' menu command.

Note that explicit colour assignment can also be applied via [Table Views](#).

3.5.12 Shading surfaces

The "Shade Surfaces" command adjusts the shade of colour filled surfaces according to the current lighting settings. The incidence of the light on the surface, the brightness and the level of ambient light all effect the shading.

The Shade Surfaces option may be set via the [Display toolbar](#). The Shade Surfaces option is also available in the [Graphic Settings dialog box](#).

Lighting Settings

The lighting settings are made up of:

- The position of a point light source, in global axes.
- The colour of a point light source.
- The colour of ambient light.
- The specular exponent, a measure of the shininess of the surface. (Higher values = more shiny)

Adjusting Lighting Settings

The "Graphics | Orientation | Shine Light From Here" (Ctrl+Alt+H) command sets the position of the point light source to the current eye point, considering both the current direction of view and the distance of the eye point from the object point, transformed into global directions.

All lighting settings can be edited in the Lighting Settings dialog box. Open the Lighting Settings dialog box either by giving the "Graphics | Orientation | Lighting Settings" menu command or from the [Graphic Settings dialog box](#).

3.5.13 Translucency

The translucency of colour filled surfaces is determined by the translucency settings.

Translucency Settings

When **Enable translucency** is selected, colour filled surfaces are displayed translucent and the following opacity factors are considered.

- **Opacity of beam elements**—when drawn as solid sections.
- **Opacity of 2D elements**—this opacity factor is also used in the display of colour filled soil zones and profiles.
- **Opacity of contouring**—on all entity types.

Adjusting Translucency Settings

Translucency settings can be edited in the Translucency Settings dialog box. Open the Translucency Settings dialog box either by giving the “Graphics | Orientation | Translucency Settings” menu command or from the [Graphic Settings dialog box](#).

3.5.14 Unwrap Graphics

The Unwrap option in Graphic Views projects the displayed structure onto the surface of a cylinder or cone and then unwraps the surface to produce a planar representation of the structure. The unwrapped image can be adorned with labels, diagrams, contours, annotation, and can be generally manipulated in the same way as the normal (wrapped) image.

How the structure is unwrapped is determined by the unwrap settings.

Unwrap Settings

There are four unwrap modes:

- **Don't unwrap**—unwrap mode is switched off.
- **Unwrap auto**—unwrap onto a cylindrical surface. The cylinder is centred on the **unwrap axes z** and the radius of the cylinder is the automatically assessed max. 'unwrap axes' r of the displayed nodes, factored by the **unwrap radius factor**.
- **Unwrap explicit cylinder**—unwrap onto a cylindrical surface. The cylinder is centred on the **unwrap axes z** and the radius of the cylinder is specified by the **unwrap radius**.
- **Unwrap explicit cone**—unwrap onto a conical surface. The cone is centred on the **unwrap axes z** and the surface of the cone is defined by two **cone surface points**.

The surface is unwrapped onto the global ZX plane. The cylindrical unwrap modes unwrap splitting at 'unwrap axes' theta = 0. The conical unwrap mode unwraps splitting at 'unwrap axes' theta = 180 to result in the apex of the cone at 0,0,0 and 'unwrap axes' theta = 0 being aligned with global Z.

The **Grab Coor.** option may be used in the 'unwrap explicit cylinder' and 'unwrap explicit cone' options to populate the **unwrap radius** or **cone surface points**, respectively, with data derived from the currently selected nodes. The coordinates of the selected nodes are transformed into 'unwrap axes', as specified at the time the 'Grab Coor.' command is given.

Accessing the Unwrap Settings

Unwrap settings can be edited in the Unwrap Settings dialog box. Open the Unwrap Settings dialog box either by giving the “Graphics | Orientation | Unwrap Settings” menu command or from the [Graphic Settings dialog box](#).

N.b. The Unwrap option is intended as a tool for clarifying the display of data on a structure. The process of unwrapping contorts 3D global space so some graphical operations that rely on 3D space being 3D may behave unexpectedly; sculpting on an unwrapped image may have bizarre consequences.

3.5.15 Highlighting element edges

Element edges can be highlighted using the “Graphics | Display | Highlight Edges” menu command. The Highlight Edges option is also available in the [Graphic Settings dialog box](#).

The “Highlight Edges” command has the following effect.

For 1D elements:

Ends of 1D elements that are attached to nodes that have just that element connecting into it are highlighted.

For 2D elements:

Bad connectivity: Brown lines are drawn along each edge of 2D elements that:

- is not connected to any other 2D element, or
- is badly connected to any other 2D element (i.e. linear v. parabolic or different mid-side nodes), or
- is upside-down with respect to other attached 2D elements in the same plane, or
- is attached to only one other element and is upside-down with respect to that.

A proper connection is where two elements are connected the same way up all along their edges.

Mesh discontinuity: Yellow lines are drawn along the edge of a 2D element where no other 2D element attached to that edge:

- has the same property reference, or
- lies in the same plane. The Facet angle [Results Preference](#) is used as the tolerance in determining whether two elements lie within the same plane.

3.5.16 Highlight coincident nodes

The “Highlight Coincident Nodes” command highlights all nodes found to be within the coincidence tolerance of another. The coincidence tolerance is defined in the [Sculpt Preferences](#).

Coincident nodes can be highlighted using the “Graphics | Display | Highlight Coincident Nodes” menu command. The Highlight Coincident Nodes option is also available in the [Graphic Settings dialog box](#).

3.5.17 Highlight coincident elements

The “Highlight Coincident Elements” command highlights all elements found to have the same topologies as another, regardless of topology sequence.

Coincident elements can be highlighted using the “Graphics | Display | Highlight Coincident Elements” menu command. The Highlight Coincident Elements option is also available in the [Graphic Settings dialog box](#).

3.5.18 Resetting the display

Resetting the display reverts all display options to their default settings.

The display can be reset using the “Graphics | Display | Reset Display Adornments” menu command. “Reset Display Adornments” is also available on the [Graphic Display toolbar](#).

“Reset Display Adornments” resets the settings to the [default view settings](#). Specifically, “Reset Display Adornments” has the following effect.

- All labels and display methods are set to the default view settings.

- All diagrams and contours are switched off.
- The image is drawn in its undeformed state.
- The entity list is set to all elements and all nodes.
- Volume clipping is switched off.

The entity lists, volumes, orientation, scaling, mid-point and object point are not affected by resetting the display adornments. The [“Reset to All Entities”](#) menu command may be used to reset to the whole model being drawn.

3.5.19 Switch layer

The display can be switched between the [Analysis Layer and the Design Layer](#) using the “Graphics | Switch Layer” (Ctrl+Alt+D) menu command. The layer may also be set by, on the [Lists toolbar](#) for the view, setting the Display Option to “Layer” and selecting the required layer in the Display List.

3.5.20 Right-click menus

Clicking the right mouse button when the cursor is in a Graphic View displays a floating menu. The commands offered in the floating menu depend on the current cursor mode and what the cursor was pointing at when the button was clicked. Some commands are always offered regardless of context other than it is a Graphic View that is current.

Most of the commands offered in the right-click menu are also available from the standard pull-down menus. However the position dependent commands are only available on the right-click menus.

Cursor mode dependent commands

These commands relate to the current [cursor mode](#). For example, “Select By List” is offered when the cursor mode is a selection mode and “Polyline Length” is offered when the cursor mode is “Polyline”.

Position dependent commands

When the cursor is pointing at a node, element or member the right-click menu offers commands that relate to the item pointed at. For example, “Node Properties” displays the attributes of the node pointed at, “Edit Property” opens the property wizard for the element or member pointed at and “Go to Material Data” opens the Materials Table View at the record relating to the element or member pointed at.

Commands always offered

Certain commands always appear on the right-click menu, such as “View Legend”, which displays the legend information in a message box.

3.5.21 Graphic Fonts and Styles

The fonts and styles of graphic entities can be adjusted from default in the following ways.

- The colours of in-fill, lines, text, symbols and background can be set to monochrome.
- All colours can be set negative on the screen, to result in a black background.
- Factors can be applied to the size of text and symbols.
- Separate settings apply to output to screen and output to printer.

The fonts and styles can be adjusted in the [Graphic Fonts and Styles dialog box](#).

3.5.22 Animation

The image can be animated to produce an apparently continuous motion of the image in the Graphic View. The animation can be by bouncing the image between its deflected and undeflected positions or by stepping through a list of cases. The image can also be automatically rotated about the vertical (global Z) axis through the object point.

When the case list for the Graphic View is a list of several load cases or several result cases (i.e. analysis cases or combination cases) then the animation is by stepping through the list of cases. Note that a case list comprising a mix of load cases and result cases does not satisfy this rule. Thus a case list “1 to 5”, which, in the typical situation, is equivalent to “L1 to L5 A1 to A5” would not produce animation by stepping through cases whereas “A1 to A5 C1 to C5” would.

When not animating cases and the deformed shape is switched on animation is of the deformation. When the deformed image is of a mode shape the animation bounces between full positive and full negative deflection.

Animation is activated by giving the “Graphics | Display | Animate” menu command. “Animate” is also available on the [Graphic Display toolbar](#). Before animation commences there is often a pause while the program assembles the images that make up the animation sequence.

While the image is animating the image can be rotated, zoomed and panned and the perspective view eye distance can be adjusted. All of these operations can be carried out using the mouse when the Rotate [cursor mode](#) is current.

The legend is not displayed during animation except when [recording](#) the animation.

Animation can be cancelled by giving either the “Animate” command or the “Stop” command on the [GSA toolbar](#). Giving any command that cannot be acted on during the animation will also cause the animation to stop.

Various animation settings can be specified using the “Graphics | Display | Settings | Animation Settings” menu command. Giving this command opens the [Animation Settings dialog box](#).

3.5.23 Printing from Graphic Views

The graphic image may be printed by giving the “File | Print” (Ctrl+P) menu command. The printed image may be previewed by giving the “File | Print Preview” menu command. “Print” and “Print Preview” are also available on the [Standard toolbar](#).

The “Page Setup” command on the [Miscellaneous tab](#) of the Preferences may be used to specify the format of the border on the printed page.

The scale at which the image is printed generally depends on how the displayed scale was achieved. Refer to “[Scaling for printed output \(and changed window sizes\)](#)” for details.

When “Print by case” in the [Graphic Settings dialog box](#) is checked each case is printed separately.

The underlying graphics code used for printing and print previewing is different to that used for display to the screen. This may occasionally cause subtle differences between the screen image and the printed image.

When printing to a monochrome device, lines, text and symbols are automatically output in black and in-fill is converted to a shade of grey by the program. (The various printers and printer drivers interpret colours differently, sometimes even to the extent of ignoring some colours.)

3.5.24 Output of the graphic image

The following options are available for outputting the image from Graphic Views.

More:

[Copying the graphic image to the clipboard](#)

[Saving the graphic image to file](#)

[Recording a moving graphic image](#)

3.5.24.1 Copying the graphic image to the clipboard

When the current [cursor mode](#) is other than a selection mode the “Edit | Copy” (Ctrl+C) menu command copies the current image to the clipboard in both bitmap and Windows Meta-File format. The application to which the image is pasted will automatically select its preferred format.

The image copied to the clipboard is as currently displayed, including orientation, scaling and adornments.

Note that the WMF image includes the whole of the currently drawn structure, regardless of the current zoom setting. If only part of the structure is required in the WMF image this should be specified either by [volume clipping](#) or by [entity list](#) rather than by zooming.

When copying an image for pasting into the Titles view it is advisable to adjust the Graphic View window size and aspect ratio to approximately the same as the Bitmap window in the Titles view.

Note that when the cursor mode is a selection mode the “Copy” command behaves differently. Refer to [“Copying sets to the clipboard”](#) for details.

3.5.24.2 Saving the graphic image to file

The graphic image can be saved to file in various file formats. All are obtained using the “Graphics | Save Image” menu command.

When “Print by case” in the [Graphic Settings dialog box](#) is checked the image for each case is saved to a separate file (except where noted below). These files are automatically given unique names that include a reference to the case.

The following file formats are available.

WMF

WMF (Windows Metafiles) files are the native format supported by Microsoft, however their file size is much greater than PNGs or JPEGs, and WMFs are not supported by web browsers. WMFs can be useful for including in Microsoft Word documents, but PNGs and JPEGs still produce a much smaller Word document when saved. A benefit is that the WMF format adds the company logo to the image, which may be desirable.

Note that the WMF image includes the whole of the currently drawn structure, regardless of the current zoom setting. If only part of the structure is required in the WMF image this should be specified either by [volume clipping](#) or by [entity list](#) rather than by zooming.

PNG

PNG (Portable Network Graphic) files are best for images that contain mainly simple lines, text and areas filled with constant colour; i.e. most GSA images including contoured images, but see [JPEG](#), below. The image quality is good, and the file size is small. PNGs are supported by professional paint packages and most web browsers. They are also useful for including in Microsoft Word documents.

The size of image may be specified in terms of resolution and physical size in the [Saved Image Settings Preferences](#).

JPEG

JPEG (Joint Photographic Experts Group) files are best for images that contain areas filled with varying colours; e.g. contoured images with large numbers of contour intervals or shaded images of curved surfaces. For this type of image the JPEG quality factor need not be set very high. A recommended value would be around 75%. For line drawings the PNG format is recommended, however if you want to use JPEG then the quality factor should be set to around 90%, otherwise colour bleeding may be visible. JPEGs are widely supported by all paint packages and web

browsers. They are also useful for including in Microsoft Word documents.

The size of image may be specified in terms of resolution and physical size in the [Saved Image Settings Preferences](#).

The JPEG quality factor setting may be adjusted in the [Saved Image Settings Preferences](#). The lower the quality setting, the smaller the file size and the poorer the picture quality.

DXF

The image saved is as currently displayed, including orientation, scaling and adornments but excluding text. The image is written to the DXF file in 3D. Note that certain information displayed in the Graphic View is displayed in the picture plane or a plane parallel to this (e.g. axial force diagrams and arrow heads). This information will be output to the DXF file in that plane; it will not adjust to the picture plane of the receiving application. DXF files can be read by various CAD packages. (N.b. "Print by case" is ignored when saving to DXF.)

3.5.24.3 Recording a moving graphic image

A Graphic View session can be recorded and saved to AVI (Audio Video Interleave) file for playback by a multimedia player such as Windows Media Player (as bundled with Windows). During a recording session each frame displayed in the Graphic View is recorded, whether animating or otherwise rotating, zooming and adorning the image. The recording session can be paused to allow the display to be adjusted without the process of adjusting the display being recorded. The session can be frozen to result in a period of still image during playback.

The following recording commands are available from the "Graphics | Recorder" pop-up menu.

- **Record**—Starts a recording session or, if a recording session has been paused, continues recording.
- **Pause**—Pauses recording.
- **Freeze**—Freezes the current image in the recording for the specified freeze time. The AVI freeze time setting may be adjusted in the [Saved Image Settings Preferences](#).
- **Stop**—Terminates the current recording session prompting for the name that the AVI file is to be saved as.
- **Abort** (Esc)—Quits the current recording session.

These commands are also available on the [Recorder toolbar](#).

The length of the current recording is displayed in the status bar.

Points to note:

- The dimensions of the recording are determined by the dimensions of the Graphic View at the time of making the recording.
- AVI files can be very large; reducing the window size significantly reduces the file size. A 12cm × 12cm window is reasonable.
- The Graphic View should be resized as required before starting the recording. Resizing during a recording aborts the recording.
- Any windows obscuring the Graphic View during recording will be included in the recording. Pause the recording before taking any action that will obscure the view.
- Only one concurrent recording session is permitted per GSA session.
- The playback speed is affected by the frame rate. The AVI frame rate setting may be adjusted in the [Saved Image Settings Preferences](#).
- If recording is in progress when animation is requested then the number of frames established for the animation is based on the AVI frame rate setting to result in the animation period being correct at the time of playback of the recording. Typically this will result in more animation frames

than would be established for animating when not recording; consequently the internal preparation of the animation will take longer and the Graphic View animation will appear slower.

- The recording may not be paused when animating.
- Creating an AVI takes a lot of processing power so it is natural for the graphics to react more slowly when recording.
- Creating videos takes a little practice. Several attempts may be required.

3.5.24.4 Capturing the graphic image in 3D PDF

Capture of the GSA Graphic View image to 3D PDF has been tested with Adobe Acrobat 3D version 8.1.2.

Note the following:

- The graphics should be rendered in the main thread. See [New Graphic View \(main thread\)](#).
- The GDI overlay (i.e. legend, axes triad, selection marks) will not be captured.
- In Acrobat 3D, if '3D Capture Settings | General | Discard stray points and lines' is 'checked' then lines in the Graphic View (as opposed to filled areas) will not be captured, and if there are no filled areas then no capture will occur!
- If Acrobat 3D crashes when converting the image then, in 'Acrobat 3D Conversion | General | Preview Image' try changing from 'Retrieve preview from default view' to 'Blank preview'. (Acrobat 3D v.8.1.2) (This may not have caused problems in earlier versions of Acrobat 3D.)

3.6 Working with Chart Views

There are a number of options in GSA which present results as charts, where some result is presented as a function of some parameter, so for example, response factor versus time. depending on the chart option selected there may be a large number of curves drawn on a single chart. Within the chart view there are many options to help organize and display individual curves.

Many of the options for controlling chart view are available on the right-click (context menu) in the chart view.

3.6.1 Chart Menus

The right-click (context) menu in chart views is tailored to the data being presented. Certain functionality will always be present, but other items will be present where appropriate.

Copy and Copy Points

There are two copy options. "Copy" copies the chart as a graphic which can then be pasted into other documents. "Copy Points" copies the points in a text format which can be conveniently pasted into spreadsheets such as Excel. This is useful where further calculations based on the GSA results are required.

Cursor Tooltip

By default no tooltip is displayed, but there are options to display a tooltip within the graph area. The options are:

- Coordinates at Cursor - the tooltip display the abscissa and ordinate values for the current cursor position.
- Curve Values at Abscissa - the tooltip displays the values of all curves for the current abscissa position. This may not be useful where there are a large number of curves on the chart.
- Coordinate of nearest point - the tooltip display the coordinate of the nearest point on any of the

displayed curves.

Chart Style

The "Chart Style" option opens the [Chart Style wizard](#) giving extensive control over the chart.

Show Symbols

This allows symbols to be switched on or off.

Hide Curves

Where there are multiple curves it is useful to be able to switch individual curves on and off. When there are multiple curves the individual curves can be toggled between hide/show. The "Hide" option opens a dialog listing all the curves. A multiple selection allows a quick switching on or off of a block of curves. Hide all and show all act as the names suggest.

Rescale & Zoom

Curves can be rescaled to fit and zoomed in or out from the menu, although zooming is more easily controlled by the mouse wheel.

Line Weight

This allows the lines weight to be increased or decreased. This can be useful to help see the individual curves more easily.

Save Chart

There are a number of options for saving the charts in a graphical form. Select the appropriate file type from dxf, jpeg, png and wmf.

Export

The export option allows either individual or all curves to be exported. These can be exported as csv curves for import to spreadsheets, cur files for import to Oasys T/HIS and crv files.

Open in Sigraph

The chart presents a set of results and while the style of the chart and curves can be changed, no post-processing of this data is permitted. When further processing of a curve or curves is required the data can be exported to Oasys Sigraph which is designed for manipulation of chart data such as summing, differentiating, fitting curves, etc.

Add Text

This allows text to be added to the chart at the position where the mouse was clicked. The text can be edited in the Chart Style Wizard on the [Background Style](#) page

3.6.2 Chart Styles

The Chart Style wizard allows the display of the chart data to be tailored to suit. The wizard gives access to five distinct aspects of the chart

- [View Style](#) - access to overall view information
- [Axis Styles](#) - access to the positioning, scaling and type of axis
- [Curve Styles](#) - access to colour, style, symbols, etc. for individual curves
- [Point Style](#) - access to colour, style, symbols, etc. for extra points
- [Background Style](#) - access to details of background curves.

3.6.2.1 Chart Styles: View Style

This page gives access to the overall style of the view

Show

Allows the user to switch on or off the following:

- **Legend** - the legend displayed on the right of the chart
- **Border** - the border around the graph area
- **Grid lines** - the grid lines in the graph area
- **Axes** - the axes on the abscissa and ordinate
- **Invert colours** - the background and foreground colours are inverted
- **Symbols** - the symbols on the individual curves
- **Fill** - the area between the curve and axis are displayed as solid colour

Font

This controls the font for basic text displayed on the chart. Both typeface and size can be selected. The buttons allow font sizes of all text to be increased or decreased by one size. This depends on the current font size selected.

Legend

The specifies the width of the legend panel and the size of the legend text with the option of bold or italic.

Title, Sub-title and Note

These are title and sub-title displayed in the header of the chart and the note is displayed at the foot of the legend. The font size and bold or italic can be selected for the title and sub-title.

Chart area to exclude width of legend panel

The chart area can be selected to exclude the width of the legend panel (default) so that the legend is not obscure by the curves. De-selecting this option allow more width for the charts but the legend may overwrite the chart curves.

Bar width

For a bar graph this defines the width of the individual bars

Margin

The margin controls the space around the chart and the extra height for the header increases the space at the top of the chart. This can be useful when other than default text sizes are used.

Cursor tooltip

By default no tooltip is display, but there are options to display a tooltip within the graph area. The options are

- **Coordinates at Cursor** - the tooltip display the abscissa and ordinate values for the current cursor position
- **Curve Values at Abscissa** - the tooltip displays the values of all curves for the current abscissa position. This may not be useful where there are a large number of curves on the chart.
- **Coordinate of nearest point** - the tooltip display the coordinate of the nearest point on any of the displayed curves.

3.6.2.2 Chart Styles: Axis Styles

This allows the display of the axes to be adjusted. Similar options exist for both abscissa and ordinate.

Label and Annotation

The **label** is the text that is displayed on the axis; the **font size** can be adjusted and both **bold** and **italic** can be selected. The **annotation** option allows the values to be included on the axis. The **major** and **minor tick marks** control the number of ticks on the axis.

The label can include the tag <UNIT>. In this case the tag will be replaced by the unit associated with this axis, for example if the axis is time and the current units are seconds, <UNIT> will be replaced by [s]. Where the units are not as clearly defined by the curve data a unit string can be forced by including the unit type explicitly in the tag, for example <FORCE>.

Scaling

Normally the chart is scaled to fit the data - **autoscale**, but can be specified explicitly - **user defined**. When autoscale is selected there is an option to include the zero value in the autoscale calculation.

Axis Type

Depending on the data in the curves they can be displayed on either **linear** or **logarithmic** axes.

Axis Position

The axis can be positioned either on the zero line or as either extremity of the axis.

3.6.2.3 Chart Styles: Curve Styles

This allows control of the individual curves in the chart. Selecting a curve in the list copies the details to the bottom of the page where they can be modified. Right-clicking in the list allows the individual curve to be switched on or off (hide) and symbols to be switched on or off.

Colour

Select a colour for the curve.

Component

Depending on the type of data different options are possible. For a scalar only value and modulus are options, but for vector and tensor curves other components can be selected.

Line Style and Weight

Normally curves are drawn with solid lines but different **line styles** with dots and dashed are available. The **weight** of the line can be adjusted, for example for emphasis.

Symbol Style, Size and Frequency

When the chart is not a bar chart the user can select **symbols** to identify the actual data points. The size can be varied from off, to any **size** from small, medium and large. The **symbol frequency** determines how often these symbols are drawn. When a suitable symbol size has been selected for one curve this can be applied to all with **Set all symbols to this size**.

Hide and Show

The **hide** check box allows individual curves to be switched on or off. The **Hide all** and **Show all** extend this functionality to all curves.

3.6.2.4 Chart Styles: Point Style

In some charts it is possible and useful to identify particular points. They can be either reference points to particular data points, which may or may not lie on particular curves.

Label

This is a label to identify a particular point on the chart.

Colour, Symbol and Weight

This identifies the **colour**, the **symbol** and the **weight** to associate with a particular point. The options for these are the same as for the curves.

Hide and Show

Individual points can be switched on or off and also the label associated with the point. Shortcuts allow all to be switched on or off.

3.6.2.5 Chart Styles: Background Style

In some charts it is useful to show the curves superimposed on a set of background curves.

Background Colour

By default the background will normally be white. This allows the background colour to be changed. The **shade** option allows for plain colour (no shading) or a gradient fill (either bottom to top or left to right). The gradient fill is defined by a **start** and **end colour**. The '**set colour wash**' option sets the end colour to be a lighter shade of the start colour. The colour can be selected from the list of a standard colour selection dialog.

Background Text

The user can add annotation to a chart. This lists all the pieces of text and allows the text to be edited or deleted.

Background Curves

Some charts are superimposed on a set of background curves. At times it is useful to be able to **show** or hide the background curves to provide greater clarity on the main curves.

Each background curve can have a **legend** to identify it. Again these can be switched off for clarity.

When there are many background curves it can be helpful to clarify to **highlight** some of the curves, for example every fifth curve.

3.7 Working with Output Views

Output Views may be used to view input data and results in tabular format. Several Output Views may be open at a time.

A new Output View may be opened using the "View | New Output View" (Alt+2) menu command. "New Output View" is also available on the [GSA toolbar](#). The "Output" button on the [Assisted Input toolbar](#) is equivalent to the "New Output View" command.

The "New Output View" command first displays the [Output Settings dialog box](#). "OK" opens an Output View containing the selected output; "Cancel" exits the dialog box without opening an Output View.

Another way of opening an Output View is via the [Gateway](#). In this case an Output View is opened containing the output selected in the Gateway with the default output specification. Note that the default output specification is "All" cases and "All" nodes, elements or members, so opening an Output View via the Gateway for a large model can cause huge quantities of output to be generated. When working with large models it is recommended that Output Views be opened using the "New Output View" command and that the output specification be carefully specified before requesting the

output to be generated.

Default view settings, preferred views and saved views are available for Output Views. Refer to [Working with Saved Views and Preferred Views](#) for details and to [Working with the Gateway](#) for accessing these.

More:

[Output Settings](#)

[Output View table format](#)

[Selecting data to output](#)

[Case and entity lists](#)

[Outputting for a selection set of entities](#)

[Enveloping](#)

[Data extents](#)

[Output summary](#)

[Output By Case, By Property, By Group](#)

[Output units](#)

[Formatting of numeric output](#)

[Output axes](#)

[Printing from Output Views](#)

[Interacting with spreadsheets](#)

3.7.1 Output Settings

All of the settings that specify the content of an Output View can be set either in the [Output Settings dialog box](#) or in dialog boxes accessible from this. The Output Settings dialog box can be accessed by giving the “Output | Output Settings” (Ctrl+W) menu command or the “Wizard” command on the [Data Options toolbar](#).

When a new Output View is opened the [default view settings](#) are adopted.

Changes to view settings may be undone and redone using the “View | Undo View” (Ctrl+Alt+Z) and “View | Redo View” (Ctrl+Alt+Y) menu commands. These commands are also available on the [Standard toolbar](#).

View settings for the current view may be applied to another Output View by clicking the “View settings painter” on the [Standard toolbar](#) to grab the current settings and then clicking in the other Output View. Click the “View settings painter” again or press <Esc> to switch off the option without applying grabbed settings.

View settings may be copied to the clipboard from within the Output Settings dialog box. View settings on the clipboard may be pasted into the view (i.e. applied to the view) either using “Edit | Paste” (Ctrl+V) or from within the Output Settings dialog box. Note that “Edit | Copy” copies the actual content of the view (see [Interacting with spreadsheets](#)), not the view settings.

There are often other ways of editing the settings, as described below.

3.7.2 Output View table format

Tables of output may be formatted by separating the fields with spaces or with tabs in a standard Output View or by displaying the output in a Grid Output View. Output that is for inspection, presentation or printing is best formatted as space separated; tab separated tables and the Grid Output View are offered to facilitate the copying of output to spreadsheets.

In standard Output Views the table format can be toggled using the “Output | Tab Separate Tables” command. New Output Views are opened in the default Output View table format. The default Output View table format is saved with the [default view settings](#).

A Grid Output View can be opened by giving the 'Output | Grid Window' when a standard Output View is open. The Grid Output View is opened with the same output as is shown in the 'parent' standard Output View and continues to be updated to reflect the 'parent' output while the parent view remains open.

There are advantages and disadvantages with each Output View table format. These are summarised as follows.

Space Separated Tables in Standard Output Views

Generally space separated tables are best for the presentation of output.

Advantages:

- Table columns are separated by spaces, not tabs.
- Long text fields in tables are wrapped within the column width.
- Table columns are correctly aligned.
- Output options may be specified directly.

Disadvantages:

- Attempting to copy a selection to the clipboard produces a message warning that data are separated by spaces.
- Pasting data to a spreadsheet that has been copied from space separated output results in all data per row being pasted into a single cell.

Tab Separated Tables in Standard Output Views

Generally tab separated tables are good for copying records into spreadsheets.

Advantages:

- Data is copied to the clipboard in tab separated format. This results in data being pasted in cells correctly when pasting to spreadsheet.
- Output options may be specified directly.

Disadvantages:

- Table columns are separated by tabs, not spaces. This can result in columns appearing misaligned in the Output View and in output printed from the Output View.
- Long text fields are not wrapped within the column width. Therefore long text fields within a table increase the likelihood of subsequent fields in a row being misaligned.
- Text may be selected by record only.

Grid Output View

Generally Grid Output Views are best for copying tables into spreadsheets.

Advantages:

- Data is copied to the clipboard in tab separated format. This results in data being pasted in cells correctly when pasting to spreadsheet.
- Data may be selected by blocks, columns or rows of cells.
- Tables may be displayed in a grid. ('Window | Settings | Grid')

Disadvantages:

- Output format is optimised for regular tables of output but is not well suited to notes and other non-

tabular output.

- Output options may not be specified directly (though the view does reflect the settings of the parent standard Output View).

Note that the Output View table format is independent of, and should not be confused with the [spreadsheet format](#) option.

3.7.3 Selecting data to output

Several tables can be displayed in a single Output View. Do this by using the Ctrl or Shift key to form multiple selections of data tables in the [Output Settings dialog box](#). Note that multiple selections may not be formed on the Output tab of the Gateway.

Refer to the [Output Settings dialog box](#) documentation for more details.

3.7.4 Case and entity lists

Where relevant, tables are output for those items identified in the case, node, element and member lists. Changing a list from the default of “All” to a specific selection refines the content of the table.

These lists can be specified both in the [Output Settings dialog box](#) and via the [Lists toolbar](#) by first setting the Display Option to the required entity type. The list of entities is then specified in the Display List. The full [GSA list syntax](#) is available when specifying the list.

See [Outputting for a selection set of entities](#) for an automated way of updating entity lists by selecting items in a Graphic View.

Note that “Reset to All Entities” (or “All” in the “[Lists](#)” toolbar) resets entities lists to result in the whole model being included thereby disabling any “<current selection>” settings.

3.7.5 Outputting for a selection set of entities

Sometimes the most convenient method of identifying the entities for which output is required is by selecting the entities in a Graphic View. One way of transporting the current selection set to the entity list for an Output View is to [copy the selection](#) in the Graphic View and then paste this into the [entity list](#) for the Output View.

An automated way of achieving this is, on the [Lists toolbar](#) for the Output View, to set the Display Option to the required entity and then setting the Display List to “<current selection>”. When an entity list is set to “<current selection>” that entity list is set to the current [selection set](#) for that entity type and is updated as the current selection is modified. If the current selection is empty then “All” is assumed. Unlike in Graphic Views, the “<current selection>” setting remains set and the list continues to be updated with the current selection until the “<current selection>” setting is switched off.

3.7.6 Enveloping

Where a case to be output is an enveloping case and the data can be enveloped the output produced consists of the maximum value and minimum value and a reference to an envelope permutation number that produced the maximum or minimum value. Note the reference to envelopes in the [Output summary](#) section.

By default the envelope is produced for all components. It is possible to specify that the envelope be of a particular “subject component” in which case other components are output as the coexistent values for the permutation that produced the maximum or minimum value for the subject component.

Refer to the [Output Settings: Further Options](#) documentation for details on how to specify this.

The scope and limitations of envelopes are described in [Program Fundamentals — Enveloping in](#)

[GSA](#).

3.7.7 Data extents

Where only a particular range of values of a component are of interest, data extents can be specified. These can be expressed in terms of minimum and/or maximum values, signed or as a magnitude, as an inclusive or exclusive range. When data extents have been activated only records for which the value of the “subject component” lies within the range are output.

Refer to the [Output Settings: Further Options](#) documentation for details on how to specify this.

3.7.8 Output summary

Most tables can be summarised in terms of the maximum and minimum values encountered in the table for each component. The table is scanned for the maximum and minimum values of each component; the whole table row in which a maximum (or minimum) value of component is found is then reported in the summary; the component under consideration in that row of the summary (i.e. the maximum or minimum value) is displayed in bold.

For some data components (e.g. element length and element strain energy) a total for that component in the specified table is included in the summary.

Whether or not the table and summary are output can be set in the [Output Settings dialog box](#).

By default the summary includes only the single worst values. It is possible to set the number of worst values in the [Output Settings: Further Options](#). This results in several worst values being output in descending sequence. Note that setting the number of worst values to a large number will result in an output table of values in descending sequence, as opposed to node or element number sequence.

3.7.9 Output By Case, By Property, By Group

The default sequence of lines in a table of output that includes an entity number and a case number per line is that the entities are output in ascending order and the cases are output in ascending order for each entity. This sequence may be altered by selecting “By Case”, “By Property” or “By Group” in the [Output Settings dialog box](#).

Selecting “By Case” causes the table to be split into separate tables per case.

“By Property” causes the table to be split into separate tables per element or member property. The element type is considered when separating the tables so, for example, if there are beam elements with property one and spring elements with property one these will appear in separate tables. However, Beams and Bars of property one will appear in the same table since they share the same property.

“By Group” causes the table to be split into separate tables per element or member group.

“By Property” and “By Group” do not affect tables of nodal data.

3.7.10 Output units

By default the units in which data is output are the [model units](#). These may be adjusted for individual Output Views from the [Output Settings dialog box](#) or by using the “Window | Settings | Units” menu command. “Units” is also available on the [Data Options toolbar](#).

3.7.11 Formatting of numeric output

By default the numeric format in which data is output is as set in the [user preferences](#). This may be adjusted for individual Output Views from the [Output Settings dialog box](#) or by using the “Window | Settings | Numeric Format” menu command. The precision of output can be adjusted using the “Window | Settings | Increase Precision” and “Window | Settings | Decrease Precision” menu commands; these increase or decrease the number of significant figures or decimal places, depending on the current 'Numeric Format' setting. “Numeric Format”, “Increase Precision” and “Decrease Precision” are also available on the [Data Options toolbar](#).

3.7.12 Output axes

By default the axes in which data is output are the default for the data being output. Typically these are global axes though certain data, like 1D element forces and stresses, is in local axes. The output of most data can be transformed to specified axes for individual Output Views from the [Output Settings dialog box](#) or by using the “Window | Settings | Axes” menu command. “Axes” is also available on the [Data Options toolbar](#).

Refer to the [Output Options](#) for details of default axes and of which data can be transformed.

3.7.13 Printing from Output Views

The output may be printed by giving the “File | Print” (Ctrl+P) menu command. The printed image may be previewed by giving the “File | Print Preview” menu command. “Print” and “Print Preview” are also available on the [Standard toolbar](#).

The “Page Setup” command on the [Miscellaneous tab](#) of the Preferences may be used to specify the format of the border on the printed page.

The Print command prints the output in the same font as is used on the screen.

The output may be printed condensed (i.e. in a small font). Typically the smallest font that is legible on printed output is smaller than that on the screen. Specify the condensed print setting using the “Window | Settings | Font” command. “Font” is also available on the Data Options toolbar. The print condensed setting is stored as a preference.

3.7.14 Interacting with spreadsheets

The “Fully populate fields” and “Output all data per entity on a single row” spreadsheet formatting options are offered to assist in the interaction between Output Views and spreadsheets. These are specified in the [Output Wizard: Further Options](#) dialog.

“Fully populate fields” arranges the output in a format that is convenient for using spreadsheet filtering options (e.g. MS Excel 'Filter' and 'PivotTable'). Column header fields and row lead fields are fully populated and the column header row is written immediately above the table.

“Output all data per entity on a single row” arranges tables such that all output for a given entity is output on one line, concatenating what would otherwise be several lines. Note that when copying data in order to paste it into a spreadsheet it is advisable either to copy from a Grid Output View or to set “Tab Separate Tables” on. See [Output View table format](#) for details.

The export option in the [Output Settings dialog box](#) writes the output as currently specified (but not necessarily currently displayed) to file in either tab or comma delimited text format or in HTML. Note that it is possible to use this option without displaying the output in an Output View. Do this by giving the “New Output View” command, then, in the Output Settings dialog box, selecting the required output, exporting, and then cancelling from the dialog box.

The “File | Export | Output” menu command writes the output as currently displayed to file in either tab or comma delimited text format or in HTML.

Copying a selection from the Output View, using “Edit | Copy” (Ctrl+C), copies the selected data onto the clipboard in a format that is acceptable by most spreadsheet programs.

3.8 Tools

This section includes topics on tools that manipulate the model, including sculpting tools that manipulate the model graphically.

3.8.1 Sculpting

Editing the model in a [Graphic View](#) is referred to as sculpting. Generally sculpting is done either by using the mouse directly on the image in one of the [Sculpt Geometry Cursor Modes](#) or by executing a sculpt command on a [selection set](#) or [polyline](#).

All sculpt commands are accessible from the Sculpt menu. Sculpt Geometry Cursor Modes are also accessible from the [Sculpt toolbar](#). Other frequently used sculpt commands are accessible from the Sculpt toolbar by selecting the required command in the Sculpt Command field and then executing this by clicking the Execute Sculpt Command button. The Sculpt Command field is initialised with a list of commonly used sculpt options. However this list is dynamic in that the most recently used command is placed at the top of the list, whether the command has been picked from the Sculpt menu or from the Sculpt Command field. Several 2D element sculpt commands are appended to the list when 2D elements are present in or added to the model. This functionality allows rapid repetition of a sculpt operation and easier access to all sculpt commands used frequently within a session. The Sculpt toolbar may be switched on and off from the [Cursor Mode toolbar](#) as well as by giving the “View | Toolbars | Sculpt” menu command.

To abort an operation, press Escape or click outside the Graphic View. This will not undo operations already carried out on nodes or elements but will abort the current operation. The “Edit | Undo” command can be used to undo operations.

More:

[Use of data defaults when sculpting](#)

[Sculpt geometry cursor modes](#)

[Creating user axes graphically](#)

[Creating grid planes graphically](#)

[Adding nodes graphically](#)

[Modifying nodes graphically](#)

[Collapsing coincident nodes](#)

[Adding elements graphically](#)

[Connecting 1D elements graphically](#)

[Splitting elements graphically](#)

[Moving and copying nodes and elements graphically](#)

[Flexing lines of nodes graphically](#)

[Flipping elements graphically](#)

[Spinning 2D elements graphically](#)

[Modifying elements graphically](#)

[Disconnecting elements graphically](#)

[Deleting nodes and elements graphically](#)

[Creating rigid constraints graphically](#)

[Creating joints graphically](#)

[Creating nodal loading graphically](#)

[Creating element loading graphically](#)

[Creating grid loading graphically](#)

[Deleting loading graphically](#)

3.8.1.1 Use of data defaults when sculpting

Many of the sculpt operations apply default values to the attributes of the entities that are being sculpted. The default values used for this purpose are the data defaults for the model. These may be edited in the Data Defaults dialog box.

Note that:

- The data defaults are also used by Table Views and may be edited from these
- Not all Table View defaults are represented in the Data Defaults dialog box
- Existing data are not affected by changes to default values
- Refer to [Program Data](#) for descriptions of the data for which these are the defaults.

Open the Data Defaults dialog box by giving the "View | Data Defaults" menu command or from the [GSA toolbar](#).

A quick way of setting the data defaults to be the same as an existing entity (i.e. node, element, member, line, area or region) is to [right-click](#) on the node or element and select the "Set <entity> Defaults to This" command.

The current default settings can be applied to an entity by right-clicking on the entity and selecting the "Apply Defaults" command, or to several entities by [selecting](#) the entities and giving the "Sculpt | Apply Defaults to Selection" command.

3.8.1.2 Sculpt geometry cursor modes

There are several [cursor modes](#) available for sculpting the geometry of a model. New nodes and elements can be created, the attributes of existing nodes can be modified and existing nodes can be moved (resulting in elements attached to these moving). These operations are carried out by first selecting the appropriate sculpt tool cursor mode and then using the cursor on the graphical image. The sculpt tools may be selected from the Sculpt toolbar or the "Sculpt | Sculpt Geometry Cursor Modes" menu.

More:

[Add Nodes Sculpt Tool](#)

[Modify Nodes Sculpt Tool](#)

[Drag Nodes Sculpt Tool](#)

[Add Elements Sculpt Tool](#)

[Modify Elements Sculpt Tool](#)

[Add Lines Sculpt Tool](#)

3.8.1.2.1 Add Nodes Sculpt Tool

Click at the position the new node is required. If an existing node is selected the option to create a new coincident node is offered. Otherwise if the construction grid is switched on a new node is created at the [grid coordinates](#). The attributes of the new node are set to the Node data defaults. The new node is displayed as an unattached node.

3.8.1.2.2 Modify Nodes Sculpt Tool

Click on an existing node. The attributes of the selected node are immediately changed to the Node data defaults. This may be useful for applying restraints. Another way of achieving the same is to use the [Modify Nodes sculpt command](#).

3.8.1.2.3 Drag Nodes Sculpt Tool

Dragging an existing node results in the node being moved to the new [grid coordinates](#). Nodes are moved within a plane parallel to the construction grid. This is achieved by the program temporarily adjusting the grid elevation to the elevation of the node while it is being dragged.

If the drag is on to another existing node then the operation takes into account the existing node, depending on the current 'drag on node' setting:

Drag on node - Collapse — the dragged node is deleted and references to the dragged node are replaced with references to the target node. So, for example, an element attached to the dragged node will end up being attached to the target node.

Drag on node - Joint — the dragged node is moved to the same position as the target node and a joint is created to connect the dragged and target nodes. The target node is made the master of the joint and the default linkage is applied.

Drag on node - Co-locate — the dragged node is moved to the same position as the target node.

Drag on node - Ignore — the target node is ignored.

The current 'drag on node' setting may be set via the 'Sculpt | Sculpt Geometry Cursor Modes' menu, or the right-click menu when in the Drag Nodes Sculpt Tool.

3.8.1.2.4 Add Elements Sculpt Tool

The type of element created by this tool is as set in the [Data Defaults dialog box](#). Any element type can be created by this method. Click on nodes in the desired topological sequence for the new element. The new element is created when the number of nodes required for the element type have been selected. Ctrl+Click when selecting the last node for an element will cause that node also to be the first node of the next new element.

When the construction grid is switched on clicking somewhere where there is no node causes a new node to be created at the grid coordinates with default attributes. This new node then becomes the next item in the topology sequence of the new element.

When creating 1D elements a new node can be created for end 2 on the line of an existing 1D element by clicking on the existing element. The position of the new node is determined by the current 'snap' setting:

Snap - Centre — the new node is located at the centre of the existing element.

Snap - Perpendicular — the new node is located such that the new element is normal to the existing.

Snap - Align with Grid Axis — the new node is located such that the new element is parallel to the most appropriate axis of the current grid plane.

The current 'snap' setting may be set via the 'Sculpt | Sculpt Geometry Cursor Modes' menu, or the right-click menu when in the Add Element Sculpt Tool.

To achieve the expected connectivity the existing element is split at the new node. In some circumstances the new node will be located beyond the end of the existing element, in which case the existing element is not split and a new element is created along the 'extension' of the existing element.

3.8.1.2.5 Modify Elements Sculpt Tool

Click on an existing element. The attributes of the selected element are immediately changed to the Element data defaults. Another way of achieving the same is to use the [Modify Elements sculpt command](#).

3.8.1.2.6 Add Lines Sculpt Tool

The type of line created by this tool can be set using the “Set Line Defaults as This” right-click menu command in a Graphic View. Any line type can be created by this method. Click on nodes in the desired topological sequence for the new line. The new line is created when the number of nodes required for the line type have been selected. Ctrl+Click when selecting the last node for an line will cause that node also to be the first node of the next new line.

When the construction grid is switched on clicking somewhere where there is no node causes a new node to be created at the grid coordinates with default attributes. This new node then becomes the next item in the topology sequence of the new line.

A new node can be created for end 2 on the line of an existing line by clicking on the existing line. The position of the new node is determined by the current 'snap' setting:

Snap - Centre — the new node is located at the centre of the existing line.

Snap - Perpendicular — the new node is located such that the new line is normal to the existing.

Snap - Align with Grid Axis — the new node is located such that the new line is parallel to the most appropriate axis of the current grid plane.

The current 'snap' setting may be set via the 'Sculpt | Sculpt Geometry Cursor Modes' menu, or the right-click menu when in the Add Line Sculpt Tool.

To achieve the expected connectivity the existing line is split at the new node. In some circumstances the new node will be located beyond the end of the existing line, in which case the existing line is not split and a new line is created along the 'extension' of the existing line.

3.8.1.3 Creating user axes graphically

New axes can be created or existing axes overwritten using the “Sculpt | Create User Axes” menu command. Two methods are available, as follows:

Create User Axes from Selected Points

Before giving the command two nodes or grid points must have been [selected](#) (or three for Space structures). These will be used to define the required axis set. Selected items will be used to define:

- The origin of the axis set.
- The x axis vector.
- The xy vector. (This is only required for Space structure types.)

Giving the Create User Axes command displays the [Create User Axes dialog box](#) in which the number, name and type of the axis set are specified and the nodes or grid points defining the origin, x axis and xy vector are identified. The user axis set is then created.

Create User Axes from Selected Elements

[Select](#) one or more elements before giving the command. New user axis sets will be created based on the element axes and the position of the first node of each of the selected elements.

3.8.1.4 Creating grid planes graphically

The “[Set Grid Plane to This](#)” command and clicking on a node when [forming a polyline](#) will both offer the option to set up a new grid plane if an appropriate one is not available. Otherwise, new grid planes can be created using the “Sculpt | Create Grid Plane” menu command. The procedure is as follows.

1. Set the [cursor mode](#) to Select Nodes or Select Elements.
2. Select some items to define the location of the plane.
3. Give the “Create Grid Plane” command.
4. If an existing grid plane is found upon which the selected items lie, the option is given to simply set that as the current grid.
5. If an existing axis set with an xy plane parallel to the plane on which the selected items lie is found, this is identified as the grid axis for the new grid plane. Otherwise an appropriate axis set is defined.
6. If elements have been selected then those elements are identified as the element list for the new grid plane.
7. The [Grid Plane Definition](#) dialog box opens. Edit as required.
8. On pressing OK the new grid plane is created and so to is the new axis, if required. The option is given to set this new grid plane as the current grid.

3.8.1.5 Adding nodes graphically

Nodes may be created one at a time using the [Add Nodes Sculpt Tool](#), or a number of nodes can be created at once using the “Sculpt | Add Nodes” menu command. The procedure is as follows.

1. The attributes assigned to new nodes are taken from the data defaults as set in the [Data Defaults dialog box](#).
2. Switch the [construction grid](#) on.
3. Adjust the construction grid such that grid points occur at desired nodal positions.
4. Set the [cursor mode](#) to Select Grid Points.
5. Select grid points where new nodes are required.
6. Give the “Add Nodes” command.

3.8.1.6 Modifying nodes graphically

The coordinates of individual nodes may be changed using the [Drag Nodes Sculpt Tool](#) and of multiple nodes using the [move nodes command](#).

The [right-click menu](#) that is displayed when the cursor is pointing at a node offers commands to view the current attributes of the node, to open the [Node Wizard](#) for that node and to open the [Nodes Table View](#) at that node.

The support conditions and constraint axes of nodes can be modified using the “Sculpt | Modify Selection” menu command when the cursor mode is set to Select Nodes. “Modify Nodes” is also available on the [right-click menu](#) when the cursor mode is set to Select Nodes. The procedure is as follows.

1. Set the [cursor mode](#) to Select Nodes.
2. Select the nodes to be modified.
3. Give the “Modify Selection” command.
4. Specify the required changes to the attributes in the [Modify Nodes dialog box](#).

3.8.1.7 Collapsing coincident nodes

Coincident nodes can be collapsed using the “Sculpt | Collapse Coincident Nodes” menu command. The procedure is as follows.

1. Set the [cursor mode](#) to Select Nodes.
2. Select the nodes to be collapsed. Note that only coincident nodes will be collapsed so there is no harm in selecting nodes that are not coincident.
3. Give the “Collapse Coincident Nodes” command.
4. Specify the coincidence tolerance, i.e. the tolerance used to establish whether two nodes are coincident. (This value defaults to the tolerance for coincidence specified on the [Sculpting page](#) of the [Preferences dialog box](#).)

In a set of coincident nodes the lowest numbered node is retained; higher numbered nodes are deleted.

Where several nodes being collapsed have differing restraints or constraints the retained node is assigned the super-set of the constraint conditions of all the coincident nodes; - and similar for mesh generation flags.

Nodes that are connected by a Joint are not collapsed.

Note that bizarre modifications to the topology of the model will occur if the coincidence tolerance is too coarse.

Note also that various other sculpt commands offer the option to collapse coincident nodes created during that sculpt operation.

3.8.1.8 Rounding nodal coordinates graphically

Nodal coordinates can be rounded to the nearest specified length using the “Sculpt | Round Coordinates” menu command. The procedure is as follows.

1. Set the [cursor mode](#) to Select Nodes.
2. Select the nodes to be rounded.
3. Give the “Round Coordinates” command.
4. Specify the length to which coordinates are to be rounded.

Note that a “Round Coordinates” command is also offered on the right-click menu when the Nodes table is displayed. This rounds coordinates for all nodes in the model.

3.8.1.9 Adding geometric entities graphically

Lines may be added directly using the [Add Lines Sculpt Tool](#). New lines can also be created in sculpt by [copying](#) and [splitting](#) existing lines.

Other [non-graphical tools](#) are also available for manipulating geometric entities.

The following specific sculpt options are available for adding geometric entities.

More:

[Create Lines from 1D Elements command](#)

3.8.1.9.1 Create Lines from 1D Elements command

Lines can be created based on the current selection of 1D elements or members using the “Sculpt | Geometric Entity Operations | Create Lines from 1D Elements” menu command. The procedure is as follows.

1. Work from the Design Layer to create from members and the Analysis Layer to create from

elements.

2. The default line attributes are assigned to new lines so set the desired [line defaults](#).
3. Set the [cursor mode](#) to Select Lines.
4. Select lines. A new line will be created for each selected element, based on the topology of the element.
5. Give the "Create Lines from 1D Elements" command.

3.8.1.10 Adding elements graphically

The most direct way of sculpting in individual new elements is to use the [Add Elements Sculpt Tool](#). New elements can also be created in sculpt by [copying](#) and [splitting](#) existing elements and by [Generating 2D element meshes](#).

Other [non-graphical tools](#) are also available for constructing a model.

The following specific sculpt options are available for adding elements.

More:

[Adding a string of 1D elements](#)

[Create 1D Elements from Lines command](#)

3.8.1.10.1 Adding a string of 1D elements

A string of 1D elements can be created using the "Sculpt | 1D Element Operations | Add String of 1D Elements" menu command. The procedure is as follows.

1. The element type and other attributes assigned to new elements are taken from the data defaults as set in the [Data Defaults dialog box](#).
2. Set the [cursor mode](#) to Select Grid Points or Select Nodes.
3. If selecting grid points then new nodes will be created at the grid point positions. The attributes assigned to the new nodes are taken from the data defaults as set in the [Data Defaults dialog box](#).
4. Select grid points or nodes.
5. Give the "Add String of 1D Elements" command.
6. Select the mode of operation:
 - o **String elements along line** – New elements will be strung generally along the line of the items selected. If the selected items do not lie vaguely along a line then the sequence in which the items are connected by elements may not be as expected.
 - o **String element to next closest** – New elements will be strung from the first node on the line of items selected to the closest item selected, and from there on to the closest unused item, and so on.

3.8.1.10.2 Create 1D Elements from Lines command

1D elements can be created based on the current selection of lines using the "Sculpt | 1D Element Operations | Create 1D Elements from Lines" menu command. The procedure is as follows.

1. Work from the Design Layer to create members and the Analysis Layer to create elements.
2. The element type and other attributes assigned to new elements are taken from the data defaults as set in the [Data Defaults dialog box](#). The element type should be set to a 1D element type, such as Beam or Bar.
3. Set the [cursor mode](#) to Select Lines.

4. Select lines. A new element will be created for each selected line, based on the topology of the line (ignoring the third node of arcs).
5. Give the "Create 1D Elements from Lines" command.

3.8.1.11 Connecting 1D elements graphically

1D elements that intersect, within a defined tolerance, can be connected by being split at the point of intersection and joined to a common node using the "Sculpt | 1D Element Operations | Connect 1D elements" menu command. (This documentation is also applicable to the "Sculpt | Geometric Entity Operations | Connect Lines" menu command.)

The standard case is where two elements that cross are connected. Sculpt creates a new node at the point of intersection and replaces the two original elements with four elements meeting at the new node.

An offset tolerance is specified for the operation. This defines the maximum separation of two elements at the point at which they "cross" for the elements to be deemed to be intersecting.

A trim tolerance is also specified. If an element crosses another within "trim tolerance" of the element end then the element is trimmed to end at the crossing point.

Generally the attributes of the original elements are adopted by the new elements and the local directions of the new elements generally align with those of the original. However, the element releases in the new elements are established to result in releases at only the existing positions; a release at end one of the original element results in a release at end one of only the element at that position.

The option is given to split loads and replace references to original elements with references to new elements. Note that the splitting of loads does not occur where a load is applied to an element because that element is included in a saved list or grid plane that is referred to by the loading record rather than being referred to directly in the loading record list. In this circumstance references to the old element in the saved list or grid plane are replaced by references to the new split elements. For example:

Element 12 has a linearly varying load applied to it and is split. If the element list for the beam load record is "12" then that load will be split correctly, whereas if the element list is "#1" and element 12 is included in saved list 1 then that load will not be split. However, in list 1 reference to element 12 will be replaced by the numbers of the new split elements, resulting in a saw-tooth load pattern along the new elements. (The "Delete Displayed Loading" command may be helpful in rectifying such problems.)

The procedure is as follows.

1. Set the cursor mode to Select Elements. Two or more elements must be selected.
2. Select the elements to be connected. Elements that are not 1D will be ignored by this operation.
3. Give the "Connect 1D Elements" command.
4. Specify how the elements are to be split in the [Connect 1D Elements](#) dialog box.

3.8.1.12 Splitting elements graphically

1D and 2D elements can be split. Various methods are available for doing this, as described below.

Generally the attributes of the existing element are adopted by the new elements and the local directions of the new elements generally align with those of the existing. However, the element releases in the new elements are established to result in releases at only the existing positions; a release at end one of the existing element results in a release at end one of only the element at that position.

Note that bizarre modifications to the topology of the model will occur if coincident nodes are collapsed during the operation and the coincidence tolerance is too coarse.

More:[Splitting 1D elements](#)[Splitting 2D elements](#)

3.8.1.12.1 Splitting 1D elements

1D elements can be split using the “Sculpt | Split 1D Elements” menu command.

The option is given to split loads and replace references to original elements with references to new elements. Note that the splitting of loads does not occur where a load is applied to an element because that element is included in a saved list or grid plane that is referred to by the loading record rather than being referred to directly in the loading record list. In this circumstance references to the old element in the saved list or grid plane are replaced by references to the new split elements. For example:

Element 12 has a linearly varying load applied to it and is split. If the element list for the beam load record is “12” then that load will be split correctly, whereas if the element list is “#1” and element 12 is included in saved list 1 then that load will not be split. However, in list 1 reference to element 12 will be replaced by the numbers of the new split elements, resulting in a saw-tooth load pattern along the new elements.

A warning is given when this circumstance arises (though it's always worth checking). The “[Delete Displayed Loading](#)” command may be helpful when rectifying such problems.

The procedure is as follows.

1. Set the cursor mode to Select Elements.
2. Select the elements to be split. Elements that are not 1D will be ignored by this operation.
3. Give the “Split 1D Elements” command.
4. Specify how the elements are to be split in the [Split 1D Elements dialog box](#).

3.8.1.12.2 Splitting 2D elements

The 2D element splitting options described below are sometimes used as a means of generating a 2D element mesh. Note that an option for [generating 2D element meshes](#) is also available in GSA.

In each of the 2D element splitting options the shape of the existing element is interpolated recognising the shape function of the element. Thus splitting a Quad 8 element, which is a parabolic element, results in new elements fitting the implied parabolic surface.

The option is given to split loads and replace references to original elements with references to new elements. Note that the splitting of loads does not occur where a load is applied to an element because that element is included in a saved list or grid plane that is referred to by the loading record rather than being referred to directly in the loading record list. In this circumstance references to the old element in the saved list or grid plane are replaced by references to the new split elements, which may have bizarre consequences. A warning is given when this circumstance arises (though it's always worth checking). The “[Delete Displayed Loading](#)” command may be helpful when rectifying such problems.

Splitting Quad elements

Quad 4 and Quad 8 elements can be split into an equal number of elements along opposite edges using the “Sculpt | Split Quad Elements” menu command. The procedure is as follows.

1. Set the [cursor mode](#) to Select Elements.
2. Select the elements to be split. Elements that are not Quad 4 or Quad 8 will be ignored by this operation.
3. Give the “Split Quad Elements” command.

4. Specify how the elements are to be split in the [Split Quad Elements dialog box](#).

Refining Quad and Tri elements

2D elements meeting at a node (or a selection of nodes) can be split to result in a mesh refinement at the node(s). The procedure is as follows.

1. Set the [cursor mode](#) to Select Nodes.
2. Select the node at which attached 2D elements are to be refined. Non-2D elements will be ignored by this operation.
3. Give the “Refine Mesh at Nodes” command.

Understandably, refining the mesh at more than one node on an element will have bizarre consequences.

Alternatively, Quad 4 and Quad 8 elements can be split into two elements along adjacent edges using the “Sculpt | Refine Quad Elements” menu command. This may be used to bring about a mesh refinement at a corner of the existing element. The procedure is as follows.

4. Set the [cursor mode](#) to Select Elements.
5. Select the elements to be refined. Elements that are not Quad 4 or Quad 8 will be ignored by this operation.
6. Give the “Refine Quad Elements” command.
7. Specify corner at which the elements are to be refined in the [Refine Quad Elements dialog box](#).

Split Quad to Tri elements

Quad 4 and Quad 8 element can be split into Triangle 3 and Triangle 6 elements respectively using the “Sculpt | Split Quad to Tri Elements” menu command. Each quad element is split into two tri elements such that the split occurs at across the corner with largest internal angle. The procedure is as follows.

1. Set the [cursor mode](#) to Select Elements.
2. Select the elements to be split. Elements that are not Quad 4 or Quad 8 will be ignored by this operation.
3. Give the “Split Quad to Tri Elements” command.

Splitting Tri elements

Triangle 3 and Triangle 6 elements can be split using the “Sculpt | Split Tri Elements” menu command. The elements can be split into either Triangle or Quad elements. When splitting into Triangle elements the split must be into an equal number of elements along each edge. The split options available when splitting into Quad elements are described in the documentation for the [Split Tri Elements dialog box](#). The procedure is as follows.

1. Set the [cursor mode](#) to Select Elements.
2. Select the elements to be split. Elements that are not Triangle 3 or Triangle 6 will be ignored by this operation.
3. Give the “Split Tri Elements” command.
4. Specify how the elements are to be split in the Split Tri Elements dialog box.

3.8.1.13 Joining 1D elements graphically

1D elements that form a linear string, within the straightness tolerance (ref. [Preferences | Sculpting | Straightness Tolerance](#)), can be joined up into one element using the “Sculpt | 1D Element Operations | Join String of 1D elements” menu command. (This documentation is also applicable to the “Sculpt | Geometric Entity Operations | Join String of Lines” menu command.)

The string is broken by:

- a node that is connected to more than two elements,
- a node that is restrained or otherwise constrained,
- a node that is loaded,
- a change in element property or group.

The attributes of the first element in the string are applied to the new element. The new element is offset by the offset amount at each end of the string of elements.

References to the original string of elements are replaced by with references to the new element.

Loads applied to the string of elements are transferred to the new element. Note that the transfer of loads does not occur where a load is applied to an element because that element is included in a saved list or grid plane that is referred to by the loading record rather than being referred to directly in the loading record list. In this circumstance references to the old element in the saved list or grid plane are replaced by references to the new split elements.

3.8.1.14 Modifying 2D elements from linear to parabolic

Linear 2D elements (i.e. Tri3 and Quad4) can be converted to parabolic (i.e. Tri6 and Quad8, respectively) using the “Sculpt | 2D Element operations | Modify Linear 2D to Parabolic” menu command. The procedure is as follows.

1. Set the [cursor mode](#) to Select Elements.
2. Select the 2D elements that are to be converted.
3. Give the “Modify Linear 2D to Parabolic” command.

The option to convert linear 2D elements to parabolic is also available in the [Modify Elements dialog box](#).

3.8.1.15 Moving and copying entities graphically

The sculpt move and copy commands operate similarly. Both can operate on nodes, elements, members, lines, areas or regions. Both operate by shifting, rotating or reflecting the selected items. The amount of shift can be specified by a [polyline](#).

When moving nodes, the selected nodes are moved. When moving other entities, the associated nodes are moved if they are associated only with entities that are being moved; if the associated nodes also have entities (of the same type) attached that are not being moved then new nodes are created at the new topology positions.

When copying nodes, only copies of the nodes are created. When copying other entities, copies of the entities and attached nodes are created.

When copying, the option is given to copy references to the copied entities to result in constraints and loading etc. being copied.

Existing entities can be moved and copied using the “Sculpt | Move Selection” and “Sculpt | Copy Selection” menu commands, respectively. The procedure is as follows.

1. If the shift vector is to be specified by a polyline:
 - 1a. Set the [cursor mode](#) to Polyline.
 - 1b. [Form a polyline](#) to define the shift vector; — the vector from the first point to the second point is used.
2. Set the [cursor mode](#) to the required selection mode.
3. Select the items to be moved or copied.
4. Give the “Move Selection” or “Copy Selection” command.
5. Specify the action to be taken in the respective [Move or Copy](#) dialog box.

Note that bizarre modifications to the topology of the model will occur if coincident nodes are collapsed during the operation and the coincidence tolerance is too coarse.

3.8.1.16 Extruding nodes and elements graphically

The sculpt extrude command generates nodes and elements by a specified number and length of increments. The extrusion is based on either a [polyline](#) or a set of nodes or elements and is generated along an axis of a specified axis set or along an alignment, if any exist.

An extrusion can be formed using the “Sculpt | Extrude Selection” menu commands. The procedure is as follows.

1. Ensure that an appropriate [axis set](#) or [alignment](#) exists.
2. Set the [cursor mode](#) to Select Nodes or Select Elements or Polyline.
3. Select the nodes or elements or polyline to form the basis of the extrusion.
4. Give the “Extrude Selection” command.
5. Specify the action to be taken in the [Extrude](#) dialog box.

Note that the generated nodes and elements may be bizarre if either coincident nodes are collapsed during the operation and the coincidence tolerance is too coarse or if the extrusion is based on a set of nodes that cannot be ordered sensibly along a line.

3.8.1.17 Transforming nodes graphically

The sculpt transform geometry command moves selected nodes to positions expressed as a function of the original positions of the nodes. Normal mathematical notation is used in [expressions](#). For example:

$$z + \text{abs}(\sin(x)) * \pi$$

Operations may be with respect to standard (e.g. global) or user defined axis sets.

Existing nodes can be transformed using the “Sculpt | Transform Geometry” menu command. The procedure is as follows.

1. Set the [cursor mode](#) to Select Nodes.
2. Select the nodes to be transformed.
3. Give the “Transform Geometry” command.
4. Specify the action to be taken in the Transform Geometry dialog box.

3.8.1.18 Flexing lines of nodes graphically

The flex command flexes a set of nodes into a specified shape. Typically the nodes are originally in a line, though this need not necessarily be the case. One of the set of nodes is identified as being the one to be shifted explicitly and the new position of this node is specified. The flex shape can be specified as linear, circular, elliptical or parabolic. When flexing linearly any node can be selected as the shift node; otherwise, the nodes at the ends of the line of nodes may not be shifted. End nodes are not flexed except when an end node is explicitly shifted in a linear flex. The flex operation moves the nodes to result in the specified shape, anchored by the end nodes and the explicitly shifted node.

Nodes can be flexed using the “Sculpt | Flex Selection” menu command. “Flex Selection” is also available on the right-click menu that is displayed when the cursor mode is set to Select Nodes. Invoking Flex via the “Sculpt | Flex Selection” menu command assumes the middle node as the node to be explicitly shifted; Flex via the right-click menu requires that a node has been right-clicked and this node is assumed as the shift node. Either way the shift node can be re-selected from within the

Flex dialog box. Since right-clicking on the shift node is the quicker method, that method is described in the following procedure.

1. Set the cursor mode to Select Nodes.
2. Select the nodes to be flexed.
3. Right-click on the node to be shifted explicitly.
4. Click on the “Flex Selection” command on the floating menu.
5. Specify the flex in the Flex dialog box.

Flex is a high level operation designed to cope with a wide variety of circumstances. Understanding the internal processes will help determine the outcome of some, apparently less predictable, flex requests. The following notes describe some of the internal flex processes.

1. On entry to the Flex dialog box the current set of nodes is organised, internally, into a line, referred to as the flex line. The flex line is established as follows.
 - The nodes are ordered along a vector pointing from the most extreme node to the centre of the set of nodes.
 - The end nodes are identified.
 - Nodes are re-ordered along a vector stretching between the end nodes.
2. This internal organisation of the nodes is the starting point of any flex operation. So, flexing a set of randomly dispersed nodes linearly by shifting an end node by nothing will result in that set of nodes being dispersed along the line between end nodes.
3. When flexing linearly:
 - Any of the set of nodes may be selected as the shift node.
 - The nodes to be flexed are dispersed along the line(s) formed between the end nodes and the shifted node.
 - “Even” spacing results in separate, even spacing on either side of the shifted node.
 - “Proportional” spacing results in spacing that is proportional to the position of the nodes **on the flex line**.
4. When flexing in circular shape:
 - Only internal nodes (not end nodes) of the set of nodes may be selected as the shift node.
 - If the circle radius is not specified the centre and radius of the circle are established from the three points on the circle: the two end nodes and the shifted node.
 - If the circle radius is specified the centre of the circle is established from the two end nodes and the radius. Note that the shifted position of the shift node is used only to indicate the general direction of the flex. The circular arc is formed in the plane of the end nodes and the shifted position of the shift node, arcing towards the shifted position. The shift node is then flexed onto the circle, which is not necessarily the specified shift position.
 - Spacing is then based upon angles around the centre.
 - “Even” spacing results in separate, even angles on either side of the shift node.
 - “Proportional” spacing results in angles that are proportional to the position of the nodes **on the flex line**.
5. The methods for flexing in elliptical and parabolic shapes are similar, as follows.
 - Only internal nodes (not end nodes) of the set of nodes may be selected as the shift node.
 - An ellipse/parabola is established symmetrical about a line normal to the flex line, through the end nodes and the position of the shift node shifted by only the component normal to the flex line.

- The nodes to be flexed are dispersed along the flex line.
 - “Even” spacing results in separate, even spacing on either side of the shift node along the flex line.
 - “Proportional” spacing results in spacing undisturbed from the ordered position on the flex line.
 - The nodes are then flexed normal to the flex line onto the parabola.
 - If the shift node has been shifted by a component parallel to the flex line then the ellipse/parabola is sheared to result in the shift node arriving at its shifted position.
 - Note that this method does not result in flexed nodes that are actually evenly or proportionally spaced.
6. Use the “Preview” command in the Flex dialog box to confirm the consequence of a Flex operation before committing to the operation.

Note that bizarre modifications to the topology of the model will occur if coincident nodes are collapsed during the operation and the coincidence tolerance is too coarse.

3.8.1.19 Straightening lines of nodes graphically

Ragged lines of nodes can be straightened using the “Sculpt | Straighten Line of Nodes | Straighten Spacing Proportionally” and “Sculpt | Straighten Line of Nodes | Straighten Spacing Evenly” menu commands. The “Straighten Spacing Proportionally” command maintains the nodal positions as projected onto the line between the two extreme nodes; the “Straighten Spacing Evenly” spaces the nodes at equidistant positions along that line. The procedure is as follows.

1. Set the [cursor mode](#) to Select Nodes.
2. Select the nodes to be straightened.
3. Give the “Straighten Spacing Proportionally” or “Straighten Spacing Evenly” command.

3.8.1.20 Flipping elements graphically

Flipping a 1D element has the effect of reversing the direction of the local x axis. Flipping a 2D element turns that element upside-down. The flip operation adjusts the position of any element releases or offsets with respect to the element topology to result in these remaining at the same position in space.

Elements can be flipped using the “Sculpt | Flip Elements” menu command.

Elements can be flipped selectively to result in consistent element axes using the following sculpt commands.

- The “Sculpt | 1D Element Operations | Align 1D Elements” menu command aligns the element x axis of the selected elements such that it points in the direction of the current grid axis to which the element x axis is most closely aligned, flipping elements as required to achieve this.
- The “Sculpt | 2D Element Operations | Align 2D Element Z Axes” menu command aligns the element z axis of the selected elements such that it points in the direction of the current grid axis to which the element z axis is most closely aligned, flipping elements as required to achieve this.

The procedure for these sculpt commands is as follows.

1. Set the [cursor mode](#) to Select Elements.
2. Select the elements to be flipped.
3. Give the respective sculpt command.

Note that these operations only affect data in the elements module, namely element topology, releases and offsets. E.g. it does not adjust loading data therefore loads are effectively flipped with the elements.

3.8.1.21 Spinning 2D elements graphically

Spinning a 2D element has the effect of rotating the element about its z axis such that the second edge becomes the first etc. The spin operation adjusts the position of any element releases or offsets with respect to the element topology to result in these remaining at the same position in space.

Elements can be spun using the “Sculpt | Spin 2D Elements” menu command.

Elements can be spun selectively to result in the first edges pointing in consistent directions using the “Sculpt | 2D Element Operations | Align 2D Element Edges” menu command. For each of the selected elements the element is spun such that the first edge (1st node to 2nd node) points in the x axis direction of the specified axis set. The axis set may be specified as either the current grid axes or the local element axes.

The procedure for these sculpt commands is as follows.

1. Set the [cursor mode](#) to Select Elements.
2. Select the elements to be spun.
3. Give the “Spin 2D Elements” command.

Note that this operation only affects data in the elements module, namely element topology, releases and offsets. E.g. it does not adjust loading data therefore loads are effectively spun with the elements.

3.8.1.22 Modifying elements graphically

The [right-click menu](#) that is displayed when the cursor is pointing at an element offers commands to view the current attributes of the element, to open the Element, Property and Material Wizards for that element and to open the Elements, Property and Materials Table Views at the record that relates to that element.

The attributes of elements, including the element type, can be modified using the “Sculpt | Modify Selection” menu command when the cursor mode is set to Select Elements. “Modify Elements” is also available on the [right-click menu](#) when the cursor mode is set to Select Elements. The procedure is as follows.

1. Set the [cursor mode](#) to Select Elements.
2. Select the elements to be modified.
3. Give the “Modify Selection” command.
4. Specify the required changes to the attributes in the [Modify Elements dialog box](#).

3.8.1.23 Disconnecting elements graphically

Elements can be disconnected from the existing structure at a specified nodal position using the “Sculpt | Disconnect Elements” menu command.

An element is disconnected by the creation of a new node, located at the same position as the original and with the same attributes (constraint axis, restraints, etc.) as the original. The topology of the selected element is then adjusted to refer to the new node instead of the original. The option is given to create a joint of default linkage that joins the new and original nodes.

Several elements can be disconnected in one operation. Where several are disconnected from the same node in one operation they are connected to the same new node.

In the situation where several elements are to be disconnected from the same node but the node appears at different nodal positions in the topology lists for some of the elements the “[Flip Elements](#)” command may be used to orientate the elements appropriately for the disconnect operation. The elements may be “re-flipped” following the disconnection to return to the original orientation.

The procedure is as follows.

1. Set the [cursor mode](#) to Select Elements.
2. Select the elements to be disconnected.
3. Give the “Disconnect Elements” command.
4. Specify the nodal position at which the elements are to be disconnected and whether joints are to be created in the [Disconnect Elements dialog box](#).

3.8.1.24 Deleting nodes and elements graphically

Nodes and elements can be deleted using the “Edit | Delete” (Del) or “Edit | Quick Delete” (Ctrl+Del) menu commands.

Unused nodes (i.e. nodes drawn red) can be deleted. When attempting to delete nodes that are in joints, rigid constraints or constraint equations but otherwise unused, the option to delete these is given. Nodes that are attached to any element or referred to as an orientation node or a grid plane elevation node cannot be deleted graphically.

The “Delete” command deletes the entity and all references to the entity, whereas the “Quick Delete” command deletes only the entity.

The procedure is as follows.

1. Set the [cursor mode](#) to Select Nodes or Select Elements, depending on whether nodes or elements are to be deleted.
2. Select the nodes or elements that are to be deleted.
3. Give the “Delete” or “Quick Delete” command.

3.8.1.25 Creating rigid constraints graphically

[Rigid constraints](#) can be created using the “Sculpt | Create Rigid Constraints” menu command. The procedure is as follows.

1. Set the [cursor mode](#) to Select Nodes.
2. Select the nodes that are to form a rigid constraint.
3. Give the “Create Rigid Constraints” command.
4. Open the [Rigid Constraints table](#) and set the type of linkage as required.

Note that this process results in all selected nodes being included in the rigid constraint node list and the highest numbered of these being the master. The fact that the master is also represented in the node list is not a problem.

3.8.1.26 Creating joints graphically

[Joints](#) can be created using the “Sculpt | Create Joints” menu command. The procedure is as follows.

1. Set the [cursor mode](#) to Select Nodes.
2. Select the nodes that are to form joints. Note that only coincident nodes will be joined; non-coincident nodes will be ignored. The tolerance used for establishing coincidence in this operation is the tolerance for coincidence on the [Sculpt page](#) of the [Preferences dialog box](#).
3. Give the “Create Joints” command.
4. Open the [Joints table](#) and set the direction of the joint as required.

3.8.1.27 Creating nodal loading graphically

The following nodal load types can be created using the respective commands in the “Sculpt | Create Nodal Loading” menu:

- Node loads
- Applied displacements
- Settlements

The “Create Nodal Loading” menu is also available on the [right-click menu](#) that is displayed when the [cursor mode](#) is set to Select Nodes. The procedure is broadly the same for creating each load type and is as follows.

1. Set the [cursor mode](#) to Select Nodes.
2. Select the nodes that are to be loaded.
3. Give the respective Create Nodal Loading command.
4. Specify the load in the respective [nodal load definition dialog box](#).

Mapping Node Loads on Nodes

The sculpt “Map Node Loads on Nodes” command allows node loads to be applied to selected nodes with loading values expressed as a function of the coordinates of the nodes. Normal mathematical notation is used in [expressions](#). For example:

$$z + \text{abs}(\sin(x)) * \pi$$

Operations may be with respect to standard (e.g. global) or user defined axis sets.

Node loads can be mapped on nodes using the “Sculpt | Create Nodal Loading | Map Node Loads on Nodes” menu command. The procedure is as follows.

1. Set the [cursor mode](#) to Select Nodes.
2. Select the nodes to be loaded.
3. Give the “Map Node Loads on Nodes” command.
4. Specify the action to be taken in the Map Node Loads dialog box.

3.8.1.28 Creating element loading graphically

The following element load types can be created using the respective commands in the “Sculpt | Create Element Loading” menu:

- Beam Loading
 - Beam Loads (note the extended patch load feature described below)
 - Pre-stress
 - Distortions
 - Thermal
- 2D Element Loading (only offered when 2D element analysis is enabled)
 - Face Loads
 - Edge Loads
 - Pre-stress
 - Thermal
- Gravity

The “Create Element Loading” menu is also available on the [right-click menu](#) that is displayed when

the [cursor mode](#) is set to Select Elements. The procedure is broadly the same for creating each load type and is as follows.

1. Set the [cursor mode](#) to Select Elements.
2. Select the elements that are to be loaded.
3. Give the respective Create Element Loading command.
4. Specify the load in the [Beam Load Definition](#), [Beam Pre-stress Definition](#), [Beam Distortion Definition](#), [Beam Thermal Load Definition](#), [2D Element Face Load Definition](#), [2D Element Edge Load Definition](#), [2D Element Pre-stress Load Definition](#), [2D Element Thermal Load Definition](#) or [Gravity Load Definition](#) dialog box respectively.

Extended Beam Patch Loads

When using the Create Beam Load command to apply a patch load, specifying the position of the load as beyond the element end will result in the patch load being “broken up” appropriately with the parts of the load that extend beyond the element being applied to the aligned and attached string of elements following. The [straightness tolerance preference](#) is used to determine whether a following element is aligned. A message is given if the load is not fully resolved.

Mapping Beam Loads on 1D elements and Face Loads on 2D elements

The sculpt load mapping commands allow beam loads and face loads to be applied to selected 1D and 2D elements, respectively, with loading values expressed as a function of the coordinates of the element topologies. Normal mathematical notation is used in [expressions](#). For example:

$$z + \text{abs}(\sin(x)) * \pi$$

Operations may be with respect to standard (e.g. global) or user defined axis sets.

Beam and face loads can be mapped on elements using the “Sculpt | Create Element Loading | Map Beam Loads on 1D Elements” and “Sculpt | Create Element Loading | Map Face Loads on 2D Elements” menu commands, respectively. The procedure is as follows.

1. Set the [cursor mode](#) to Select Elements.
2. Select the elements to be loaded.
3. Give the respective Map Loads command.
4. Specify the action to be taken in the Map Beam Loads or Map Face Loads dialog box, respectively.

3.8.1.29 Creating grid loading graphically

Grid point, line and area loads can be created using the respective commands in the “Sculpt | Create Grid Loading” menu. The “Create Grid Loading” menu is also available on the [right-click menu](#) that is displayed when the [cursor mode](#) is set to Polyline. The procedure is broadly the same for creating each grid load type and is as follows.

1. Set the [cursor mode](#) to Polyline.
2. [Define a polyline](#) to locate the loading. The polyline will be used according to the grid load type being created.
 - Grid point loads will be created at each point on the polyline. A single point polyline will suffice.
 - A grid line load will be created along the polyline. At least two polyline points are required.
 - A grid area load will be created within the closed polyline or across the plane defined by the first three polyline points. Either way at least three polyline points are required.
3. Give the “Create Grid Point Loads”, “Create Grid Line Load” or “Create Grid Area Load” command.
4. Specify the grid load in the respective [grid load definition dialog box](#).

3.8.1.30 Deleting loading graphically

Loading applied to nodes or elements and grid loading can be deleted using the “Sculpt | Delete Displayed Loading” menu command. The procedure is as follows.

1. Set the [Cases list](#) to the load case(s) in which the loads that are to be deleted are specified. Make sure that none of the load cases in the list have been analysed; the program will not allow cases referred to by analysed, analysis cases to be deleted.
2. Display [diagrams](#) of the loads that are to be deleted.
3. If the loads to be deleted are applied to nodes or elements then select the nodes or elements. When deleting grid loading set the [cursor mode](#) to other than Select Nodes or Select Elements.
4. Give the “Delete Displayed Loading” command.

The consequence of this operation is that the lists in the affected loading records have the deleted entities removed unless the resulting list is empty, in which case the loading record is deleted.

3.9 Convert Orientation Node to Angle

The Convert Orientation Node to Angle command may be used to replace specified orientation nodes with orientation angles that result in unchanged orientations for a set of beam (etc.) elements or members.

This command may be found in the 'Sculpt | 1D Element Operations' menu.

The entities that are to be modified must be [selected](#) prior to giving this command.

3.10 Create Mass from Element/Member Mass

The tools allow the mass of 1D elements and members to be converted to point masses at the ends. Once these have been created the mass of the original elements should be ignored.

3.11 Undo and Redo

In GSA it is possible to undo both edits to the data and view settings. [Undoing an edit](#) results in the last edit carried out on the data being cancelled, regardless of the view in which that edit was carried out. [Undoing a view](#) results in the view reverting to the previously displayed view in that window. Undoing views is only possible in Graphic Views and Output Views. The undo edit and undo view mechanisms operate independently. Redo is available in both.

3.11.1 Undoing edits

The “Edit | Undo” (Ctrl+Z) menu command undoes the last edit on the data.

The “Edit | Redo” (Ctrl+Y) menu command redoes edits that have been undone.

Data edits are preserved in an undo buffer (i.e. remembered) regardless of how the edits were applied (i.e. whether via tables, sculpt or batch operation).

Edits that are undone are preserved in a redo buffer.

The undo buffer is cleared when the file is closed. It is frozen when an analysis is done and only unfrozen when all results are deleted.

The redo buffer is cleared when any edits are carried out on the data and when the file is closed.

Edits are preserved in the undo and redo buffers as individual edit operations, though several edit operations may be associated with one undo step. So, for example, modifying 10,000 elements in one sculpt command constitutes 10,000 undo operations but just one undo step. The Undo

command undoes one step.

Multiple undo and redo steps are possible.

Undo edit limit

There is a limit to the number of operations that are preserved in the undo and redo buffers. This limit may be adjusted in the [Input Data Preferences](#). GSA preserves at least one undo step and one redo step if called upon to do so. In our example, even if the maximum number of preserved undo operations is set to 5,000, the 10,000 modified elements will be preserved in the undo buffer and one Undo command will undo all of the element modifications.

Undoing Analyses

The option to undo analyses may be set in the [Input Data Preferences](#). When this is set, an analysis becomes an undo step and the first undo after an analysis will cause the results and any analysis cases generated during the analysis to be deleted. A subsequent undo will undo the last edit done before the analysis, etc. Redo is not available for an analysis.

When the undo analysis preference is not set, undoing an analysis is not available **and** the undo and redo buffers are cleared upon commencing any analysis.

Enabling / Disabling Undo

Undo for edits may be enabled or disabled in the [Advanced Preferences](#). When Undo is disabled then the undo and redo buffers are cleared and no edits are preserved in the undo buffer until Undo is enabled again.

Undo Buffer and Computer Memory

The undo and redo buffers reside in the memory of the computer. The larger these buffers are, the greater the demand will be on the memory resources. This should be considered when adjusting the undo limit and undo analysis preferences, especially when working on a machine with limited memory available.

Note that:

- Setting the undo limit preference to a huge number will result in data for that many operations being preserved in the undo buffer.
- Switching the undo analysis preference on will result in the undo and redo buffers coexisting in memory with the results generated during the analysis.

3.11.2 Undoing views

The “View | Undo View” (Ctrl+Alt+Z) menu command undoes the last change in view settings in the current Graphic View or Output View. Undo View is also available on the [Standard toolbar](#).

The “View | Redo View” (Ctrl+Alt+Y) menu command redoes the last Undo View.

All changes to the view settings are preserved in the undo view buffer. In the case of a Graphic View this means changes in orientation (e.g. dragging to rotate), scale (e.g. zoom), content (e.g. volume and entity list), adornments (e.g. labels, diagrams and contours), case list etc.

View settings that are undone are preserved in a redo view buffer.

The undo view buffer is associated with the view. Each Graphic View and each Output View has its own undo view buffer which is cleared when the view is closed.

The redo view buffer is cleared when any changes are made to the view settings and when the view is closed.

Multiple undo view and redo view steps are possible, currently set at 32 steps. The limit is not adjustable.

3.12 Working with Saved Views and Preferred Views

Specifying the settings for Graphic Views and Output Views can be laborious. To remove the need to repeatedly specify these, GSA offers the option to save these views as:

- default view settings (saved as preference data),
- preferred views (saved as preference data) and as
- saved views (that are saved with the model).

View lists are sequenced collections of preferred views and saved views. These may be used to print particular views in a particular order.

More:

[Default View Settings](#)

[Preferred Views](#)

[Saved Views](#)

[Auto Views](#)

[Units and Numeric Format](#)

[View Lists](#)

[Batch Printing of Views](#)

[View Management](#)

3.12.1 Default View Settings

Opening a new Graphic View or Output View opens the view with the default view settings. The default view settings are saved as preference data and may be adjusted to suit personal preferences.

This is done by, first, displaying a Graphic View or Output View with the required settings and then using the “View | Save Default View Settings” menu command (also on the right-click menus for the views). Thereafter, whether in the current GSA session or subsequent sessions, new views will adopt the adjusted default settings.

Stored Data

Since the default view settings are applied to all new Graphic and Output Views, no references to specific data or results modules are stored.

In the case of Graphic Views this means that no diagrams or contours are stored with the settings. Furthermore, the orientation of the structure is not stored, leaving the Graphic View to default to an orientation that is appropriate for the structure.

While the default Graphic View settings exclude any diagrams or contours, they do include diagram / contour settings, i.e. the settings specified in the [Diagram Settings dialog](#) and the [Contour Settings dialog](#) other than the reference to the module that is the subject of the diagram or contour. Such settings may be saved as default by displaying a Graphic View that includes one diagram or contour with the desired settings and saving the default settings for that view; — the diagram or contour will be ignored but the related settings will be saved with the default settings.

The Output View table format, whether space separated tables or tab separated tables, is included with the default Output View settings.

Using Default View Settings

Default view settings are applied whenever a New Graphic View or New Output View is opened. These may be opened from the “View” menu, [GSA toolbar](#) and from the [Views tab](#) on the Gateway.

3.12.2 Preferred Views

A displayed Graphic View or Output View can be saved as a preferred view. This may be done using the “View | Save As Preferred View” menu command (also on the right-click menus for the views). The preferred view is then given a name and is saved with the preferences. Preferred views may be applied to any model.

For example, a preferred Graphic View might be saved that has “node dots” labels, “element x” labels and element “shrink” switched on. This might then be used for the sculpting of skeletal models.

Up to five preferred Graphic Views and five preferred Output Views may be saved.

Stored Data

Since preferred views can be applied to any model, only those settings that are model independent are stored. So, for example, a preferred Graphic View includes the state of the legend and the diagram annotation method but excludes the entity lists and scales.

Preferred Graphic Views exclude:

- Structure scaling
- Case list
- Entities lists
- Node and element sets to which diagrams and contours are applied
- Node and element sets to which annotation is applied
- Diagram / contour scaling

and include:

- Structure orientation
- Labels (but only those applied to All of the model)
- Display methods
- One diagram or contour (taking the earliest specified for the view)
- Diagram / contour settings (but only up to 10 specified contour values)

Preferred Output Views exclude:

- Case list
- Entities lists

and include:

- Data options, including multiple selection of modules

The option to save the window size and position is given at the time of saving the preferred view.

Using Preferred Views

Preferred views may be opened from the [Views tab](#) on the Gateway. Preferred view settings may also be applied to an existing view by, on the [Lists toolbar](#) for the view, setting the Display Option to “Views” and selecting the required preferred view in the Display List. The “Views” Display Option is not offered when no preferred views or saved views or auto views are present in the model.

3.12.3 Saved Views

A displayed Graphic View or Output View can be saved as a saved view. This may be done using the “View | Save Graphic View” and “View | Save Output View” menu commands, respectively (also on the right-click menus for the views).

There is no limit to the number of views that may be saved.

Stored Data

Saved views are stored with the model and all settings are stored, including the current print size and orientation as specified in the Print Setup dialog and, optionally, the window size and position. Note the special treatment of [units and numeric format](#), below.

If, when a saved view is displayed, the saved window position is found to be outside the extents of the GSA window then the saved window position is ignored.

Using Saved Views

Saved views may be opened from the [Views tab](#) on the Gateway. Saved view settings may also be applied to an existing view by, on the [Lists toolbar](#) for the view, setting the Display Option to “Views” and selecting the required saved view in the Display List. The “Views” Display Option is not offered when no preferred views or saved views or auto views are present in the model.

3.12.4 Auto Views

Auto views are views automatically generated on-the-fly, based on data that exists in the model.

Using Auto Views

Auto view settings may be applied to an existing view by, on the [Lists toolbar](#) for the view, setting the Display Option to “Views” and selecting the required auto view in the Display List. The “Views” Display Option is not offered when no preferred views or saved views or auto views are present in the model.

Auto View Options

The following auto views are available.

Storey Views — Storey views are offered when storey grid planes are present in the model. A storey view is a plan view of the elements in the storey grid plane, the vertical elements attached beneath the storey grid plane, with section shape labels and shrink on.

3.12.5 Units and Numeric Format

Units and the numeric format are not saved with default preferred views and are only saved with named preferred views and saved views if they have been explicitly specified for the view being saved.

When a Graphic View or Output View is opened, if the units are not specified the model units are assumed. If the numeric format is not specified the numeric format specified in the [Preferences Dialog](#) is assumed.

3.12.6 View Lists

A view list is a sequenced collection of preferred views and saved views. The main purpose of view lists is to enable a list of views to be printed in sequence and for that list to be preserved to allow the same list of views to be printed again later.

There is no limit to the number of view lists that may be saved.

Stored Data

A view list is stored as a list of view names coupled with view type, whether saved or preferred Graphic View or Output View.

Using View Lists

Options to edit, print, rename and delete view lists are accessible from the [Views tab](#) on the Gateway.

To create a new view list double click “View Lists” on the Views tab of the Gateway or right-click on an item in the View Lists branch and select “New View List”. Existing view lists may be edited by right-clicking on the view list in the Gateway and selecting “Edit “<view list name>”. Both of these open the [View Lists dialog](#).

See also [Batch Printing and Saving of Views](#).

3.12.7 Batch Printing and Saving of Views

Saved and preferred Graphic Views and Output Views can be printed and saved to file without opening the view in GSA.

When saving to file the format of the saved file is determined by the respective 'batch output file format' setting in the [Miscellaneous Preferences dialog](#).

The options are to print or save to file:

- a selected view,
- all views of a type (e.g. all saved Graphic Views),
- all marked views of a type (e.g. all tagged Graphic Views),
- a selected [view list](#)
- all view lists and
- all marked view lists.

The commands to invoke these options may be given either:

- from the right-click menu for the [Views tab](#) on the Gateway or
- from the 'View | Batch View Options' menu or
- remotely (i.e. without running GSA interactively) by [COM Automation](#) or by [Command File](#).

View Lists may also be printed and saved from the [View Lists dialog](#).

Marking Views for Printing and Saving to File

Saved and preferred Graphic Views and Output Views and view lists can be marked for printing or saving to file by right-clicking on the view in the Gateway and giving the “Mark <view name>” command. Views can be unmarked by the same process. Marks on saved views are saved with the model. Marks on preferred views are saved with the preferred view settings.

3.12.8 View Management

Options to rename and delete views and to reset default settings to GSA default are available on right-click menus for the [Views tab](#) on the Gateway.

3.13 Working with Analysis Tasks and Cases

Once a model has been created GSA is then used to analyse the model. Analysis tasks and analysis cases are used to define the analysis.

The basic analysis unit in GSA is the analysis task. This gives the solver information about the

analysis that is to be carried out such as whether static or modal analysis is requested. Associated with each analysis task is one or more analysis cases. Each analysis case holds details that are particular to the case such as the case description. An analysis case may also be thought of as a container for a set of results.

So, for example, a static analysis of a dead load case, a live load case and a wind load case would form a single analysis task consisting of three analysis cases. Multiple analysis tasks can be set up, so, for example, task 1 may be a static analysis, task 2 a static P-delta analysis and task 3 a modal dynamic analysis. The [Task View](#) gives a way of inspecting and working with these tasks.

An analysis task can have one of the following states:

- **pre-analysis** – the task is set up but has not yet been analysed
- **marked** – the task has been selected for analysis
- **post-analysis** – the task has been analysed and the model contains results

When a static analysis of each load case is requested by simply giving the “Analysis | Analyse” command (or “Analyse” on the [GSA toolbar](#)) a static analysis task and analysis cases for each load case are set up automatically. For every other type of analysis it is necessary to define the analysis task and cases explicitly using the options described in this section.

See also the Step By Step Guide section on [Requesting Analysis](#).

More:

[Task View](#)

[Task View Right-click Menu](#)

[Tasks, Cases and the Analysis Wizard](#)

[Copy and Paste Tasks and Cases](#)

3.13.1 Task View

The Task View displays analysis tasks and cases in a tree view, with tasks at the top level and the related analysis cases on a branch from the parent task. Various options for manipulating tasks and cases are available on the [Task View right-click menu](#). These include options to edit, rename, delete and mark for analysis.

The state of an analysis task or case is represented in the Task View by an icon:

- **pre-analysis** – the “empty” icon
- **marked** – the “sigma” icon
- **post-analysis** – the “filled” icon

Only pre-analysis tasks can be edited. It is useful to edit a task when, say, a solution has not converged and some minor adjustment is required in the task details.

With the exception of the “Delete all results” command, all deleting of analysis tasks and cases should be carried out from the Task View.

By using the “Mark Task for Analysis” option several tasks may be identified as being ready for analysis. The “Analyse All Marked Tasks” command may then be used to execute the analysis of these tasks.

3.13.2 Task View Right-click Menu

Clicking the right mouse button when the cursor is pointing at an item in the Task View displays a floating menu that relates to that item.

This menu allows new tasks to be created, tasks to be edited, results deleted and analyses selected

and run. So for example a modal analysis task can be edited so that the maximum number of iterations can be increased if convergence has not been achieved.

The context sensitive menu splits the options into three main categories:

- general options – this allows new tasks to be created, all results to be deleted and analyses to be started.
- task specific options – these allow tasks to be edited, renamed, deleted, selected for analysis or their properties inspected.
- case specific options – these allow cases to be renamed, their results to be deleted or their properties inspected.

General Options

In the context sensitive menu the general options and paste are always present (although some options may be disabled):

- New Analysis Task – opens the Analysis Wizard to allow definition of a new analysis task.
- Manage Tasks – opens a dialog which allows tasks to be quickly marked for analysis and for deletion of the task/task results.
- Paste as New Task – pasting a new task (or tasks), appending to the existing tasks (see [Copy and Paste Tasks and Cases](#))
- Check Data – does a data check
- Analyse – starts the analysis of all unanalysed tasks
- Analyse All Marked Tasks – starts the analysis of all marked tasks
- Terminate Analysis
- Delete All Tasks
- Erase All Results
- Renormalize Modes – for a modal analysis this renormalizes the mode shapes so that the maximum displacement is 1m.
- Batch Analysis – this allows the solver to run without asking for any user intervention. If the 'Save file after batch analysis' setting in the [Miscellaneous Preferences](#) is set then the data file is automatically saved to disk upon successful completion of the analysis.
- Show Description – this displays the case description rather than the case name in the Task View.

Task Specific Options

When a task is selected the context sensitive menu is expanded to include:

- Edit *task* – opens the Analysis Wizard to allow the task to be modified. This can be used to completely redefine the task, but it is more often used to modify parameters or add new analysis cases. If the task is post-analysis the task cannot be edited but there is an option on the last page of the wizard to delete results for specific cases.
- Copy *task* – copy the selected task (see [Copy and Paste Tasks and Cases](#))
- Delete *task* – deletes the selected task and any associated tasks.
- Erase *task* results – erases the results from the selected task and any associated tasks.
- Mark *task* for analysis – allows a subset of the tasks to be selected for analysis.
- Rename *task* – equivalent to in-place editing of the task name.
- Properties for *task* – to display a summary of this task in the [Properties tab](#) of the Object Viewer.

Case Specific Options

When a case is selected the context sensitive menu is expanded to include:

- Edit case – opens the Case Definition dialog to allow the case to be modified. This can be only be used for static analysis options when no results are present.
- Delete case / Delete case results – depending on the status of this case one or other option will be offered. It is not possible to delete results when other results are dependent on these (e.g. results referred to by an analysed analysis envelope cannot be deleted).
- Rename case – equivalent to in-place editing of the case name.
- Properties for case

3.13.3 Tasks, Cases and the Analysis Wizard

With the exception of simple linear static analysis, the [Analysis Wizard](#) is used to set up the data required for an analysis – both the task and case information. The wizard hides the complexity of the inter-relationship between these from the user.

The starting point is to select the type of analysis required, then depending on the analysis type a different path is followed through the wizard, setting up the task and a case parameters.

Setting up static cases

The most common analysis in GSA is a static analysis and this can be the most tedious analysis to set up. This is particularly so when there are many analysis cases or other issues such as stages or load combinations that have to be considered. Consequently, for a static analysis (either linear, P-delta or full non-linear) the Static Analysis Cases page is probably the most significant page in the Wizard.

When a simple analysis of all the load cases is required the “Create default cases” option gives an easy way to set up the analysis cases. Likewise the “copy cases from task” option allows the loading conditions from an existing task to be copied directly to the current task.

The Add and Edit buttons allow quick assembly of new or editing of existing analysis cases. These buttons bring up a separate dialog that displays all the load cases that exist along with the load case titles (if they exist). Double-clicking in this list append the load case to the load description. This makes it straightforward to define loading such as $1.4 \times \text{"dead"} + 1.6 \times \text{"imposed"}$.

In addition to these there is a set of buttons that allow case names and descriptions to be copied to and pasted from the clipboard. This means that a set of analysis cases can be set up externally, for example in a spreadsheet:

dead → L1

imposed → L2

dead+imposed → $1.4L1 + 1.6L2$

and then copied and pasted directly into the wizard.

The up/down arrows give a convenient way to reorder the analysis cases in the task.

3.13.4 Copy and Paste Tasks and Cases

Gateway

Data can be copied and pasted from the Tables tab on the Gateway, including the task and case information. The information is copied to the clipboard or pasted from the clipboard in GWA format. When copying an analysis task several GWA records are copied: the analysis task and the analysis case associated with the task. When pasting into the Gateway the task information overwrites the task information that may exist in the model. The “Paste Special” option allows the data to be

appended to or to overwrite the existing task data.

Task View

The Task View also allows copy and paste operations for task and case data. As with the Gateway the information is copied to the clipboard or pasted from the clipboard in GWA format. The copy option in this case is for the currently selected task. The paste option in this case can paste more than one task at a time but pasted tasks and cases will always be appended and are subject to renumbering.

GWA records

At its simplest a task is defined by a task record (TASK keyword) and a set of analysis case records (ANAL keyword). So a simple static analysis might look like:

TASK → 1 → ULS → 0 → GSS → STATIC → ...

ANAL → 1 → dead → 1 → L1

ANAL → 2 → imposed → 1 → L2

ANAL → 3 → dead+imposed → 1 → 1.4L1 + 1.6L2

For the case of simple static or modal analysis the TASK record can be omitted and a default GSS static analysis task is assumed. The stage number (if required) is then appended to the record.

So a static analysis might be:

ANAL → 1 → dead → 1 → L1 → 0

ANAL → 2 → imposed → 1 → L2 → 0

ANAL → 3 → dead+imposed → 1 → 1.4L1 + 1.6L2 → 0

... or a modal analysis:

ANAL → 1 → mode 1 → 1 → M1 → 0

ANAL → 2 → mode 2 → 1 → M2 → 0

ANAL → 3 → mode 3 → 1 → M3 → 0

In the situation of a paste that appends the data, GSA will reorder the tasks and cases to avoid conflict with existing tasks and cases. For multiple tasks this can mean reordering within the set of the cases being pasted.

3.14 Working with Grid Planes and Grid Loading

Grid planes and grid loading give a way of defining load in a structure in geometric terms rather than having load tied to the analysis model attributes (elements). As such it is possible to revise the layout of the structure without a need to redefined the loading.

[Grid loading](#) is mapped to beam elements on a grid plane. The [grid plane](#) defines a surface in terms of a plane – the xy plane of an axis set at a given elevation in z. As well as the basic grid plane definition a grid plane also has a list of elements and a tolerance $\pm\Delta z$ forming a box. The surface is formed by the elements in the element list that exist inside the box. This definition means that the surface need not be flat (by careful definition of element list and tolerance could be hemispherical) so grid loading can be used to apply loading to complex roof geometries.

The grid plane also determines how the loads span on to the supporting beams – either one-way, two-way or multi-way.

The basic unit considered in grid loading is termed a panel. A panel is a closed loop of elements that would typically represent a bay in a structure.

3.14.1 Specifying Grid Planes

[Grid planes](#) lie in an xy plane of an axis set so an appropriate axis set must be specified to which the grid plane can refer. The global axis set will be appropriate for all horizontal grid planes.

The axis set can be set up either explicitly by entering the [axes data](#) in the axes table or graphically by using the “[Sculpt | Create User Axes](#)” menu command.

Note that an appropriate axis set is automatically set up in the “Create Grid Plane” and “Set Grid Plane to This” options described below.

Next the grid plane should be specified. This can be done either by explicitly entering the [grid planes data](#) in the grid planes table or graphically by using the “[Sculpt | Create Grid Plane](#)” menu command.

Another less direct way of specifying a grid plane is to right-click on a node in a Graphic View and to select the “[Set Current Grid to This](#)” menu command. The [current grid](#) is set to a grid on which the node lies, first looking for one parallel to the existing current grid, then for any. If none exists then the program offers to create one parallel to the existing current grid.

3.14.2 Specifying Grid Loading

Grid loading

[Grid loading](#) is applied to grid planes. Any number of grid loads can be applied to a grid plane. Note that the grid plane element list can be used to exclude elements that lie in the plane from the grid plane and thereby from the elements that attract load.

2D polylines are used to locate grid loads on the grid plane as follows.

- Grid point loads – When using the “[Sculpt | Create Grid Loading | Create Grid Point Loads](#)” command a load is applied at each point on the current polyline.
- Grid line loads – The load line is defined by a reference to a 2D polyline. When using the “[Sculpt | Create Grid Loading | Create Grid Line Load](#)” command a polyline is created from the current polyline (if an identical one does not already exist) and this 2D polyline is referred to.
- Grid area loads – The area is either defined as being the whole plane or as being bounded by a 2D polyline. When using the “[Sculpt | Create Grid Loading | Create Grid Area Load](#)” command a polyline is created from the current polyline (if an identical one does not already exist) and this 2D polyline is referred to.

2D polylines can be set up either explicitly by entering the [2D polylines data](#) in the 2D polylines table or graphically in the [Polyline](#) cursor mode.

2D polylines used for grid loading can be generally located. They do not need to align with elements. They can be wholly within panels or spanning several panels.

Having set up an appropriate 2D polyline the grid loading can be set up either explicitly by entering the [grid point loads data](#), the [grid line loads data](#) or the [grid area loads data](#) in the grid loading table or graphically using the sculpt options for [creating grid loading graphically](#).

Grid load expansion

Since its original implementation the grid loading algorithm has been improved. The original implementation is referred to as 'legacy'. The more recent algorithms are variant on a single method, which treats the load panel as a rigid plane, hence the name 'plane approximation'. This assumes a load intensity which is a function of the (x,y) coordinates of the grid panel. There is a subsequent adjustment that takes into account the aspect ratio of the panel. The basic variant on this gives intensities which are higher closer to the centre of load – this has the effect of the load distribution being dependent on the element refinement around the panel. The 'smooth' option removes this anomaly by replacing the uniform loads on each element by a single linearly varying load. The 'corner' option takes a similar approach to the smooth option but in addition assume that there is a lower load intensity at internal corners, so the load at these corners tapers to zero.

Grid loading restrictions

There are a number of situations where grid loading cannot be applied to the structure. Grid loading works by searching for areas enclosed by a string of elements. These areas are called panels and loading can only be applied to panels. So if a model has a grid plane with pair of cantilevers that are unconnected at the ends the area between these cantilevers cannot be loaded by grid loads. This situation can be resolved by adding a dummy element between the ends of the cantilevers. When dummy elements are present the grid load is distributed on all the panel elements as normal but then the load which is assigned to dummy elements is transferred to the ends of the adjacent real elements.

Panels which contain re-entrant edges cannot be loaded with the legacy option. A re-entrant panel is defined as one where the internal angle between two adjacent elements is greater than 180°. This tends to arise in one of the two following circumstances. Firstly the panel is L shaped: in this case the most expedient solution is to add a dummy element from the re-entrant vertex to the opposite vertex splitting the L shape into it two arms. Secondly the panel has a number of elements defining an edge and these are not straight: in this case the best solution is to adjust the Grid edge straightness tolerance (in [Analysis Specification | Tolerances](#)).

3.14.3 Displaying Grid Planes and Grid Loading

Grid planes

The following options are available for displaying grid planes in Graphic Views.

- The [grid](#) is displayed on the [current grid plane](#).
- The [Labels and Display Methods | Display Methods](#) dialog 'Draw current grid panels' option displays panels on the current grid plane.

Grid loading

Grid loading may be displayed as [diagrams](#). Grid Area Loads may also be [contoured](#).

If the [current grid plane](#) is not 'Global' then only loads that are applied to the current grid plane are displayed.

Loads are not displayed on invalid panels. Use the [Draw current grid panels](#) option to verify that panels are valid for the grid plane to which the loads are applied.

3.15 Working with Geometric Entities

Geometric entities are as follows:

- [lines](#)
- [areas](#)
- [regions](#)

See also the Step By Step Guide section on [Generating 2D element meshes](#).

3.16 Data Management

[Data Management Tools](#)

[Create storeys](#)

[Create grid planes from storeys](#)

[Create Rigid membranes from storeys](#)

[Import GWA data](#)

[Comparing models](#)

3.16.1 Data Management Tools

The following tools are available in the [Manage Data dialog box](#):

- Delete blank nodes
- Delete references to blank nodes
- Renumber nodes incrementally
- Delete blank elements
- Delete references to blank elements
- Renumber elements incrementally
- Delete loads for a list of load cases

Sculpt tools that manage node and element references

The following sculpt operations include the option to manage references associated with the nodes or elements being operated on:

- [Split elements](#)
- [Copy nodes or elements](#)
- [Delete nodes or elements](#)

Sculpt tools that delete loads

As well as the “manual” method of deleting records in the loading tables, loads can be deleted as follows:

- When deleting nodes or elements in Graphic Views the option is given to delete references, including references in the loading tables, to the nodes or elements. Where this results in a load being applied to no nodes or elements, the load record is deleted.
- The sculpt “[Delete Displayed Loading](#)” can be used.

Other ways of deleting loads

When deleting load case titles in the Load Case Titles table the option is given to delete all loading in the respective load cases.

3.16.2 Create Storey Grid Planes

This is available from the “Tools | Create Storey Grid Planes” menu command, when no storeys are defined in the structure. The process works using the default storey tolerance properties. The program scans through the nodes. If no storey exists for a node a new storey is created. At the end of the procedure the storeys are sorted into ascending order (of elevation).

3.16.3 Create Grid Planes from Storeys

This is available from the “Tools | Create Grid Planes from Storeys” menu command. A new grid plane is created corresponding to each storey. Default grid plane properties (such as span type and direction) are used.

3.16.4 Create Rigid Membranes from Storeys

This is available from the “Tools | Create Rigid Membranes from Storeys” menu command. A rigid constraint is created that corresponds to each storey. The constraint type is set to xy plane so that the “floor” is considered rigid in-plane but can deform in the out-of-plane direction.

3.16.5 Import GWA data

The “File | Import | Text (GWA file)” menu command allows GWA data to be imported into the existing model. Before the data is imported the [GWA Import Options dialog box](#) is displayed to offer the selection of GWA data to be imported and to give the option to append nodes, elements and load cases.

When the option to “append” is not selected data records that are identified by record number (and often referred to by that number) such as nodes, elements and properties, overwrite existing data; other data append to existing data. When “append” is selected, the node numbers, element numbers and load cases numbers, and references to these, are renumbered in the imported data to follow on from the existing data.

3.16.6 Comparing models

The recommended method for finding the differences between two versions of the same structure is to use the “File | Export | Input data (GWA file)” menu command to save the input data for each model in GWA format. Then use a proprietary file-difference analysis tool such as CSDiff to highlight the differences. (At the time of writing, CSDiff is available as a free download.)

3.17 Miscellaneous

[Unlock file](#)

[File backups](#)

[Delete results from files](#)

[Edit text file](#)

[User Modules](#)

3.17.1 Unlock file

It is possible for files to be left in a locked state if GSA terminated in an abnormal manner. In such a situation the file can be unlocked prior to use with the “Tools | Unlock File” menu command. This option is only available when there is no GSA model open.

3.17.2 File backups

There are two file backup systems that are in operation while a model is open in GSA: the manual backup and the timed backup. Both systems result in a temporary copy of the model (input data and results) being preserved in the Oasys temporary folder in GWB file format.

When a model is opened, names for the two backup files are assigned. These are the filename with “_M” or “_T” appended, for the manual backup and timed backup respectively, followed by a number to uniquely associate the backup name with the GSA session. For example, a timed backup file might be called MyFile_T9.gwb.

A manual backup is made immediately on opening a model. This may be overwritten by the “File | Manual Backup” menu command.

A timed backup is automatically made periodically. The timed backup interval may be adjusted on the [Miscellaneous tab](#) of the Preferences. Note that setting the interval to zero results in no timed backups being made; this may be desirable when working with large models that take a long time to save. A timed backup is only made if there has been a change to the data since the last timed backup. The timer is reset when the model is saved or 'saved as' a GWB file.

The “File | Restore Manual Backup” command may be used to reset the model in your current GSA session to the state as when the manual backup was taken. The backup data is applied to all views currently open without changing the view settings. For example a Graphic View will continue to display the image as currently orientated with the current adornments set but with the backed-up data applied. The “File | Restore Timed Backup” commands operates similarly when a timed backup exists for the model.

The two backup files are automatically deleted when the model is properly closed. However, if the program shuts down prematurely (as a result of program malfunction or power failure, etc.) then the backup files remain on disk. In this circumstance, the next time GSA is run the [Welcome dialog](#) invites you to open a backup file. Note the “Delete all backup files” button on this dialog box that is provided to encourage you to tidy up your Oasys temporary folder.

3.17.3 Delete results from files

It is sometimes convenient to reduce the size of files by deleting results. It is tedious to have to do this manually so the “Tools | Delete Results From Files” menu command is provided to do this. This allows the user to select a set of files from which the results are to be deleted. This option is only available when there is no GSA model open.

3.17.4 Edit text file

The edit text file is provided to provide a quick means of view a text file. The user selects a file which is then opened as a separate application in Notepad.

3.17.5 User Modules

User modules are the means by which users can feed their own data into a model for display by GSA either as contours in a Graphic View or tables in an Output View. Nodal, element and member user modules are available.

A nodal user module consists of one value per node in the model.

An element user module consists of values per position on the element. For 1D elements positions are expressed in terms of millionths of the way along the element. So for example position 0 is at end 1 of the element, position 1000000 is at end 2, and position 333333 is a third of the way along the element. Values must be specified at both ends of 1D elements (i.e. positions 0 and 1000000). For 2D elements a value is expected at each topology position plus a centre value, though if the 'centre only 2D' flag is set only the centre value is expected. User modules also have a name, a dimension (i.e. whether length, moment, etc. or dimensionless) and a flag for discrete or continuous contours. A discrete contour is one that represents the value that exists for that position and does not vary continuously along the element.

A member user module is similar to an element user module except that all members are 1D.

User modules may be supplied to GSA in text file (GWA) format. Refer to GSA Text (ASCII) File: Keywords for a definition of the format. Data should be supplied in SI units. User modules can be appended to a GSA model that has been saved in text file format and then opened or they can be imported into an existing model using the “File | Import | Text (GWA file)” menu command.

Once a user module has been fed into a model it remains part of the model, being saved in and read from the GWB file. The [user module management dialogs](#) allow user modules to be deleted and renamed.

When user modules are present in a model they are offered as contour and diagram options for Graphic Views and output options for Output Views. User module data are converted to the current units for display. Note that contouring of element and member user module data requires values to have been specified at all nodes on the element.

As well as outputting nodal user module data per node this data can be contoured on elements and can be output per topology item per element.

Reading user modules from GWA files

When a user module is read from GWA file if the user module ID is specified as zero then the user module is appended to the current user modules: the new user module is created by the user module title record and the following user module records (with ID=0) supply values to the new user module.

If the specified imported user module ID references an existing user module and the imported user module title data are compatible with the existing user module then imported user module values are written into the existing user module, preserving the existing values except where overwritten by imported values. However if the title data are incompatible then the existing values are nulled. Title data are compatible if the 'dimension' and 'is_centre' fields are the same.

Generating user modules

User modules can be generated from contoured data in a Graphic View, using the “Graphics | Generate User Module” command, and from displayed data in an Output View, using the “Output | Generate User Modules” command. An option to generate user modules is also offered in the [Output Settings dialog box](#). (Hint: It is possible to generate user modules without first displaying the output by giving the [“New Output View” command](#) to open the Output Settings dialog directly.) When generating from an Output View separate user modules are generated for each component and for each case currently displayed.

3.17.6 Evaluating expressions

GSA allows geometry and loading to be modified or defined using mathematical expressions. These are mathematical expressions which can define a variation as a function of position (x,y,x)

Normal mathematical notation is used in expressions:

e.g. $1000 \cdot \sin(x) + y^2$

Arithmetic operations

The arithmetic operators are

+	addition
-	subtraction
*	multiplication
/	division
^	raise to power of

Normal operator precedence is followed. The constants pi (3.14...) and g (9.81...) are defined. Parenthesis can be used to clarify the order of operations.

Mathematical functions

The following functions can be used:

sqrt(x)	square root
abs(x)	absolute value
exp(x)	e raised to power of

$\ln(x)$	natural logarithm
$\log(x)$	base 10 logarithm
$\sin(x)$	sine (in radians)
$\cos(x)$	cosine (in radians)
$\tan(x)$	tangent (in radians)
$\text{asin}(x)$	inverse sine (in radians)
$\text{acos}(x)$	inverse cosine (in radians)
$\text{atan}(x)$	inverse tangent (in radians)
$\text{atan}(y,x)$	inverse tangent of (y/x) (in radians)
$\sinh(x)$	hyperbolic sine
$\cosh(x)$	hyperbolic cosine
$\tanh(x)$	hyperbolic tangent
$\text{asinh}(x)$	inverse hyperbolic sine
$\text{acosh}(x)$	inverse hyperbolic cosine
$\text{atanh}(x)$	inverse hyperbolic tangent
$\text{radians}(x)$	conversion of degrees to radians
$\text{degrees}(x)$	conversion of radians to degrees
$\text{floor}(x)$	round a number down to integer value
$\text{ceil}(x)$	round a number up to integer value
$\text{sign}(x)$	return sign of number
$\text{max}(x,y)$	maximum of two numbers
$\text{min}(x,y)$	minimum of two numbers

Conditional expressions

Conditional expressions can be specified in the form:

`if( condition, true_expression, false_expression )`

The conditional operators are `>`, `<`, `!=` (or `<>`), `>=`, `<=`, `==` (or `=`) and the logical operators are `&&` (and) and `||` (or). Conditional expressions can be nested.

3.17.7 Touch Gestures

When operating GSA using gestures on a touch sensitive screen the following gestures are recognised.

- tap - click
- drag - drag
- pinch - zoom
- double finger drag - pan (in Graphic Views)
- press and tap - right-click; press on target and click using a second finger (in Graphic Views)

Part IV

4 Program Fundamentals

This chapter describes some of the fundamental concepts on which the program is based.

Refer to the [Program Data chapter](#) for definitions of the data that may be specified to describe a model.

More:

[User Preferences](#)

[Units](#)

[Axes](#)

[Grid Planes](#)

[Element types](#)

[Element axes](#)

[Beam Sections and Section Database](#)

[Spring Supports and Ground Springs](#)

[Sets and Lists](#)

[Use of Constraints](#)

[Applying Load](#)

[Cases and tasks](#)

[Cursor modes in Graphic Views](#)

[Numeric formats](#)

4.1 User Preferences

There are many ways in which the operation of the program can be customised to suit the requirements of the particular user or the particular circumstance in which the program is being used. These customisation settings are referred to as “preferences”. Most preferences are stored in the system registry for the current user. Usually the preferences for a particular user are picked up whichever machine in a domain the user is logged in at.

Many preferences are specified in the [Preferences dialog box](#) which is opened by using the “Tools | Preferences” (Ctrl+F7) menu command. Most of these preferences are acted upon immediately though some serve as default settings for operations or new files. Those that serve as defaults may be adjusted for the particular operation or file. Details of how each particular preference behaves are identified in the [Preferences dialog box](#) documentation.

Some preferences, such as the toolbar selection and positioning and window sizes are automatically set each time the program is closed.

Default settings for new Graphic Views and Output Views are saved as preferences. In addition to these default view settings there is the option to save several named preferred views. Details are given in the [Working with Saved Views and Preferred Views](#) documentation.

The preferences may be reset to their default “factory” setting by using the “Tools | Reset Preferences” menu command. The option is given to reset all view settings. Specific view settings may be reset from the [Gateway](#).

More:

[Advanced features](#)

4.1.1 Advanced features

The [Advanced Features](#) tab on the [Preferences dialog box](#) displays the optional features that are currently switched on. When a feature is not switched on then it is neither possible to enter data for that feature nor request that that feature be invoked. Reading a data file that includes data that relates to a particular feature implies that that particular feature is switched on while that model is open, whether or not the feature is actually set in the [Advanced Features preferences](#).

If a feature has not been licensed then that feature cannot be switched on and data files containing data that relates to that feature cannot be read.

If no advanced features are highlighted a static linear analysis using the Gss solver and a mass analysis using the Gss solver are available.

Feature	Data options enabled	Analysis options enabled
Modal dynamic analysis		GSS Modal dynamic
P-delta and buckling analysis		GSS P-delta GSS Buckling
Modal dynamic analysis P-delta and buckling analysis		GSS Modal p-delta analysis
Dynamic response analysis	Response spectrum data .	GsSpec response spectrum analysis , harmonic analysis and linear time history analysis
Non-linear analysis	Non-linear elements and materials exhibit full behaviour	GsRelax non-linear analysis
Fabric analysis	Fabric materials	
Form-finding analysis	Spacer elements Form-finding properties	GsRelax non-linear form-finding analysis
Raft analysis	Soil-structure interaction data	GsRaft solver
Bridge analysis	Vehicle , Alignment Paths and Bridge Loading	
Footfall induced vibration analysis		GsSpec footfall induced vibration analysis

4.2 Units

Throughout GSA the user is able to change the units he is using. This means that appropriate units can be selected depending on the context.

All data in GSA is stored in SI units and then converted to the user units as required. The particular user units will depend on which view or dialog the user is in and if the user has read an existing file or created a new file.

More:

[Unit set](#)

[Preferred Units](#)

[Units in the model](#)

[Units in views](#)

[Units in Sections](#)

4.2.1 Unit set

The set of units that are used by GSA are summarised in the table below.

unit	default
Force	N
Length	m
Mass	kg
Time	s
Temperature	K
Displacement	m
Stress	Pa
Ground acceleration	m/s ²

Other units are derived from these units, so for instance moments are derived from the force and length units giving in the SI units set N m.

Note that data considered to be in units of 'pressure' (e.g. Grid Area Load values) are reported in units of Force/Length², not Stress.

4.2.2 Preferred Units

In many cases there is a set of units which users will want to use for most of their work. In GSA these are termed preferred units. These can be edited and set from the [Input page](#) in the [Preferences](#). This brings up a dialog box to allow individual units to be selected.

A selection of units for each of force, length, etc are offered. For these units the conversion factor are known. If the required units are not in the list, the user can enter the name and must then set the conversion factor. The pre-programmed units are:

- **Force**—N, kN, MN, lbf, kip
- **Length**—m, cm, mm, ft, in
- **Mass**—kg, t, kt, g, lb, Ton, slug
- **Time**—s, ms, min
- **Temperature**—°C, K, °F
- **Displacement**—m, cm, mm, ft, in
- **Stress**—Pa, kPa, MPa, GPa, N/m², N/mm², kip/in², psi, psf
- **Acceleration**—m/s², cm/s², mm/s², ft/s², in/s², g

Also displayed is the value for gravity, which is fixed at 9.80665m/s².

Along with the units that can be set here the user can also set the preferred currency units. No conversion takes place between different currencies!

The preferred units are stored in the registry against the current user so that the next time GSA is used the same units are offered as the preferred units.

4.2.3 Units in the model

Each GSA model has a set of units associated with it. These units are saved along with the data in the GSA file. So when an existing model is opened the default model units will be those saved with the model. When a new model is created the units used for that model are determined by looking at the users preferred units. The base units for a model are changed in the Units Specification. This is accessed either from the [Units Specification](#). This gives the Units dialog, which allows the user to change the units individually or for “standard” unit sets to be selected. The standard unit sets are summarised in the table below:

unit set	SI	kN-m	kip-ft	kip-in
force	N	kN	kip	kip
length	m	m	ft	in
mass	kg	t	slug	slug
time	s	s	s	s
temperature	K	°C	°F	°F
displacement	m	mm	in	in
stress	Pa	N/mm ²	kip/in ²	kip/in ²
ground acceleration	m/s ²	m/s ²	ft/s ²	ft/s ²

Setting the preferred units has no effect on the current model units.

4.2.4 Units in views

In many cases it can be useful to adjust the units in individual views. In these cases the starting units are the current model units, and any changes made in that view do not change the model units. In tables and output views the units can be edited by selecting the units button on the [Data Options toolbar](#).

When the units are selected from the tables only the units relevant to that table are enabled. So for example in the sections table only length units are relevant, so all others are displayed greyed.

In the Graphic View the units relate to data that is being displayed in a diagram or contoured. In this case the units dialog is accessible from a button on the diagrams/contour dialogs.

4.2.5 Units in Sections

The most convenient units used for defining beam sections will most often be different from those used to define the structure geometry. On entering the [Section Wizard](#) the user is given a set of unit for defining the section. If the model units are SI based the section dimensions will default to mm and if the units are non-SI based the default will be in.

4.3 Axes

[Axis sets](#)

[Use of axis sets](#)

[Projected axes](#)

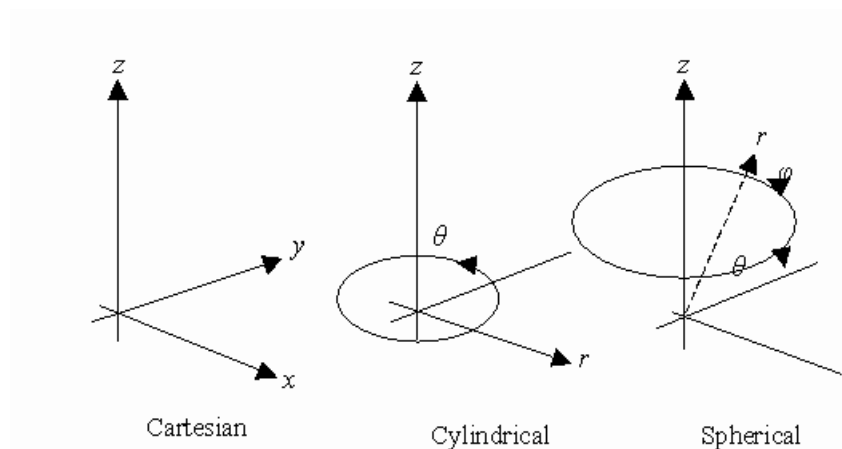
[Grid axes and the current grid](#)

[Constraint axes](#)

[Element and member axes](#)

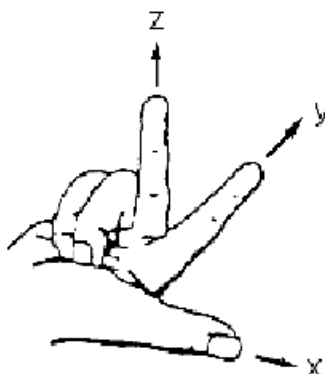
4.3.1 Axis sets

In order to make the most of GSA it is important to understand the axis systems available in GSA and how they are used. The term axis is used to refer to axis sets or coordinate systems. The axis sets can be either Cartesian, cylindrical or spherical.

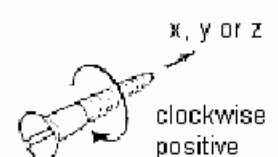


The basic axis system is the Global Cartesian axis system, normally referred to as the **Global** axis system. All other axis systems are located relative to the Global axis system.

All the axes systems in GSA are right handed axes systems.



Rotations about the axes follow the right hand screw rule



Several other standard axes are provided in GSA. These are all centred on the global origin.

- X elevation – this is a Cartesian axis set with the x axis parallel to the global Y axis in the negative direction, the y axis parallel to the global Z axis, and the z axis parallel to the global X axis.
- Y elevation – this is a Cartesian axis set with the x axis parallel to the global X axis, the y axis parallel to the global Z axis, and the z axis parallel to the global X axis in the negative direction.

- Vertical – this is a Cartesian axis set with the x axis parallel to the global Z axis, the y axis parallel to the global x axis and the z axis parallel to the global Y axis.
- Global cylindrical – this is a cylindrical axis set with the radial and hoop directions in the global XY plane and the meridional direction parallel with the global Z axis.

In addition to these are user defined axes. User defined axes can be either Cartesian, cylindrical or spherical and are located in space by an origin. The x axis is defined by a vector direction in the x axis (x_a) direction and the xy plane (y_p) of the axis is defined by any vector lying in the xy plane.

The axis set is then constructed from these vectors.

$$\mathbf{x} = \frac{\mathbf{x}_a}{|\mathbf{x}_a|}$$

$$\mathbf{z} = \frac{\mathbf{x} \times \mathbf{y}_p}{|\mathbf{x} \times \mathbf{y}_p|}$$

$$\mathbf{y} = \mathbf{z} \times \mathbf{x}$$

4.3.2 Use of axis sets

Axis sets are used widely in GSA. Some of the uses are

- locating construction grids in space
- node definition
- applying constraint to nodes
- orientation of spring, mass and 2D elements
- orientation of loads
- examination of results

Note that cylindrical and spherical user defined axes cannot be used in some situations. The following table summarises the use of cylindrical and spherical axes in GSA.

Feature	Cylindrical & Spherical Axes Available
Constraint Axes:	
Restraints	yes
Spring Supports	yes
Joints (in GSS)	yes
Joints (in GsRelax)	
Rigid Constraints	
Constraint Equations	
Link Elements	
Settlements	yes
Grid Planes	-
Spring Properties	yes
Mass Properties	yes
2D Element Properties	yes
Node Loads	yes
Applied Displacements	yes

Beam Loads	-
2D Element Loads	-
Current Grid Axes	yes
Output Axes	yes (but data dependent)

4.3.3 Projected axes

When dealing with 2D elements there is still a need to use user axis sets, but their axis sets may not coincide with the plane of the element. Some way is needed to give a consistent axis system in the plane of the surface. The principle is to take some user defined axis set and project this on to the element.

If the axis system selected is Cartesian the projection is based on projecting the **x** axis on to the element, if cylindrical by projecting the **z** axis on to the element. This is achieved as follows for a Cartesian system, where **n** is a unit vector normal to the element and the subscripts p and g refer to the projected and global axis systems:

$$\mathbf{x}_p = \frac{\mathbf{y}_g \times \mathbf{n}}{|\mathbf{y}_g \times \mathbf{n}|} \quad \mathbf{y}_p = \frac{\mathbf{n} \times \mathbf{x}_p}{|\mathbf{n} \times \mathbf{x}_p|} \quad \mathbf{z}_p = \mathbf{n}$$

Using this method to project axes fails when the element lies in a vertical plane. In these situations the axes are then projected so that the **x** axis is normal to the **z** axis and the element normal, **n**.

$$\mathbf{x}_p = \frac{\mathbf{z}_g \times \mathbf{n}}{|\mathbf{z}_g \times \mathbf{n}|} \quad \mathbf{y}_p = \frac{\mathbf{n} \times \mathbf{x}_p}{|\mathbf{n} \times \mathbf{x}_p|} \quad \mathbf{z}_p = \mathbf{n}$$

When the axis is cylindrical or spherical a similar procedure is followed except that at the point considered a local Cartesian axis system is set up, and the projected directions are hoop, meridional and radial (rather than x, y and z).

4.3.4 Grid axes and the current grid

The coordinates of all nodes are stored in global axes, however they are displayed and edited in the axes of the current grid.

The current grid is a reference to a [grid plane](#) and the axes of the current grid are the axes referred to by that grid plane elevated to the grid elevation defined for that grid plane. The default current grid is the global grid plane (which refers to global axes and has the elevation set to zero).

The current grid is defined using the “Sculpt | Define Current Grid” menu command or the “Axes” command on the [Data Options toolbar](#) when the Nodes table is the current view.

The coordinates displayed in the status bar for a Graphic View are displayed in the axes of the current grid (and also in global axes where these differ). Details of the use of the current grid in Graphic Views is given in the [Working with Graphic Views, Current Grid](#) documentation.

The current grid axes can be Cartesian, cylindrical or spherical.

The term “grid axes” is sometimes used to refer to the xy axes of the grid plane. Since the grid plane is planar there is no z component, though the z direction may be taken as being orthogonal to the x and y.

The term “grid coordinates” refers to the 2D (x and y) coordinates in a grid plane.

4.3.5 Constraint axes

The constraint axes of a node are the axes in which the directions of constraints, such as restraints, joints, rigid constraints and constraint equations, act. Constraint axes are also referred to as the local axes of a node.

If the axis set is cylindrical or spherical a local Cartesian axes is calculated at the location of the node and the x, y and z directions of this local axis are used.

4.3.6 Element and member axes

Each element has its own unique element axes based on the orientation information. The particular definition of the [element axes](#) depends on the element type. Element axes are also referred to as the local axes of an element.

4.4 Grid Planes

[Grid Plane Definition](#)

[Use of Grid Planes](#)

4.4.1 Grid Plane Definition

A grid plane is a planar surface.

A grid plane is defined as being parallel to the xy plane of a specified axis set. The elevation of the grid plane is specified as a distance in the z direction from the axis origin. (See [Grid axes and the current grid](#) above.)

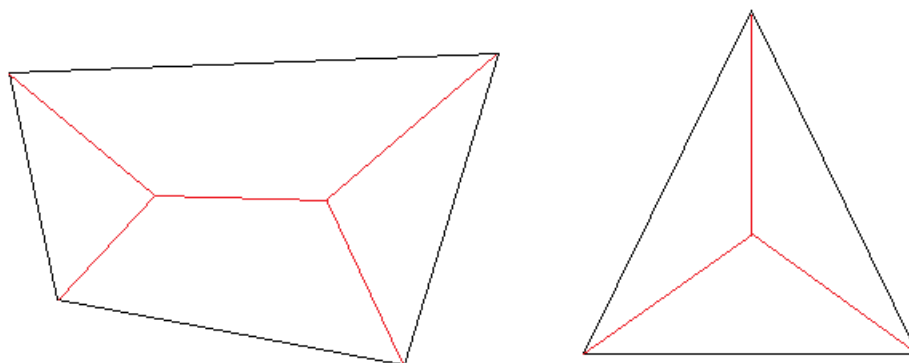
Depending on context, a grid plane may be used to define a set of entities (i.e. nodes, elements or members). In this circumstance an entity is included in the set if it is within the [grid plane tolerance](#) of the grid plane in the local axis z direction.

Where a grid plane is used to define a set of elements, the set may be further refined by use of the element list. Only elements that both lie within the grid plane tolerance of the grid plane and are in the element list are included. Note that grid planes can be referred to in [element lists](#) so the “Edit | Find” Graphic View command can be used to identify elements that are included in a Grid Plane.

One of the main uses of grid planes is for the application of grid loading. The grid plane allows the span type to be defined as one-way or two way. For one-way spanning grid planes a span direction is required. For two-way spanning there are a number of different expansion algorithms:

- **Legacy** - Legacy option: loads assumed to ripple out to edges of panel (does not work on re-entrant panels)
- **Plane approx.** - Uses a plane approximation to distribute the load & adjusts for aspect ratio (load distribution depends on model refinement)
- **Plane approx. (smooth)** - Uses a plane approximation to distribute the load, adjusts for aspect ratio and smooths loads along edges
- **Plane approx. (corner)** - Uses a plane approximation to distribute the load, adjusts for aspect ratio and smooths loads along edges, adjusting for corner effects (recommended)

In addition there is an option to use a modified (simplified tributary areas) algorithm, which gives a simple distribution on fully loaded triangular and quadrilateral panels as shown below



4.4.2 Use of Grid Planes

Grid planes are used as follows:

- To define the current grid. (See [Grid axes and the current grid](#) above.)
- Grid loading is applied to grid planes, positioned in grid plane coordinates.
- Bridge alignments are defined with respect to grid planes.

4.5 Element types

There are a wide range of element types available in GSA. The selection of the correct element type can improve both the efficiency and accuracy of the model.

More:

[Beam element](#)

[Bar type elements](#)

[Spring type elements](#)

[Mass elements](#)

[Link elements](#)

[Spacer elements](#)

[Cable elements](#)

[2D element types](#)

4.5.1 Beam element

A beam element is a two-noded element that models axial, bending and torsional effects. Shear deformations are included when calculating deflections, but can be excluded by setting the shear area factors to 0. The orientation of a beam is defined by its topology, an orientation node and an orientation angle as described in the [Axes section](#). Releases allow some of the degrees of freedom to be treated as pinned. If all the degrees of freedom are to be considered as pins it is preferable to use a bar element. Offsets can be applied to beam elements where the flexible part of an element has to be offset from the node position.

4.5.2 Bar type elements

[Bar](#)

[Tie](#)

[Strut](#)

4.5.2.1 Bar

A bar element is a two-noded element with axial stiffness only. A bar can sustain compression and tension forces. The orientation of bars is the same as for a beam element with an orientation node and angle. Releases and offsets are not allowed.

4.5.2.2 Tie

A tie element is a two-noded element with axial stiffness only. A tie will have zero stiffness under compressive forces.

If a modal analysis is requested the model must behave linearly. In this case the tie behaves exactly like a bar element. A warning is given if tie elements go into compression.

4.5.2.3 Strut

A strut element is a two-noded element with axial stiffness only. A strut will have zero stiffness under tensile forces.

If a modal analysis is requested the model must behave linearly. In this case the strut behaves exactly like a bar element. A warning is given if strut elements go into tension.

4.5.3 Spring type elements

[Spring](#)

[Grounded spring](#)

4.5.3.1 Spring

A spring is a 2 noded element in which stiffnesses are specified explicitly rather than derived from geometrical properties. The orientation of a spring is defined either as being in global or user defined axis directions or, when local axis is specified, by its topology, an orientation node and an orientation angle as described in the [Axes section](#). A spring can be either a lateral or a rotational spring, as defined by the spring property. Releases are not allowed for springs, as springs cannot have both lateral and rotational properties. A spring may be offset if required.

4.5.3.2 Grounded spring

A grounded spring is a single-noded element that acts as an indirect restraint to the model. It is specified in a similar way to a spring element, except that the orientation is defined by the axis specified in the spring properties module.

4.5.4 Mass elements

A lumped mass element is a single-noded element that associates a mass and inertia with a node. No orientation is allowed for a lumped mass element. Offsets can be applied to mass elements. These elements are normally used only for dynamic analysis, as a node load is usually used in static analysis. Only if the gravity module is used, will lumped masses be included in linear and non-

linear static analysis as equivalent nodal forces.

Mass elements have no effect on the masses and inertias used in the iterative process of non-linear analysis.

4.5.5 Link elements

A link is a two-noded element which links two nodes rigidly in the directions specified in the Link Properties module. The element is assumed infinitely stiff in the link direction. Most link types preserve equilibrium but in the case of a custom link the same restriction apply as for a joint and equilibrium is not enforced.

See also [Link elements and rigid constraints](#).

4.5.6 Spacer elements

Spacer elements are only for use in [soap-film form-finding analysis](#) with the GsRelax solver. Spacer elements are ignored by the Gss solver. This element type will only be offered if Form-Finding analysis is selected as an [Advanced Feature Preference](#). Each Spacer chain joins a line of nodes and is, in effect, the opposite of the sliding Cable element. Spacer behaviour is described in detail in the [Form-Finding theory section](#). Spacer chains do not apply any force to their nodes in a direction normal to the surface of the adjoining triangles and quads. Spacers therefore do not affect the surface that is form-found; they only control the position of nodes on the surface that is formed. Spacer chains do not exert any force onto the end nodes of the chain. Because of this, an isolated Spacer element will have no structural effect. To define the multi-noded Spacer chain input a list of 2 noded individual spacer elements with the same property number. Each individual spacer chain must have a unique property number. The program connects individual elements together to form a continuous chain of linked elements by linking common node numbers. A spacer cannot be branched.

4.5.7 Cable elements

Cable elements are used with the GsRelax solver. In linear analysis (i.e. using the Gss solver) they are treated as individual tie elements.

In a dynamic relaxation, non-linear analysis (i.e. using the GsRelax solver), cable elements model a sliding cable. The cable chain is multi-noded and takes tension only. Between the two end nodes of the cable chain forces are transferred to adjoining elements and nodes in a direction perpendicular to the cable only. Thus maintaining a constant tension along the length of the cable. To define the multi-noded cable chain input a list of 2 noded individual cable elements. The program connects individual cable elements together to form a continuous chain of linked elements by linking common node numbers. A cable cannot be branched.<<add figures here>> Each individual length of cable must have a unique property number. An isolated (2 noded) cable element will behave as a [tie](#).

4.5.8 2D element types

Well shaped 2D elements are important if accurate results are to be produced. In particular the aspect ratio (length-to-width), corner angles and flatness are checked. If these suggest badly formed elements, [warnings, severe warnings or errors](#) are reported during the data checking.

More:

[Quad 4](#)

[Quad 8](#)

[Triangle 3](#)

[Triangle 6](#)

4.5.8.1 Quad 4

Quad 4 elements are 4-node 2D elements. These element types are offered if 2D elements analysis is selected as a [Preference](#) and the structure type (in the [Analysis Specification](#)) is not a “Plane” or “Grid”. They can be specified for use with the Gss and GsRelax solvers but are treated completely differently by the two solvers.

The Gss solver treats the Quad 4 as a 4-node finite element. The GsRelax solver converts each Quad 4 element during the analysis into four triangular elements (Triangle 3) by creating an additional dummy node in the centre of the quadrilateral.

The GSS solver can use one of two methods for their out-of-plane formulation (Mindlin or MITC) and two methods for their in-plane behaviour (Bilinear or Allman-Cook). See the [Advanced Solver Settings](#) for more details on the differences.

The behaviour of the Quad 4 element in the GsRelax solver depends on the definition of the Property Type in the [2D Element Property](#) data. If Non-linear analysis is set as an [Advanced Feature Preference](#) the 2D property type definition for a Quad 4 must be Plane Stress, Plane Strain or shell. If Fabric Analysis is also a Preference the element may be either the Fabric or Plane Stress property type. For [Form-Finding](#) the element must be a Fabric type.

4.5.8.2 Quad 8

Quad 8 elements are 8-node 2D elements. These element types are offered if 2D elements analysis is selected as a [Preference](#) and the structure type is specified as a space or grid in the [Analysis Specification](#). They can only be specified for use with the Gss solver and are the preferred Quadrilateral element type, as they do not have the same limitations as the Quad 4 elements.

4.5.8.3 Triangle 3

Triangle 3 elements are 3-node 2D elements. These element types are offered if 2D elements analysis is selected as a [Preference](#) and the structure type is specified as a space or grid in the [Analysis Specification](#). They can be specified for use with the Gss and GsRelax solvers but are treated completely differently by the two solvers.

The Gss solver treats the Triangle 3 as a 3-node finite element. This element does not perform as well as the 6-node finite element, and in general quad elements give better performance than triangular elements so this element is not recommended for general use. The Triangle 3, like the linear Quad 4, can use either the Mindlin or MITC method for their out-of-plane stiffness formulation and either the Bilinear or Allman-Cook for their in-plane stiffness. See the [Advanced Solver Settings](#) for more details on the differences.

The 3-node triangular element is the basic 2D element for use in non-linear analysis with the [GsRelax solver](#).

The behaviour of the Triangle 3 element in the GsRelax solver depends on the definition of the Property Type in the [2-D Element Property](#) data. If Non-linear analysis is set as an [Advanced Feature Preference](#) the 2D element property type must be Plane Stress, Plane Strain or shell. If Fabric Analysis is also a Preference the element may be Fabric or Plane Stress property type. For [Form-Finding](#) the element must be a Fabric type.

4.5.8.4 Triangle 6

Triangle 6 elements are 6-noded 2D elements. These element types are offered if 2D elements analysis is selected as a [Preference](#) and the structure type is specified as a space or grid in the [Analysis Specification](#). They can only be specified for use with the GSS solver but should only be used where it is not feasible to use Quad 8 elements.

4.6 Element axes

It is important for the user to know how an element is oriented, element orientation is defined by element axis that depends on the element type and its definition. Each element has an associated axis that is used both to establish the element stiffness relative to the structure as a whole and to establish the direction of element loading as well as the analysis results.

More:

[Beam element axes](#)

[Spring axes](#)

[Grounded spring and mass axes](#)

[Cable axes](#)

[Link axes](#)

[2D element axes](#)

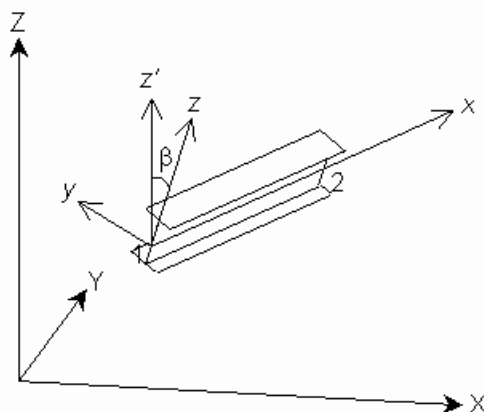
4.6.1 Beam element axes

Beam elements including bars, ties and struts are defined by two nodes locating the ends of the element. The x axis of the element is along the axis of the element (taking account of any offsets) from the first topology item to the second.

The definition of the element y and z axes then depends on the element's orientation, verticality, orientation node and orientation angle. The element is considered vertical in GSA if the element is within the "[vertical element tolerance](#)".

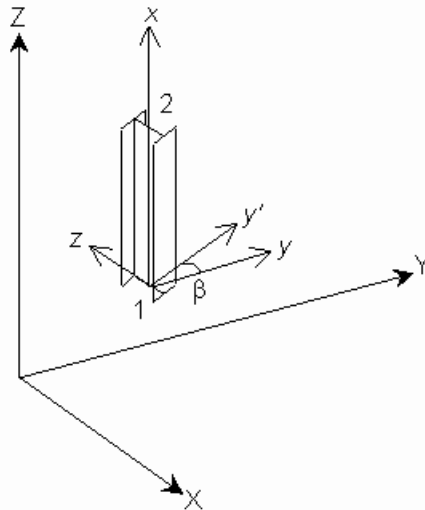
Non-vertical elements

If an orientation node is not specified, the element z axis of a non-vertical element defaults to lying in the vertical plane through the element and is directed in the positive sense of the global Z direction. The element y axis is orthogonal to the element z and x axes. The element y and z axes may be rotated out of this default position by the orientation angle.



Vertical elements

If an orientation node is not specified, the element y axis of a vertical element defaults to being parallel to and is directed in the positive sense of the global Y axis. The element z axis is orthogonal to the element x and y axes. The element y and z axes may be rotated out of this default position by the orientation angle.



Orientation node

If an orientation node is specified, the element xy plane is defined by the element x axis and a vector from the first topology position to the orientation node, such that the node has a positive y coordinate. The element z axis is orthogonal to the element x and y axes. Specifying an orientation node overrides the “vertical element” and “non-vertical element” definitions described above. The element y and z axes may be rotated out of this default position by the orientation angle.

Orientation angle

The element y and z axes are rotated from their default positions about the element x axis by the orientation angle in the direction following the right hand screw rule. This occurs regardless of whether or not the element is vertical and of whether or not an orientation node is specified.

4.6.2 Spring axes

The definition of the element axes for a spring depends on the axis system set in the spring property. If the axis system is “local” then the rules for orientating bars and beams apply. If the axis system is “global” or user defined, these are used and the topology, orientation node and orientation angle are ignored. User defined axes can be Cartesian, cylindrical or spherical.

4.6.3 Grounded spring and mass axes

The definition of the element axes for a grounded spring or mass depends on the axis system set in the spring or mass property. If the axis system is “global” or user defined, these are used. If the axis system is “local” the element axis is taken as global. User defined axes can be Cartesian, cylindrical or spherical.

4.6.4 Cable axes

Cable elements act only in the axial direction, so only the x axis is defined following the same definition as the x axis of a [beam element](#).

4.6.5 Link axes

The local axes of link elements are the same as those of the master node constraint axis .

4.6.6 2D element axes

2D element local axes may be defined either by reference to an axis set or topologically. This is determined by the axis system defined in the 2D element property. If the axis system is "global" or user defined then the specified axis set is used. If the axis system is "local" then the topological definition is applied. User defined axes can be Cartesian, cylindrical or spherical.

Typically defining 2D element local axes by reference to an axis set results in more consistent local axes in the mesh.

The local axes for flat 2D elements are chosen so that the plane of the element is the local xy plane. The normal to the element is defined as

$$\mathbf{n} = (\mathbf{c}_3 - \mathbf{c}_1) \times (\mathbf{c}_4 - \mathbf{c}_2)$$

Where $\mathbf{c}$ is the coordinates on a point on the element, i.e. the coordinates of the node, $\mathbf{n}$, plus any offset, $\mathbf{o}$, at that topology position.

$$\mathbf{c} = \mathbf{n} + \mathbf{o}$$

2D element axes defined by axis set

If the 2D element property axis is set to other than 'local' then the specified axis system is projected on to the element. For Cartesian axes the x axis of the axis set is projected onto the element

$$\mathbf{y} = \mathbf{n} \times \mathbf{x}_{axis} \quad \mathbf{x} = \mathbf{y} \times \mathbf{n} \quad \mathbf{z} = \mathbf{n}$$

The exception to this rule is when the x axis of the axis set is within 1° of the element normal in which case a vector for an interim y axis is defined as

$$\tilde{\mathbf{y}} = \mathbf{n} \times \mathbf{z}_{axis} \quad \tilde{\mathbf{x}} = \tilde{\mathbf{y}} \times \mathbf{n} \quad \mathbf{z} = \mathbf{n}$$

This axis set is then rotated about the element normal equivalent to an orientation angle of 90°.

For Cylindrical and Spherical axes the z axis of the axis set is projected on to the element to become the local y axis.

$$\mathbf{x} = \mathbf{z}_{axis} \times \mathbf{n} \quad \mathbf{y} = \mathbf{n} \times \mathbf{x} \quad \mathbf{z} = \mathbf{x} \times \mathbf{y}$$

Topological definition of 2D element axes

If the 2D element property axis is set to 'local' the local x and y axes are based on the topology of the element.

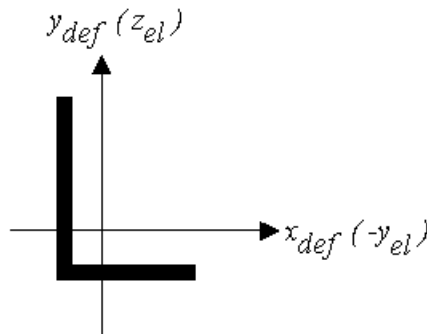
$$\mathbf{x} = \mathbf{c}_2 - \mathbf{c}_1 \quad \mathbf{y} = \mathbf{n} \times \mathbf{x} \quad \mathbf{z} = \mathbf{n}$$

If an orientation angle is defined these axes are rotated by the orientation angle in a positive direction about the element z axis.

4.7 Beam Sections and Section Database

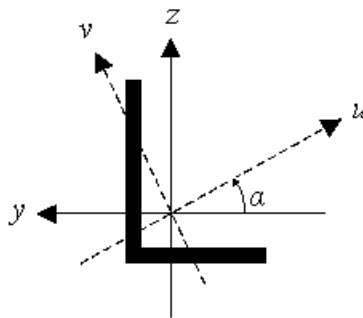
Beam Sections define the properties for beam, bar, tie and strut elements. Only a subset of these properties are used for analysis and another subset are used for design. Sections can be defined by reference to a catalogue, by definition of the section geometry or by explicit definition of the section properties.

Section Definition Axes and Element Axes



Sections are defined with respect to section definition axes, shown in this diagram as x_{def} and y_{def} . The section is used in GSA such that the section appears as defined, when viewed with the element x axis pointing into the page. This has the, perhaps unintuitive, consequence of GSA having to translate the specified (x_{def}, y_{def}) section definition coordinates to $(0, -x_{def}, y_{def})$ [element axes](#) coordinates. (I.e. (x_{def}, y_{def}) translates to $(-y_{el}, z_{el})$.) The translation occurs automatically within GSA, requiring no action from the user. The distinction between section definition axes and element axes is particularly relevant when considering [geometric sections](#). Display “Section Shapes” [labels](#) in a Graphic View to confirm the orientation of sections.

Principal Axes



For a doubly non-symmetric section principal axes are not aligned with element axes. In this circumstance loading applied in element axis directions will produce deflections in the plane orthogonal to the loading unless there are constraints on the section to prevent this behaviour. If there are such constraints the user can choose, when defining the section, to prevent this effect by ignoring the I_{yz} term in which case the I_{yy} , I_{zz} , K_y and K_z values would be used to calculate the element stiffness as if they were principal 2nd moments of area.

More:

[Section types](#)

[Naming convention for sections](#)

[Design section information](#)

[Section Database](#)

4.7.1 Section types

The section types have the following hierarchy:

Section

Catalogue Section

Standard Section

Rectangular Section
 Circular Section
 I Section
 Tee Section
 Channel Section
 Angle Section
 Taper
 Ellipse
 General I section
 Taper T section
 Taper angle section
 Recto-circular
 Recto-ellipse
 Taper I section
 Geometric Section
 Perimeter Section
 Line Segment Section
 Explicit Section

4.7.2 Naming convention for sections

The description of a section depends on the type of the section but is used to identify the particular section. The general syntax of the description is

<Definition type> <Description>

where the description is defined below.

The definition types are

Definition

catalogue

standard

geometric

explicit

Definition Type

CAT

STD

GEO

EXP

More:

[Catalogue sections](#)

[Standard sections](#)

[Geometric sections](#)

[Explicit sections](#)

4.7.2.1 Catalogue sections

For catalogue sections the descriptions based on the catalogue, type and date to provide a string uniquely identifying the section. This can be constructed as follows:

CAT <Type ID> <Sec name> <Date>

So for example a British Universal beam of 457 x 152 and mass of 82 kg/m of 15 June 1987 would be:

CAT UB UB457x152x82 19870615

The date field is normally used only to identify superseded sections, so normally the section description would be abbreviated to

CAT UB 457x152x82

4.7.2.2 Standard sections

The standard sections are identified by a code for the section shape, followed by a list of dimensions defining the section. The section shape descriptors can be followed by an optional unit if the section is not defined in mm, where the shapes are:

SecShape code	Dimensions
rect R ang ular	d, b
rect RHS ang ular hollo w	d, b, t_w , t_f
circ C ular	d
circ CHS ular hollo w	d, t
I I secti on	d, b, t_w , t_f
tee T	d, b, t_w , t_f
cha CH nnel	d, b, t_w , t_f
angl A e	d, b, t_w , t_f
tape TR r	d, b_t , b_b
ellip E se	d, b, k
gen Gl eral	d, b_t , b_b , t_w , t_f , t_{fb}

I section	
tapered T section	$d, b, t_{wt}, t_{wb}, t_f$
tapered TA angle section	$d, b, t_{wt}, t_{wb}, t_f$
rectangular RC	d, b
rectangular RE ellipse	d, b, d_f, b_f, k
tapered I section	$d, b, b_t, b_b, t_{wt}, t_{wb}, t_{ft}, t_{fb}$
secant pile section	d, c, n
secant pile wall	d, c, n
oval OVAL	d, b, t

Note:

b	width
b_b	bottom width
b_f	flat width
b_t	top width
c	distance between pile centres
d	depth, diameter
d_f	flat depth
k	order
n	number of piles
t	thickness
t_f	flange thickness

t_{ft}	top flange thickness
t_{fb}	bottom flange thickness
t_w	web thickness
t_{wt}	top web thickness
t_{wb}	bottom web thickness

so the section description is as follows:

STD <Shape(unit)> <Dimension> <Dimension>...

Thus a 320 × 120 rectangular section would be

```
STD R 320 120
```

or an 8in diameter circular section would be

```
STD C(in) 8
```

4.7.2.3 Geometric sections

There are two types of geometric section: perimeter and line segment. Both are described in terms of 2D vertices. A perimeter section is specified by a polygon to define the section outline and, optionally, one or more further polygons to define voids within the outline. A line segment section is specified by a number of lines of specified thickness.

The syntax for geometric sections gives instructions for constructing the shape of the section. The section must start with a flag to give the type of geometric section. This followed by a series of instructions defining the shape of the section, separated by spaces. The instructions that are used are:

Type	Type Code
Perimeter	P
Line segment	L

and

Instruction	Values	Code
Move to position	(x, y)	M(x y)
Line to position	(x, y)	L(x y)
Set thickness	t	T(t)

The (x,y) coordinates referred to here are in section definition axes. These translate to (-y,z) coordinates in element axes. (See [Beam Sections and Section Database](#) for more information on this.) This translation happens automatically requiring no action by the user.

Thickness is only required for line segment section.

In general a geometric section is defined as

GEO <Type(unit)> <Instruction> <Instruction>...

The available units are m, cm, mm, ft and in, defaulting to mm.

So for example a rectangular section 200mm wide and 400mm deep defined as a perimeter section would be:

```
GEO P M(0|0) L(200|0) L(200|400) L(0|400) L(0|0)
```

or a hollow rectangular section 200mm wide and 400mm deep with wall thickness of 10mm defined as a perimeter:

GEO P M(0|0) L(200|0) L(200|400) L(0|400) L(0|0)
 M(10|10) L(190|10) L(190|390) L(10|390) L(10|10)

or as line segments:

GEO L T(10) M(5|5) L(195|5) L(195|395) L(5|395) L(5|5)

A rectangular section 12.5in by 4.5in defined as perimeter section would be:

GEO P(in) M(0|0) L(4.5|0) L(4.5|12.5) L(0|12.5) L(0|0)

More:

[Geometric section properties](#)

4.7.2.3.1 Geometric section properties

The section properties are first calculated with respect to the section definition coordinate axes. From these the centroid of the section is calculated and then the properties with respect to the centroid.

For a perimeter section the section properties are calculated with respect to the coordinate axes using the following formulae.

$$A = \frac{1}{2} \sum (xy' - x'y)$$

$$A_x = \frac{1}{6} \sum (xy' - x'y)(y - y')$$

$$A_y = \frac{1}{6} \sum (xy' - x'y)(x - x')$$

$$I_{ox} = \frac{1}{12} \sum (xy' - x'y)(y^2 - yy' + y'^2)$$

$$I_{oy} = \frac{1}{12} \sum (xy' - x'y)(x^2 - xx' + x'^2)$$

$$I_{oxy} = \frac{1}{24} \sum (xy' - x'y)(2xy + x'y + xy' + 2x'y')$$

For a line segment section the section properties relative to the coordinate axes are calculated using the following formulae.

$$\begin{aligned}
A &= \frac{1}{2} \sum (lt) \\
A_x &= \frac{1}{2} \sum lt(x + x') \\
A_y &= \frac{1}{2} \sum lt(y + y') \\
I_{ox} &= \sum lt \left[\frac{l^2}{12} - \frac{l^2 - t^2}{12l^2} (x - x')^2 + \frac{(y + y')^2}{4} \right] \\
I_{oy} &= \sum lt \left[\frac{l^2}{12} - \frac{l^2 - t^2}{12l^2} (y - y')^2 + \frac{(x - x')^2}{4} \right] \\
I_{oxy} &= \sum lt \left[\frac{l^2 - t^2}{12l^2} (x - x')(y - y') + \frac{(x - x')(y - y')}{4} \right] \\
J &= \begin{cases} \frac{4a^2}{3} & \text{when it is a single closed section} \\ \sum \frac{l}{t} & \text{(a - area enclosed by the line segments)} \\ \frac{\sum lt^3}{3} & \text{when it is not a single closed section} \end{cases}
\end{aligned}$$

Then, for both perimeter and line segment sections, the centroid of the section and the section properties with respect to the centroid are calculated as follows.

$$\begin{aligned}
x_c &= \frac{A_y}{A} \\
y_c &= \frac{A_x}{A} \\
I_{xx} &= I_{ox} - A y_c^2 \\
I_{yy} &= I_{oy} - A x_c^2 \\
I_{xy} &= I_{oxy} - A x_c y_c
\end{aligned}$$

where (x, y) are the coordinates of a point and (x_c, y_c) are the coordinates of the previous point. The summation is carried out for each point round the section if it is a perimeter section or for each segment if it is a line segment section.

Note that the torsion constant, J of Perimeter sections, and shear factors, K_y and K_z of both perimeter and line segment sections are not calculated and are set to zero. Values for these can be entered from the [Section Modifiers dialog](#).

4.7.2.4 Explicit sections

No description beyond the definition type is given for an explicit section. Thus an explicit section is

EXP

Note that the principal bending axes are assumed to coincide with the local axes and the shear centre is assumed to coincide with the centroid for explicit sections.

4.7.3 Design section information

For design purposes it is necessary to define some extra information about the section. Thus two extra fields are added which are primarily intended for steel structures.

The section type flags if the section is rolled or welded. The cost per unit mass is used to give an approximate cost for the steelwork.

4.7.4 Section Database

Beam sections can also be selected from a section catalogue. Oasys supply a the most widely used catalogues of sections.

4.8 Spring Supports and Ground Springs

When a node is required to be stiff there are two options. Either a stiffness can be assigned directly to the node or a grounded spring element can be attached to the node. The main differences between the two are that:

- Grounded springs refer to a spring property; spring supports have stiffnesses defined directly.
- Grounded springs are of a specified type with a maximum of three stiffness terms (either translational or rotational). Spring supports can have six stiffnesses (three translational and three rotational).
- The stiffness of grounded springs can be specified explicitly in matrix form. The stiffness of spring supports cannot be specified explicitly.
- The stiffness of grounded springs can vary per analysis stage. The stiffness of spring supports is constant across all analysis stages.

Thus in general a grounded spring will be more useful when a number of nodes are to be supported with the same stiffness (so that the same spring property can be used). The spring support is more useful when the stiffnesses under different nodes are likely to be quite different.

4.9 Sets and Lists

[Overview of sets and lists](#)

[Lists and embedded lists](#)

[List syntax](#)

4.9.1 Overview of sets and lists

There are many instances in GSA where there is a need to work with collections of nodes, elements, members or cases. Depending on the context these can be either "Selection Sets" or "Lists".

Lists are used where the user is required to specify a collection of entities. A list is expressed as a string of text in a specific syntax.

Sets are only specified and displayed in Graphic Views though they can be translated into lists and used elsewhere in the program. Forming and working with selection sets is described in detail in ["Working with the Program"](#).

4.9.2 Lists and embedded lists

Lists can be defined directly in the List table and then referred to in other tables where a list is required. In the List table the user must define the list type and give the list a name. The name can then be used to identify the list.

Embedded lists are lists included as part of another data record (e.g. the constraint and load records). In this case the type of list can be inferred from the context and the name of the list is not required, so only the list definition is required.

4.9.3 List syntax

[Type](#)

[Definitions](#)

4.9.3.1 Type

A list can be defined for the following entities:

- **Node**
- **Element**
- **Member**
- **Case**

See also:

[List Definitions](#)

[Lists Toolbar](#)

4.9.3.2 Definitions

The definition of a list uses a set of rules so that a list can be defined in a clear and unambiguous manner. The vocabulary used is case insensitive and consists of:

- *n*—item number by default node/element/member/case depending on the list type
- **to**—to specify a range
- **not**—to specify exclusions
- **step**—to specify the granularity of a range
- **all**—to specify all
- **none**—to specify none
- **and**—to specify items that match both of two conditions
- **or**—to specify items that match one of two conditions (implied if omitted)
- (...)—to clarify the order in which expressions are evaluated
- *#n*—to include list number “n” in a list

Alternatively a list can refer to another list by name. In this case the list name is enclosed in quotes:

- "name"—to include the list named “name” in a list

An element list can refer to a grid plane by name (enclosed by quotes). This specifies all elements included in the grid plane.

The following letters are used as prefixes in:

Node lists

- **N**—node (optional)

Element lists

- **E**—element (optional)
- **P**—property
 - identifies elements that reference the specified property, ignoring stage properties (typical)
- **PB**—beam property
- **PS**—spring property
- **PM**—mass property
- **PA**—area (2D) property
- **PL**—link property
- **PR**—rigid property
- **PG**—spacer property (chain)
- **PC**—cable property (chain)
- **DCS**—RC slab design property
- **G**—group
- **M**—user material
- **MS**—standard material

Member lists

- **E**—member (optional)
- **P**—property
- **PB**—beam property
- **DS**—steel beam design property
- **DSR**—steel restraint property
- **DCB**—RC beam design property
- **G**—group
- **M**—user material
- **MS**—standard material

Case lists

- **L**—load case
- **M**—mode
- **A**—analysis case
- **C**—combination case
- **C_np_p**—permutation p of combination case n
 - ranges of permutations can be specified, such as c4p1 to p5 or c4p1 to p* to specify all permutations in a combination.
- **T**—analysis task

- n (with no prefix) is only valid when there is an unambiguous relationship between load or mode case and analysis case.

Geometry based lists

- **X**—nodes, elements or members on global X line passing through the node following
- **Y**—ditto for Y
- **Z**—ditto for Z
- **XY**—nodes, elements or members on global XY plane passing through the node following
- **YZ**—ditto for YZ
- **ZX**—ditto for ZX

Notes: The items in a list should be separated by one or more space characters, except where there is a parenthesis to separate items. Lists should not be self-referential. Self reference either directly or indirectly would result in an error in interpreting the list. A set or list that is undefined or of the wrong type is interpreted as an empty set of list.

Examples

Examples of lists are:

1 to 100 not (51 to 54) PB1 PA5

or

E1 to E100 not (51 to 54) or PB1 or PA5

A list of nodes on the XY plane passing through node 50 but excluding 43 to 49 would be

XY50 not (43 to 49)

The following associations apply:

- **a and b**
- **not a**
- **a or b**

Notes:

a and or b — not valid

not not a — valid (but not particularly useful)

not a and not b — equivalent to **(not a) and (not b)**

not (a or b) — equivalent to **not a and not b**

a b — equivalent to "**a or b**"

The expression "**a and b or c or not d**" is evaluated as follows:

(a and b) or c or not d

((a and b) or c) or not d

((a and b) or c) or (not d)

((a and b) or c) or (not d))

4.10 Use of Constraints

In the most general sense constraints are where the user is putting some restriction on the free displacement of the structure. Some constraint is required on all structures to prevent pure rigid body motions. Thus in general the minimum number of constraints on a 3D structure is six, three suppressing the translational rigid body modes and three suppressing the rotational rigid body

modes.

In most cases there will be considerably more constraints than the minimum to suppress rigid body motion. These constraints consist of:

- [Restraints and generalised restraints](#)
- [Settlements](#)
- [Element offsets](#)
- [Link elements and rigid constraints](#)
- [Joints](#)
- [Constraint equations](#)
- Tied interfaces

These are discussed in more detail below.

More:

[Conflicting constraints](#)

[Automatic constraints](#)

4.10.1 Restraints and generalised restraints

Restraints are a constraint on the model where the displacement at a node in a particular direction is set to zero. Restraints can be applied to a model in several ways and it is important to understand the different restraint methods.

Implied restraint due to structure type

For a “Plane” structure type the active degrees of freedom are x, z and yy. So the remaining three degrees of freedom are ignored in the analysis. This has the same results as restraining all the out-of-plane degrees of freedom. This feature is set by the program when the user selects a structure type from the “[General Specification](#)”.

Global restraints

In some it is useful to restrain all the degrees of freedom in model in a particular direction. For example modelling a slab with plate elements, where the plate elements have no stiffness in the x and y directions, it may be useful to apply global restraints in the x and y directions. These are also set on the “[Analysis Specification](#)”.

Nodal restraints

Nodal restraints are the most commonly used restraint type. Here particular degrees of freedom at a node are restrained. So, for example, to create a pinned support, the user can apply restraints to a node in the x, y and z directions. These are set in the Restraints page of the “[Nodes](#)” table view or in the [Node Wizard](#).

Generalised Restraints

Generalised restraints are similar to nodal restraints except that they are applied to a list of nodes. It is therefore possible to use a list specifying all the nodes on a plane so as to apply restraint across all of the base of a structure. Generalised restraints are set up in the “[Generalised Restraints](#)” table view.

The total restraint on a node is the sum of all of the restraints acting on it.

4.10.2 Settlements

Settlements are used in conjunction with [restraints](#). A restraint fixes a node so that it is no longer free to move and a settlement determines the displacement imposed on that node for the particular load case. In the load cases where there are no settlements the nodes are treated as simply restrained (i.e. fixed). The restraint must be specified explicitly by the user.

Typically settlements are used to examine differential settlement of the structure. The distortion of the structure can be calculated by applying a series of displacements or settlements (typically in the z direction) at the base of the structure.

Settlements should not be confused with [Applied Displacements](#). It is useful to think of settlements as “constraints”, and applied displacements as “loads”.

A settlement is applied in the constraint axis directions of the node to which it is applied.

4.10.3 Element offsets

Another type of constraint that can be applied is in the form of element offsets. These are used where the ends of the element are effectively rigid and it is required to offset the flexible part of the element. The actual element can then be thought of as shorter than the node to node length of the element with the gaps filled in with “rigid links”. Element offsets are most useful where the span of a beam is less than the column to column centre-line distance or where the effect of eccentricity on the element is important.

4.10.4 Link elements and rigid constraints

At times there are very stiff sections in a model and generally it is not good to model these simply by creating very stiff elements as this can lead to [ill-conditioning](#) problems. It is better to replace the elements in these regions with link elements or rigid constraints. The action of link elements and rigid constraints is substantially the same, the main difference being that a rigid constraint can include many nodes while a link element can only include two nodes, but has some additional linkage options.

The basic idea behind link elements and rigid constraints is that one node is selected as the master node, and the others are the slave nodes. Using the geometrical relationship between the slave and master nodes it is possible to define the displacements and rotations at the slave node(s) in terms of the displacement and rotation of the master. Likewise the forces and moments at the slave nodes can be transferred to equivalent forces and moments at the master. Thus it is possible to remove the slave nodes from the solution and avoid the potential ill-conditioning problems.

The link element can also act as tension link, maintaining the geometrical relationship while the element is in tension; a compression link, maintaining the geometrical relationship while the element is in compression or a bar maintaining the fixed length of the link but not applying any constraint on the rotation of the link. These link types can only be used in [GsRelax](#).

A custom link works in the same way as a joint by simply coupling degrees of freedom at the slave to the same degree of freedom at the master. As this does not take account of the geometry of the element it is possible to create a custom link which does not enforce equilibrium.

The master node of a link element is taken as the first node in the topology list for the element. The master node of a rigid constraint must be specified.

The constraint axes of nodes in a link element or rigid constraint must be the same and may only be Cartesian; not cylindrical or spherical.

4.10.5 Joints

Joints are a facility to allow two nodes to be coupled in particular directions. Thus to create a pin joint, two nodes can be defined at the two parts of the structure to be joined. These are included in a joint, which is linked in the required directions (e.g. x, y and z). In a joint as in a link element there is a slave and master node and the linked degrees of freedom are simply shared between the slave and the master node. The axis directions of a joint are the constraint axis directions of the constituent nodes. This means that if a direction is linked this may be different for the two nodes in the joint; in this circumstance a warning is given, though in the case of GsRelax analyses this is treated as an error.

While a link element or rigid constraint uses the geometrical relationship between slave and masters to ensure equilibrium, this is not true for joints. Thus joints in which the nodes are not coincident will not in general be in equilibrium. To guard against this problem the user is warned if the nodes are not coincident.

4.10.6 Constraint equations

Constraint equations are the most fundamental way of specifying a constraint on a node. This slave node is related to one or more master nodes. The displacement at the slave node in a particular direction is a function of the displacements in a particular direction at a set of master nodes. The basic relationship can be specified in the form:

$$u_{s,i} = \sum_m a_m \times u_{m,j}$$

Rigid constraints and joints can be thought of as particular instances of constraint equations.

The axis directions of a constraint are the constraint axis directions of the constituent nodes. While a link element or rigid constraint uses the geometrical relationship between slave and masters to ensure equilibrium, this is not true for the more general constraint equation. Thus care should be taken when defining constraint equations to ensure that no unwanted constraint is applied to the model. Also the factors in the constraint equation are dimensionless, so if these are to represent actual relative positions of nodes care should be taken with units.

4.10.7 Tied Interfaces

A tied interface works by connecting the slave nodes or the nodes associated with the slave elements to the elements on the master side to form a continuous entity.

A typical use of a tied interface is for joining two meshes with dissimilar mesh densities; this is achieved by establishing a set of constraint equations to "stitch" the nodes on the slave side to those on the master side. A tied interface can be used to attached elements along two edges or elements on an edge to a face, or to connect a beam element to the face of a 2D element, or to connect a node to the face of a 2D element.

The masters must be 2D elements, but the slaves can be nodes or elements of any element type (although link elements should be avoided as these will give rise to conflicting constraints). The master retains all its degrees of freedom.

When 2D element slaves are specified the elements on the master side should be of the same size as or larger than those on the slave side for the tied interface to work well. The accuracy of the tied interface will be controlled by the mesh size on the master side.

4.10.8 Conflicting constraints

There are various rules that must be followed when specifying constraints. Failure to abide by these rules results in constraints that are in conflict. Such conflicts are reported by the pre-analysis data checking.

The rules are as follows:

- a slave degree of freedom may be slave to only one master node.
- a degree of freedom may not be constrained by more than one rigid constraint, link element, joint or constraint equation.
- a restraint applied at a slave node may not conflict with the restraint conditions at its master.
- a master degree of freedom may not be a slave degree of freedom to another constraint. (GSS only; does not apply to GsRelax analysis.)

4.10.9 Automatic constraints

One further constraint that users should be aware of is the Automatic Constraint. These are constraints set up by the Gss solver to enable the solution of shell models where the elements do not have stiffness in all directions. Shell elements do not have a local zz degree of freedom. Thus when the elements are transformed into a non-global plane they appear to have stiffness in all directions but one of the principal stiffnesses at the nodes is zero. To avoid problems of zero stiffness at the solution stage these degrees of freedom have to be removed from the solution. The method of doing this is termed applying Automatic Constraints.

There are a number of ways to apply Automatic Constraints but the default method is to look at how flat the surface is at a node. If all the elements at a node seem to be co-planar then the number of degrees of freedom at that node are reduced to five rather than the normal six. Gss builds a list of automatic constraints, which can then be examined by the user after the analysis stage.

4.11 Applying Load

The term load is used to for a number of different effects that can be applied to the model.

Mechanical loads

Forces or moments applied to the structure.

Distortion loads

Loading caused by displacement or slope discontinuity in elements.

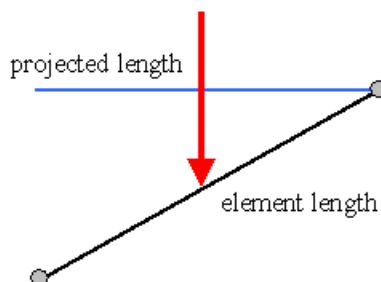
Pre-stress loads

Loading that comes from pre-stress forces or initial strains in elements.

Thermal loads

Loading due to temperature changes either along or across elements.

Irrespective of the type of loading applied all loading results in a set of forces and moments at nodes which are assembled into a structure force vector. For a number of load types loads can be projected. When projected the load intensity is applied to the projected length of the element as shown below.



In general, input data may not be edited when a model has been analysed and results are present. Loading is an exception to this rule in that loads, of any type, are only locked against editing if they are applied in a load case which is referred to by an analysis case that has been analysed. This means that un-analysed load cases can be edited and appended to and new load cases can always be created.

More:

[Node Loads](#)

[Applied Displacements](#)

[Beam Loading](#)

[2D Element Loads](#)

[Grid Loading](#)

[Gravity Loads](#)

[Selecting entities for loading](#)

4.11.1 Node Loads

Node loads are the most fundamental type of load. A node load is a force or moment applied to a particular node or set of nodes. Node loads can be applied in local (i.e. node constraint axis), global or user defined axis directions. User axes can be Cartesian, cylindrical or spherical.

4.11.2 Applied Displacements

Applied displacements are different to the other load types and are also distinct from [settlements](#).

Applied displacement works differently in linear analysis and nonlinear analysis:

- In linear analysis, an applied displacement allows a fixed displacement to be applied to a node in the structure and calculates the deformation of the structure which results in the specified displacement at the specified degree of freedom. Thus applied displacements may be used to investigate the stiffness of a structure by constraining certain degrees of freedom to move by a fixed amount and noting the forces involved.

Applied displacements apply to a particular load case and the node is free in all other load cases. Warning — applied displacements should not be used in conjunction with settlement load in the same load case as it may result in wrong analysis results. If applied displacements have the same load case number with any other loads, the results may not be intuitive as the nodal displacements with applied displacement may not be the same as the applied displacement.

Applied displacements are handled by using [Lagrange multipliers](#).

Applied displacements can be applied in local (i.e. node constraint axis), global or user defined axis directions. User axes can be Cartesian, cylindrical or spherical.

- In nonlinear analysis (GsRelax solver), an applied displacement is taken as initial imperfection, i.e. the node is moved away from its original position by the specified direction and amount before the nonlinear analysis starts. If the structure is stable under the imposed loads in this analysis, the node may move back to its original position, otherwise, the node may continue to move away from its original position.

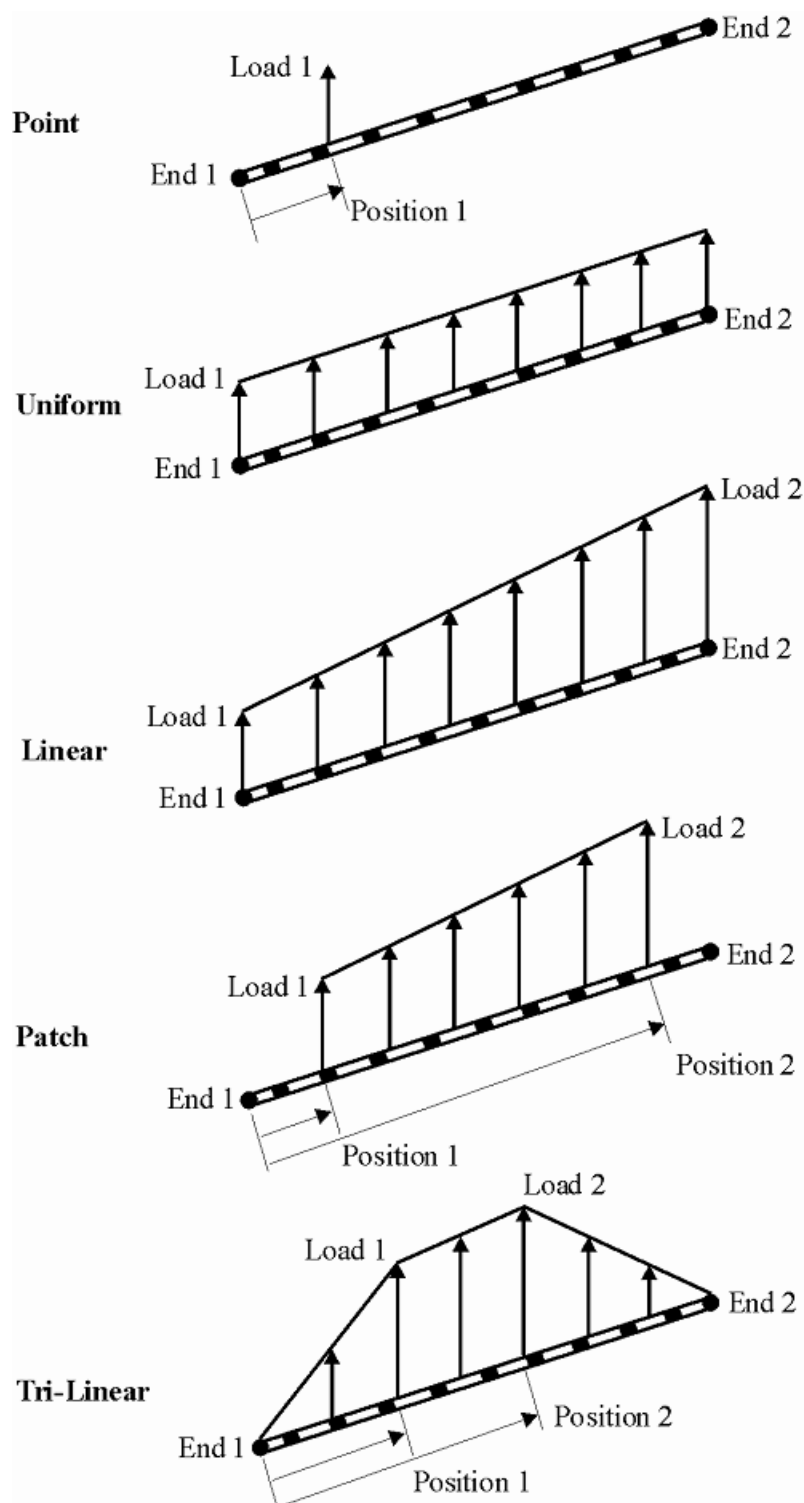
4.11.3 Beam Loading

Beam loads can be applied to beam, bar, tie and strut elements. There are a number of ways in which loads can be applied but in all cases the loading is resolved into an equivalent set of nodal forces and moments. When a position along an element is required this may be specified as an actual distance (e.g. 2.46) or as a percentage (25%).

Beam Loads

Beam Loads are a ways of applying load to beam elements offering the commonly used load patterns as different types. All these loads except the Beam Loads point load type are distributed along the elements. If the beam has offsets the load positions relate to the flexible part of the beam, not the node positions.

The various types of beam loads are illustrated as follows.



Beam Loads offer the option to specify the load as a projected load. When “projected” is set the distributed load is specified as the intensity applied to the projection of the element on the surface normal to the direction of the load; the load intensity actually applied to the element is then

$$W \times \cos \theta$$

where θ is the angle between the element and the surface normal to the direction of the load, and W is the specified load intensity. A typical application of projected loads is for snow loads on inclined

rafters.

Beam loads can be applied in local (i.e. element axis), global or user defined axis directions. User axes can only be Cartesian; not cylindrical or spherical.

Beam pre-stress

There are four kinds of pre-stress load types to choose from in GSA.

- **Pre-stress Force** – This is the force in the element with undeflected nodes, before the structure is allowed to relax into its deflected shape. The resultant force will not necessarily be the same, since the attached structure's stiffness and applied loading also affect the position of the nodes. This option is equivalent to the Initial Strain and Lack of Fit options: The three pre-stress types are related by the relevant element's stiffness and length alone. A positive pre-stress force is equivalent to a negative Initial Strain or lack of Fit, and will result in a tension in the element for normal structures.
- **Initial Strain** – This allows the natural length of an element to be longer (positive) or shorter (negative) than the distance between its undeflected node positions. The difference in length is the initial strain multiplied by the node-to-node distance. The eventual length of the element depends on the stiffness of the attached structure, and the other applied loading.
- **Lack-of-Fit** – Similar to Initial Strain, with the two being related by the element's undeflected node-to-node length.
- **Tendon Pre-stress** – Concrete elements are commonly pre-stressed using tendons or wires. This option allows the simulation of a tendon by the application of equal and opposite forces to the ends of the element. Since the tendon's stiffness is much lower than the beam's, the tendon's elastic properties do not need to be modelled, and the tendon force is assumed to remain constant over the elastic range of beam deflections. The tendon can be offset from the element's centroid to induce a moment as well as an axial force. A positive tendon pre-stress force causes compression in the element. The element forces output from the model are those for the "concrete" section alone: e.g. an unloaded cantilever element that had a pre-stress applied would show a negative axial force despite having no load applied to the tip.

Distortion

The main application of beam distortion is to generate influence diagrams. This makes use of Muller-Breslau's principle to determine influence coefficients from the deflected shapes resulting from member distortions.

Beam thermal loads

In many cases it is useful to consider the effect of thermal loading on a structure. Thermal loads can have two effects. A uniform temperature change causes the element to expand axially but induces no bending. However, the thermal gradient option defines a linearly varying strain through the thickness of the element so resulting in both axial expansion and bending. The positions of the temperatures are used to define the temperature gradient.

More:

[Beam loads in non-linear analysis](#)

4.11.3.1 Beam loads in non-linear analysis

Element loads are converted into nodal forces and moments and added to other nodal forces, GsRelax analysis being performed for nodal forces only. Only torsion loads are converted into moments; all other loads are converted into forces only, with pinned end connections assumed. Obviously this means that beam loads should only be used as a convenient means of generating statically equivalent nodal loads and not in the expectation that the analysis will treat local effects accurately.

The vector of the equivalent nodal load will not change orientation as the element deflects, even if a local axis has been specified for the load direction. This can lead to inaccuracies in the case of

large deflections and results should be checked.

4.11.4 2D Element Loads

2D element loads can be applied to 2D elements but there are some restrictions as to which loads make sense for different 2D element types.

Face loads and edge loads

Face loads should be used where a load distributed over the face of a 2D element is required and edge loads where it is the edges of a 2D element. The nodal forces resulting from face and edge loads are generally non-intuitive.

Face loads and edge loads can be applied in local, global or user defined axis directions. For face loads local is interpreted as element axis directions. For edge loads, local is interpreted as follows:

- x is the direction along the edge in the topology order direction.
- y is the direction into the element, in the plane of the element and normal to the edge of the element at the point at which it is applied.
- z is orthogonal to the x and y directions.

User axes can only be Cartesian; not cylindrical or spherical.

Face loads cannot be used for plane stress, plane strain or axisymmetric elements. Face loads are interpreted differently when the element is a [load panel](#). Edge loads cannot be used for plate elements.

Face loads offer the option to specify the load as a projected load. When “projected” is set the distributed load is specified as the intensity applied to the projection of the element on the surface normal to the direction of the load; the load intensity actually applied to the element is then

$$W \times \cos \theta$$

where θ is the angle between the element and the surface normal to the direction of the load, and W is the specified load intensity. A typical application of projected loads is for snow loading on an inclined roof.

2D element pre-stress

Pre-stress loads can be thought of either as pre-stress forces or tendon pre-stresses applied to the element or an initial strain. The tendon pre-stress option allows for the pre-stress to be offset by a constant distance from the centre of the element in the thickness direction to induce bending.

The pre-stress or initial strain is applied in either the local x or y direction or in both directions.

An element with a positive pre-stress tends to contract, as in a pre-stressed slab.

2D element thermal loads

In many cases it is useful to consider the effect of thermal loading on a structure. Thermal loads can have two effects. A uniform temperature change causes the element to expand axially but induces no bending. However, the thermal gradient options define a linearly varying strain through the thickness of the element so resulting in both in-plane expansion and bending.

More:

[2D loads in non-linear analysis](#)

4.11.4.1 2D loads in non-linear analysis

Element loads are converted into nodal forces and moments and added to other nodal forces, GsRelax analysis being performed for nodal forces only. No nodal forces are applied to quad dummy nodes.

The vector of the equivalent nodal load will not change orientation as the element deflects, even if a local axis has been specified for the load direction. This can lead to inaccuracies in the case of large deflections and results should be checked.

4.11.4.2 2D loads on load panels

Linear 2D elements can be defined as load panels. A load panel element is not structural and so does not contribute to the stiffness or mass of the structure. The purpose of the load panel is to allow loads to be defined as a face load on 2D elements and then distributed to the beam elements surrounding the 2D element. In many ways face loads on load panels gives an alternative to [grid load](#).

A 2D element is specified as a load panel in the properties. A load panel element does not have a material, thickness, etc. Instead a support type and reference edge are specified.

- **Automatic** – this is a special case. The default is that all edges are supported but if some edges are defined by dummy elements the load pattern will be adjusted so that forces are only applied to the supported edges.
- **All edges supported** – the load is distributed to the edges in proportion to the contributing areas.



- **Three edges supported** – the load is distributed to the supporting edges so that as far as possible equilibrium is maintained. The edge opposite the reference edge does not take any load



- **Two edges supported** – the load spans from the sides adjacent to the reference edge.



- **Two adjacent edges supported** – the load is taken on the reference edge and the "next" edge on the element.



- **One edge supported** – the load is all attributed to the reference edge. The moment due to the offset of the load is ignored.



- **Cantilever support** – the forces are distributed as for one edge supported but the moment is applied to the edge so that equilibrium is achieved.



Quad 8 or tri 6 element cannot be selected for load panels.

Setting the 'Load panels' labelling option in the [Labels and Display Methods: On Elements dialog tab](#) displays the load distribution pattern on load panels.

The loads from the load panels are distributed by searching for beam/bar elements that bound the panel and attributing load to these elements based on the support condition. There must be elements

along all edges of the load panel element, but provided the edge is not loaded these may be dummy elements. An appropriate support condition should be selected to ensure that no load is attributed to the dummy elements. In cases where there may be some confusion as to what defines the edge of the panel any element which spans from one corner to the next will be reckoned to define the edge.

For elements with rigid offsets the basic load distribution is modified to avoid loads on the rigid offsets.

4.11.5 Grid Loading

Grid loads are loads located in space in terms of [grid plane coordinates](#). Three types of grid load are possible – point loads, line loads and area loads. The loads are not applied directly to the structure but by comparing the location of the loads relative to elements in the structure. The advantage of grid loads is that the loads can remain unchanged if the local definition of the structure changes, with the disadvantage that the loads are not tied to the structure.

The structure is considered as a collection of polygonal regions bounded by elements. These regions are referred to as panels. The panel associated with the loading is found and provided a number of checks are satisfied the load can be considered as a load on the structure. It is possible, though neither necessary nor desirable, to expand grid loading into an equivalent set of beam loads. Alternatively the grid loads are expanded as required by the solver.

Application of grid loading

The way in which the grid loads are applied to the structure depends on the span type associated with the grid plane.

Loads on a one-way spanning grid plane are assumed to span from one side of the panel to the other as is carried on a plank (or set of unconnected planks) spanning across the panel in the span direction. Thus a point load on a one-way spanning panel produces a pair of point loads on the panel edges.

Area loads on a two-way spanning grid plane are treated in a special way if the panel is triangular or quadrilateral, otherwise two-way spanning panels are treated as multi-way spanning. For two-way spanning panels the angle at each corner is bisected and the intersection of these lines is used to split the panel up into three or four regions. For a rectangular panel this leads to the typical “back of an envelope” pattern. The load on each of these regions is then assumed to be applied to the nearest edge. (Note that grid load expansion for two-way spanning panels is less numerically intensive than that for multi-way spanning panels and the resulting expanded loads are generally more appropriate so two-way spanning is recommended over multi-way spanning.)

Loads on multi-way spanning panels are assumed to radiate in all directions from the point where the load is applied to result in a load which varies around the edge of the panel. Line loads are considered by dividing the line into a series of short segments each of which can be considered as an equivalent point load. Integrating the loading from each of these points gives the loading from the line load. Area loads are considered in a similar way but in this case a grid is superimposed on the loaded area. The cells of the grid are trimmed (if required) for panel and load boundaries and then the load is considered as set of equivalent point loads. As with the line loads the total load is the integral of all the point load contributions.

In all cases the load distributions are chosen to result in a set of loads which are in equilibrium with the defined grid loading.

Setting the 'Draw current grid panels' display method on the [Labels and Display Methods: Display Methods dialog tab](#) displays the panels that have been identified for the current grid plane.

See also [Technical note – Grid Loading](#).

Face loads on 2D load panels

An alternative to grid area loads is to apply face loads to 2D elements defined as [load panels](#).

4.11.6 Gravity Loads

Gravity loads are one way to account for the dead weight of the structural elements, but excluding any non-structural loads. However gravity loads can be applied in any global direction so it is possible to use gravity loads to model transverse effects as well as vertical effects. The gravity load considers all the mass of the structure that is modelled. Non-structural elements, floor slabs etc which are not modelled explicitly are excluded. The gravity loads are defined in terms of “g”, thus a normal gravity loading on the structure is modelled with an acceleration of $-1g$ ($1g$ in the negative Z direction).

Normally gravity is thought of applied to the whole structure. In some cases it is convenient to look at gravity applied to a subset of the structure. In this case gravity loads are applied to elements specified by a list.

This is useful where design checks which require different gravity factors on different spans. So for example if elements 1, 3, 5, ... are assigned gravity loading in load case 1 and elements 2, 4, 6, ... are assigned gravity loading in load case 2. Then it is possible to examine the effects of adverse and beneficial loading using $L1 + 1.4L2$ and $1.4L1 + L2$.

4.11.7 Selecting entities for loading

Node and element loads apply to lists of entities. These lists can be entered directly in the loading tables or selections can be made graphically using the “Select Nodes” or “Select Elements” options on the [Cursor Mode toolbar](#). “Copy” (Ctrl+C) will copy these to the clipboard and they can then be pasted directly into the appropriate loads table. No check is made to see if a list of elements is being pasted into a node load table or vice versa.

Checks can be made that the loading has been applied correctly by requesting a [load diagram](#).

4.11.8 Patterned Load

Several tools are available to assist in defining patterned floor loading. These tools are focused on the method described in the technical note '[An Efficient Alternative Approach for Patterned Load Analyses of Arbitrary Multiple-ways Spanning Floor Systems](#)'. There are three tools, grouped under the Tools | Building Analysis menu item.

[Create Load Panels on Storeys](#) – examines the storey grid planes and creates load panel elements

[Create Patterned Loads](#) – allows specification of adverse and beneficial load factors and creates elements loads for ‘mean’ and ‘difference’ loads.

[Create Patterned Load Combinations](#) – allows analysis cases to be selected and sets up the combinations of ‘mean’ and ‘difference’ analysis cases to give maximum and minimum envelope cases.

Patterned load - typical workflow

Build a GSA model as normal. In order to simplify the loading use storey grid planes to define the floor levels. The ‘Create Load Panels on Storeys’ tool can then be used to create load panel elements where the floor loading is to be applied.

Define the dead and imposed loads as normal, then use the ‘Create Patterned Loads’ tool to create a new set of loads on the model. At this stage adverse/beneficial factors can be specified for the loads and there is the option to include any of beam loads, face loads and gravity loads. These are then used to create a mean load case and a set of difference load cases.

$$f_{mean} = \frac{(\gamma_{G,sup} + \gamma_{G,inf})}{2} f_G + \frac{(\gamma_{Q,sup} + \gamma_{Q,inf})}{2} f_Q$$

$$\Delta f_i = \frac{(\gamma_{G,sup} - \gamma_{G,inf})}{2} f_{Gi} + \frac{(\gamma_{Q,sup} - \gamma_{Q,inf})}{2} f_{Qi}$$

Where subscripts G and Q relate to permanent (dead) load and variable (imposed) load respectively, and sup and inf relate to unfavourable (adverse) and favourable (beneficial) effects respectively.

These are then used in a linear static analysis as normal.

The final stage is to use the 'Create Patterned Load Combinations' tool which then uses the mean load case and difference load cases to set up combination cases that will give an envelope of the results for the patterned loading.

4.12 Cases and Tasks

Once a model is set up the user then wants to analyse the model. Before analysing the user should have a clear idea of the differences between load cases, analysis cases and combination cases. For anything other than simple linear static analysis it is also useful to have some idea of the relationship between analysis cases and analysis tasks.

More:

[Load cases](#)

[Analysis cases](#)

[Analysis tasks](#)

[Analysis wizard](#)

[Combination cases](#)

[Enveloping in GSA](#)

4.12.1 Load cases

Loading is applied in load cases, so load cases are a convenient way of grouping together a collection of loads that are to be considered together. Thus a load case may be set up to contain all the loads associated with dead loads, another to contain all the loads associated with live loads etc.

In GSA the load cases are only used to group loading and are not used directly for analysis. The case type information is used when exporting concrete design data in establishing which analysis cases may be relevant.

4.12.2 Analysis cases

Analysis cases are one of the basic concepts that needs to be understood for anything other than a linear static analysis. It is important to understand the difference between a load case and analysis case. This is best achieved by using an example.

Consider setting up a model with two load cases, one for dead load and one for live load. It can be useful to keep these in separate load cases so let us assume the dead load is load case 1 and the live load is load case 2. If we want to look at the deflections and forces from both of these loads, there are two approaches: either carry out an analysis with the load in load case 1 and an analysis with the loads in load case 2 and then combine the results post-analysis, or carry out an analysis with the loads in both load cases 1 and 2. Provided the analysis is linear these will give the same results but for a non-linear analysis they will be different. Using the prefix L for load case and A for analysis case we can summarise the different approaches. In the first case we construct two analysis cases one corresponding to each load case:

analysis case	load case
A1	L1
A2	L2

and in the second, one analysis case corresponding to the combined loading in cases 1 and 2

analysis case

A1

load cases

L1 + L2

The description of the analysis case for static loading has the general form

$$a_1 L_1 + a_2 L_2 + \dots$$

The concept of analysis cases becomes more useful when applied to modal dynamic analysis where the results relate to mode shapes of the structure and not to loading. Thus for a modal analysis we may have ten analysis cases corresponding to modes 1 to 10 and these have nothing to do with loading.

Analysis cases cannot exist in isolation but are defined with respect to analysis tasks.

The types of analysis case are:

- **load**—cases that relate to static loading e.g. 1.4L1 + 1.2L2
- **mode**—modal results from dynamic or buckling analysis e.g. M1
- **response**—response spectrum results e.g. R1
- **envelope**—e.g. C1 or C2

See also [Analysis tasks](#).

4.12.3 Analysis tasks

An analysis task is a package of work for the solver. Thus we can have a static analysis task, a modal analysis task, etc. Each analysis task has one or more analysis case(s). The distinction is that the cases corresponds to result sets and define items such as loading (in the static case) while the task describes what the solver has to do. So for example the task will determine which solver has to be used, what solution option in that solver is required and other control parameters such as the number of iterations, convergence tolerances etc.

With the exception of a simple linear static analysis the analysis task is set up in the Analysis Wizard. Once set up these analysis tasks can be managed in the Task View, where the analysis tasks and associated analysis cases are displayed in a tree form.

See also:

[Analysis Wizard](#)

[Working with the Task View](#)

4.12.4 Analysis wizard

For anything other than a simple analysis the analysis wizard is the way to set up the analysis cases and tasks. Depending on the solver chosen the user will be guided through the process of setting up an analysis. Once all the data is set up the user is given a choice of analysing **now** or **later**. Choosing to analyse now will carry out the data check and start the analysis. Choosing later will leave the analysis cases ready for analysis at a later date.

See also [Analysis Wizard](#).

4.12.5 Combination cases

Combination cases are similar to analysis cases but differ in two respects:

- Results for combination cases are inferred from analysis case results and not calculated explicitly.
- Combination cases can be enveloping cases, as described in [“Enveloping in GSA”](#).

The syntax of combination cases is similar to that of analysis cases, except that they are based on analysis cases rather than load cases, thus

$$a_1A_1 + a_2A_2 + \dots$$

However, the syntax of [combination case descriptions](#) is extended to allow specification of enveloping cases.

As combination cases are linear combinations of analysis cases the results of a combination are only valid if all the analysis cases are linear. This means that combination cases should not, in general, be used with P-delta, non-linear analysis cases or when tie or strut elements are present in the model; in this circumstance the desired effect can be achieved by specifying a combination of load cases in an [analysis case](#). In addition modal results represent mode shapes rather than a specific solution so combinations of modal results are unlikely to be helpful.

4.12.6 Enveloping in GSA

Most results produced by GSA can be enveloped. A result envelope is the worst (maximum and minimum) results encountered in a series of analysis or combination cases. Result envelopes are produced for enveloping combination cases.

Envelopes may be specified in the Combination Cases table. The syntax of the [combination case description](#) allows the use of various keywords for describing enveloping combination cases. Typically an enveloping case will include the keyword “or”, though other syntax is available that is interpreted as such. An enveloping case represents a number of permutations of simple cases; during the enveloping process GSA compares the results for each of these simple cases to arrive at an envelope.

Envelopes described in the Combination Cases table are extracted from the permutations that the case describes at the time of requesting output. This can be a time consuming operation especially when the enveloping combination case identifies large numbers of permutations. (See below.) Alternatively [Analysis Envelopes](#) may be calculated as a batch operation to store the envelope results for subsequent, immediate display.

In Output Views the output for an enveloping case reports the maximum and minimum values, each coupled with the permutation number of the permutation that produced that value. The permutation numbers are listed against the interpreted simple cases in the Combination Case and Envelope Details output. Output of envelopes may be either for all components listed or for a defined “subject component” reported alongside coexistent values for other components.

Output of a particular permutation of an enveloping case may be requested.

Note: Where different components being enveloped in a table are related (and a component has a direction as well as magnitude which can vary between cases – e.g. resolved forces and displacements, 2D element derived stresses) the relationship between the components will not be satisfied where the values come from different permutations.

In Graphic Views enveloping is offered as follows.

- The maximum and minimum deformed shape is drawn.
- The maximum and minimum diagram is drawn and there is an option to annotate just the maximum or minimum or both.
- Either the maximum, minimum, absolute or signed absolute value is contoured.

Note: If a diagram of 2D element derived stresses is requested, what will be plotted is the most positive Max and Min stresses at the most positive Angle, and the most negative Max and Min stresses at the most negative Angle. If the Angle is from a different permutation to the Max and Min stresses this is not an accurate representation of the state of stress.

An enveloping case yields maximum and minimum values, as discussed above. The syntax of the [combination case description](#) also allows operators to be applied to enveloping cases to yield just the maximum, just the minimum, the absolute or the signed absolute values. These operators are useful for both specifying the required output and minimising the processing involved in calculating large enveloping cases. Examples of their use are given in the [Syntax of combination case descriptions](#) section.

4.12.7 Syntax of combination case descriptions

Combination cases may be used to combine the factored results of a number of analysis and combination cases. They may also be used to identify a number of permutations of factored cases for the enveloping of results. The former are referred to as “simple” combination cases and the latter as “enveloping” combination cases.

Simple combination cases

The description of a simple combination case is made up of factored analysis and combination cases, ‘+’ (plus), ‘-’ (minus) and parentheses.

Rules:

- ‘+’ (plus) or ‘-’ (minus) causes the cases to be added or subtracted.
- ‘0’ (zero) represents a data value of 0.0. I.e. $0.0 \times (\text{any case})$.
- A missing factor implies a factor of 1.0.
- A case factored by 0.0 is set to ‘0’ (zero) on validation.
- A factor outside parentheses implies that the case references within the parentheses each have that factor applied.
- Case names within double quotes may be used instead of case reference numbers.
- Recursion is not permitted. I.e. if combination A refers to combination B then combination B cannot refer to combination A.

A simple combination case identifies a single permutation of factored analysis cases.

Operators on combination cases

The operator $\text{srss}(A1, C2, \dots)$, where the factors are simple combinations, evaluates to the square root of the sum of squares

$$\sqrt{A1^2 + C2^2 + \dots}$$

The operator $\text{sign}(C1, A2)$ returns the results for the first argument with the sign of the second analysis case.

Enveloping combination cases

Keywords ‘or’, ‘to’, ‘#n’ and “name” are available for describing enveloping combination cases.

Rules:

- ‘A1 or A2 or C3’ causes A1, A2 and C3 to be enveloped.
- ‘(1.4 or 1.0) A1’ causes 1.4A1 and 1.0A1 to be enveloped.
- ‘A1 to A5’ is shorthand for ‘A1 or A2 or A3 or A4 or A5’. ‘C2 to C5’ is valid but ‘A1 to C5’ is not valid.

- References to Case Lists can also be used as shorthand for enveloping; each case in the list is enveloped. For example, if List 3 is named "complex" and is a case list and has the description 'C3 A1 to A9 step 2 not A5' then entering '#3' or "complex" in the combination description is shorthand for 'C3 or A1 or A3 or A7 or A9'.
- Where the same name is associated with both a List and a Case the List takes precedence. For example, if "complex" is the name of both a case list and a combination case then a reference to "complex" is a reference to the case list.
- Otherwise, there is no rule of precedence so '+' and '-' may not be placed within the same parentheses as 'or'.
- You can also envelope all of the analysis cases in a task by the task number, for instance 'T2' will envelope all of the cases in task 2.
- An enveloping combination case evaluates a number of permutations of factored analysis cases. The permutations derived from an enveloping combination case are listed in the Combination Case and Envelope Details output. The output from an enveloping combination case typically reports the maximum and minimum values occurring in these permutations, and identifies which permutation produced this value. The user can choose to maximise/minimise a single "subject component" only; the remaining values in the output will then be the coexistent values (e.g. maximum M_{yy} with coexistent F_x , F_z etc.).

Operators on enveloping combination cases

In the case where C1 and C2 are both enveloping combinations with 4 permutations then $C1 + C2$ produces 16 permutations; i.e. $C1,P1 + C2,P1$; $C1,P1 + C2,P2$; ... $C1,P4 + C2,P4$. The amount of processing involved in calculating the envelope of C1 and C2 can be reduced by enveloping just the maximum and minimum values of C1 with the same of C2. This can be achieved by applying the following operators to enveloping combinations so as to extract a single value from the envelope, as if the case were a simple combination case.

Rules:

- 'C3max' yields just the maximum value from envelope C3.
- 'C3min' yields just the minimum value from envelope C3.
- 'C3abs' yields the greater of the absolute maximum and the absolute minimum value of envelope C3.
- 'C3signabs' is the same as "C3abs" except that the original sign is retained.
- 'C3p2' yields permutation 2 from envelope C3.

For example:

Case	Value
A1	1
A2	-1.5
A3	-2
C3 = A1 to A3	1(p1) -2(p3)
C3max	1
C3min	-2
C3abs	2
C3signabs	-2
C3p2	-1.5

Note the following limitations when results are produced for a case that includes an enveloping

operator, directly or indirectly:

- The permutation references identifying the permutations that cause the results are not output.
- “Envelope subject component only” (i.e. envelope of one component reported with coexistent values) is not available.
- Resolved values (e.g. |U|) are not output.
- The deformed shape is not available.
- Angles for 2D element principal stresses, forces and moments are not output. Principal diagrams are not available. The values given for derived stresses, forces and moments are the numerical sum not the vectorial sum (which is conservative).

Optimising envelopes

The enveloping operators may be used to minimise the processing involved in generating results for a complex envelope which could otherwise require the calculation of thousands of permutations, slowing down diagram and output view production dramatically.

For example:

Case	Description	Permutations Calculated	Case Type
C1	A1 to A4	4	Envelope
C2	A5 to A10	6	Envelope
C3	C1 + C2	24	Envelope
C4	(C1max or C1min) + (C2max or C2min)	10	Envelope
C5	C1max + C2max	10	Simple
C6	C1min + C2min	10	Simple
C7	C5 or C6	20	Envelope
C8	C1abs + C2abs	10	Simple

Note that:

- The “Permutations Calculated” are the total number of permutations for which derived results have to be calculated and compared to arrive at a value.
- C3, C4 and C7 give the same results but C4 will produce them quickest.
- The “Envelope” case type yields both maximum and minimum values.
- The “Simple” case type yields a single value, even though several permutations may have been calculated and compared to arrive at that value.

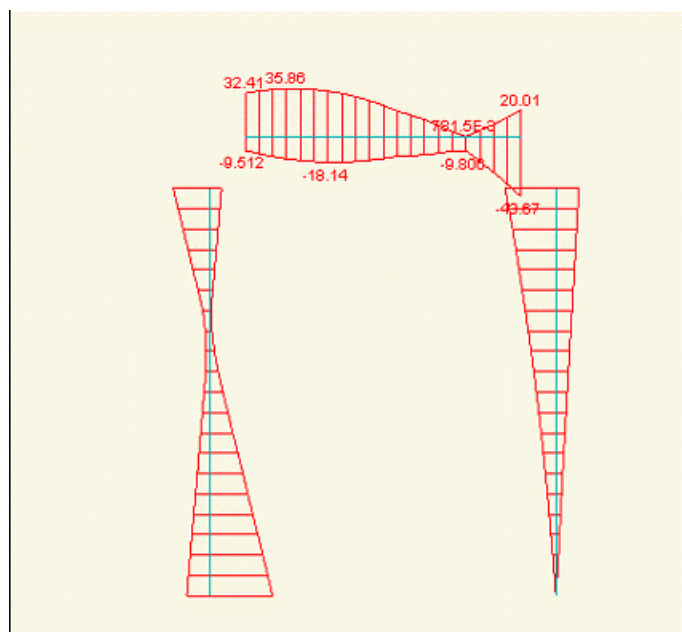
Combination examples

Examples of valid combination case descriptions:

Case	Description	Permutations
C1	$1.4A1 + 1.6A2 - A3$	(Not an envelope)
C2	$1.4A1$ or $1.6A2$ or $A3$	$1.4A1$ $1.6A2$ $1.0A3$
C3	$(A1 \text{ or } -1.4A1) + (0 \text{ or } -1.6A2)$	$1.0A1$ $1.0A1 - 1.6A2$

		-1.4A1
		-1.4A1 - 1.6A2
C4	$(1 \text{ or } -1.4)A1 + (0 \text{ or } -1.6)A2$	(Same as C3)
C5	$C2 + (A4 \text{ or } A5)$	1.4A1 + 1.0A4
		1.4A1 + 1.0A5
		1.6A2 + 1.0A4
		1.6A2 + 1.0A5
		1.0A3 + 1.0A4
		1.0A3 + 1.0A5
C6	A1 to A4	1.0A1
		1.0A2
		1.0A3
		1.0A4
C7	A5 to A10	1.0A5
		...
		1.0A10
C8	$(C6_{\text{max}} \text{ or } C6_{\text{min}}) + (C7_{\text{max}} \text{ or } C7_{\text{min}})$	1.0C6max + 1.0C7max
		1.0C6max + 1.0C7min
		1.0C6min + 1.0C7max
		1.0C6min + 1.0C7min
C9	C6abs + C7abs	(Not an envelope)

The following image shows a Graphic View bending moment diagram envelope.



4.13 Mass and Weight

Mass in the structure is represented by the mass of the structural elements and by lumped mass elements. Weight is the effect of gravity on this mass. Mass is the key aspect for dynamic analysis while weight is key for static loads.

Mass modification

There is scope to modify the basic mass of the elements, which can have an effect on both static and dynamic analysis.

For a beam, bar, tie or strut elements the mass is the area $\times$ length $\times$ density. There is an option in the section properties to modify the effective area of the section and this modification can either be applied or ignored in the mass (and weight) calculations.

For a 2D element the mass is the area $\times$ (thickness $\times$ density + additional mass). There is an option to modify the effective thickness for mass and weight calculations.

For mass elements the mass is specified directly. There is also an option to modify the mass in different directions. This is only available for dynamic analysis and is primarily intended for fluid structure interaction calculations.

Additional mass in dynamic analysis

When building a model it is convenient to ignore non-structural parts of a building and represent these as loads: usually beam loads and 2D face/edge loads. If these are ignored in a dynamic analysis the response will be incorrect. The 'additional mass' option for dynamic is intended to allow for these to be properly included. In general these loads represent the effect of gravity of the non-structural aspects of the building, so to convert these to mass the load vector is assembled and the mass is calculated (assuming z is vertical) from

$$w = f_z$$

$$m = \frac{w}{g}$$

Any 'gravity' loads are ignored in the additional mass calculation as the mass of these loads is defined explicitly.

Weight from mass in static analysis

In a static analysis the weight of the structural elements is accounted for by gravity loading. The general case is

$$\begin{Bmatrix} f_x \\ f_y \\ f_z \end{Bmatrix} = \begin{Bmatrix} a_x \\ a_y \\ a_z \end{Bmatrix} mg$$

but the weight calculation corresponds to the factor being 0 for x and y and -1 for z so

$$f_z = -mg$$

This is always based on a scalar mass so the mass modifiers applied to a mass element are excluded.

The weight calculated from the mass is likely to be an underestimate as the mass of connections, stiffening plates, etc. is ignored. This can be adjusted for by modifying the gravity factor, so for a 10% allowance the gravity factor can be set to -1.1.

4.14 Assemblies

Assemblies are way to define an entity that is formed from a collection of elements and can be thought of as a 'super-element'. This is not an analysis entity but just a convenience for post-processing. Typical uses of assemblies include cores, where the core is modelled with 2D finite elements or wall elements, or trusses where the truss is modelled with top and bottom chords and bracing. In both these cases the assembly is identified by a list of included elements.

Assemblies can be straight or curved. In the case of a straight assembly it can be defined either by an axis and offset or by a set of topology. This defines the position of the x axis of the assembly. The axis or an orientation node defines the xy plane of the assembly. Curved assemblies can be defined, but only with the topology option.

Assembly results can be in the form of tabulated output, diagrams and contours of displacements, drifts and force/moments, for analysis and combination cases, including envelopes.

The 'Tools | Investigate Model | Check Assemblies' tool checks all assemblies for missing topology items, and invalid geometry configurations.

4.15 Cursor modes in Graphic Views

There are various modes of operation of the cursor in Graphic Views. The following table summarises the scope of these cursor modes.

	Rotate	Zoom	Volume	Omit	Select	Polyline	Sculpt Geometry
Click	-	zoom in & pan	delete last volume	omit related entities depending on 'colour by' mode	clear & select picked item	add vertex	create node if necessary, add node to topology list
Drag	rotate	zoom box	new inclusive volume	-	clear & select items	drag existing vertex	drag existing node on grid plane
Ctrl+Click	pick object point	pan	delete last volume	undo last omission	toggle picked item	-	as Click but use node as start of next element
Ctrl+Drag	Vert.: zoom - Horz.: distance		new exclusive volume	-	toggle items	-	-
Shift+Click	reset object point	zoom out & pan	delete all volumes	omit selected entity	select picked item	-	-
Shift+Drag	pan	-	-	omit selected entities	select items	-	-
Ctrl+Shift+ Click	re-scale	re-scale	delete all volumes	undo all omissions	re-scale	re-scale	re-scale

Note:

1. Either Escape or clicking outside the Graphic View will cancel most operations.
2. The selection point for elements is an [adjustable preference](#).

4.16 Numeric formats

The numeric format governs the way that numbers are output in GSA. This is best illustrated by an example of output of E , ν and ρ for a standard material in the three different formats.

		E (N/m ²)	ν	ρ (t/m ³)
Engineering	3 sig. figures	205.E+9	300.E-3	7.85
Decimal	2 decimal places	2.0E+11	0.30	7.85
Scientific	4 sig. figures	2.050E+11	3.000E-1	7.850

The preferred numeric format is set in the [Miscellaneous Preferences](#). This can be adjusted locally for Output Views using the Numeric Format button on the [Data Options toolbar](#).

4.17 String IDs

Most GSA objects can have an associated string ID (sID). Typically these are used to associate GSA objects with other objects, whether GSA or third party. String IDs are of little interest to the typical GSA user.

An example of the use of sIDs in GSA is in 2D mesh generation: in that process a unique sID is assigned to the region that defines the mesh and the same sID is assigned to all of the data that are generated for that region.

GSA objects that cannot have sIDs assigned are:

- Specification modules
- Result modules

Loosely speaking, a string ID is a string and it is up to the author, or authoring application, to devise a syntax for the string that makes sense to the author. However, there will be a rapid decline to mayhem if some rules in the use of sIDs are not adhered to.

Rules in the use of String IDs

- The typical format of a sID is a 'tag, value' pair, enclosed by curly braces. e.g. "{tag:value}"
- An object may have several 'tag, value' pairs, possibly written by different authoring applications. e.g. "{tag1:value1}{tag2:value2}..."
- 'tag, value' pairs may be nested. e.g. "{tag1:value1}{tag2:value2}{tag3:{sub-tag1:sub-value1}{sub-tag2:sub-value2}...}"
- An authoring application will devise a sID tag that will stand a good chance of being unique. e.g. "MyApp"
- Nested 'sub-tag, sub-value' pairs are owned by the authoring application of the enclosing tag.
- It is the duty of the authoring application not to destroy or corrupt any sID 'tag | value' pairs not written by itself.

Part V

5 Program Data

This chapter describes the different types of data that can be used to describe the model. The data is organised in modules and displayed for input and editing in a number of dialogs and tables. It is also possible to input and edit the data graphically (see “[Sculpting](#)” in the “Working with the Program” section). All the tables can be accessed from the Data pull down menu, or from the Tables tab in the Gateway. The most commonly used tables can be accessed from the [Assisted Input toolbar](#).

This toolbar is displayed by default along the lower edge of the view. It contains buttons to facilitate access to the most frequently used data tables and views. It may be removed or restored through the “View | Toolbars | Assisted Input” (Alt+F7) menu command.

More:

[Axes](#)

[Nodes](#)

[Elements](#)

[Grid Planes](#)

[2D Polylines](#)

[Members](#)

[Material Properties](#)

[Beam Sections](#)

[Properties](#)

[Form-Finding Properties](#)

[Design Properties](#)

[Constraints](#)

[Nodal Loading](#)

[Beam Loading](#)

[2D Element Loading](#)

[Grid Loading](#)

[Gravity](#)

[Dynamic Response](#)

[Raft](#)

[Bridge](#)

[Bridge Loading](#)

[Cases and Tasks](#)

[General Data](#)

[Analysis Stages](#)

5.1 Axes

This feature enables new user-defined axes systems to be specified e.g. to specify an inclined roller support or to assist in interpretation of results, or to specify the Construction Grid. Axes systems may be Cartesian, cylindrical or spherical. A user axes set is created by specifying the following:—

Name

Used as a convenient way of identifying an axis.

Origin

The location of this axis system in the global axis system.

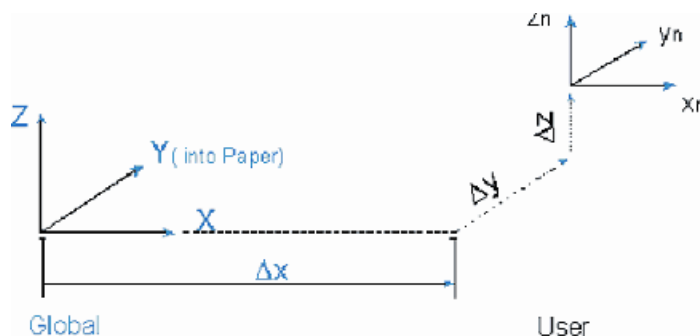
X Vector

A vector, defined in the global axis system, in the direction of the local x-axis.

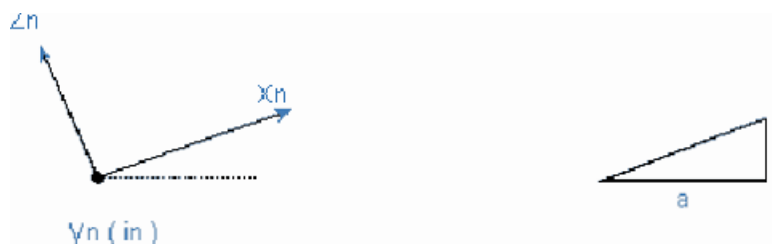
XY Vector

A vector, defined in the global axis system, which lies in the local xy-plane.

An example of the definition and application of user-defined axes is given below.



The user axis **origin** coordinates are thus defined as (x, y, z) above. The rotational orientation is defined in terms of a vector along the positive x_n -axis and a vector in the $x_n y_n$ plane. These two vectors are defined in the laterally shifted axes system shown above.



E.g. consider the simple case of rotation about the y_n axis alone to give an inclined plane for a support

The **x vector** could be expressed as $(a, 0, 1)$. This defines the position of the x_n axis.

The rotation of the user axis system about this axis must be defined by specifying any **xy vector** in the $x_n y_n$ plane, apart obviously from the x_n axis itself. In our case here, there is no rotation and the vector can be taken as $(0, 1, 0)$ i.e. the y_n axis.

See also [Program Fundamentals — Axes](#).

5.2 Nodes

Nodes are used to locate points in space. Elements are located in space by referring to nodes in their topology lists. Loads may be applied to nodes. The [Node Definition](#) dialog may be used to define node attributes of nodes.

Node Coordinates

How node coordinates are specified depends on whether the node is defined as a standard node or grid node:

- **Standard nodes** — Coordinates are defined with respect to the current grid plane, which

defaults to the global grid (i.e. global axis directions). Coordinates are stored in global axes, regardless of the current grid plane. For structures specified as “Plane”, only X and Z are required; for “Grid” only X and Y.

- **Grid nodes** — Coordinates are defined with respect to the grid plane that is specified for the node, and may be with defined respect to grid line intersections on the specified grid plane. Coordinates are stored as grid coordinates (i.e. relative to the grid plane origin or the grid line intersection on the grid plane, if so specified). In the Nodes table the coordinates of grid nodes are transformed to the same directions as standard nodes (i.e. with respect to the current grid plane) and are displayed in maroon; the coordinates may be edited in the table but are transformed to grid coordinates internally, to maintain the grid node status.

The 'Sculpt | Node Operations | Convert Nodes to Standard' and '... | Convert Nodes to Grid' commands may be used to convert the node status, maintaining their position in global space.

Constraint Axis

Constraint axes are used for specifying restraints, joints, constraint equations and applied displacements and results at nodes, in **non-global** directions.

The constraint axes default is global, but user-defined axes can be specified by typing the axis record number as specified in the Axes Table.

Restraint Direction

Any combination of translational and rotational restraint can be specified for each node. Restraints will be in global or user-specified axis directions, depending on the constraint axis.

Nodal Support Stiffnesses

Nodes can be assigned a stiffness in any of the translational or rotational directions provided there is not a restraint in that direction. Supports will be in global or user-specified axis directions, depending on the constraint axis.

See also:

[Program Fundamentals — Definition Axes](#)

[Program Fundamentals — Restraints and generalised restraints](#)

[Program Fundamentals — Conflicting constraints](#)

[Program Fundamentals — Spring Supports and Ground Springs](#)

5.3 Elements

An Element is an entity that is analysed. Its topology and position is defined using Nodes and Offsets. Its orientation is defined by the [element axis set](#) which depends on the topology of the element and the orientation node and angle. Its end fixity is defined partly by the behaviour of the Element Type and partly by the element Releases.

The types of element offered depend on the [structure type specified](#) and the [Advanced Feature Preference settings](#). Also the definition of the element behaviour varies with the solver chosen in the [Analysis Wizard](#).

Each element is assigned a property number and a group number. The property number is used to assign stiffnesses, material and other relevant properties to the element in the appropriate properties table. The group number can be used for data management.

In general Elements are treated as individual entities. However there are two element types that are treated as a linked set of entities during analysis. These are Spacer and Cable Elements. When Non-linear analysis and/or Form-Finding Advanced Features are set as Preferences, the GsRelax solver will link adjacent Spacer or Cable elements together to form a chain.

Any element can be made into a dummy element to result in it being ignored during analysis. The dummy attribute can be set in the [Element Wizard](#) or graphically using the Sculpt command to

[Modify Elements](#).

Elements should not be confused with [members](#), which are used for design.

More:

[Elements — Definition](#)

[Elements — Releases](#)

[Elements — Offsets](#)

5.3.1 Elements - Definition

Element Type

Sets the [element type](#). The element types are:

- [Beam](#)
- [Bar, Tie, Strut](#)
- [Spring, Grounded spring](#)
- [Mass](#)
- [Link](#)
- [Cable](#)
- [Spacer](#)
- [Quad 4, Quad 8, Triangle 3, Triangle 6](#)

Property

The property number defines which properties relate to a particular element. The property table to which this item points will depend on the element type. Each of the property tables starts with Property Number 1.

- **Beams, Bars, Ties and Struts** reference [Beam Sections](#)
- **Springs and Grounded springs** reference [Spring Properties](#)
- **Lumped Mass elements** reference [Mass Properties](#)
- **2D elements** reference [2D Element Properties](#)
- **Link elements** reference [Link Element Properties](#)
- **Cable elements** reference [Cable Properties](#)
- **Spacers** reference [Spacer Properties](#)

Note that each complete linked chain of spacer elements should have a unique spacer property number. This number should be given to each of the individual 2-noded spacer elements that make up the chain.

If Form-finding is set as an [Advanced Feature Preference](#), an element can have more than one property definition. These are all defined using the same Property number.

For example if a bar is given property number 3 in the Elements Table all analysis solvers will normally look for Property 3 in the Beam Sections property table for the required physical properties of the bar. However, if the analysis selected is soap film form-finding, Property 3 in the [1-d soap-film form finding properties table](#) will be used instead if information has been input. Similarly, if the analysis selected is force-density form-finding, Property 3 in the [1-d force-density form finding properties table](#) will be used instead if information has been input.

Beams, bars, ties, struts and 2D elements that require material definitions reference these through

the relevant property table.

Group

The user can split the structure into Groups of elements, which can be used to:

- attribute element loads to particular group(s).
- reduce the quantity of tabular output or clarify plots, when only nominated group(s) are shown.

Orientation

The orientation information is used to establish the [local axes](#) of an element. The definition of the element axes depends on the element type, the orientation node and the orientation angle.

Both orientation node and angle are used to define the orientation of beam, bar, tie, strut and spring elements for which the spring property axis is set to local. The orientation angle is used to define the orientation of the 2D elements, but this can be modified by the 2D element axis defined in the “[2D Element Properties](#)”. Mass, grounded spring, link, cable and spacer elements use neither orientation node or angle to define the orientation.

Note for fabric 2D elements the warp axis aligns with the element local x axis.

Topology

All elements are located in space through the nodes to which they attach. The number of nodes defining the topology depends on the element type, so mass and grounded spring elements have a single topology item; beams, bars, ties, struts and springs have two topology items; and for 2D elements the number of topology items, depending on the element type, will be 3, 4, 6 or 8.

For 1D elements the order in which the topology is defined will determine the local x direction. For link elements the first node in the topology list becomes the master node. In the Gss solver this implies a constraint on the other (slave) node.

For 2D elements the topology order is anti-clockwise when looking on the top surface of the element. All the corner nodes are identified first, followed by the mid-side nodes. The local axes of a 2D element will depend on the topology order unless overridden by the axis in the “2D Element Properties”.

See also:

[Program Fundamentals — Element types](#)

[Program Fundamentals — Element axes](#)

5.3.2 Elements - Releases

By default, Beam and 2D elements transfer moment to nodes and adjoining elements; releases may be applied to these elements to allow free rotational movement.

Releases may be set for each element axis direction, for each topology item.

Releases may be applied only to Beams and 2D elements.

A release on an offset element is located at the offset position, not at the node.

Note that releasing the x axis at both ends of a beam element will result in an element free to spin about its axis. Where a fully released beam is required it is usually preferable to use a Bar element.

5.3.3 Elements - Offsets

The flexible part of an element may be offset from the nodal positions by specifying an offset.

Offsets may be used where the flexible part of an element is less than the full length of the element or where the centreline of the element does not coincide with the line between the nodes at each end. For example where a beam is connected to a large column, it may be desirable to offset the flexible part of the beam by half the column width from the node. Offsets are treated as rigid arms

between the node and the flexible part of the element.

Offsets are specified in global axis directions.

Care should be exercised before using offsets to ensure that this feature is appropriate. Offsets cannot be used in GsRelax and attempts to use them will result in an error message.

See also [Program Fundamentals — Element offsets](#).

5.4 Members

A member is an entity that is used in design. A member's topology and position is defined using Nodes and Offsets. Its orientation is defined by the [member axis set](#) which depends on the member topology and the orientation node and angle. Its end fixity is defined partly by the behaviour of the member type and partly by the member releases.

Members cannot be used for the 2D element structure types.

Each member is assigned a property number, a design property number and a group number. The property number is used to assign the section and material information in the beam section table. The design property number is used to assign information specific to steel design in the steel beam design properties table. The group number can be used for data management.

Members should not be confused with [elements](#), which are used for analysis.

Geometry

The geometry of the member may be any of the following

- Linear

Member span between the two nodes.

- Arc with third point on arc

The arc spans from the first node to the second passing through the third.

- Arc with radius and point.

The third node determines the plane of the arc and the side of the line between the first and second nodes towards which the arc is 'pulled'.

- Explicit

Member will span between the end nodes through ordered list of internal nodes.

Member Type

The type identifies if these are beam, column or other members and if they are of concrete, steel or another material. For steel members the beam, column or other distinction is not important but for concrete members beam and columns are treated very differently. Concrete members which are neither beams nor columns are not considered in the RC member design.

The orientation of beams uses an orientation node and angle as described in the [Axes section](#). Releases allow some of the degrees of freedom to be treated as pinned. If all the degrees of freedom are to be considered as pins it is preferable to use a bar member. Offsets can be applied to beam members where the actual end of the member has to be offset from the node position.

Section Property

The property number defines which beam section relates to a particular member.

Design Property

The property number defines which steel beam design property or RC member property relates to a particular member, depending on the member type. Non steel and concrete members do not have a design property.

Restraint Property

The restraint property is applicable only to steel members and allows the restraint on the member to be specified which influences the effective length of the member.

Bar Arrangements

The bar arrangements are applicable only to concrete members. Beam members have three bar arrangements corresponding to the ends and the middle of the section. Column members are assumed to have the same bar arrangement along the whole length, so only a single bar arrangement is required. The definition of beam and column arrangements is different and take account of the typical bar layouts for the appropriate condition. When setting up members it is convenient to set the bar arrangement to "<design>" which indicates that no bar arrangement has been assigned to this member.

Group

The group number allows the user to split the structure into groups of members.

Orientation

The orientation information is used to establish the local axes of a member. The definition of the [member axes](#) is identical to that for a beam or bar element.

Topology

All members are located in space through the nodes to which they attach. The order in which the topology is defined will determine the local x direction.

Radius

Radius of arc, only used when the geometry is 'Arc with radius and point'.

Internal Nodes

List of nodes defining the intermediate topologies, only used when the geometry is 'Explicit'. The list of nodes need not be ordered and may be entered as a GSA list (e.g. '11 to 20 step 2') but will be stored as a simple list.

Restraint Nodes

List of nodes defining additional restraint points along the member. The list of nodes need not be ordered and may be entered as a GSA list (e.g. '11 to 20 step 2') but will be stored as a simple list. The restraint nodes must lie along the member for the member to be valid.

Nodal Releases

A member can be fully fixed to an adjacent one, allowing full transfer of moment; or pinned, where full rotational movement can freely occur. These conditions are set for each member axes at both ends.

Nodal Offsets

Where the actual member is less than the full length between nodes or where the centreline of the member does not coincide with the line between the nodes at each end it may be convenient to offset the member from the nodes. For example where a beam is connected to a large column, it may be better to offset the member by half the column width from the node.

See also:

[Program Fundamentals — Element types](#)

[Program Fundamentals — Element axes](#)

5.5 Grid Planes

A grid plane defines a surface.

Refer to [Program Fundamentals: — Grid Planes](#) for a general description of grid planes and their use.

The [Grid Plane Definition](#) dialog documentation describes the meaning of the data in this table in more detail.

Name

The name (or number) may be used to refer to the grid plane.

Type

A grid plane can be of one of two types. A general grid plane is typically used for loading. A storey grid plane is a special grid plane that is used to define a storey in the model. For a story grid plane the axis must be global, so that the grid plane is in the global XY plane.

Axis

The grid plane is parallel to the xy plane of this axis set.

Elevation

The elevation defines the height of the grid plane above the origin of the axis.

Element List & Grid Plane Tolerance

Where a grid plane is used to identify a set of elements, elements not in this element list will be excluded from the identified set. The grid plane tolerance is used to determine whether an element lies in the grid plane.

Span Type

The span information is associated with a grid plane for loading purposes. There are two aspects to the span – a type and information that relates to the selected type. The type can be one of:

- One-way spanning
- Two-way spanning

Span Direction

When the span type is one-way spanning a span direction is required. This is the angle of the span relative to the grid x axis.

Load Expansion & Simplified Tributary Areas

When the span type is two-way spanning there are a choice of expansion algorithms. In addition there is the option of assigning simplified tributary area. This option allows for a simpler load distribution on triangular and quadrilateral panels where a whole panel on the grid plane is loaded.

See also:

[Program Fundamentals — Grid Planes](#)

[Program Fundamentals — Grid Loads](#)

5.6 2D Polylines

2D polylines are used mainly in conjunction with grid loading to define lines or area that are loaded. Polylines are 2D entities that are interpreted with respect to the x-y plane of a grid plane.

Name

The name used to refer to the polyline.

Description

2D polylines are defined by a series of x-y coordinates. These are entered as a description of the form:

(a, b) (c, d) (e, f)

Note that they are defined by Cartesian x-y coordinates even when used with a cylindrical grid plane.

A 2D polyline does not need to be closed to represent a polygon: closure will be inferred from the context.

5.7 Material Properties

Materials can be either [standard](#), where the properties are known by GSA, or [user defined](#).

5.7.1 Standard materials

Default values for steel, concrete (long and short-term), aluminium and glass are available. These materials can be referenced by name in the properties tables.

No changes can be made to standard materials. To modify standard material properties copy a standard material to a [user-defined material](#), where the name can be changed and the properties adjusted.

The standard materials are:

Name	E (kN/m ²)	ν	ρ (t/m ³)	α (1/°C)
Steel	205×10^6	0.3	7.85	12×10^{-6}
Concrete (short term)	28×10^6	0.2	2.4	10×10^{-6}
Concrete (long term)	14×10^6	0.2	2.4	10×10^{-6}
Aluminium (typical)	70×10^6	0.34	2.71	2.31×10^{-6}
Glass (typical)	70.3×10^6	0.22	2.47	9.35×10^{-6}

5.7.2 User-Defined Materials

For each non-standard material property name mentioned in the [Beam Sections](#) or [2D Properties](#) modules. The [Material Wizard](#) should normally be used to define the material properties.

The detailed properties are defined here:

Name

The name is only used as a convenient way of identifying a material.

Material Model

The following material models can be defined

- Elastic isotropic
- Elastic orthotropic
- Elastic-plastic isotropic

- Fabric

Material Type

The material type is used for design purposes to determine if a material is steel, concrete, aluminum, glass, FRP, rebar and timber.

Young's modulus, Poisson's ratios, Shear Modulus

Elastic isotropic material - Young's modulus, Poisson ratio and shear modulus have the following relationship

$$G = \frac{E}{2(1+\nu)}$$

However the shear modulus may be specified independently to allow for the non-isotropic nature of material such as timber. Note that the shear modulus can only be edited using the Material Wizard.

Elastic orthotropic material - there are three Young's moduli, three Poisson's ratios and three shear moduli need to be defined and they are:

The material properties are

E_x - Young's modulus in x direction

E_y - Young's modulus in y direction

E_z - Young's modulus in z direction

ν_{xy} - Poisson's ratio, y direction strain generated by unit strain in x direction

ν_{yz} - Poisson's ratio, z direction strain generated by unit strain in y direction

ν_{zx} - Poisson's ratio, x direction strain generated by unit strain in z direction

The other three Poisson's ratios (ν_{yx} , ν_{zy} , ν_{xz}) can be calculated from the following relationships:

$$\frac{E_x}{E_y} = \frac{\nu_{xy}}{\nu_{yx}} \quad \frac{E_y}{E_z} = \frac{\nu_{yz}}{\nu_{zy}} \quad \frac{E_z}{E_x} = \frac{\nu_{zx}}{\nu_{xz}}$$

G_{xy} - Shear modulus in xy plane

G_{yz} - Shear modulus in yz plane

G_{zx} - Shear modulus in zx plane

Note: If an element uses an orthotropic material, the parameters used by the element are:

Beam elements

$$E_x$$

$$G_{xy}, G_{zx}$$

Bar, Tie & Strut elements

$$E_x$$

2D shell and plate elements

$$E_x, E_y$$

$$\nu_{xy}, \nu_{yx}^*$$

$$G_{xy}, G_{yz}, G_{zx}$$

2D plane stress elements

$$E_x, E_y$$

$$\nu_{xy}, \nu_{yx}^*$$

$$G_{xy}$$

2D plane strain elements

$$E_x, E_y, E_z$$

$$\nu_{xy}, \nu_{yx}^*, \nu_{yz}, \nu_{zy}^*, \nu_{zx}, \nu_{xz}^*$$

$$G_{xy}$$

2D axisymmetric elements

$$E_x, E_y, E_z$$

$$\nu_{yz}, \nu_{zy}^*, \nu_{zx}, \nu_{xz}^*$$

$$G_{xy}$$

* these values are calculated by GSA from the E values and the complementary Poisson's ratio.

Elastic-plastic isotropic material - the Young's modulus, Poisson ratios and shear modulus are the same as elastic isotropic material.

Yield stress is the tensile and compressive strength of the material, in non-linear analysis, the material stress will not exceed the yield strength in the analysis. **Yield stress is only used for beam, bar, tie and strut elements in non-linear static analysis. It is ignored in all other analyses in GSA.**

Ultimate stress, hardening Modulus, Hardening Parameter are not used in this version of GSA and they do not need to be defined. Even though they are defined, they will not be used.

Fabric material - Fabric material is only used for 2D membrane element, since the element has no thickness, the unit of Young's modulus and shear modulus of fabric material is force per unit length. Fabric material is orthotropic and the following properties need to be defined:

E_x - Young's modulus in x (warp) direction

E_y - Young's modulus in y (weft) direction

ν_{xy} - Poisson ratio, y direction strain generated by unit strain in x direction

The other Poisson's ratio (ν_{yx}) can be obtained from the following relationship:

$$\frac{E_x}{E_y} = \frac{\nu_{xy}}{\nu_{yx}}$$

Fabric material can be defined to take tensile force only (the default option) or take both tensile and compressive forces. If the check box "Allow compression" is checked, fabric will take compressive force, otherwise, it will take tensile force only.

Density

The mass density of the material used in dynamic analyses and to calculate gravity loads.

Temperature coefficient

The temperature coefficient of expansion which is used in conjunction with thermal loading. For orthotropic material, three coefficients need to be defined for x, y & z directions. There is no need to define temperature coefficient for fabric material.

Damping Ratio

The damping ratio is used during a dynamic analysis to calculate an estimate of the modal damping ratio.

5.8 Beam Sections

This defines the geometrical cross-section properties (e.g. area, inertia) ascribed to each Beam Property Number. Sections are most conveniently edited using the [Section Wizard](#). This allows the user to either select sections from a catalogue, to define standard section shapes or explicitly define the section properties.

See also [Program Fundamentals — Beam Sections and Section Database](#).

More:

[Use of section data in standard \(GSS\) analysis](#)

[Use of section data in GsRelax analysis](#)

[Section Properties](#)

5.8.1 Use of section data in standard (GSS) analysis

Bending action is about the principal axes unless the user has chosen for the stiffness calculation in the "uv axes" of the section term. For all symmetric sections principal and local axes are coincident.

The shear centre is assumed to be at the section centroid.

5.8.2 Use of section data in GsRelax analysis

The following points are only valid if the GsRelax solver is used.

If a section shape has been specified, the effect of plasticity on the interaction between axial force and biaxial bending is modelled. The net bending moment is limited to the plastic capacity of the section for the current axial force and direction of applied moment. The bending stiffness of a section is halved at first yield.

If explicitly defined section properties are specified, the interaction between axial and bending yield is ignored and the plastic section moduli used are approximations.

Bending action is in local element directions; the principal directions of the section are ignored.

Shear action is only modelled elastically and the shear centre is assumed to be at the section centroid.

5.8.3 Section Properties

There are three sets of values associated with each property. These are:

Section Properties

These are the actual section properties of the chosen section, so for example a 400×200mm

rectangular section would have an area of 0.08m^2 .

Modifiers

Each base section property can be modified so that a different value is used in the analysis. So for example the user may want to increase the axial stiffness of the $400 \times 200\text{mm}$ rectangular section defined above, so he can either factor the value or replace the value with a new value. So for example a factor of 2 on the area of the section would give an analysis value of 0.16m^2 .

Modifiers on the I and k values apply to either the xy or uv values depending on the axes selected for the stiffness calculation.

There are flags to say if modified properties should be used in the mass/weight calculations and in the stress calculations.

Analysis properties

The analysis properties are the base section properties modified as above. These are the values used in the stiffness calculation. Note that the modified area may or may not be used to determine the mass or gravity load, so in the above example the stiffness would be based on an area of 0.16m^2 but the mass and gravity loads would be based on either an area of 0.08m^2 or 0.16m^2 .

Name

The name is only used as a convenient way of identifying a particular section. It does not correspond to a specification of the cross-section.

Material

The material from which this section is made; typically one material will be used for many sections. Either a [standard material](#) or a [user-defined material](#) may be assigned.

Description

This gives a description of the section from which the section properties can be determined. The description is explained in the "[Beam Sections](#)" section.

Area

The area of the beam cross-section.

I_{yy} , I_{zz} or I_{uu} , I_{vv}

The second moment of area of the beam about the yy and zz axes or uu and v axes respectively.

J

The torsion constant for the section; note this is not the polar moment of inertia unless the section is circular.

s

Spacing between web in case of back to back sections.

K_y , K_z or K_u , K_v

The shear factor for shear in the y and z or u and v directions respectively. It is the proportion of the area of the beam which is effective in carrying shear (i.e. the $[\text{Shear Area}]/[\text{Total Area}]$). This can be defined as zero if the shear deformations are to be ignored.

Note: shear area factors are not supplied for all section shapes.

Type

The type of the section applies only to steel sections and is used in the design checks.

Cost

This allows the user to assign a cost per unit mass to the section. This is primarily intended for steel sections and is used in design to give an approximate cost to steel work.

5.9 Properties

[Spring Properties](#)

[Mass Properties](#)

[2D Element Properties](#)

[Link Properties](#)

[Cable Properties](#)

[Spacer Properties](#)

5.9.1 Spring Properties

Name

The name is only used as a convenient way of identifying a spring property.

Axis

The axis used to orientate a spring element. This may be global, local or a user defined axis. A local axis uses the element local axis as defined by the topology together with the orientation node and angle. If a global or user defined axis is specified the topology, orientation node and orientation angle are ignored.

Type

The type defines the spring is one of:

- Translational
- Rotational
- Matrix
- Tension only
- Compression only

x/xx: Non-linear curve ref

Non-linear spring curve reference for the spring property in x direction (or rotation about x axis). If this is greater than 0, the referred non-linear spring curve in non-linear spring curve table will be used, otherwise, the linear spring stiffness defined in the next column will be used.

x/xx: Stiffness

The stiffness of the spring in x direction (or rotation about x axis). If "non-linear curve ref" is greater than 0, the non-linear spring curve will be used, this is not required and ignored.

y/yy: Non-linear curve ref

Non-linear spring curve reference for the spring property in y direction (or rotation about y axis). If this is greater than 0, the referred non-linear spring curve in non-linear spring curve table will be used, otherwise, the linear spring stiffness defined in the next column will be used.

y/yy: Stiffness

The stiffness of the spring in y direction (or rotation about y axis). If "non-linear curve ref" is greater than 0, the non-linear spring curve will be used, this is not required and ignored.

z/zz: Non-linear curve ref

Non-linear spring curve reference for the spring property in z direction (or rotation about z axis). If this is greater than 0, the referred non-linear spring curve in non-linear spring curve table will be used, otherwise, the linear spring stiffness defined in the next column will be used.

z/zz: Stiffness

The stiffness of the spring in z direction (or rotation about z axis). If "non-linear curve ref" is greater than 0, the non-linear spring curve will be used, this is not required and ignored.

Matrix

The reference to the Spring Matrix that explicitly defines the spring stiffness.

Damping Ratio

The damping ratio is used during a dynamic analysis to calculate an estimate of the modal damping ratio.

See also:

[Program Fundamentals — Spring Supports and Ground Springs.](#)

[Spring Curves](#)

[Spring Matrices](#)

5.9.2 Non-linear Spring Curves

Name

The name is only used as a convenient way of identifying a spring curve.

Type

The type defines the spring is translational or rotational.

Description

A table defining the force/displacement or moment/rotation relationship for a non-linear spring.

See also:

[Spring Properties](#)

[Spring Matrices](#)

5.9.3 Spring Matrices

A spring matrix allows the user to define the stiffness terms explicitly for a grounded spring. The matrix option cannot be used with internal springs.

Name

The name is only used as a convenient way of identifying a spring matrix.

K(i,j)

The stiffness terms defining the upper triangle of the 6 x 6 stiffness matrix. The stiffness matrix must be positive definite.

See also:

[Spring Properties](#)

[Spring Curves](#)

[Spring Matrices](#)

5.9.4 Mass Properties

In a static analysis (linear or non-linear), only the Mass term is used when gravity loads are applied to the structure. In a dynamic analysis both mass and inertia are used.

Name

The name is only used as a convenient way of identifying a mass property.

Axis

In most cases a global axis will be used. A user defined axis is generally only of interest for orientation of the inertia properties so that only the principal inertias are required.

Mass, Inertia

This defines the mass and a (symmetric) inertia tensor. The terms that make up the inertia tensor are described more fully in the theory section.

Care should be taken in defining the off-diagonal terms to ensure that the resulting principal inertias are positive.

Mass Modifiers

The mass modifiers allow the mass to be varied in the different directions. These can be specified as a mass value or as proportion of the actual mass. This only applies to mass and in dynamic analysis. Typically this is used to add mass in some directions when a structure is in contact with a fluid.

5.9.5 2D Element Properties

This defines the type and properties ascribed to each 2D Element Property Number. Properties can be edited using the ["2D Property Wizard"](#).

Name

The name is only used as a convenient way of identifying a 2D element property.

Type

The types of 2D properties available will depend on the [Structure Type](#). For the 2D structure types (plane stress, plane strain and axisymmetric) only the type corresponding to the structure type is available. Likewise for non 2D structure types the plane strain and axisymmetric types are not available, although the plane stress option is. The types are:

- **Plane Stress**—in plane effects only (no out of plane stress)
- **Plane Strain**—in plane effects only (no out of plane strain)
- **Axisymmetric**—in plane effects only (the out of plane direction is the hoop direction)
- **Fabric**—in plane effects only (no thickness associated with fabrics)
- **Flat Plate**—out of plane effects only
- **Shell**—in-plane and out-of-plane effects
- **Curved Shell**—a general shell element which may be curved out of plane (this is not available in Gss or GsRelax at present)
- **Load Panel**—an element which is used only for the distribution of loads. It has no stiffness, so does not need material or thickness parameters.
- **Wall**—a super element which has both in-plane and out-of-plane stiffness. The wall element is meshed internally, so a single wall element can be used to model a wall between two floors.

Axis

Users may assign an axis to be used to define the orientation of 2D elements. If the axis is set to local the orientation of the element will be determined from the topology and the orientation angle. If another axis (global or user defined) is assigned here then that axis will be projected on to the element to determine its [orientation](#).

Material

The material from which the 2D elements are made. Either a standard material or a user defined material may be assigned. User defined materials can be Elastic isotropic, Elastic orthotropic, Elasto-plastic or Fabric. Fabric material can only be used for 2D fabric elements.

Thickness

This specifies the thickness of the elements. A uniform thickness is assumed over all the elements with the same property. No thickness is associated with fabric elements, and a unit thickness is assumed for plane strain and axisymmetric structure types.

This is used when determining the element in-plane stiffness and mass.

Thickness factors

There are three stiffness factors - bending, in-plane and mass/weight. The bending and in-plane thicknesses are used to determine the stiffness of the element may be specified separate from the actual thickness of the element. The mass/weight thickness is used to determine the mass of the element of the weight in self-weight calculations. These allow for modelling situations such as cracked concrete slabs (reduced bending thickness to allow for cracking) or hollow slabs where the stiffness of the slab in bending and in-plane will be less than that for a solid slab.

The thickness can be specified as a factor on the actual thickness (expressed as a percentage) or as an actual thickness value. For Plane Strain, Axisymmetric or Fabric property types this is not relevant so it is not enabled. The bending thickness is also used for transverse shear stiffness.

The stresses and forces in 2D elements are based on the modified thickness.

Support pattern & reference edge

The support pattern and reference edge are applicable only to load panel elements. The support pattern defined the number of edges to which the load is transferred and the reference edge sets the selection of the free/loaded edges.

Mass

Normally the gravity loads on a 2D element are calculated from the density and thickness. Fabric elements have no thickness but it may still be desirable to assign a mass, or where a slab is modelled which has a non-structural screed that adds mass. Both of these situations are provided for by the mass per unit area. This mass is in addition to the mass implied by the thickness and density.

5.9.6 Link Properties

Name

The name is only used as a convenient way of identifying a link property.

Type of Linkage

The type should be set to "All" to make a link completely rigid. There are however many cases in which it is useful to make a link rigid in a plane (for example in modelling floor slabs). The following planes can be specified:

Plane	Linked degrees of freedom		
yz	y	z	xx
zx	x	z	yy

xy	x	y	zz

The “pin” linkage types do not include the rotational degrees of freedom at the slave node in the rigid body displacements. Note that the “pin” condition applies only to the slave node - the rotational degrees of freedom at the master are always retained.

The above options can be used in both the Gss and GsRelax solvers. A further set of options are available for use only in the GsRelax solver:

- **Tension**—the link supports tension only
- **Compression**—the link supports compression only
- **Bar**—the link works in both tension and compression
- **Custom**—the link works like a joint linking the nodes in the specified coupled directions

See also:

[Program Fundamentals — Link elements and rigid constraints](#)

[Program Fundamentals — Conflicting constraints](#)

5.9.7 Cable Properties

Cable properties are most useful for GsRelax non-linear analysis, where all the [cable elements](#) with the same property form a cable that is allowed to slide. Cable elements can also be used in the Gss solver where they are treated in the same way as ties.

Name

The name is only used as a convenient way of identifying a cable property.

Stiffness

The stiffness of cables is specified directly rather than through the area and Young’s modulus, which are less clearly defined for cables. This stiffness is equivalent to AE (area $\times$ elastic modulus).

Mass

For gravity loading and dynamic analysis it is necessary to be able to associate a mass with cables. This is specified as a mass per unit length.

Temperature Coefficient

To allow for thermal effects in cables the temperature coefficient of expansion is required.

Damping Ratio

The damping ratio is used during a dynamic analysis to calculate an estimate of the modal damping ratio.

5.9.8 Spacer Properties

This defines the properties of a spacer chain made up of a series of spacer elements. Each individual spacer chain has its own unique spacer property number. The properties determine how the nodes along the spacer chain are controlled during the form-finding process.

Name

The name is only used as a convenient way of identifying a spacer property.

Spacer Axis

The axis number, which is only used when the spacer leg length type is “Projected Ratio”. (Global or user defined axis, local axis is not available.)

Type

The types are

- **Geodesic**—Geodesic spacers will reposition the nodes in 2 directions within the tangent plane. The tangent plane is defined by the normal vector of the node on a 2D fabric surface.
- **Free**—Free spacers will reposition the node in only one direction; the tangent direction is defined by the two legs of the spacer connected to the node.
- **Bar**—Bar spacers will reposition the nodes in all three directions. Bar spacers can be imagined as Tie elements except the spacer element length will be adjusted during form-finding analysis to meet the nodal spacing requirement.

Spacer Leg Length Type

The spacer leg length types can be defined as

- **Proportional**—Final spacer leg length will be proportional to the original leg length.
- **Ratio**—Final leg length of the spacer elements will form a geometric series using the ratio defined in the Spacer leg ratio column.
- **XY plane projected ratio**—Final projected leg length of the spacer on the xy plane of the Spacer Axis will form a geometric series with the ratio defined by the spacer leg ratio.
- **X axis projected ratio**—Final projected leg length of the spacer on the x Spacer Axis will form a geometric series with the ratio defined by the spacer leg ratio.
- **Projected proportional**—Final projected leg length of the spacer on the xy plane of the Spacer Axis will be proportional to the original projected leg length.

Spacer stiffness

If the spacer is imagined as a Tie element, the spacer stiffness is equal to EA (where E is the Young's modulus and A is section area). If zero is specified, a default value will be used in form-finding analysis.

Spacer leg ratio

This is only used when spacer leg length type is Ratio or Projected Ratio. For example,

$$\frac{l_{i+1}}{l_i} =$$

where r is the Spacer leg ratio ($i = 1, 2, 3 \dots$).

5.10 Form-Finding Properties

During form-finding analysis with the GsRelax solver the program will search for form finding properties with the property number assigned to the element. If none are found, the normal beam section or 2D element properties will be used instead.

More:

[Force Density 1D](#)

[Soap Film 1D](#)

[Force Density 2D](#)

[Soap Film 2D](#)

5.10.1 Force Density 1D

Name

The name is only used as a convenient way of identifying a form-finding property.

Force density

The final axial force (f) and the final length (l) of the element that uses this property will satisfy the following equation

$$f = l \times FD$$

where FD is the force density.

5.10.2 Soap Film 1D

Name

The name is only used as a convenient way of identifying a form-finding property.

Axis

If a local axis is used, the force in the Bar, Tie and Strut elements that use this property will be equal to that defined in Force column.

If a user defined axis number is input, the projected element force along the X axis direction of this axis set will be constant and equal to the force defined. This can be used to model pre-stressing using hanging weights supported by inclined ties. Here the force is constant in a projected direction, and the element force varies as the element local axis moves.

Force

The constant force (in the x-axis direction of the axis defined by column) of Bar, Tie and Strut elements that use this property in soap-film form-finding analysis. The Young's modulus of these elements will be set to zero in the form-finding analysis in order to achieve constant axial force.

5.10.3 Force Density 2D

Name

The name is only used as a convenient way of identifying a form-finding property.

Force density

The final axial stress (σ) and the final area (A) of the 2D element that uses this property will satisfy the following equation

$$\sigma = A \times FD$$

where FD is the force density.

5.10.4 Soap Film 2D

Name

The name is only used as a convenient way of identifying a form-finding property.

Stress in x

The constant stress in x direction of the 2D elements that use this property in soap-film form-finding analysis. The Young's modulus of these elements will be set to zero in the form-finding analysis in order to achieve constant in-plane force.

Stress in y

The constant stress in y direction of the 2D elements that use this property in soap-film form-finding analysis. The Young's modulus of these elements will be set to zero in the form-finding analysis in order to achieve constant in-plane force.

5.11 Design Properties

[Steel Beam Design Properties](#)

[Steel Restraint Properties](#)

[RC Beam Design Properties](#)

[RC Slab Design Properties](#)

RC Member Bar Layouts

5.11.1 Steel Beam Design Properties

These properties are associated with steel members (not elements) and are used in steel design checks. Together with [Steel Restraint Properties](#) and the [section property](#) (as used in normal 1D beam elements), it describes a steel beam's design attributes.

The [Steel Beam Design Property Definition](#) dialog documentation describes the meaning of the data in this table in more detail.

Name

The name is used only as a label for the beam design property.

Effective Lengths Override

This can be "Yes" or "No". If set to "No", then the effective lengths may be calculated automatically.

Effective Lengths

These are the effective lengths to use in member checks, only if the decision is made to use Effective Lengths Override. Otherwise they are greyed out.

Max Plastic:Elastic Ratio

This represents the maximum plastic bending capacity that can be used in relation to the elastic bending capacity ($p_y Z$).

Net:Gross Area Ratio

The net:gross area ratio is used to define the reduction in area for tension due to holes in the member. For welded structures this value should be 1.

Effective Net Area Factor (Beta)

The Effective net area factor (Beta) is used to compute the design tensile strength of the member. This represents the effective net area in tension for the member.

Temperature

The temperature is used in 'EN 1993-1-2:2005 Eurocode 3 Fire (UK)' design code only.

Exposure Type

The exposure type is used to define the adaptation factors defined 'EN 1993-1-2:2005 Eurocode 3 Fire (UK)' design code. The following exposure types can be defined.

- **All sides:** All four sides of the member are exposed to fire.

- **3 sides protected:** Protected member exposed on three sides.
- **3 Sides unprotected:** Unprotected member exposed on three sides.

5.11.2 Steel Restraint Properties

The steel restraint property table is a multi-page table. Each page represents a single restraint property. The different pages are selected using the tabs on the bottom of the window. The tabs (and consequently the restraint properties) can be renamed by either double clicking on them or by right clicking on them and selecting "rename" in the pop-up menu.

The [Steel Restraint Property Definition](#) dialog documentation describes the meaning of the data in this table in more detail.

Each restraint property page consists of four columns:

Node/Span

Accepts either "node" or "span", and determines whether the restraint being applied is distributed along a span within a member or concentrated at a node point.

Ref.

The number of the node or span, "end1", "end2" or "all".

Load Height

The loading position perpendicular to the member. A positive distance implies that the load is destabilizing, and induces destabilizing moments in the beam.

Restraint Description

The mnemonic for the design restraint at that span or node. If you type an invalid mnemonic, a dialog box summarizing the correct syntax is displayed.

5.11.3 RC Member Design Properties

These properties are associated with RC beams and are used in reinforced concrete design checks. These properties are only relevant for elements whose material is flagged as "concrete".

Note that RC Member Design Properties are currently only used in the options to Export to ADC and to Export to AdSec.

Name

The name is used only as a label for the beam design property.

Type

This specifies if this property applies to a beam or a column.

Concrete Grade

The strength of the concrete to be used in the design checks.

Reinforcement Grades

The strength of the reinforcement for the main reinforcement and for the links (or ties and stirrups).

Link Diameter and Aggregate

The link diameter and aggregate are specified as these will have an influence on the position of the bars in the section.

Cover

The minimum cover to the reinforcement at the top, bottom and sides. For columns read "back and front" for "top and bottom".

Bar Layout

The number of bar layouts depends on the property type, with three layouts for beam and a single layout for columns. These reference bar layouts defined in the bar layouts table.

5.11.4 RC Slab Design Properties

These properties are associated with 2d elements and are used in reinforced concrete design checks. These properties are only relevant for elements whose material is flagged as “concrete”.

Additional properties may be modified in the property wizard. For details refer to the “See also” links at the bottom of this page.

Name

The name is used only as a label for the slab design property.

Reinforcement Direction w.r.t. Local Axes

The angle, in degrees, between reinforcements and the x direction of the element local axes.

Reinforcement Axis-to-Surface Distance

The distance from the centre of the reinforcements to the surface of the concrete slab.

Override Analysis Thickness

Flag to indicate whether to use the **2D Element Property Thickness** or to override this with the **Slab Thickness**.

Slab Thickness

Enabled only when the **Override Analysis Thickness** flag is yes. In this circumstance, this thickness will be used as the slab thickness in the design.

Concrete Grade

The grade of concrete. The standard grades of concrete that are available for the specified RC Slab design code are available in this droplist. A “User Defined” option is also available. If this option is chosen, then direct specification of a non-standard concrete strength may be made in the [RC Slab Design Properties: Concrete](#) property page.

If the Wizard has been opened and the 'Modify code parameters' option of the concrete properties has been checked, then the concrete grade will display 'User Defined'. Changing the grade from this setting to a standard grade will cause the 'Modify code parameters' option to become unchecked. The concrete parameters will then be derived from the concrete strength, as defined by the design code. A warning will be issued before this action is taken.

Reinforcement Grade

The grade of reinforcement. The standard grades of reinforcement that are available for the specified RC Slab design code are available in this droplist. A “User Defined” option is also available. If this option is chosen, then direct specification of a non-standard reinforcement strength may be made in the [RC Slab Design Properties : Reinforcement](#) property page.

If the Wizard has been opened and the 'Modify code parameters' option of the reinforcement properties has been checked, then the reinforcement grade will display 'User Defined'. Changing the grade from this setting to a standard grade will cause the 'Modify code parameters' option to become unchecked. The reinforcement parameters will then be derived from the reinforcement strength, as defined by the design code. A warning will be issued before this action is taken.

See also:

[RC Slab Design Properties: Basic Properties](#)

[RC Slab Design Properties: Concrete](#)

[RC Slab Design Properties: Reinforcement](#)

5.11.5 RC Member Bar Layouts

RC Member Bar Layouts are referenced by RC Member Design Properties to define the reinforcement in a reinforced concrete section.

Note that RC Member Bar Layouts are currently only used in the options to Export to ADC and to Export to AdSec.

Name

The name is used only as a label for the bar layout.

Type

This defines the type of layout. The options are

beam - this is used for beams and is defined in terms of rows of top and bottom bars

circular column - this is used for columns and is defined in terms of outer (and inner rings) in a circular section

rectangular column - this assumes a rectangular columns and a particular number of bars and a specific arrangement is chosen

perimeter - this assumes a column of any shape and places bars around the perimeter of the section.

Number of bars, size and number of layers

This specifies the bars in a beam or in a circular column. While this is intended primarily for rectangular beams, it can also be used for other beam sections, where the reinforcement is assume to be in the 'web'.

Number of bars, size and arrangement

This specifies the bars in a rectangular column. The arrangements are most easily selected from the wizard. A number of bars and an arrangement determined the reinforcement in a column.

Definition, number of bars/spacing and size

The perimeter reinforcement can be specified either in terms of the minimum number of bars or the maximum spacing of the bars along with a bar size.

5.12 Constraints

[Generalised Restraints](#)

[Rigid Constraints](#)

[Joints](#)

[Constraint equations](#)

[Tied interfaces](#)

5.12.1 Generalised Restraints

Generalised Restraints allow a set of restraint conditions to be applied to a list of nodes, saving the effort of specifying restraints for individual nodes in the "Nodes | Restraints" table.

Name

The name is only used as a convenient way of identifying a restraint.

Stage List

This specifies a stage list using one any of the forms detailed in "[Lists](#)". If this is set to "all" the restraint applies to the whole model, irrespective of stage. Otherwise it applies only to the specified stage(s).

This field is only available when the analysis stage feature is switched on. Refer to [Step By Step Guide — Analysis Stages](#) for more details.

Node List

This specifies a node list using any of the forms detailed in "[Lists](#)". These include a list definition to select nodes on a specified plane.

Restraints

Specifies the directions that are to be restrained. Restraints are applied in the **constraint axes** of each individual node, as specified in the [Nodes](#) module.

Generalised Restraints act in addition to Nodal Restraints. The user should remain aware of this to avoid duplication and general confusion.

Generalised restraints are not affected by the Sculpt "Modify Nodes" operation — it adjusts the Nodal Restraints. A warning is given when the user is about to modify Nodal Restraints in Sculpt when Generalised Restraints are specified.

In Output Views, the Nodes table reports the nodal restraints and the generalised restraints separately. The Supports table reports the total restraints (= nodal + generalised).

See also:

[Program Fundamentals — Restraints and generalised restraints](#)

[Program Fundamentals — Conflicting constraints](#)

5.12.2 Rigid Constraints

Rigid constraints are sets of nodes may be restrained to move as a rigid body. They are similar to link elements except that the number of nodes is not limited.

Name

The name is only used as a convenient way of identifying a rigid constraint.

Stage List

This specifies a stage list using one any of the forms detailed in "[Lists](#)". If this is set to "all" the constraint applies to the whole model, irrespective of stage. Otherwise it applies only to the specified stage(s).

This field is only available when the analysis stage feature is switched on. Refer to [Step By Step Guide — Analysis Stages](#) for more details.

Node List

This specifies a node list using any one of the forms detailed in "[Lists](#)". These include a list definition to select nodes on a specified plane.

Master

The reference node to which the "slave" nodes are attached.

Linkage Type

This should be set to "All" to make the constraint completely rigid. There are however many cases in which it is useful to make a constraint rigid in a plane (for example in modelling floor slabs) or a plate. The following planes can be specified:

Linkage	Linked degrees of freedom
all	x, y, z, xx, yy, zz
xy plane	x, y, zz
yz plane	y, z, xx
zx plane	x, y, zz
xy plate	z, xx, yy
yz plate	x, yy, zz
zx plate	y, xx, zz
custom	any

The “pin” linkage types do not include the rotational degrees of freedom at the slave node in the rigid body displacements. Note that the “pin” condition applies only to the slave node - the rotational degrees of freedom at the master are always retained. The degrees of freedom linked are always in the constraint axis of the nodes and all nodes must be assigned the same constraint axis.

Coupled directions

For most constraint types the couple directions are fixed, but for custom links they must be specified explicitly. This can lead to behaviour that is not consistent with a 'rigid' condition.

See also:

[Program Fundamentals — Joints](#)

[Program Fundamentals — Link elements and rigid constraints](#)

5.12.3 Joints

Joints allow nodes to be tied in the specified translational or rotational directions.

Name

The name is only used as a convenient way of identifying a joint.

Stage List

This specifies a stage list using one any of the forms detailed in “[Lists](#)”. If this is set to “all” the joint applies to the whole model, irrespective of stage. Otherwise it applies only to the specified stage().

This field is only available when the analysis stage feature is switched on. Refer to [Step By Step Guide — Analysis Stages](#) for more details.

Master

The reference node to which the “slave” node is attached.

Slave

The node coupled to a “master” node so that it shares the coupled degrees of freedom.

Directions

Specifies the translational and rotational directions that are linked by this joint.

If the slave and master nodes are coincident there are no problems with joints, however if the slave and master nodes are not coincident there can be moment imbalance since forces are transferred directly to the master node and no account is taken of the moment implied by the force times offset. If this is to be modelled, rigid constraints should be used instead.

See also:

[Program Fundamentals — Joints](#)

5.12.4 Constraint Equations

Constraint equations allow a node, in a particular direction to be constrained relative to a set of other nodes.

Name

The name is only used as a convenient way of identifying a constraint equation.

Stage List

This specifies a stage list using one any of the forms detailed in "[Lists](#)". If this is set to "all" the constraint equation applies to the whole model, irrespective of stage. Otherwise it applies only to the specified stage(s).

This field is only available when the analysis stage feature is switched on. Refer to [Step By Step Guide — Analysis Stages](#) for more details.

Slave

The node coupled to a set of "master" nodes so that it shares the coupled degrees of freedom

Direction

This is the direction in which the slave node is constrained.

Equation

The equation is a set of factors, nodes and directions. The displacement of the "slave" node is the summation of the factored displacements at the masters in the specified directions.

Any linear equation can be set up and there is no requirement that the constraint equation preserves equilibrium in model, so care should be taken that no artificial constraint is introduced into the model.

See also:

[Program Fundamentals — Constraint Equations](#)

5.12.5 Tied Interfaces

[Tied interfaces](#) allow two sets of elements that do not have connectivity to be joined without the need for complex mesh refinement.

Name

The name is only used as a convenient way of identifying a tied interface.

Stage List

This specifies a stage list using one any of the forms detailed in "[Lists](#)". If this is set to "all" the tied interface applies to the whole model, irrespective of stage. Otherwise it applies only to the specified stage(s).

This field is only available when the analysis stage feature is switched on. Refer to [Step By Step Guide — Analysis Stages](#) for more details.

Slave type

The slave side of the tied interface can be defined either as nodes or elements. When elements are selected GSA will find the nodes on the slave side that need to be connected.

Slave list and master list

The nodes in the slave list are tied to elements in the master list to form a continuous entity. The

master retains all its degrees of freedom. The master elements must be 2D elements, but the slave elements can be of any element type (although link elements should be avoided as these will give rise to conflicting constraints). So for example a beam element can be connected to the face of a 2D elements.

The master list should contain the larger elements.

Type of Linkage

There are two option for the type of linkage if the slave type is "nodes":

- **Auto** – the tied interface behaves as tied if the slave type is element and rigid if the slave type is node.
- **Tied** – the tied interface connects the node on the slave side to the element faces on the master side using constraint equations.
- **Rigid** – the tied interface take the node on the slave side as a master and creates a rigid constraint making the element on the master side rigidly connected with the node.

The third of these options is intended for situations where the node represent a line entity such as a column that is to be attached to a slab. In this case, if the beam element cross section is a similar size to the 2D element on the other face the effect of the connection will be to make the 2D element behave in a manner close to being rigid. This option will not work where there are nodes to connect to the same element face or adjacent element faces.

The rigid linkage cannot be used when the slave type is element as this would constraint conflict.

Surface Tolerance

This is used to limit the nodes that are connected to only those along the edge. Nodes outside this tolerance are excluded from the tied interface.

See also:

[Program Fundamentals – Tied Interfaces](#)

5.13 Nodal Loading

[Node Loads](#)

[Applied Displacements](#)

[Settlements](#)

5.13.1 Node Loads

A force or moment is applied to a node or list of nodes.

Name

The name is only used as a convenient way of identifying a load. Optional.

Node list

This specifies a list of nodes to be loaded using any one of the forms detailed in "[Lists](#)". The list may be a single item.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Axis

The axis in which the load is applied (global by default).

Direction

The direction of the loading, in the specified axis direction.

Value

The magnitude of the load.

See also:

[Program Fundamentals — Node Loads](#)

5.13.2 Applied Displacements

An applied displacement forces a node to move a specified distance in a certain global or local direction in a particular load case. The node should not be restrained in the displacement direction. A force at this node is implied which depends on the stiffness of the structure and other loads applied in that load case. In other load cases the node is free to move.

If the node is to be restrained in all load cases use [settlements](#).

Name

The name is only used as a convenient way of identifying a load. Optional.

Node list

This specifies a list of nodes to be displaced using any one of the forms detailed in "[Lists](#)". The list may be a single item.

Load case

This is the load case in which the displacement applies. The load case gives a way of grouping load effects together.

Axis

The axis in which the displacement is applied (global by default).

Direction

The direction of the loading, in the specified axis direction.

Value

The magnitude of the displacement.

Warning — applied displacements should not be used in conjunction with settlement load in the same load case as it may results in wrong analysis results. If applied displacements have the same load case number with any other loads, the results may not be intuitive as the nodal displacements with applied displacement may not be the same as the applied displacement.

See also:

[Program Fundamentals — Applied Displacements](#)

5.13.3 Settlements

A settlement forces a node to move a specified distance in a specified global or local direction in a particular load case, where the node has been [restrained](#) in the relevant direction.

It has no effect on any other load cases. Note that no axis is specified for settlements are the settlement is in the node constraint axis.

Name

The name is only used as a convenient way of identifying a load. Optional.

Node list

This specifies a list of nodes to settle using any one of the forms detailed in "[Lists](#)". The list may be a single item.

Load case

This is the load case in which the settlement applies. The load cases give a way of grouping load effects together.

Direction

The settlement of the loading, in the node constraint axis direction.

Value

The magnitude of the settlement.

Warning — Settlements should not be used in conjunction with applied displacement loads in the same load case as it may results in wrong analysis results.

See also:

[Program Fundamentals — Use of Constraints — Settlements](#)

5.14 Beam Loading

Beam loading applies to beam, bar, tie and strut elements and to a more limited extent to cable elements.

More:

[Beam Loads](#)

[Pre-stress Loads](#)

[Distortion Loads](#)

[Thermal Loads](#)

5.14.1 Beam Loads

The Beam Loads are the most common type of beam loading. Refer to [Program Fundamentals — Beam Loading](#) for more information.

Name

The name is only used as a convenient way of identifying a load. Optional.

Beam list

This specifies a list of beam, bar, tie, strut or link elements to load using any one of the forms detailed in "[Lists](#)". The list may be a single item. Note that beam loads applied to cable elements are ignored.

Load case

The load case in which the load applies. The load case gives a way of grouping load effects together.

Type

The [type of load](#) on the element, which can be: Point, Uniform, Linearly varying, Patch or Tri-Linear.

Axis

The axis in which the load is applied.

Projected

Normally, by default, loads are applied along the length of the element. In some cases it may be more appropriate to project the loads on to the element, e.g. (vertical) snow loading on an inclined roof may be considered more easily as a projected load.

Direction

The direction of the loading.

Position 1

This is only relevant for point loads and patch loads. For a point load this is the location of the load measured from end 1 of the beam; for a patch load it is the start of the patch load measured from end 1 of the beam. Position 1 can be specified as a length or as a percentage of the length of the element, e.g. either 1.2 or 15%.

Load 1

Load magnitude. For a point load this is the load value; for a uniform load this is the load intensity; for a linear load this is the load intensity at end 1; for a patch load this is the load intensity at Position 1.

Position 2

This is only relevant for patch loads and is the end of the patch load measured from end 1 of the beam. Position 2 can be specified as a length or as a percentage of the length of the element, e.g. either 2.4 or 85%.

Where it is intended that a position should refer to end 2 of the beam 100% should be used instead of entering the actual length. Otherwise a rounding error in the element length calculation may result in an error being reported for the load.

Load 2

Load magnitude. For a linear load this is the load intensity at end 2 and for a patch load this is the load intensity at Position 2.

See also:

[Program Fundamentals — Beam Loading](#)

5.14.2 Pre-stress Loads

Pre-stress is a general description covering pre-stress loads, initial strains and initial lengths. In all cases the result is a pre-stress condition in the element.

Name

The name is only used as a convenient way of identifying a load. Optional.

Beam list

This specifies a list of beam, bar, tie, strut or cable elements to load using any one of the forms detailed in "[Lists](#)". The list may be a single item.

Load case

The load case in which the load applies. The load case gives a way of grouping load effects together.

Type

The type of load on the element, which can be Pre-stress Force, Initial Strain, Lack of Fit or Tendon Pre-stress.

Pre-stress Force & Tendon Pre-stress

The magnitude of the pre-stressing load. This applies only to the pre-stress force and tendon pre-stress load types. A positive force implies a tensile pre-stress force or a tensile tendon pre-stress.

Pre-stress Offsets

The offsets in local (y, z) directions of the tendon pre-stress. This allows for the introduction of pre-stressing moments in addition to pre-stressing force. This applies only to the tendon pre-stress load type and can only be applied to beam elements. Note that the pre-stress offset cannot be used in GsRelax.

Initial Strain

The initial strain in the element used to define a lack-of-fit. The value specified should be the factor of the element length by which the element is too long or too short. A positive value means the element is too long compared with the nodal distance and will induce compression in the element, a negative value will introduce tension in the element.

Lack of Fit

The length by which the element is too long or too short. A positive value means the element is too long compared with the nodal distance and will induce compression in the element, a negative value will introduce tension in the element.

See also:

[Program Fundamentals — Beam Loading](#)

5.14.3 Distortion Loads

Distortions are used to model the effect of introducing a cut in an element and applying a translational or rotational displacement across the cut.

The GsRelax solver can only deal with rotational distortions (excluding torsional distortions) and only when applied to the ends of elements.

Name

The name is only used as a convenient way of identifying a load. Optional.

Beam list

This specifies a list of beam, bar, tie or strut elements to which the distortion loads apply using any one of the forms detailed in "[Lists](#)". The list may be a single item. Note that distortion loads applied to cable elements are ignored.

Load case

The load case in which the distortion applies. The load case gives a way of grouping load effects together.

Direction

The direction of the distortion.

Position

The position along the element at which the member distortion is applied. Position can be specified as a length or as a percentage of the length of the element, e.g. either 1.2 or 15%.

Displacement

The magnitude of the distortion.

See also:

[Program Fundamentals — Beam Loading](#)

5.14.4 Thermal Loads

Either constant temperatures or temperature gradients can be applied to an element. In either case the temperature values specified are changes in temperature from ambient.

The temperature gradient is defined by applying temperatures at notional lower and upper surfaces. These need not coincide with the actual lower and upper surfaces of the element (which may not be known). The temperature gradient is then calculated from the top and bottom temperatures (T_{top} and T_{bot}) and the top and bottom locations (p_{top} and p_{bot}) as follows:

$$\frac{dT}{dn} = \frac{T_{top} - T_{bot}}{p_{top} - p_{bot}}$$

This temperature gradient applies to the section as a whole.

Name

The name is only used as a convenient way of identifying a load. Optional.

Beam list

This specifies a list of beam, bar, tie, strut or cable elements to which the thermal loads apply using any one of the forms detailed in [“Lists”](#). The list may be a single item.

Load case

The load case in which the thermal load applies. The load case gives a way of grouping load effects together.

Type

The type of thermal load on the element which can be; Uniform temperature, Temperature gradient in the local y direction, or Temperature gradient in the local z direction.

Uniform temperature

The temperature rise relative to ambient applied to the whole element.

Lower surface location, temperature

For temperature gradients the temperature profile is defined by the temperature at two points, referred to as the lower and upper surfaces. This defines the location and temperature rise that applies in the negative y or z directions relative to the neutral axis. A negative value denotes a position “below” the neutral axis.

Upper surface location, temperature

This defines the location and temperature rise that applies in the positive y or z directions relative to the neutral axis.

See also:

[Program Fundamentals — Beam Loading](#)

5.15 2D Element Loading

2D element loads apply to 2D element types, though not all the load types can be applied to each 2D element type. In general element loads should be used in preference to loads distributed to nodes.

More:

[Face Loads](#)

[Edge Loads](#)

[Pre-stress Loads](#)

[Thermal Loads](#)

5.15.1 Face Loads

Face Loads are loads applied to the face of 2D elements. These cannot be applied if the structure type is one of the 2D types (plane stress, plane strain or axisymmetric). In general face loads should be applied with care to ensure that the loading applied to the element is in a direction in which the element is stiff.

Name

The name is only used as a convenient way of identifying a load. Optional.

2D element list

This specifies a list of 2D elements to load using any one of the forms detailed in [“Lists”](#). The list may be a single item.

Load case

The load case in which the load applies. The load case gives a way of grouping load effects together.

Axis

The axis in which the load is applied.

The axis may be **Global**, **Initial Local**, **Deformed Local** or the number of an axis set specified in the Axes module.

Loads applied in **Initial Local** directions are applied in the element axis directions with the element axes being based on the undeformed state of the element.

Loads applied in **Deformed Local** directions are applied in the element axis directions with the element axes being based on the deformed state of the element. Loads applied in Deformed Local directions may only be specified for GsRelax analysis.

Projected

By default, loads are applied at the specified intensity over the element. In some cases it may be more appropriate to project the loads on to the element, e.g. (vertical) snow loading on an inclined roof may be considered more easily as a projected load.

Type

The type of load on the element, which can be **Uniform** or **Variable**.

Direction

The direction in which the load applies in the specified axis.

Pressure

The pressure to apply to the element. For a uniform face load a single pressure value is required and for the variable face load a value is required for each corner node on the element. The pressure over the face is interpolated from the corner values.

See also:

[Program Fundamentals — 2D Element Loads](#)

5.15.2 Edge Loads

Edge Loads are loads applied to the edge of 2D elements. In general edge loads should be applied with care to ensure that the loading applied to the element is in a direction in which the element is stiff. Edge loads may not be applied to Flat Plate or Fabric type 2D elements.

Name

The name is only used as a convenient way of identifying a load. Optional.

2D element list

This specifies a list of 2D elements to load using any one of the forms detailed in "[Lists](#)". The list may be a single item.

Load case

The load case in which the load applies. The load case gives a way of grouping load effects together.

Axis

The axis in which the load is applied.

The axis may be **Global**, **Initial Local**, **Deformed Local** or the number of an axis set specified in the Axes module.

Loads applied in **Initial Local** directions are applied in a direction that relates to the element edge. The x direction is the direction along the edge in the topology order direction. The y direction is the direction into the element, in the plane of the element and normal to the edge of the element at the point at which it is applied. The z direction is orthogonal to the x and y directions. The element is taken as being in its undeformed state.

Loads applied in **Deformed Local** directions are in directions as described for Initial Local directions except that the element is taken as being in its deformed state. Loads applied in Deformed Local directions may only be specified for GsRelax analysis.

Edge

The edge of the element that is loaded. The edges as defined by the topology items are

Edge	Topology items	
edge 1	1	2
edge 2	2	3
edge 3	3	4
edge 4	4	1

Direction

The direction in which the load applies in the specified axis.

Stress at ends

The stress to apply to the element. The stress along the edge is interpolated from the end values.

See also:

[Program Fundamentals — 2D Element Loads](#)

5.15.3 Pre-stress Loads

These are the 2D equivalent to pre-stress loads on beam elements. They can only be applied to elements that carry in-plane loads, or shell elements if the offset is specified.

Name

The name is only used as a convenient way of identifying a load. Optional.

2D element list

This specifies a list of 2D elements to load using any one of the forms detailed in "[Lists](#)". The list may be a single item.

Load case

The load case in which the load applies. The load case gives a way of grouping load effects together.

Type

The type of pre-stress on the element, which can be pre-stress force, initial strain or tendon pre-stress. Only the latter option induces forces and moments in the element.

Direction

The direction in which the load applies in element local axes. This can be x, y or Both.

Pre-stress Force

The pre-stress force (load per unit length) to apply to the element.

Initial strain

The initial strain in the element.

Tendon Pre-stress, Offset

The pre-stress force (load per unit length) and offset (in the local z direction) to apply to the element. When the offset is specified the element is loaded by an in-plane force and a moment.

See also:

[Program Fundamentals — 2D Element Loads](#)

5.15.4 Thermal Loads

Either constant temperatures or temperature gradients can be applied to an element. In either case the temperature values specified are changes in temperature from ambient.

Name

The name is only used as a convenient way of identifying a load. Optional.

2D element list

This specifies a list of 2D elements to which the thermal loads apply using any one of the forms detailed in "[Lists](#)". The list may be a single item.

Load case

The load case in which the thermal load applies. The load case gives a way of grouping load effects together.

Type

The type of thermal load on the element which can be; Uniform temperature, a temperature Gradient in the local z direction, or General temperatures specified at all corner nodes on top and bottom surfaces.

Uniform temperature

The temperature rise relative to the background temperature applied to the whole element.

Temperature at position

For temperature gradients the temperature profile is defined by two values, the temperature at the

top and bottom surfaces. For general temperatures the temperature is defined at the top and bottom surface for each corner of the element.

See also:

[Program Fundamentals — 2D Element Loads](#)

5.16 Grid Loading

Grid loads are loads that are not applied directly to a node or element, but are located in space or, more precisely, on a [grid plane](#) which is located in space. Internally, the load is applied to the structure by considering the location of the load relative to adjacent structural elements.

Grid loads are applied only to beam type elements.

More:

[Grid Point Loads](#)

[Grid Line Loads](#)

[Grid Area Loads](#)

5.16.1 Grid Point Loads

These are point loads located in space or, more precisely, on a grid plane which is located in space. The grid plane is also used to identify the elements that are considered during the grid load expansion. Refer to [Program Fundamentals: — Grid Planes](#) for details.

Grid point loads must be located so as, either, to coincide with an element or to be bounded by a closed polygon of elements. The [grid loading edge tolerance](#) is used to determine whether a load is applied directly on an element.

Name

The name is only used as a convenient way of identifying a load. Optional.

Grid plane

The plane in which the load is located and the plane that defines the elements that attract the load. Note that the load is not necessarily applied in grid plane axis directions (see [Axis](#), below).

Grid coordinates

The grid coordinates of the load point. Only the x and y coordinates (or r and theta if the grid plane is cylindrical) are defined, the z axis is taken as zero.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Axis

The axis in which the load is applied. Setting this to local specifies grid plane axis directions.

Direction

The direction of the loading, in the specified axis direction.

Value

The magnitude of the load.

See also:

[Program Fundamentals — Grid Loading](#)

5.16.2 Grid Line Loads

These are line loads located in space or, more precisely, on a grid plane which is located in space. The grid plane is also used to identify the elements that are considered during the grid load expansion. Refer to [Program Fundamentals: — Grid Planes](#) for details.

Grid line loads must be located so as, either, to coincide with an element or to be bounded by a closed polygon of elements. The [grid loading edge tolerance](#) is used to determine whether a load is applied directly on an element.

Name

The name is only used as a convenient way of identifying a load. Optional.

Grid plane

The plane in which the load is located and the plane that defines the elements that attract the load. Note that the load is not necessarily applied in grid plane axis directions (see [Axis](#), below).

Load type

Line loads apply along a polyline. This can either be a reference to a 2D polyline defined outside the grid loading or a polygon embedded in the load record.

2D polyline

If the area loads are applied to a reference to a polygon the polygon is defined by reference to a 2D polyline. If a polygon is embedded in the load record it should be of the form ' $(x_0, y_0), (x_1, y_1), \dots (x_n, y_n)$ (unit)', where the '(unit)' part is optional.

The polyline is defined in grid coordinates (i.e. x and y coordinates in the grid plane) and is assumed to close back to the starting point – it is not necessary to explicitly close the polyline. Note that the vertices are defined by Cartesian x-y coordinates even when the grid plane is cylindrical.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Axis

The axis in which the load is applied. Setting this to local specifies grid plane axis directions.

Direction

The direction of the loading, in the specified axis direction.

Projected

When the load and the structure are not orthogonal the load applied to the structure depends on whether the load is taken as applying along the length of the polyline or the length of the polyline projected on to the structure in the load direction.

Values

The magnitude of the load at the ends of the polyline. A linear variation between the ends is assumed.

See also:

[Program Fundamentals — Grid Loading](#)

5.16.3 Grid Area Loads

These are area loads located in space or, more precisely, on a grid plane which is located in space. The grid plane is also used to identify the elements that are considered during the grid load expansion. Refer to [Program Fundamentals: — Grid Planes](#) for details.

Grid area loads must be located so as to be bounded by a closed polygon of elements.

Name

The name is only used as a convenient way of identifying a load. Optional.

Grid plane

The plane in which the load is located and the plane that defines the elements that attract the load. Note that the load is not necessarily applied in grid plane axis directions (see [Axis](#), below).

Load type

Area loads can either apply to the whole plane, in which case the load boundary is the structure boundary, or to a polygon on the grid plane. If the loaded area is defined by a polygon this can either be a reference to a 2D polyline defined outside the grid loading or a polygon embedded in the load record.

2D polygon

If the area loads are applied to a reference to a polygon the polygon is defined by reference to a 2D polyline. If a polygon is embedded in the load record it should be of the form ' $(x_0, y_0), (x_1, y_1), \dots (x_n, y_n)$ (unit)', where the '(unit)' part is optional.

The polyline is defined in grid coordinates (i.e. x and y coordinates in the grid plane) and is assumed to close back to the starting point – it is not necessary to explicitly close the polyline. Note that the vertices are defined by Cartesian x-y coordinates even when the grid plane is cylindrical.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Axis

The axis in which the load is applied. Setting this to local specifies grid plane axis directions.

Direction

The direction of the loading, in the specified axis direction.

Projected

When the load and the structure are not orthogonal the load applied to the structure depends on whether the load is taken as applying to the area of the polygon or the area of the polygon projected on to the structure in the load direction.

Value

The magnitude of the load.

See also:

[Program Fundamentals — Grid Loading](#)

5.17 Gravity

Gravity loads are a special case of body loads i.e. loads that apply internally throughout the body rather than being applied externally to the body. The most common use of gravity loads is to model the self weight of the structure.

Gravity loads apply to the following element types:

- **Beams, bars, ties and struts**—through the area, density and length.
- **Masses**—through the element mass
- **Cables**—through the mass per unit length and mass
- **2D Elements**—through the thickness, density and area. plus the additional mass per unit area and area

Spring, grounded spring, link and spacer elements are considered massless, so they attract no gravity load.

Name

The name is only used as a convenient way of identifying a load. Optional.

Element list

This specifies a list of elements to which the gravity loads apply using any one of the forms detailed in "[Lists](#)". The list may be a single item, but is typically "all".

Load Case

The load case in which the gravity load applies. The load case gives a way of grouping load effects together.

Gravity factors

These are the factors that apply on "gravity". They will nearly always be used with gravity factors of (0, 0, -1) for normal static structures with the global Z-axis vertically upwards. However to model seismic effects it may be useful to apply a static horizontal loading of, for example, 0.25g in the X direction in which case the factors may be (0.25, 0, 0).

See also:

[Program Fundamentals — Gravity Loads](#)

5.18 Dynamic Response

These modules are required for dynamic response analysis. The basic requirement for response spectrum analysis is a response spectrum which can be either user defined or coded specific. In addition to the parameters which define the spectrum other related seismic parameters are defined here.

More:

[Response Spectra](#)

[Damping Table](#)

[Load Curve](#)

[Dynamic Load Factor](#)

5.18.1 Response Spectra

This where the basic type of response spectrum is defined.

Name

The name is used only as a label for the response spectrum.

Spectrum type

The user has a choice of the type of spectrum to use. This can be code defined to one of

- ASCE 7-05
- ASCE 7-10
- Eurocode 8 : 1994
- Eurocode 8 : 2003 (draft)
- Eurocode 8 : 2004
- FEMA 356
- GB 50011-2001
- GB 50011-2010
- IBC 2000
- IBC 2006 (ASCE 7-05)
- IBC 2009 (ASCE 7-05)
- IBC 2015 (ASCE 7-10)
- IS 1893 (Part 1) : 2002
- Ordinanze PCM 3274
- UBC 1994
- UBC 1997

or

- User defined

Depending on the spectrum type a different set of information is required to define the spectrum. The spectra are defined via the Response Spectrum Wizard. Details of the parameters are given in the [Data and Analysis Dialogs and Wizards section](#).

5.18.2 Damping Table

The damping table defines the total critical damping ratio for the given modes of modal dynamic analysis. It is used by harmonic analysis and footfall induced vibration analysis etc when the damping ratios for some modes need to be defined individually. **It is only necessary to define the damping ratio for those modes that have different damping ratio from the rest of the modes.** If damping ratios of the modes have not been defined in this table, they will be given on the analysis wizard of harmonic and footfall analysis.

Name

The name is used only as a label for this damping table definition.

Description

Damping table consists of a series pair of data that gives the mode number of modal dynamic analysis and its corresponding critical damping ratio. The description is a string in the form of "(a,b)

(c,d)(e.f)(g,h)..." to describe each pair of the mode and damping ratio. The description can be typed directly in the table or open the wizard to input the mode and damping ratio numerically.

5.18.3 Load Curve

The load curve defines the variations of the dynamic loads or the base accelerations against time. The load curve defines the load factor versus time and it is used by time history analysis. The load curve will be normalized before being used in time history analysis, i.e. the magnitude of the maximum load factor in the load curve will be unit and all other load factors on the curve are factorised accordingly.

Name

The name is used only as a label for this load curve definition.

Description

Load curve consists of a series pair of data that defines the load factor at the corresponding time. The description is a string in the form of "(a,b)(c,d)(e.f)(g,h)..." to describe each pair of the load factor and time. The description can be typed directly in the load curve table or open the wizard to input the load factor and time numerically.

5.18.4 Dynamic Load Factor

This defines the dynamic load factors (DLF) for each of the harmonic components of periodic loads against the periodic load frequencies. Dynamic load factors are used by periodic load analysis. The dynamic load factors are limited for maximum of 4 harmonic components of the periodic loads.

Name

The name is used only as a label for this dynamic load factor definition.

Description

Dynamic load factors consists of a series points of data that defines the periodic load frequency, DLFs for first, second, third and fourth harmonic components. The description is a string in the form of "(a,b,c,d,e)(f,g,h,i,j)(k,l,m,n,o)(p,q,r,s,t)..." to describe each point of the periodic load frequency and dynamic load factors. The description can be typed directly in the dynamic load factor table or open the wizard to input the frequencies and dynamic load factors numerically.

5.18.5 Frequency weighting curve

This defines Frequency weighting curve (FWC), FWC is used for calculating response factors of footfall analysis. The response factor is calculated using the product of Root-Mean-Square (RMS) acceleration response and the frequency weighting factor from FWC, then divided by 0.005 m/s².

Name

The name is used only as a label for this frequency weighting curve.

Description

Frequency weighting curve consists of a series points of data that defines the relationship between vibration frequency and weighting factor. The description is a string in the form of "(a,b) (c,d) (f,g) (h,i)..." The description can be typed directly in the frequency weighting curve table or open the wizard to input the frequencies and weighting factor.

5.19 Raft

These modules are required for soil-structure interaction analysis with GsRaft.

More:

[Pdisp Data](#)

[Raft Interaction](#)

[Pile Interaction](#)

5.19.1 Pdisp

The GsRaft solution makes use of an embedded version of [Pdisp](#). Click [Pdisp manual](#) to open the manual of Pdisp program.

5.19.2 Raft Interaction

The Raft interaction module defines the interface between the nodes on raft and the soil. Each raft interacting node in the structure is assumed to interact with a rectangular area of soil.

Node list

A list of nodes that interact with soil.

Dimension of interaction area

If Automatic is selected, the interaction areas will be calculated by GSA automatically. Automatic can only be used for raft defined by grillage. The rectangular interaction area defined by X and Y dimensions are associated global axis.

Interaction elevation

The elevation of the raft nodes interacts with soil. If Automatic is selected, the Z coordinate of the interaction will be used as the interaction elevation. The user defined interaction elevations are relative to elevation used in soil profile.

Minimum pressure (positive in compression)

The minimum pressure that can be exerted on the soil. If a negative minimum pressure is specified then this is the maximum tensile pressure that can be exerted on the soil.

Maximum pressure (positive in compression)

The maximum pressure that can be exerted on the soil. If the soil pressure is higher than this limit during raft analysis, the soil pressure will be constant and equal to this maximum pressure.

5.19.3 Pile Interaction

The Pile interaction module defines the interface between the nodes on pile and the soil.

Node list

A list of nodes that interact with soil.

Dimension of interaction area

If Automatic is selected, the interaction areas will be calculated by GSA automatically, otherwise, X, Y and Z represent the dimension of the beam section of the pile covered by the interaction nodes. The perimeter is the perimeter length of the pile section that are used by pile-soil vertical shear interaction.

Interaction elevation

The elevation of the pile nodes interacts with soil. If Automatic is selected, the Z coordinate of the interaction will be used as the interaction elevation. The user defined interaction elevations are relative to elevation used in soil profile.

5.20 Bridge

[Alignments](#)

[Paths](#)

[Vehicles](#)

5.20.1 Alignments

An Alignment can be used to define the position of a vehicle in the [Generate Vehicle dialog](#) in chainage and offset terms, and is needed to define Paths. Each alignment is a curve defined by radii of curvature at chainages.

Name

The name is used to identify the alignments where they are referred to. The name is displayed on the tab in the Alignment table. This can be adjusted by double clicking on the tab or right-clicking on the tab and selecting the modify option.

Chainage

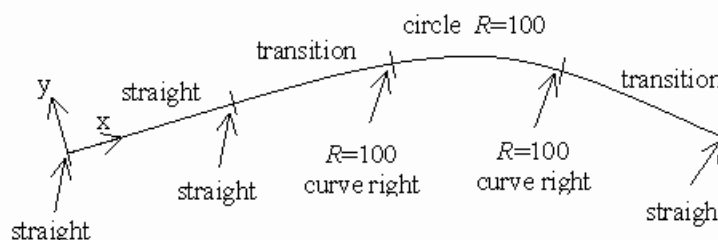
Distance along the Alignment from the origin. The origin of the alignment (chainage 0) is at the origin of the axis system of the Grid Plane, see below.

Curvature

Straight, Right Curve or Left Curve (looking in direction of increasing chainage).

Radius

The instantaneous radius curvature of the curve at that chainage. A circular curve is defined by two chainages with the same curvature, and a transition curve by two chainages with different radii (or a radius and a straight. Curvature varies linearly along the transition as in a standard highways transition). A typical alignment is illustrated below.



The formulae for the transition curves are taken from the "County Surveyors' Society" Tables.

If a change of curvature without a transition is needed then a very short transition length needs to be used.

Grid plane

The alignment has to be positioned in space relative to the structure. The grid plane provides this positioning with zero chainage at the origin of the grid plane and the alignment at this point aligned with the positive x axis of the grid plane.

5.20.2 Paths

Paths define a cross section through a bridge deck. Different path types are used for the different ways of using the tools available for bridge loading described in [Step-by-Step guide to Bridge Loading](#).

For highway loading, if a design code has been specified in the [Bridge Loading Analysis Specification](#) dialogue then the full width of a carriageway between kerbs is entered (the notional width for HK carriageways without raised kerbs), along with any footways, and the programme will automatically divide this into lanes and vehicle paths for analysis. In all other cases the user must specify Lanes individually, plus Vehicle paths if vehicles are not to be placed central in Lanes.

Paths must be defined before carrying out an [Influence Analysis](#), or using the [Bridge Loading table](#) to set up load cases, and can be used to position vehicles in the [Generate Vehicle dialog](#). Paths are used in conjunction with an Alignment to define the road or railway track geometry.

Path name

The name of the path will be used in the [Bridge Loading table](#) and the [Generate Static Vehicle dialog](#) to identify the path.

Path type

The deck section is split up into paths. The paths can be of the following types:

- **Lane**—for roads where automatic path generation is not available.
- **Footway**—for pedestrians.
- **Track**—for railways.
- **Vehicle**—to define a vehicle path straddling lanes, or to place vehicles off centre within a lane, where automatic path generation is not available.
- **Carriageway 1 way** – for roads where automatic lane generation is available, carrying 1 way traffic, minimum width 5m.
- **Carriageway 2 way** – for roads where automatic lane generation is available, carrying 2 way traffic, minimum width 5m.

No checks are made on whether Footway and Lane paths overlap, but a severe warning is given if Footway and Carriageways defined from a single alignment overlap. In this case (or if there are overlaps between carriageways defined from different alignments, which are not identified by the program) the user will need to review the Static Bridge Loads and delete any overlapping loads. Any gaps between lanes will be considered as central reserves and left unloaded.

Path Group

Group number is only used by the program in automatic path generation.

The deck can be considered as a series of groups of paths. Thus a two lane dual carriageway consisting of six paths could be considered as four groups — two carriageway groups (each containing two paths), and two footway groups. The central reserve is formed by the gap between the central lanes. Vehicle paths should be given the same group number as the lanes they straddle.

Alignment

The Alignment which this path follows. For highways this will normally be the same for all paths.

Offsets / Gauge

The values entered here define different things depending on the path type.

For footways and lanes: offsets from the Alignment to the left and right sides of the path. Vehicles will always be placed centrally in the path.

For tracks: offset to the centreline of the track, and track gauge (rail spacing, standard gauge — 1.435m).

For vehicle path: offset to centreline of vehicle.

A positive offset is to the right when looking along the alignment.

Left Rail Factor

When the path type is track it is possible to specify how the load will be split between the left and right rails by specifying the percentage of the load taken on the left rail.

5.20.3 Vehicles

User defined arrangements of wheel loads to form a vehicle can be defined here for use in the [Bridge Loading table](#) or the [Generate Static Vehicle dialog](#). Wheels are always positioned in pairs symmetrically about the central axis, but can have different intensities of load to allow for lurching, wind loading etc.

Name

The name is used to identify the vehicles where they are referred to. The name is displayed on the tab in the Vehicle table. This can be adjusted by double clicking on the tab or right-clicking on the tab and selecting the modify option.

Axle location

The position relative to vehicle origin, i.e. x coordinate.

Wheel offset

The distance of each wheel from centreline i.e. $\pm y$ coordinate.

Wheel loads

Magnitude of wheel load to the left and right respectively of the centreline looking in the direction of increasing x.

Width

This specifies the width of the vehicle used when checking vehicle paths.

5.21 Bridge Loading

[Bridge VUDL](#)

[Node Influence Effects](#)

[Beam Influence Effects](#)

[Path Loading](#)

[Static Bridge Loads](#)

[Moving Bridge Loads](#)

[Standard Bridge Loading](#)

5.21.1 Bridge VUDL

To optimize bridge loading based on an influence line analysis, any uniformly distributed load to be applied to a path needs to be specified either as a standard VUDL or a user VUDL.

The expansion of VUDL loads into grid point loads uses a coarse grid (typically 2–3m square); this will not give a good representation of the effects of a UDL if the mesh of the grillage is much finer than this. In such situations replacing the grid point loads with UDLs applied directly to the beam elements should be considered.

A VUDL is a uniformly distributed load that has an intensity (Force/Length/path) which can depend

on the loaded length L .

A number of [standard VUDLs](#) implement the requirements of specific design codes.

The user can specify additional VUDLs, this is most easily done using the Bridge VUDL Wizard.

Each segment of the VUDL curve between transition lengths has intensity per unit length per path given by the equation

$$\text{VUDL} = A \times \left(\frac{1}{L} \right)^n$$

Up to three segments of VUDL can be specified. The user needs to ensure the intensity matches either side of the transition length.

A constant UDL can be entered as one segment with the intensity as Factor(A) and Index(n) = 0, or obtained by factoring the UDL(1kN) in the Path Loading module.

5.21.2 Node Influence Effects

Node Influence Effects can be defined here for use in an influence analysis. The effect of a moving load will be found for any of the following situations:

- Displacement or rotation in the specified direction at an unrestrained node.
- Reaction in the specified direction at a restrained node.

The total effect is the sum of the factored individual effects. Note that only the Effect columns in the Static Bridge Loads table will be affected in this way. The total effect is used to find the critical load pattern but the magnitude of the loads (and so of the moments and forces produced by analysing them) is independent of the effect factors.

Name

The name from the first line with a given effect number is used to identify the effect. Names from subsequent lines with the same effect number are ignored.

Effect

The effect number will normally be the same as the line number, unless the user chooses to combine effects (for example to maximise the sum of the reactions from a series of supports on a single column).

Node

The node number where the effect is to be calculated.

Factor

The effect to be calculated can be factored, but will normally be 1. Non-unity factors can be used to combine effects (e.g. to maximise the weighted average of a number of reactions), but note, this is an advanced feature.

Type

- **Displacement**—nodal displacement will be optimized.
- **Force**—reaction will be optimized.

Axis & Direction

The axis and direction relative to that axis to be used for calculation.

5.21.3 Beam Influence Effects

Beam Influence effects can be defined here for use in an influence analysis. The effect of a moving load will be found for any of the following situations:

- Displacement, rotation, force or moment in the specified direction at the specified position on any single beam element.

The total effect is the sum of the factored individual effects. Note that only the Effect columns in the Static Bridge Loads table will be affected in this way. The total effect is used to find the critical load pattern but the magnitude of the loads (and so of the moments and forces produced by analysing them) is independent of the effect factors.

Name

The name from the first line with a given effect number is used to identify the effect. Names from subsequent lines with the same effect number are ignored.

Effect

The effect number will normally be the same as the line number, unless the user chooses to combine effects (for example to maximise the sum of the bending moments from a series of elements across the width of a deck).

Beam

The beam number where the effect is to be calculated.

Position

The point on the element where the effect is to be calculated.

Factor

The effect to be calculated can be factored, but will normally be 1. Non-unity factors can be used to combine effects (e.g. to maximise the average moment either side of a node, two effects with factors of 0.5 could be used), but note, this is an advanced feature.

Type

- **Displacement**—element displacement will be optimized.
- **Force**—element force or moment will be optimized.

Direction

The direction, in the beam local axis, to be used for calculation.

5.21.4 Path Loading

Path Loading can be used with the results of an influence line analysis to derive optimized bridge loading load cases.

Path

The path name to be loaded. A number of restrictions apply to combinations of path and load types: Vehicle paths can only be loaded with vehicles.

Footway paths can only be loaded with KEL or UDL loads (to place accidental wheel loads on footways use the [Generate Static Vehicle dialog](#).)

Lane and Track paths can accept any loading.

Load Type

A number of [standard path loadings](#) implement the requirements of specific design codes.

The user can define other loading which consists of one of the following:

- User Vehicle: any standard or user defined vehicle on its own.
- User Vehicle + VUDL: any standard or user defined vehicle combined with any standard or user defined VUDL. An exclusion length can be specified, see below.
- User VUDL + KEL: any standard or user defined VUDL combined with a single Knife Edge Load.
- EC1:LM3 Generic: any standard or user defined vehicle, combined with a UDL outside the exclusion length (see below) and a second vehicle placed within the UDL in accordance with EN 1991-2 (EC1). Note that UDL in GSBridge is defined as kN/m/lane, while UDL in EC1 is defined in kN/m².

The components of the Standard loading can be factored independently (to allow for 25 units of HB for example). Note that the VUDL factor specified for Standard loading is also applied to any KEL included in the Standard loading. For User VUDL + KEL the factor applies to the VUDL component only and any factored KEL must be entered directly.

Variable UDL

Details of the Variable UDLs (VUDLs) are defined in the VUDL module. The factor can be used to modify the intensity of standard VUDLs (to allow for lane factors).

Standard footway loading intensity is for a unit width footway and the VUDL factor is the footway width (modified by reduction factors for wide footways).

UDL(1kN) is a 1kN/m/lane UDL which can be factored to give any intensity of constant UDL.

KEL

A knife edge load (KEL) of the specified intensity can be placed at one point only on the lane.

The KEL will always be placed normal to the alignment. To place a KEL at a skew the Grid Point loads will need to be modified after optimisation and expansion.

Vehicle

Details of the Vehicles are defined in the Vehicle module and referred to here.

The factor can be used to modify the intensity of standard Vehicles (for example to reduce the standard 45 unit HB vehicle to a lower intensity).

The exclusion length is measured from the midpoint of the vehicle to the point where the VUDL starts on either side of the vehicle (so an exclusion length of 0 gives an uninterrupted VUDL).

5.21.5 Static Bridge Loads

This table allows the user to define static bridge loads. To apply these loads to the structure they have to be expanded to [Grid Point Loads](#) using the “Tools | Bridge Analysis | Expand Bridge Loading” menu command.

All bridge loading defined in this table is defined relative to paths specified in the Paths table. Single vehicle load cases can also be defined in the “[Generate Static Vehicle](#)” dialog from the “Tools | Bridge Analysis Generate Static Vehicle Load” menu command. Note that once the load cases have been generated changes to this table or the data to which it refers will **not** affect the generated load.

Name, Group

Each Group forms a single load case when expanded into grid point loads, with Name from first line in group used as the load case title.

Path

The path name to be loaded. A number of restrictions apply to combinations of path and load types:

Vehicle paths can only be loaded with vehicles.

Footway paths can only be loaded with KEL or UDL loads (to place accidental wheel loads on footways use the [Generate Static Vehicle dialog](#).)

Lane and Track paths can accept any loading.

Load Type

This can be one of the following:

- **Static vehicle**—standard vehicle from the list below, or a vehicle defined in the Vehicle table, placed centrally in the path with the datum (normally the centre of the rear axle) at the chainage specified. A single grid point load will be generated for each wheel of the vehicle. The magnitude of the wheel loads can be factored (for example the standard 45 unit HB vehicle can be factored to represent smaller vehicles). Note that vehicles can also be positioned in space and wheel loads represented by patches rather than point loads using the [Generate Static Vehicle dialog](#).
- **Static KEL**—a series of six grid point loads (or two point loads for tracks) will be generated to simulate a line load, equally spaced across the width of the path at the chainage specified. The intensity of the KEL is the total line load. The KEL will always be placed normal to the alignment. To place a KEL at a skew the Grid Point loads will need to be modified after expansion.
- **Static UDL**—a series of grid point loads will be generated to simulate a UDL over the full width of the path. Lines of six point loads (or two point loads for tracks) equally spaced across the path at regular intervals will be generated in a single load case (combined with any other static loads with the same bridge case number). The intensity of the UDL is defined per unit length of the path, **not** per unit area.

Start and End Chainage

Define first and last positions of the UDL on the path.

Vehicle

Selected from the standard and user defined [vehicles](#).

Load Intensity / Load factor

Multiplier on vehicle wheel loads.

Magnitude of KEL (positive).

Magnitude of UDL (positive) per unit length (**not** per unit area) along the alignment. Note that with large offsets on curved alignments the intensity per metre of path will not correspond exactly to this value.

Effect, Group Total Effect

Effect is given where the static bridge load has been derived by the [Optimiser](#). The effect is the magnitude (in SI units) of the effect being optimised resulting from the current line of static bridge loading, derived from the influence line. The group total effect is the sum of the effects for a single group of static bridge loads, and can be used by the user to decide how to combine the different groups to maximise a particular effect.

5.21.6 Moving Bridge Loads

This table allows the user to define loads which move along the structure. To apply these loads to the structure they have to be expanded using to [Grid Point Loads](#) using the “Tools | Bridge Analysis | Expand Bridge Loading” menu command.

All bridge loading defined in this table is defined relative to paths specified in the Paths table. Note that once the load cases have been generated changes to this table or the data to which it refers will **not** affect the generated load. Each entry forms a series of load cases when expanded into grid point loads.

Name

The name is used as the basis of the load case title for the expanded bridge loads. The individual load cases are identified as title.1, title.2, etc.

Path

The path name to be loaded. A number of restrictions apply to combinations of path and load types:

Vehicle paths can only be loaded with vehicles.

Footway paths can only be loaded with KEL or UDL loads (to place accidental wheel loads on footways use the [Generate Static Vehicle dialog](#).)

Lane and Track paths can accept any loading.

Load Type

This can be one of the following:

- **Moving vehicle**—vehicle as above positioned at successive intervals along the path. Each vehicle position appears in a separate load case.
- **Moving KEL**—a series of six grid point loads (or two point loads for tracks) will be generated to simulate a line load, equally spaced across the width of the path at the chainage specified, positioned at successive intervals along the path. The intensity of the KEL is the total line load. Each KEL appears in a separate load case. The KEL will always be placed normal to the alignment. To place a KEL at a skew the Grid Point loads will need to be modified after expansion.

Start and End Chainage

Define first and last positions of the KEL or of the origin of the vehicle (normally the back axle) on the path.

Interval

The space between successive KEL or vehicle positions.

Vehicle

Selected from the standard and user defined [vehicles](#).

Load Intensity / Load factor**Multiplier on vehicle wheel loads.**

Magnitude of KEL (positive).

5.21.7 Standard Bridge Loading

Standard Vehicles, VUDLs and Path Loadings have been set up to simplify calculations for different loading codes and types of loading.

- [UK or HK Highway Design Loading](#) (HA and HB)
- [UIC or UK Railway Loading](#) (UIC71, RU, RL, SW/0, SW/2)
- [US Highway Loading](#) (H20-44, HS20-44, and HL-93)
- [Eurocode Loading](#) (LM1 and LM3)
- [Australian Highway Loading](#) (M1600, S1600)
- [UK Assessment Loading](#) (SV)

More:

[UK or HK Highway Design Loading](#)

[UIC or UK Railway Loading](#)

[US Highway Loading](#)

[Eurocode Loading](#)

[Australian Highway Loading](#)

[UK Assessment Loading](#)

5.21.7.1 UK or HK Highway Design Loading

To BD37/01 (UK) or HK SDM (1997 edition)

Note this loading on constant width carriageways is best dealt with by specifying the design code in the [Bridge Loading Analysis Specification](#) dialogue when paths and loading will be generated automatically. For varying width carriageways or local analysis where the loaded area under individual wheels is important the following components are available:

Vehicles

- **HB45-6**—45 unit HB vehicle to BD37 with 6m between central axes (to generate other weights of HB vehicle use the load factor column).
- **HB45-11, HB45-16, HB45-21, HB45-26**—45 unit HB vehicles with larger spacing between central axes.

VUDL

- **HA (UK)**—full HA loading to BD37
- **HA (HK)**—full HA loading to HK SDM dated 1997
- **Foot (UK)**—full footway loading on a 1m strip to BD37
- **Foot (HK)**—full footway loading on a 1m strip to BD37 (using HAHK)

Path Loading

- **HA (UK)**—VUDL + 120kN KEL
- **HA (HK)**—VUDL + 120kN KEL
- **HB**—most onerous of the standard 45 unit vehicles HB45-6, HB45-11 ... HB45-26
- **Foot (UK)**—load on a 1m wide footway
- **Foot (HK)**—load on a 1m wide footway
- **HB + HA(UK)**—Vehicle+VUDL—exclusion length for HB vehicles is 25.2m from the end axes.
- **HB+HA(Hong Kong)**—Vehicle+VUDL—exclusion length for HB vehicles is 25.2m from the end axes.
- **Shadow HB + HA(UK)**—HA VUDL outside the exclusion length for an HB vehicle, to use when the vehicle straddles two lanes
- **Shadow HB + HA(HK)**—HA VUDL outside the exclusion length for an HB vehicle, to use when the vehicle straddles two lanes

These loads should normally be applied to Lane Paths between 2.5m and 3.65m wide. Where lanes are less than 3.5m wide then the HB vehicle does not fit entirely within a single lane. In this case vehicle paths can be used to place the vehicle eccentrically, with Shadow HB + HA loading on the lane paths.

5.21.7.2 UIC or UK Railway Loading

Vehicles

- **UIC 71/RU axles**—locomotive axles for RU loading to BD37. These are the same as the axles for UIC 71. Note that the UDL to be applied either side of these axles is not included.
- **SW/0**—works train, two 15m long UDLs 133kN/m separated by 5.3m gap.
- **SW/2**—works train, two 25m long UDLs 150kN/m separated by 7m gap.

Path loads

- **RL**—UDL [50kN/m] + KEL [200kN]
- **UIC71/RU**—80kN/m UDL, UIC71/RU axes, 3.2m exclusion length

These loads will normally be applied to Track Paths with the default standard gauge of 1.435m. Effects of lurching and centrifugal effects can be modelled using the Left Rail Factor when specifying the Track Path.

Note that dynamic factors are not included in the above loads and need to be applied separately.

5.21.7.3 US Highway Loading

Loading for two different US Loading codes has been included as detailed below.

The older AASHTO 17th Edition loading, H20-44 or HS20-44 consisted of either a UDL+KEL or a Truck load, and has been retained since it forms the basis for other load criteria. It can also be factored by 0.75 to give H15-44 or HS15-44 loading.

The newer AASHTO LRFD loading HL-93 uses the HS20-44 truck, but applies one or two trucks simultaneously with a UDL.

In both codes the influence effect being considered affects the magnitude of loading. Enveloping different load cases for different effects will therefore be overconservative.

To match the requirements of the codes, Lane paths for AASHTO loading should normally be 10 ft (3m) wide and placed within 12ft (3.6m) notional lanes (central in the 12ft lanes for internal lanes with HL-93 loading).

Because the AASHTO vehicles are not symmetrical the direction of travel needs to be considered. (...+ vehicle travels in direction of increasing chainage, ...- vehicle travels in direction of decreasing chainage). Path Loadings take the vehicle travelling in the most onerous direction.

Vehicles

- **H20-44+, H20-44-** — Truck load
- **AASHTO Tandem** – Two axle load
- **HS20-44/14+, HS20-44/22+, HS20-44/30+, HS20-44/14-, HS20-44/22-, HS20-44/30-** —Truck load. The suffixes /14 etc. refer to the variable axle spacing, v , in feet.
- **AASHTO Two Truck+, AASHTO Two Truck-** – 90% of two HS20-44/14 trucks with 15m between them

VUDL

- **Sidewalk (AASHTO)** – full sidewalk loading on a 1 foot strip (ie 85lb/ft for short lengths)

Path Loadings

- **Sidewalk (AASHTO)**
- **H20-44**—Vehicle H20-44 or UDL + 1 or 2 KEL (magnitude and number of KEL depends on type of effect – Moment / Shear / Hogging Moment)
- **HS20-44**—Worst Vehicle (HS20-44/14 or /22 or /30) or UDL + 1 or 2 KEL
- **HL-93**—UDL + tandem or 1 or 2 truck loads—zero exclusion length. (magnitude and number of trucks depends on type of effect – Normal / Hogging Moment / Internal Reaction)

Note that dynamic factors are not included in the above loads and need to be applied separately.

5.21.7.4 Eurocode Loading

Load Model 1 (LM1) from EN 1991-2 (EC1) and Load Model 3 (LM3) from the UK National Application document have been allowed for as follows (LM1 load types are controlled by national parameters which are selected when choosing the list of vehicles in the Bridges Specification wizard).

Note: EC1 UK loading on constant width carriageways is best dealt with by specifying the design code in the Bridge Loading Analysis Specification dialogue when paths and loading will be generated automatically. For varying width carriageways or local analysis where the loaded area under individual wheels is important the following components are available:

Vehicles

LM1

- **EU-TS300**—2 axles 300kN each
- **EU-TS200**—2 axles 200kN each
- **EU-TS100**—2 axles 100kN each

LM3 UK

- **SV80EC-1.2, 5 and 9** - SV80 vehicle including dynamic effect with alternative central axle spacing.
- **SV100EC-1.2, 5 and 9** - SV100 vehicle including dynamic effect with alternative central axle spacing.
- **SV196EC-1.2 To High Chainage, 5 and 9** - SV196 vehicle with tractor at high chainage end including dynamic effect with alternative central axle spacing.
- **SV196EC-1.2 To Low Chainage, 5 and 9** - SV196 vehicle with tractor at low chainage end including dynamic effect with alternative central axle spacing.
- **SOV250 to SOV600**, 28 vehicles, all with tractor at high chainage end, including dynamic effect, with 1.5, 5, 9, 14, 20, 30, 40m central axle spacing as specified in the UK NA.

Path Loads

These loads should normally be applied to 3m wide lanes, (except the EC1:Remainder path loadings which should be applied to narrower paths to fill in the remaining width of a carriageway). Note that for very short spans, less than about 7m, Load Model 2 (single axle loading) may govern. This is not included in standard bridge loading in GSA.

LM1

EC1 allows LM1 loading to be factored in the National Annex. Where national parameters have been included in GSA the country can be chosen in the Bridge Specification, otherwise EC1 Generic can be chosen and national parameters entered as factors in the Path Loading table.

- **EC1:Lane1** – LM1 Lane1 characteristic loading, UDL and TS300 vehicle, incorporating national factors
- **EC1:Lane2** – LM1 Lane2 characteristic loading, UDL and TS200 vehicle, incorporating national factors
- **EC1:Lane3** – LM1 Lane3 characteristic loading, UDL and TS100 vehicle, incorporating national factors
- **EC1:Lane4** – LM1 Lane4 characteristic loading, UDL incorporating national factors
- **EC1:Remainder** – LM1 Remaining area characteristic loading, UDL incorporating national factors
- **EC1:Lane1comb** – LM1 Lane1 combination loading, UDL and TS300 vehicle, incorporating national factors

- **EC1:Lane2comb** – LM1 Lane2 combination loading, UDL and TS200 vehicle, incorporating national factors
- **EC1:Lane3comb** – LM1 Lane3 combination loading, UDL and TS100 vehicle, incorporating national factors
- **EC1:Lane4comb** – LM1 Lane4 combination loading, UDL incorporating national factors
- **EC1:Rem comb** – LM1 Remaining area combination loading, UDL incorporating national factors
- **EC1:Lane1freq** – LM1 Lane1 frequent loading, UDL and TS300 vehicle, incorporating national factors
- **EC1:Lane2freq** – LM1 Lane2 frequent loading, UDL and TS200 vehicle, incorporating national factors
- **EC1:Lane3freq** – LM1 Lane3 frequent loading, UDL and TS100 vehicle, incorporating national factors
- **EC1:Lane4freq** – LM1 Lane4 frequent loading, UDL incorporating national factors
- **EC1:Rem free** – LM1 Remaining area frequent loading, UDL incorporating national factors
- **EC1:Foot on verge** – footway loading

LM3 UK

UK LM3 loading can have SV or SOV vehicles placed within a lane or straddling two lanes and these are combined with the frequent LM1 loading. Each loading type includes all vehicles up to the heaviest specified. Shadow loading applies the LM1 loading without the associated SV vehicle for cases where the vehicle displaces two lanes of LM1 loading:

EC1UK:SV80&Lane1freq

EC1UK:SV100&Lane1freq

EC1UK:SV196&Lane1freq

EC1UK:SOV250&Lane1freq

EC1UK:SOV350&Lane1freq

EC1UK:SOV450&Lane1freq

EC1UK:SOV600&Lane1freq

EC1UK:shadow SV80&Lane2freq

EC1UK:shadow SV100&Lane2freq

EC1UK:shadow SV196&Lane2freq

These loads should normally be applied to 3m wide lanes, (except the EU-LM1-Other path loading which can be applied to any path width).

Note that for very short spans, less than about 7m, Load Model 2 (single axle loading) may govern. Load Model 3 (abnormal load) is defined in National Annexes to the code. These models need to be considered separately.

5.21.7.5 Australian Highway Loading

Loading from AS5100:2 - 2004 has been allowed for as follows:

Vehicles

- **M1600 tri-axle** – 3 axle vehicle 162kN/axle
- **M1600-3.75**—6 axle vehicle 156kN/axle 3.75m centre spacing
- **M1600-5**—6 axle vehicle 156kN/axle 5m centre spacing

- **S1600-3.75**—6 axle vehicle 80kN/axle 3.75m centre spacing
- **S1600-5**—6 axle vehicle 80kN/axle 5m centre spacing
- **HLP320-1.8, HLP320-6, HLP320-10, HLP320-15** – 16 axle vehicle, centre spacing as shown
- **HLP400-1.8, HLP400-6, HLP400-10, HLP400-15** – 16 axle vehicle, centre spacing as shown

Path Loading

- **M1600** tri-axle UDL 8.1kN/m + M1600 tri-axle
- **M1600**—UDL 7.8kN/m + Two Vehicles, M1600-3.75 and M1600-5, at least 6.25m clear between them, placed in worst positions, zero exclusion length
- **S1600**—UDL 24kN/m + Two Vehicles, S1600-3.75 and S1600-5, at least 6.25m clear between them, placed in worst positions, zero exclusion length
- **SM1600** – most onerous of M1600 tri-axle, M1600 and S1600 loading.
- **AS 5100 Footway constant** 5kN/m² (note that lower intensities are allowed for larger loaded areas)

Note that impact factors **have** been included in the vehicles and path loadings as described above.

5.21.7.6 UK Assessment Loading

To BD86/04 (UK)

Note this loading on constant width carriageways is best dealt with by specifying the design code in the [Bridge Loading Analysis Specification](#) dialog when paths and loading will be generated automatically. For varying width carriageways or local analysis where the loaded area under individual wheels is important the following components are available:

Vehicles

NOTES: See BD86 for details of vehicles, Dynamic Factor has been included in axle load definition. Slow speed vehicles have lower axle loads but smaller exclusion lengths. Overload factor, OF, has been applied to a particular axle in these vehicles, this can be slightly unconservative since BD requires factor to be applied to the most onerous axle.

- **SV-TT-Norm-ToHighChain**—Normal speed travelling in direction of increasing chainage
- **SV-TT-Norm-ToLowChain**—Normal speed travelling in direction of decreasing chainage
- **SV-TT-Slow-ToHighChain**—Slow speed travelling in direction of increasing chainage
- **SV-TT-Slow-ToLowChain**—Slow speed travelling in direction of decreasing chainage
- **SV-Train-1.2ToHighChain**—Normal speed travelling in direction of increasing chainage, 1.2m between trailer axle groups
- **SV-Train-1.2ToLowChain**—Normal speed travelling in direction of decreasing chainage, 1.2m between trailer axle groups
- **SV-Train-5ToHighChain**—Normal speed travelling in direction of increasing chainage, 5m between trailer axle groups
- **SV-Train-5ToLowChain**—SV-Train-9ToHighChain, SV-Train-9ToLowChain
- **SV150-1.2Norm, SV150-1.2Slow, SV150-5Norm, SV150-5Slow, SV150-9Norm, SV150-9Slow**
- **SV100-1.2Norm, SV100-1.2Slow, SV100-5Norm, SV100-5Slow, SV100-9Norm, SV100-9Slow**
- **SV80-1.2Norm, SV80-1.2Slow, SV80-5Norm, SV80-5Slow, SV80-9Norm, SV80-9Slow**

VUDL

- **HA (UK)**—full HA loading to BD37
- **Foot (UK)**—full footway loading on a 1m strip to BD37

Path Loading

NOTES: These loadings select the most onerous of the vehicles listed, regardless of traffic direction. HA + loadings include full HA(UK) Loading and should have VUDL factored to suit BD21 lane factors. Because ULS factor on SV vehicles is 1.1 while on HA loading it is 1.3 an additional factor is needed to account for this (in full carriageway optimization the SV vehicle loading is factored by $1.1/1.3$). Exclusion lengths are measured from end axes and are 25m for normal vehicles and 5m for slow vehicles.

- **SV-TT, HA + SV-TT**—SV-TT-Norm-ToHighChain, SV-TT-Norm-ToLowChain, SV-TT-Slow-ToHighChain, SV-TT-Slow-ToLowChain
- **SV-Train, HA + SV-Train**—SV-Train-1.2ToHighChain, SV-Train-1.2ToLowChain, SV-Train-5ToHighChain, SV-Train-5ToLowChain, SV-Train-9ToHighChain, SV-Train-9ToLowChain.
- **SV-150, HA + SV-150**—SV150-1.2Norm, SV150-1.2Slow, SV150-5Norm, SV150-5Slow, SV150-9Norm, SV150-9Slow
- **SV100, HA + SV-100**—SV100-1.2Norm, SV100-1.2Slow, SV100-5Norm, SV100-5Slow, SV100-9Norm, SV100-9Slow
- **SV-80, HA + SV-80**—SV80-1.2Norm, SV80-1.2Slow, SV80-5Norm, SV80-5Slow, SV80-9Norm, SV80-9Slow

These loads should normally be applied to Lane Paths between 2.5m and 3.65m wide. Where lanes are less than 3.5m wide then the SV vehicles do not fit entirely within a single lane. In this case vehicle paths can be used to place the vehicle eccentrically, Shadow HB + HA loading placed on the lane paths will approximate to lane loading in this case, but will not give correct exclusion lengths.

5.22 Analysis Stages

The analysis stages modules are used to specify analysis stages. Refer to [Step By Step Guide — Analysis Stages](#) for more details.

These modules are only available when the analysis stages feature is switched on.

More:

[Stage Definition](#)

[Analysis Stage Properties](#)

5.22.1 Stage Definition

Stage definitions are used to define analysis stages. An analysis stage exists when it has been defined in this module.

Stage Name

The name is used to identify the stage.

Element List

The list of elements that are included in this stage. The [list syntax](#) is described in the [“Sets and Lists”](#) section of [“Program Fundamentals”](#).

Concrete Creep Coefficients

This option allows the concrete properties to be modified for a long term analysis by specifying a creep coefficient. The properties are modified as follows:

$$E \rightarrow \frac{E}{(1 + \phi)}, \quad G \rightarrow \frac{G}{(1 + \phi)}$$

5.22.2 Analysis Stage Properties

Stage properties are used to specify element properties for analysis stages where these are to differ from the whole model element properties.

Stage No.

The analysis stage for which the properties are to be adjusted.

Type

The type of property, whether Beam Section, 2D Element Property, etc.

Element Property No.

The property number as given in the Elements table. I.e. the whole model element property number.

Property Record

The record number in the relevant property table.

Example

Stage No.	Type	Element Property No.	Property Record
1	Beam Section	2	15

Elements that have Beam Section properties (i.e. beams, bars etc.) that have property number 2 in the Elements module (i.e. have Beam Section 2 properties in the whole model) are to have Beam Section 15 properties in analysis stage 1.

5.23 Cases and Tasks

The use of cases is covered in detail in the section on [case](#) in the [Program Fundamentals](#) chapter.

More:

[Load cases](#)

[Analysis Tasks](#)

[Combination cases](#)

5.23.1 Load Cases

When loads are applied to the model they are assigned to a load case. Load cases should be assigned to simplify the management of the data. Load cases, when referred to in lists, are prefixed with an "L" (e.g. L2). See [Lists](#) for more details.

The Load Cases are simply titles for these load cases defined elsewhere.

Name

The name for a load case where the loads are defined elsewhere.

Case Type

Defines the type of load case. The options offered are specific to the selected load design code, but include dead, live, snow, seismic, etc.

Source

A way of grouping load cases with the same action.

Category

Specified the category of loading. The options offered are specific to the selected load design code, but include domestic office, storage, roof, etc.

Direction

For loading such as seismic the direction in which it is applied

Include

Specified if adverse (unfavourable) and/or beneficial (favourable) effects should be included.

See also:

[Program Fundamentals — Load Cases](#)

5.23.2 Analysis Tasks and Cases

Any analysis carried out in GSA is controlled by a task. The task hold the information that controls the solver with details of the type of analysis, control parameters, etc.

Each analysis task has associated with in one or more analysis case(s). The analysis cases contain a description of the type of result that they represent, so for example in a static analysis these represent different loadings while in a modal analysis these represent mode shapes.

See also:

[Working with the Task View](#)

[Program Fundamentals — Cases and Tasks](#)

5.23.3 Combination Cases

It is often useful to combine results from more than one analysis case. This is achieved by defining combination cases, which combine a number of factored analysis cases (e.g. $1.4A1 + 0.8A3$).

Note combinations are only valid for linear solutions and should not be used to combine results from non-linear analyses.

Name

The name is only used as a convenient way of identifying a combination case.

Description

Defines the analysis cases and factors that constitute the combination case (e.g. $1.4A1 + 0.8A3$). An envelope of any number of cases can be specified as part of the combination, for example to view the worst case results for a number of cases. See "[Cases](#)" in "Program fundamentals" for details of the syntax for combination cases.

A1, A2 ...

In the case of a simple combination (e.g. $1.4A1 + 0.8A3$) these are the factors that apply to the particular analysis cases.

See also:

[Program Fundamentals — Combination cases](#)

[Program Fundamentals — Enveloping in GSA](#)

5.24 General Data

[Lists](#)

[Case Descriptions](#)

[Assemblies](#)

5.24.1 Lists

Lists (of nodes, elements, members and cases) are used, for example, when a particular load is to be applied to one or several elements. To define a series of items the list can either specify each individually or, if applicable, use a more concise syntax.

Lists may be pre-defined separately by name and stored in the Lists module and later referenced by that name in e.g. the beam loads module; alternatively they may be specified directly in that module itself.

Name

The name is used as a convenient way of referring to lists.

Type

A list can be defined for the following entities:

- **Node**
- **Element**
- **Member**
- **Case**

Definitions

The entities that make up the list. The list syntax is described in the "[Sets and Lists](#)" section of "[Program Fundamentals](#)".

See also:

[List Definitions](#)

[Lists Toolbar](#)

5.24.2 Case Descriptions

Case descriptions are used to store commonly used case descriptions. For example, the dead load may have contributions from several load cases. The combination of the loads can be entered as a case description and this can then be used when a case is required in an analysis task.

Name

The name is used as a convenient way of referring to the case description.

Description

This can be any valid case description, so will typically be of the form "1.6A1 + 1.4A2".

5.24.3 Assemblies

Assemblies are a way of grouping elements for further processing such as cut section forces. These give a convenient way of working with parts of the model. Unlike the analysis elements as assembly does not have a clearly define orientation and location of reference point so these must be defined explicitly.

Name

The name is used as a convenient way of referring to assemblies.

Element list

This defines the elements that make up the assembly.

Assembly definition

There are two ways in which assemblies can be defined: by node or by axis. When the node option is selected the definition uses topology (nodes) to define the assembly. When the axis option is selected the definition uses an axis and offsets to define the assembly.

Topology

This defines the nodes at the ends of the element.

Orientation node

The plane formed by the topology items and the orientation node is the xy plane of the assembly. The orientation node cannot be colinear with the topology nodes

Internal topology

This is a list of nodes that define the curve of the assembly. The order of the nodes is not important as the nodes are ordered along a vector between ends one and two.

Curve fit

For a curved element there is a choice of how the curve is fitted to the points. The default is to use Lagrange interpolation to fit a curve through all the intermediate points. The other option which is only valid if there is just a single internal topology item is for a circular arc from end one passing through the internal node to end two.

Axis

This defines the local axis system of the assembly, so for a vertical assembly the x axis should be aligned with the global z axis.

Extents

The extents of the assembly can be specified directly as end 1 and end 2 or GSA can determine this from the elements that define the assembly.

Transverse offset

This is used to locate the centre line of the assembly. 'Set from element list' sets the transverse offset to be at the average extents of the assembly in the y and z directions of the axis.

Point definition, points, spacing and offsets

This is how the assembly is defined internally (as used for assembly forces and moments). There are three options

- **Number of points** – a number of equally spaced points along the axis direction of the assembly
- **Spacing of points** – the assembly has points spaced along its length
- **By storey** – the assembly is divided by storey (the assembly local x axis must be aligned with global z for this to be valid).
- **Explicit** – the assembly has points at the explicitly defined positions. This can be a list of values or can be specified with keywords to and step (eg 2 to 8 step 2 12).

These points can be offset from the ends of the assembly. The sign of the offset is ignored and the offset always acts to reduce the active length of the assembly for calculation of intermediate points.

Part VI

6 Toolbars and Keyboard Accelerators

[Toolbars](#)

[Keyboard Accelerators](#)

6.1 Toolbars

Toolbars provide a short cut to the more commonly used commands. The List and Layer toolbars are attached to the views to which they relate. All other toolbars except the Assisted Input toolbar can be docked (attached to the application frame) or floating (free to be positioned by the user). The toolbars can be switched on and off as required from the “View | Toolbars” menu command.

More:

[Assisted Input](#)

[Standard](#)

[GSA](#)

[Data Options](#)

[Lists](#)

[Orientation](#)

[Cursor Mode](#)

[Graphic Display](#)

[Sculpt toolbar](#)

[Command](#)

[Display Favourites](#)

[Recorder](#)

6.1.1 Assisted Input



The assisted input toolbar is displayed along the bottom of the GSA window attached to the frame. This is intended to give quick access to the more commonly used features of GSA. More advanced users may wish to switch off this toolbar.

[Specification](#)—open the general specification dialog or create a new model if none open.

[Nodes](#)—open the Node Coordinates table

[Elements](#)—open the Element table

[Material](#)—open the Material table

[Sections](#)—open the Beam Sections table

[Gateway](#)—show or hide the Gateway

[Obj. Viewer](#)—show or hide the Object Viewer

[Data Defaults](#)—show or hide the Data Defaults dialog

[Graphics](#)—bring the top-most Graphic View into focus or, if none is open, open a new Graphic View

[Output](#)—open an new Output view

6.1.2 Standard



The standard toolbar handles the common Windows options:

[New](#)—create a new model

[Open](#)—open an existing file

[Close](#)—close the current model

[Save](#)—save the model to file

Cut—cut the data and place on clipboard

Copy—copy the data and place on the clipboard

Paste—paste the data from the clipboard into the model

Print—print the current view

Print Preview—preview the current view

[Undo](#)—undo the last edit

[Redo](#)—redo a previously undone edit

[Undo view](#)—undo the last change to the view settings

[Redo view](#)—redo a previously undone change to view settings

[View settings painter](#)—click once to grab the current view settings then click in a view of the same type to apply the grabbed settings, or click again to switch off

6.1.3 GSA



The GSA toolbar provides access to some of the main GSA options:

[Gateway](#)—show or hide the Gateway

[Object Viewer](#)—show or hide the Object Viewer

[Data Defaults](#)—show or hide the Data Defaults dialog

[New Graphic View](#)—opens a new Graphic View

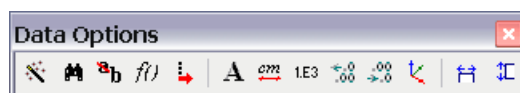
[New Output View](#)—opens a new Output View

[Analyse](#)—analyse the model

[Delete All Results](#)—delete all the results and optionally the analysis cases

Stop—stop the activity in the current view

6.1.4 Data Options



The Data Options toolbar gives access to various methods for modify the way data can be viewed or edited:

Wizard—access the data wizard

Find—search data for particular value

[Replace](#)—search and replace values

[Modify](#)—modify the values in tables

[Go To](#)—go to particular record in table

Font—change the font

Units—change the current units

Numeric Format—change the current numeric format

Increase Precision—increase the number of significant figures or decimal places

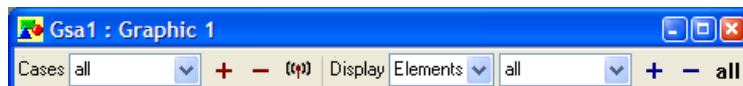
Decrease Precision—decrease the number of significant figures or decimal places

Axes—change the current axes

[Size Columns to Fit](#)—change displayed column widths

[Size Rows to Fit](#)—change displayed row heights

6.1.5 Lists



The Lists toolbar allows the user to select which cases are to be reported and how the content of the display is to be adjusted in a Graphic or Output View. The width of the toolbar automatically adjusts with the width of the window thereby maximising the size of the Case List and Display List.

Case List—the list of cases that are currently selected.

Next Case—select the next item in the case list pull-down.

Previous Case—select the previous item in the case list pull-down.

[Condense Cases](#)—when checked, the case list pull-down is automatically trimmed to offer only relevant cases for the current display.

Broadcast Case List—set the case list in other Graphic Views and Output Views to what is specified in the current view.

Display Option—the method by which the display content is to be adjusted. This can be by adjusting the list of entities, whether nodes, elements or members, or by selecting a saved view, in which case the settings of the saved view are applied to the current view. In Graphic Views the current grid, layer and stage can also be adjusted.

Display List—the entity list, saved view, layer or stage, depending on the Display Option setting.

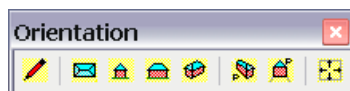
Next Display—select the next item in the current display list pull-down.

Previous Display—select the previous item in the current display list pull-down.

Reset To All Entities—the lists are reset to “all” entities. In Graphic Views any specified volumes are also cleared.

PinDisplay—when pinned selecting a grid plane of elements also sets that as the current grid plane, and vice versa.

6.1.6 Orientation



The orientation toolbar is associated with the aspects of a Graphic View related to orientation and scaling:

Graphic View Regenerate—regenerate and redraw the view without changing the scale, orientation, adornments or any other view settings

Plan—draw a plan view

X elevation—draw an x elevation

Y elevation—draw a y elevation

Isometric—draw an isometric (orthographic) view

Skew View—draw a skew (perspective) view

Ground View—draw a ground (perspective) view

Scale to Fit—scale the model to fit the view

6.1.7 Cursor Mode



The cursor mode toolbar is used to identify what the cursor should do in Graphic Views:

Rotate—dynamic viewing — rotation, zoom, pan, etc

Zoom—zoom in or out

Volume—select a volume — entities outside the volume are ignored

Omit From Display—select elements that are to be omitted from display — the display list is modified to exclude the selected elements

Select Grid Points—select grid points

Select Nodes—select nodes

Select Elements—select elements

Select Lines—select lines

Select Areas—select areas

Select Regions—select regions

Polyline—create a polyline

[Select For Annotation](#)—select entities for annotation

[Sculpt Toolbar](#)—display the “[Sculpt](#)” toolbar

6.1.8 Graphic Display



The graphic display toolbar governs the appearance of the graphical display:

[Shrink](#)—the elements are displayed “shrunk” from the nodes

[Section Display](#)—displays solid beam sections and solid 2D elements

[Labels and Display Methods](#)—accesses the “Labels and Display Methods” dialog

[Deformed Image](#)—draw the deformed image

[Animate](#)—animates the model

[Shade Surfaces](#)—shade surfaces according to current lighting conditions

[Contour Settings](#)—accesses the “Contour Settings” dialog

[Diagram Settings](#)—accesses the “Diagram Settings” dialog

[Reset Display](#)—resets the display, only scale and orientation are retained

[Rescale Data](#)—rescales the data for the current diagram

[Double Size of Diagram](#)—increases the size of the diagrams

[Halve Size of Diagram](#)—reduces the size of the diagrams

[Recorder Toolbar](#)—displays the “[Recorder](#)” toolbar

6.1.9 Sculpt toolbar



The sculpt toolbar is associated with the sculpting options:

[Draw Grid](#)—draw the construction grid points

[Snap to Grid Points](#)—snap on to the grid points

[Orient About Grid Axes](#)—perform orientation operations with respect to the current grid axes

[Add Nodes Sculpt Tool](#)—use the cursor to add nodes

[Modify Nodes Sculpt Tool](#)—use the cursor to modify nodes

[Drag Nodes Sculpt Tool](#)—use the cursor to drag nodes

[Add Elements Sculpt Tool](#)—use the cursor to add elements and nodes, if required

[Modify Elements Sculpt Tool](#)—use the cursor to add elements and nodes, if required

Move / Copy Selection—move the current selection or copy to create new nodes and elements

Split 1D Elements—split the selected 1D elements

Sculpt Command—the sculpt command to be carried out when Execute Sculpt Command is pressed

Display Favourites



Label Node Dots—display dots at the nodes

Label Restraints—label the restraints at nodes

Label Element Releases—display the element releases

U Displacement Diagrams—display diagram of “Displacements”

Fx Axial Force Diagrams—display diagram of “Axial Force” (F_x)

Fz Shear Force Diagrams—display diagram of “Shear Force” (F_z)

Myy Bending Moments Diagrams—display diagram of “Bending Moment” (M_{yy})

Copyright © Oasys 1985–2017

6.1.12 Recorder



The Recorder toolbar contains commands to record Graphic Views:

[Record](#)—Starts a recording session or, if a recording session has been paused, continues recording

[Pause](#)—Pauses recording

[Freeze](#)—Freezes the current image in the recording for a predetermined number of seconds

[Stop](#)—Terminates the current recording session prompting for the name which the AVI file is to be saved as

6.2 Keyboard Accelerators

Key	Action
Alt+0	Gateway
Alt+1	New Graphic View
Alt+2	New Output View
Alt+3	Object Viewer
Alt+4	Data Defaults
Ctrl+Num 1	Window bottom-left
Ctrl+Num 2	Window bottom
Ctrl+Num 3	Window bottom-right
Ctrl+Num 4	Window left
Ctrl+Num 5	Window middle (full)
Ctrl+Num 6	Window right
Ctrl+Num 7	Window top-left
Ctrl+Num 8	Window top
Ctrl+Num 9	Window top-right
A	Select for annotation
Ctrl+A	Select All
Ctrl+Alt+A	Reset to All Entities
BG	Colour elements by group
BM	Colour elements by material
BP	Colour elements by property
BS	Colour elements by initial stage
BT	Colour elements by type
Ctrl+B	Manual Backup
C	Case list
Ctrl+C	Copy
Ctrl+Alt+C	Size Columns to Fit
	Copy Selection

D	Display option: select how to adjust display content
Ctrl+Alt+D	Switch Layer
Ctrl+Alt+Shift+E	Add Elements Sculpt Tool
Ctrl+Alt+E	Modify Elements Sculpt Tool
Ctrl+F	Find
Ctrl+Alt+F	Display Excluded Entities Faint
G	Ground View
Ctrl+G	Go To
Ctrl+Alt+G	Grid Window
Ctrl+H	Replace
I	Isometric
K	Skew View
L	Polyline
Ctrl+Alt+Shift+L	Add Lines Sculpt Tool
Ctrl+Alt+L	Legend display
Ctrl+M	Modify
Ctrl+Alt+M	Move Selection
Ctrl+N	New
Ctrl+Alt+Shift+N	Add Nodes Sculpt Tool
Ctrl+Alt+N	Modify Nodes Sculpt Tool
Ctrl+Shift+N	Drag Nodes Sculpt Tool
Ctrl+O	Open
P	Plan
Alt+P	Reverse Plan
Ctrl+P	Print
Ctrl+Alt+P	Perspective
R	Rotate
Ctrl+Alt+R	Size Rows to Fit
S A	Select areas
S E	Select elements/members
S G	Select grid points
S L	Select lines
S N	Select nodes
S R	Select regions
Ctrl+S	Save
Ctrl+Shift+S	Save As
Ctrl+Alt+S	Snap to Grid Points
V	Volume
Ctrl+V	Paste
Ctrl+W	Wizard

X	X elevation
Alt+X	Reverse X elevation
Ctrl+X	Cut
Y	Y elevation
Alt+Y	Redo View
Ctrl+Y	Redo
Ctrl+Shift+Y	Reverse Y elevation
Z	Zoom
Ctrl+Z	Undo
Ctrl+Alt+Z	Undo View
F1	Help
Shift+F1	Context help
F2	Highlight
F5	Regenerate Graphic View
Ctrl+F7	Preferences
Alt+F7	Assisted Input
Esc	Quit
Tab	Next Cell
Return	Next Cell
Insert	Insert
Delete	Delete
Ctrl+Delete	Quick Delete
Home	Beginning of Table
Shift+Home	Reset Pan
Ctrl+Home	Move cell to Beginning; Scale to fit
Alt+Home	Rescale data
End	End of Table
Ctrl+End	Move cell to End
Page Up	Scroll up
Page Down	Scroll down
↑Up	Row Up; Up 7.5°
Alt+↑Up	Fast rotate up
Shift+↑Up	Pan Up
Ctrl+↑Up	Zoom In
←Lft	Column Left; Left 7.5°
Alt+←Lft	Fast rotate left
Shift+←Lft	Pan Left
Ctrl+←Lft	Eye Distance Out
→Rt	Column Right; Right 7.5°
Alt+→Rt	Fast rotate right

Shift+→Rt

Ctrl+→Rt

↓Dn

Alt+↓Dn

Shift+↓Dn

Ctrl+↓Dn

[Pan Right](#)[Eye Distance In](#)[Row Down; Down 7.5°](#)[Fast rotate down](#)[Pan Down](#)[Zoom Out](#)

Part VII

7 Data and Analysis Dialogs and Wizards

[Welcome to GSA](#)

[New Model and Data Generation Wizards](#)

[Data Definition Dialogs and Wizards](#)

[Analysis Dialogs and Wizards](#)

7.1 Welcome to GSA

The Welcome to GSA dialog is displayed on entry to GSA and is designed to assist the user to get started quickly. In addition to allowing the user to select what he wants to do the dialog displays a “Did you know...” tip.

Create a new structure from a template

This option takes the user to the New Model Wizard to allow simple structures to be defined with the minimum effort.

Work on your own

This allows the user to exit from the dialog and work on his own. This is the same as selecting “Cancel”.

Open an existing file

This activates the “File Open” dialog, allowing the user to select a file to open.

Load a backup file

This option is visible if program backup files are found. This will normally be the case if the program has previously crashed. This gives the opportunity to recover the model from a backup file.

Delete all backup files

This option is visible if program backup files are found. This will normally be the case if the program has previously crashed. If these files are not required the gives the opportunity to delete them.

Show this dialog on StartUp

The “Welcome to GSA” dialog can be disabled if required. This can also be switched from the [Miscellaneous page](#) of the Preferences dialog.

7.2 New Model and Data Generation Wizards

[New Model Wizard](#)

[Data Generation Wizard](#)

7.2.1 New Model Wizard

The titles dialog can be accessed from the “File | New” (Ctrl+N) menu command or from the new button on the [Standard toolbar](#). The new model wizard is intended to ensure that the basic settings for a model are correct before any data is generated.

Cancelling at any time will result in no new document.

More:

[New Model Wizard : Titles](#)

[New Model Wizard : Structure Type](#)

7.2.1.1 New Model Wizard : Titles

This allows the user to enter the job details. By default the job details of the previous job are used.

Job Number

This is the job number, which can be any alphanumeric string.

Initials

The initials of the user used on printed output.

Job Title

The title of the job.

Subtitle

The subtitle that this model relates to.

Calc Heading

Specific to this model.

Notes

Notes on the model can be stored here. This is a text field that can be appended to as required.

7.2.1.2 New Model Wizard : Structure Type

This page sets up the basic details of the structure. Cancelling at any time will result in no new document.

Structure Type

The structure type describes the type of structural model to be considered. The options are

- **Space**—3D structure type. This is the most general structure type, and all the other can be considered as subsets of this.
- **Plane**—planar structure type in the XZ plane. This is used for modelling for example a 2D frame.
- **Grid**— grid structure in the XY plane. This is used for modelling grillages.
- **Plane stress**—2D analysis option with plane stress conditions.
- **Plane strain**—2D analysis option with plane strain conditions.
- **Axisymmetric**—2D analysis option with axisymmetric conditions.

Units

Opens the [Units dialog](#) to allow the user to set a default set of units for the structure. The current units are reported on the page.

Generate data for the selected structure type

Closes the New Model Wizard and opens the [Data Generation Wizard](#) to allow the definition of simple structural forms.

7.2.2 Data Generation Wizard

The data generation wizard is only accessible from the New Model Wizard and is designed to provide a quick way of generating simple structural forms. The route through the data generation will depend on the structure type chosen.

More:

[Data Generation Wizard : Structure types](#)

[Data Generation Wizard : Portal / Orthogonal Frame](#)

[Data Generation Wizard : Grid](#)

[Data Generation Wizard : Pitched portal](#)

[Data Generation Wizard : Roof truss](#)

[Data Generation Wizard : Truss / Vierendeel / Pratt truss](#)

[Data Generation Wizard : 2D element orthogonal grid](#)

[Data Generation Wizard : 2D element grid](#)

[Data Generation Wizard : Generate](#)

7.2.2.1 Data Generation Wizard : Structure types

For space and plane structures the user is able to select the type of structure to generate. The full set of options is:

- [Portal](#)
 - [Grid](#)—a 2D grid structure available only in space structures.
 - [Orthogonal frame](#)—a standard 3D orthogonal frame available only in space structures.
-

- [Pitched portal](#)
 - [Roof truss](#)
-

- [Truss](#)
 - [Vierendeel](#)
 - [Pratt truss](#)
-

- [2D element orthogonal grid](#)
- [2D element grid](#)
- [2D element circular grid](#)

Selecting one of these will lead directly to the page to define this structure.

7.2.2.2 Data Generation Wizard : Portal / Orthogonal Frame

The information supplied on these pages provides a description of an portal or orthogonal frame, which is subsequently used to generate the structural model. The units of length are reported on the dialog, alongside a sketch of the structural form.

Generation

The user can choose to create analysis and/or design models with

- **Generate elements**
- **Generate members**

Bay width, Number of bays

The width of the structure is defined by the number of bays of equal bay width.

Storey height, Number of storeys

The height of the structure is defined by the number of storeys of equal storey height.

Column

Opens the [Section Wizard](#) to allow the column section to be selected. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Beam

Opens the Section Wizard to allow the beam section to be selected. This is marked as unspecified until a section is assigned, when the actual section will be displayed. On the Orthogonal Frame page, there are separate buttons for the beams spanning in the x direction and for the beams spanning in the y direction.

Include supports

Encastre supports are created at the base of the columns. By default no restraints are applied to the model.

Include self weight

A gravity load is set up in load case 1 applied to all elements. By default no loading is applied to the model.

Generate grid planes & storeys (Orthogonal Frame)

If this box is ticked, generate a storey [grid plane](#) at each level.

7.2.2.3 Data Generation Wizard : Grid

The information supplied on this page provides a description of a grid, which is subsequently used to generate the structural model. The units of length are reported on the dialog, alongside a sketch of the structural form.

Generation

The user can choose to create analysis and/or design models with

- **Generate elements**
- **Generate members**

Bay width, Number of bays, Skew angle

The plan layout is defined by the number of bays of equal bay width in each of the two directions, with the option of including a skew of the elements or members in the y direction.

Beam — x

Opens the [Section Wizard](#) to allow the section to be selected for the beams spanning in the x direction. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Beam — y

Opens the Section Wizard to allow the section to be selected for the beams spanning in the y direction. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Include self weight

A gravity load is set up in load case 1. By default no loading is applied to the model.

Generate grid planes

If this box is ticked, generate a [grid plane](#) at each level.

7.2.2.4 Data Generation Wizard : Pitched portal

The information supplied on this page provides a description of a pitched portal frame, which is subsequently used to generate the structural model. The units of length are reported on the dialog, alongside a sketch of the structural form.

Generation

The user can choose to create analysis and/or design models with

- **Generate elements**
- **Generate members**

Bay width, Number of bays

The width of the structure is defined by the number of bays of equal bay width.

Eaves height

Specify the height from ground to the eaves.

Eaves-ridge height

Specify the height from the eaves to the ridge.

Include supports

Encastre supports are created at the base of the columns. By default no restraints are applied to the model.

Column

Opens the [Section Wizard](#) to allow the column section to be selected. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Beam

Opens the Section Wizard to allow the beam section to be selected. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Include self weight

A gravity load is set up in load case 1. By default no loading is applied to the model.

7.2.2.5 Data Generation Wizard : Roof truss

The information supplied on this page provides a description of a roof truss, which is subsequently used to generate the structural model. The units of length are reported on the dialog, alongside a sketch of the structural form.

Generation

The user can choose to create analysis and/or design models with

- **Generate elements**
- **Generate members**

Width

Specify the width of the truss between supports.

Depth

Specify the depth of the truss at the ridge.

Use bars for bracing

Bar elements should be used for bracing. By default beam elements will be used.

Rafter

Opens the [Section Wizard](#) to allow the rafter section to be selected. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Tie

Opens the Section Wizard to allow the section to be selected for the tie. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Bracing

Opens the Section Wizard to allow the section to be selected for the bracing members. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Include supports

Pin supports are created at the ends of the span. By default no restraints are applied to the model.

Include self weight

A gravity load is set up in load case 1. By default no loading is applied to the model.

7.2.2.6 Data Generation Wizard : Truss / Vierendeel / Pratt truss

The information supplied on these pages provides a description of a truss, a Vierendeel truss, or a Pratt truss, which is subsequently used to generate the structural model. The units of length are reported on the dialog, alongside a sketch of the structural form.

Generation

The user can choose to create analysis and/or design models with

- **Generate elements**
- **Generate members**

Width, Number of bays

Specify the width of the truss and the number of bays between supports.

Depth

The distance between the neutral axes of the top and bottom chords.

End style (Truss)

The end style of the truss can be selected from

- **Cornice**—overhanging.
- **Facia**—vertical.
- **Mansard**—sloping.

Use bars for bracing (Truss, Pratt truss)

Bar elements should be used for bracing. By default beam elements will be used.

Top Chord

Opens the [Section Wizard](#) to allow the top chord section to be selected. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Bottom Chord

Opens the Section Wizard to allow the bottom chord section to be selected. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Diagonals (Truss, Pratt truss)

Opens the Section Wizard to allow the section to be selected for the bracing elements. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Verticals

Opens the Section Wizard to allow the section to be selected for the vertical members at the end of the truss. On the Truss page, this is only relevant if the facia option is selected, otherwise this item is disabled. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Include supports

Pin supports are created at the ends of the span. By default no restraints are applied to the model.

Include self weight

A gravity load is set up in load case 1 applied to all elements. By default no loading is applied to the model.

7.2.2.7 Data Generation Wizard : 2D element orthogonal grid

The information supplied on this page provides a description of a regular grid of 2D elements, which is subsequently used to generate the structural model. The units of length are reported on the dialog, alongside a sketch of the structural form.

The spacer and edge element options are only enabled when the “Form-finding analysis” option on the “Advanced Features” page of the [Preferences](#) dialog is enabled.

Width, Number of elements

Specify the overall width of the grid in the two directions. This width is subdivided by the number of elements in the two directions.

2D Property

Opens the [2D Element Property dialog](#) to allow the element properties to be defined. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Include spacers for form finding

This generates spacer elements along the 2D element edges for use in form finding.

Internal

If spacers are to be generated this opens the spacer properties wizard to allow properties to be defined for internal spacer elements (those between 2D elements).

Edge

If spacers are to be generated this opens the spacer properties wizard to allow properties to be defined for spacer elements around the edge of the 2D elements.

Generate edge elements

This generates 1D elements around the edge of the grid of the type selected.

Section

If edge elements are to be generated this allows the properties for the edge elements to be specified.

7.2.2.8 Data Generation Wizard : 2D element grid

The information supplied on this page provides a description of a regular warped grid of 2D elements, which is subsequently used to generate the structural model. The units of length are reported on the dialog, alongside a sketch of the structural form.

Coordinates

Specify the coordinates of the four corners of the grid. The program will generate a mesh with in the boundary defined by straight lines between the corners, but the resulting mesh may be warped since the corner nodes are not required to lie in a plane.

Mesh Density

This width is subdivided by the number of elements in the two directions.

2D Property

Opens the [2D Element Property dialog](#) to allow the element properties to be defined. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Include spacers for form finding

This generates spacer elements along the 2D element edges for use in form finding.

Internal

If spacers are to be generated this opens the spacer properties wizard to allow properties to be defined for internal spacer elements (those between 2D elements).

Edge

If spacers are to be generated this opens the spacer properties wizard to allow properties to be defined for spacer elements around the edge of the 2D elements.

Generate edge elements

This generates 1D elements around the edge of the grid of the type selected.

Section

If edge elements are to be generated this allows the properties for the edge elements to be specified.

7.2.2.9 Data Generation Wizard : 2D element circular grid

The information supplied on this page provides a description of a regular warped grid of 2D elements, which is subsequently used to generate the structural model. The units of length are reported on the dialog, alongside a sketch of the structural form.

Origin

Specify the global X, Y and Z coordinates of the grid origin.

Radius

Specify the bottom and top radiuses of a frustum, or the outer and inner ones of a hollow circular grid. Leave the top radius at 0 to create a solid circle or a cone, or use the same value for the bottom and top of a cylinder.

Height

Input 0 to form a solid or hollow circle; if the height is non-zero, the grid forms a cone, frustum or cylinder of the given height.

Blend to square at bottom

If selected, the bottom of a frustum or outside edge of a planar grid is square.

Mesh Density

This width is subdivided by the number of elements in the two directions. To create a solid or hollow polygon, pyramid or pyramidal frustum, set the circumferential density equal to the number of sides in the polygon.

2D Property

Opens the [2D Element Property dialog](#) to allow the element properties to be defined. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Include spacers for form finding

This generates spacer elements along the 2D element edges for use in form finding.

Internal

If spacers are to be generated this opens the spacer properties wizard to allow properties to be defined for internal spacer elements (those between 2D elements).

Edge

If spacers are to be generated this opens the spacer properties wizard to allow properties to be defined for spacer elements around the edge of the 2D elements.

Generate edge elements

This generates 1D elements around the edge of the grid of the type selected.

Section

If edge elements are to be generated this allows the properties for the edge elements to be specified.

7.2.2.10 Data Generation Wizard : Generate

This page confirms that you have entered the data to generate a model if you select Finish, the model will be generated.

7.3 Data Definition Dialogs and Wizards

[Titles](#)

[Analysis Specification](#)

[Units Specification](#)

[Currency Specification](#)

[Tolerances](#)

[Design Code Specification](#)

[Bridge Loading Analysis Specification](#)

[Environmental Impact Specification](#)

[Environmental Impact Wizard](#)

[Axis Definition](#)

[Modify Axis](#)

[Current Grid Definition](#)

[Grid Plane Definition](#)

[Grid Layout Definition](#)

[Grid Line Definition](#)

[Line Definition](#)

[Area Definition](#)
[Region Definition](#)
[2D Polyline Definition](#)
[Node Definition](#)
[Node Grid Settings Definition](#)
[Node Stiffness Definition](#)
[Node Mesh Attribute Definition](#)
[Element Wizard](#)
[Beam Element Releases](#)
[Element Releases](#)
[Member Wizard](#)
[Material Wizard](#)
[Section Wizard](#)
[Section Modifiers](#)
[Section — Multiple Sections](#)
[Spring Property Wizard](#)
[Spring Curve Definition](#)
[Spring Matrix Definition](#)
[Non-linear Spring Curve Definition](#)
[Mass Definition](#)
[2D Property Wizard](#)
[Link Property Definition](#)
[Cable Definition](#)
[Spacer Definition](#)
[Steel Beam Design Property Definition](#)
[Steel Restraint Property Definition](#)
[Member Restraint](#)
[RC Beam Design Properties](#)
[RC Slab Design Properties](#)
RC Member Bar Layouts
[Generalised Restraint Definition](#)
[Rigid Constraint Definition](#)
[Joint Definition](#)
[Constraint Equation Definition](#)
[Tied Interfaces](#)
[Node Loading Definition](#)
[Beam Load Definition](#)
[Beam Pre-stress Definition](#)
[Beam Distortion Definition](#)

[Beam Thermal Load Definition](#)
[2D Element Face Load Definition](#)
[2D Element Edge Load Definition](#)
[2D Element Pre-stress Load Definition](#)
[2D Element Thermal Load Definition](#)
[Grid Loading Definition](#)
[Gravity Load Definition](#)
[Response Spectrum Wizard](#)
[Variable UDL Wizard](#)
[Path Loading](#)
[Load Case Titles](#)
[Analysis Wizard](#)
[Combination Cases](#)
[Append Analysis Case, Append Combination Case, Append List Case](#)
[Append Envelope](#)
[Assemblies](#)

7.3.1 Titles

The titles dialog can be accessed from the Gateway and from the “Data | Titles” menu command. This sets the title information for the model. This is the information that is read and displayed by the “Columbus” document management system.

The title entries, excluding the Notes and Bitmap, are printed at the top of each page of output. For a new file, certain entries default to those in the last file that was saved, but can be changed here.

Job Number

This is the job number, which can be any alphanumeric string.

Initials

The initials of the user.

Job Title

The title of the job.

Subtitle

The subtitle that this model relates to.

Calc Heading

Specific to this model.

Notes

Notes on the model can be stored here. This is a text field that can be appended to as required.

Paste Bitmap

Allows a bitmap graphic to be stored for quick identification of the model. This is useful when viewing the file in Columbus.

Remove Bitmap

Removes the bitmap and leave a blank area.

Copy Bitmap

This places a copy of the bitmap graphic on the clipboard.

7.3.2 General Specification

This can be accessed from the Gateway or from the “Data | Specification | General Specification” menu command. The general specification gives information about the model as a whole.

Structure type & global restraints

The structure type describes the type of structural model to be considered. The options are

- **Space** – 3D structure type. This is the most general structure type, and all the others can be considered as subsets of this.
- **Plane** – planar structure type in the XZ plane. This is used for modelling for example a 2D frame.
- **Grid** – grid structure in the XY plane. This is used for modelling grillages.
- **Plane stress** – 2D analysis option with plane stress conditions.
- **Plane strain** – 2D analysis option with plane strain conditions.
- **Axisymmetric** – 2D analysis option with axisymmetric conditions.

For space and plane stress structure types the edge load specification can be selected to be either pressure or force per unit length. Only the pressure option is available for plane strain and axisymmetric analyses

Global restraints are applied to all the nodes in a model, e.g. to model a plane frame in the XY plane, the user should select a space structure type and apply global restraints in the Z translational, and X and Y rotational, directions. Global restraints do not need to be specified for directions with implied restraint due to the structure type. These directions are displayed greyed.

Location information

This gives information about the location of the site. Those that are used in GSA are the angle to north for setting wind loading and elevation of ground for various storey based calculations. The latitude and longitude are not used by GSA and are only for information. The Map options shows the location specified by the latitude and longitude on a map.

Some advanced specification items can be set. These do not normally need to be changed from the default values.

Miscellaneous

This determines the way various features are interpreted in GSA.

The edge load specification determines if edge loads on 2D elements are interpreted as force per unit length or as pressures. (For axisymmetric only the pressure option is available.)

The tolerances allows the engineer to adjust various tolerances that are preset in GSA. These do not normally need to be changed from the default values.

Units

A base set of units can be selected for the model. These become the default set of units for tables and graphical views. The current units are displayed in the “Units” group box.

7.3.3 Units Specification

When creating a new model the user's "[Preferred Units](#)" are used. These may be changed in the units specification at any time. The units chosen here will be the default units used in GSA, however, local changes of units can be made in individual views.

The use of units is discussed in "[Program Fundamentals](#)".

This can be accessed from the Gateway, from the "Data | Specification | Unit Specification" menu command, from the [General Specification dialog](#) or from the "[Structure type](#)" page of the [New Model Wizard](#). The [units specification](#) sets the default units for the model.

In addition the Units dialog is accessible from the "[Diagram Settings](#)", "[Contour Settings](#)" and "[Wizard: Output Settings](#)" dialog boxes, and the units button on the [Data Options toolbar](#) where there is scope to modify units locally. In this case units which are not relevant are disabled.

Force

Used for force and force derived units such as moment, etc., but not for stress.

Sets the force units to be used and also used in units derived from force, such as moment. Stress units are handled separately.

Length (large)

Used for length and length derived units such as area, etc. but not for displacements.

Sets the "large" length units to be used and also used in units derived from length, such as moment. "Large" length units are used for data that are typically large such as coordinates. "Small" length, stress and acceleration units are handled separately.

Length (small)

Sets the "small" length units to be used. "Small" length units are used for data that are typically small such as displacements and cross-sectional areas. "Small" length units are considered as distinct from "large" length units.

As displacements are one of the main results produced by a structural analysis package they are assigned their own units. This means that, for example, the model geometry may be specified in metres and the model displacements in millimetres.

Length (sections)

Mass

Sets the mass units to be used and also used in units derived from mass, such as density.

Used for mass and mass derived units such as inertia, etc.

Time

Used for time and time derived units, such as frequency, etc.

Sets the time units to be used and also used in units derived from time, such as frequency.

Temperature

Sets the temperature units to be used and also used in units derived from temperature, such as the coefficient of thermal expansion.

Used for temperature and temperature derived units such as coefficients of expansion.

Stress

Like displacements, stresses are one of the main results produced by a structural analysis package, so they are assigned their own units. These units are also used for stress related quantities like the elastic modulus.

Sets the stress units to be used. Stress units are also used for material properties such as the Young's modulus. Stress units are considered as distinct from force and length units.

Acceleration

Sets the acceleration units to be used. Acceleration units are considered as distinct from length and time units.

It is common for accelerations to be specified in g so accelerations are given a separate unit.

Energy

Sets the energy units to be used. Energy units are considered as distinct from force and length units.

Reset units

This allow the user to reset the units to

- **Preferred**—those selected in the [Preferred Units](#) accessible from the [Miscellaneous Preferences](#).
- **SI**—standard SI units.
- **kN-m**—a more useful set of units for structural analysis based on the SI system of units.
- **kip-ft**—a set of units based on those commonly used in the USA.
- **kip-in**—an alternative set of units based on those commonly used in the USA.

Modify Currency

The currency can be specified but no values are modified when the currency is changed. The currency is reported using the ISO 4217 currency codes.

See also:

[Program Fundamentals — Units](#)

7.3.4 Currency Specification

Currencies are stored using the ISO 4217 currency codes (e.g. GBP for United Kingdom pounds or USD for United States Dollars). The currency can be changed to any of those supported by GSA or none.

7.3.5 Tolerances

The Tolerances dialog box allows tolerances that are used in the production of results to be adjusted.

Open this dialog box from the “Tolerances” command on the [Analysis Specification](#).

OR

Open this dialog box from the [Input Data page](#) of the [Preferences dialog box](#).

When opened from the Preferences dialog box this dialog box specifies the tolerance preferences that are used as default for new models. They do not automatically update currently opened models.

When opened from the Analysis Specification the Preferred button may be used to update the tolerances for the model with the tolerance preferences.

Refer to Program Data: — Tolerances Specification for more details.

Vertical element tolerance

Elements are considered to be vertical if the element x axis is within the angle from the global z axis. This affects the interpretation of the element orientation information.

Grid edge straightness tolerance

When identifying panels on a grid plane (sets of elements that enclose a region) the elements will be considered to form a single edge if the change in alignment is less than this tolerance.

Grid loading edge tolerance

If a beam or bar element is found inside the radius defined by the edge tolerance, all of the load is assumed to apply directly to that element at the closest point.

Grid plane tolerance

Entities are deemed to be on a grid plane if they are within this distance of the grid plane.

(This option is only available when this dialog box is opened from the Preferences dialog box.)

Grid loading refinement tolerance

Where a [grid plane](#) is multi-way spanning the algorithm used to distribute the load uses a number integration scheme. The grid loading refinement tolerance determines the density of the points used in the integration. Low tolerances will be adequate where a load is spread over a large number of panels and it is the behaviour of the structure is of interest. Where a single panel or a small number of panels are loaded and local effects are of interest a higher tolerance may be preferred.

A high loading refinement tolerance may cause analyses to slow significantly.

Notes: The algorithm used is based on equilibrium considerations and not on satisfying the flexibility conditions, so higher tolerance levels do not necessarily give better results. A high tolerance may slow the expansion of the grid loading during the analysis.

Member straightness tolerance — maximum difference of element orientation angle

The acceptable difference of the orientation angles of all the elements constitute a member.

Member straightness tolerance — maximum distance from the centre line

The allowable distance of deviation of the intermediate nodes of a member from the centre line.

Bar spacer leg length tolerance for form-finding

The acceptable deviation from the specified spacer leg length in form-finding analysis.

Preferred

Update the tolerances with the tolerance preferences.

(This option is not available when this dialog box is opened from the Preferences dialog box.)

7.3.6 Design Code Specification

The [design codes](#) for both steel and concrete design.

Steel Member Code Checking

This is the code that will be used for steel member utilisation checks.

Concrete Design Code

The code that will be used for slab design and for any beams exported to ADC/AdBeam. With design codes like Eurocode 2 the implementation of the code can vary from one country to another. The country option allows for these variations to be considered.

The codes displayed are all the concrete design codes that GSA has available. Not all of these may be available for both slab and beam design.

Load Design Code

The design code used for load combinations.

7.3.7 Bridge Loading Analysis Specification

As discussed in the [Step-by-Step guide to Bridge Loading](#), there are three different ways of using the tools available for bridge loading. This specification dialog is used to distinguish between these methods and set up the parameters for the first method.

Bridge Design Loading Code

Unless working with UK or HK loading (HA and HB or EC1) on constant width carriageways, the code must be set to "Undefined". Otherwise the user can select the appropriate code from the options below:

- **UK_BD37** – For HA and HB loading to the United Kingdom Design Manual for Roads and Bridges.
- **HK_SDM** – For HA and HB loading to the Hong Kong Structures Design Manual.
- **EC1_UK** – UK implementation of EC1, LM1 and SV.
- **UK_BD21_HB** – For HA to BD21 and HB to BD37
- **UK_BD86_SV-TT** – For HA to BD21 and SV-TT to BD86
- **UK_BD86_SV-Train** – For HA to BD21 and SV-Train to BD86
- **UK_BD86_SV-150** – For HA to BD21 and SV-150 to BD86
- **UK_BD86_SV-100** – For HA to BD21 and SV-100 to BD86
- **UK_BD86_SV-80** – For HA to BD21 and SV-80 to BD86

Using these options where possible makes the most efficient use of the tools available for bridge loading in GSA. The SV loading types are an approximate implementation of the requirements of BD86 requirements. Intensity of SV vehicle loading is adjusted to compensate for the different load factors (1.1 for vehicles, 1.3 for associated lane loading). SV loading is applied in a similar way to HB loading to BD37, with HA loading applied over the full width of lanes, not on 2.5m widths within lines as required by BD86/BD21; note that this is unconservative if there is little or no transverse distribution in the bridge. All HA loading is factored by 0.91 (equivalent to Heavy Traffic / Poor Surface to BD21).

UK BD37/HK SDM and UK BD21/BD86 Loading

To allow for different load factors in different combinations, the user can specify one or more of the following:

- **ULS Comb1:** for ULS Load Combination 1 (dead and live load only)
- **ULS Comb3:** for other ULS load combinations (dead, live and temperature or dead, live and wind)
- **SLS Comb1:** for SLS Load Combination 1

Specifying more than one combination increases the number of load cases generated. Having specified the combinations then the number of HB units to be applied is entered for ULS and/or SLS combinations. Note that for BD37 loading on a concrete bridge the default is not to apply HB units at SLS, but 30 units are needed for certain slabs (see BD49). For HK loading the default is to apply 25 units at SLS (see BS5400:4). At ULS the default is to apply 45 units, but 30 or 37.5 units may be acceptable.

The BD86 - SV loading types are an approximate implementation of the requirements of BD86 requirements. Intensity of SV vehicle loading is adjusted to compensate for the different load factors (1.1 for vehicles, 1.3 for associated lane loading). SV loading is applied in a similar way to HB loading to BD37, with HA loading applied over the full width of lanes, not on 2.5m widths within lines as required by BD86/BD21; note that this is unconservative if there is little or no transverse distribution in the bridge. All HA loading is factored by 0.91 (equivalent to Heavy Traffic / Poor Surface to BD21).

EC1_UK Loading

To allow for the different forms of loading specified in EC1 the user can specify one or more of the following:

- Characteristic (LC1) : gr1/gr5 (traffic load leading action)
- Combination (LC3): gr1 (other load leading action)
- Frequent (LC4): gr1 (with accidental action / reversible SLS)

Specifying more than one form of loading increases the number of load cases generated. For gr5 the user can choose between the following options for the SV/SOV vehicle (each loading type includes all vehicles up to the heaviest specified):

- None,
- SV80,
- SV100,
- SV196,
- SOV250
- SOV350
- SOV450
- SOV600

Undefined Loading

To reduce the length of lists of vehicles and path loading available the user needs to identify which sorts of loading might be applied from the following list. EC1 allows LM1 loading to be factored in the National Annex. Where national parameters have been included in GSA the country can be chosen here, otherwise EC1 User defined can be chosen and national parameters entered here.

- [UK or HK Highway Design Loading \(HA and HB\)](#)
- [UIC or UK Railway Loading \(UIC71, RU, RL, SW/0, SW/2\)](#)
- [US Highway Loading \(H20-44, HS20-44, and HL-93\)](#)
- [Eurocode Loading Generic \(LM-1\)](#)
- Eurocode Loading UK (LM-1, LM-3 UK)
- [Australian Highway Loading \(M1600, S1600\)](#)
- [UK Assessment Loading \(SV\)](#)

7.3.8 Environmental Impact Specification

As discussed in the [Step-by-Step guide to Environmental Impact](#), global environmental impact parameters may be specified in the Environmental Impact Specification dialog.

Global environmental impact parameters are listed for each material type. Highlighting a material type and clicking **Edit** (or double-clicking the material type) opens the [Environmental Impact Wizard](#) for that material type.

The **Reinforcement % for environmental impact** is used to combine the concrete values with the rebar values in determining values for reinforced concrete.

7.3.9 Environmental Impact Wizard

As discussed in the [Step-by-Step guide to Environmental Impact](#), environmental impact parameters for a material type may be specified in the Environmental Impact Wizard. This wizard can be accessed either from the [Environmental Impact Specification](#) dialog to specify global parameters, or from the [Material Wizard](#) to specify values for a particular user material.

Specifying a **Country/Region**, **Variant**, and **Grade** for the material type being considered results in **Typical values** being displayed for each environmental impact parameter. Note that the typical values are based on information available at the time of publication. The data will become out-dated rapidly as environmental circumstances change and as the understanding of the industry evolves. The **date of reference data** is reported to give a measure of confidence.

Click the > button to use the Typical values. Otherwise enter **Specified values** for each environmental parameter.

When the Environmental Impact Wizard is accessed from the Material Wizard for a concrete material type, separate tabs are offered to enable environmental parameters to be specified for both concrete and rebar.

7.3.10 Axis Definition

The axis definition dialog is used to define or modify an axis set. This dialog can be activated from [Axes table view](#) using the wizard button or from a Graphic View by selecting the “Edit user axes” button on the [Sculpt toolbar](#). The units are displayed in the top right corner of the dialog. The z axis direction is derived from the x axis vector (x) and the vector in the xy plane (y) as follows:

$$\mathbf{z} = \mathbf{x} \times \hat{\mathbf{y}}$$

Name

The name is used only as a label for the axis set. Where axes have to be selected this is the name that will be displayed.

Type

Axes can be [Cartesian](#), [Cylindrical](#) or [Spherical](#).

Origin

The origin locates the axis set in space. This is in the current units.

Vector in x

This defines a vector that points in the direction of the x-axis. As this only defines a direction it is assumed to be dimensionless.

Vector in xy plane

To complete the definition of the axis set a second vector is required which along with the vector in the x direction defines the xy plane. This is also dimensionless.

Modify

In many cases an axis system is most easily defined by copying an existing axis and then applying a transformation to that axis system. The modify option opens the “Modify Axis” dialog to allow this.

7.3.11 Modify Axis

The modify axis dialog is used to modify an axis set. This dialog can be activated from “[Axis Definition](#)” and modifies the axis definition. To apply multiple transformations use this dialog to apply the modifications one after the other.

Translate by/Rotate by

The user can choose to modify the axis either by translating along one of the current axis directions or rotating about one of the current axes by the amount specified.

Along/about

This specifies which axis is to be used for the translation or rotation. These are always the current axis direction of the axis system.

7.3.12 Current Grid Definition

The current grid definition dialog is used to modify the current grid. This dialog can be activated from the “Define Current Grid” (Ctrl+Alt+W) menu command on the “Edit” and “Sculpt” menus and on the Graphic View right-click menu.

The current grid is used as follows:

- GSA reports coordinates transformed to the (elevated) axes of the current grid plane.
- Grid points are displayed in Graphic Views in the xy plane of the current grid plane and according to the current grid layout.
- Grid lines are displayed in Graphic Views projected onto the xy plane of the current grid plane

See [Grid axes and the current grid](#) for details of the axes of the current grid.

Grid plane

A reference to a [grid plane](#). If the grid plane does not exist the “Global grid” is assumed.

Click the **Grid Plane...** button to edit the currently selected grid plane, or a copy of it, if the current one is a standard grid plane, or a new grid plane if “<new>” is selected.

Grid layout

A reference to a [grid layout](#). If the grid layout does not exist the “Default” layout is assumed.

Set the grid layout to <disabled> to remove the option from the Draw Grid command.

Click the **Grid Layout...** button to edit the currently selected grid layout, or a copy of it, if the current one is a standard grid layout, or a new grid layout if “<new>” is selected.

Grid Lines

Click the **Grid Lines...** button to edit the [grid lines](#) for the model.

7.3.13 Grid Plane Definition

The grid plane definition dialog is used to define or modify a grid plane. This dialog can be activated from the Grid Planes table view by giving the “Edit | Wizard” menu command. “Wizard” is also available on the Data Options toolbar and on the right-click menu for the table view. It can also be activated from the [Current Grid Definition dialog](#).

Name

The name is used only as a label for the grid plane. Where grid planes have to be selected this is the name that will be displayed.

Type

The grid plane can be either a general grid plane or a storey grid plane. A storey grid plane must exist in a global XY plane.

Grid axis

Grid planes reference an axis set. This may be global or any user defined axis system. The grid plane is parallel to the xy plane of this axis set.

Grid elevation

For situations where grid planes are to be stacked at different levels the grid elevation allows the grid plane to be offset from the axis origin in the local z direction. The elevation defines the height of the grid plane above the origin.

Element list

In some situations only some of the element are to be considered to be part of the grid plane (for example secondary bracing) so the element list can be used to exclude these.

Grid plane tolerance

The grid plane tolerance is used in conjunction with the element list to determine whether an element lies in the grid plane.

Span type

The span type describes the way in which loading on the grid plane will be transferred from space to the surrounding elements. The options are

- **One-way** – the load is transferred in the span direction defined by the angle (see below).
- **Two-way** – this option is for an area load on a complete panel and allows the load to be distributed in a pattern resembling the back of an envelope.

Angle

The angle of the span direction relative to the x-direction of the axis defining the grid plane.

The element list limits the elements that are considered to be in the grid plane. However there are likely to be many elements that may be included in the list which do not form part of the grid plane. The grid plane tolerance allows the user to define a volume (above and below the actual grid plane) within which elements are considered as candidates for the grid plane. Thus a floor may use a relatively small tolerance while a roof structure may use a much larger tolerance.

Load expansion & Simplified tributary areas

For two-way spanning there are a number of different expansion algorithms:

- **Legacy** - Legacy option: loads assumed to ripple out to edges of panel (does not work on re-entrant panels)
- **Plane approx.** - Uses a plane approximation to distribute the load & adjusts for aspect ratio (load distribution depends on model refinement)
- **Plane approx. (smooth)** - Uses a plane approximation to distribute the load, adjusts for aspect ratio and smooths loads along edges
- **Plane approx. (corner)** - Uses a plane approximation to distribute the load, adjusts for aspect ratio and smooths loads along edges, adjusting for corner effects (recommended)

In addition there is an option to use a modified (simplified tributary areas) algorithm, which gives a simple distribution on fully loaded triangular and quadrilateral panels as shown below

The grid load expansion option determines which algorithm is used to expand the grid loads. The legacy option treats loads as a series of point loads applied across the grid plane which 'ripple' out to the elements bounding closed areas. This doesn't work for re-entrant panels, eg L shaped areas. The plane approximation options treat the load on a panel as a single load applied at the centre of loads and a set of loads applied around the perimeter that give equilibrium. The loads are determined so that the aspect ratio of the panel results in a greater load intensity applied to the 'long' edges than the 'short' ones. The basic version of this gives a mesh dependent load distribution and can lead to the mid-span load intensities being too large. The 'smooth' variant takes the udl's on the individual elements and aggregate these along the edges resulting in a linear variation along the panel edges. The 'corner' variant assumes that the loads on the ends of edges at internal corners should be reduced so an adjustment is made to taper the load intensity to zero at these corners.

7.3.14 Grid Layout Definition

The grid layout definition dialog is used to define or modify a grid layout. This dialog can be activated from the [Current Grid Definition dialog](#).

A grid layout defines a distribution of grid points. Grid layouts are referred to by the [current grid](#) and are positioned with respect to the axes of the current grid plane at z=0 elevation. Grid layouts can be displayed in Graphic Views and can be operated on using sculpt commands.

Name

The name (or number) may be used to refer to the grid layout.

Spacing

The spacing of grid points, defined separately for the x and y directions.

Automatic extents

When “automatic extents” is set the extents of the region of grid points are set such that the grid points fill the Graphic View for which they are being calculated (subject to a limit imposed by the program).

Min extent, Max extent

When “automatic extents” is not set, the minimum and maximum x and y extents of the region of grid points.

7.3.15 Grid Line Definition

The grid line definition dialog is used to define or modify the grid lines for the model. This dialog can be activated from the [Current Grid Definition dialog](#).

Grid lines are defined as being in the global XY plane. Grid lines form part of the [current grid](#) and, when displayed, are projected onto the current grid plane (unless it is vertical). Grid lines can be displayed in Graphic Views and can be operated on using sculpt commands.

Grid lines may be straight or circular arcs.

- Straight grid lines are defined by a start position, a length and an angle
- Circular arc grid lines are defined by a centre of the arc, a radius, a start angle and an end angle - where the angles are measured from global X, anti-clockwise about global Z.

Grid Lines table

Label

The name by which the grid line is referred. Labels must be unique.

Arc

Specifies whether the grid line is a straight “Line” or circular “Arc”.

X, Y

The start of a straight line or the centre of a circular arc, in global coordinates.

Length

The length of a straight line or the radius of a circular arc.

Theta 1

The angle of inclination of a straight line or the start angle of a circular arc, measured from global X, anti-clockwise about global Z.

Theta 2

The end angle of a circular arc, measured from global X, anti-clockwise about global Z. (Not required for straight grid lines.)

Add Grid Lines

This is a tool for generating new grid lines using as a basis the grid line that is currently selected in the table. If the increment value is positive the new grid lines are inserted below the current grid line; if negative, above.

The new grid lines are given unique labels, based on the label of the current grid line.

Move Grid Lines

This tool moves the existing grid lines that are currently selected in the table. The grid lines may be translated in the x or y directions or rotated about the Z axis; the transformation occurs in the currently defined axes.

Note that only axes that have a vertical z axis are offered in the list of available axes.

7.3.16 Line Definition

Name

The name is used only as a label for the line.

Type

The type specifies whether the line is a line or an arc. Choose from:

- Line
- Arc defined by third point on arc
The arc spans from the first node to the second passing through the third.
- Arc defined by radius and point.
The third node determines the plane of the arc and the side of the line between the first and second nodes towards which the arc is 'pulled'.

Nodes

The nodes defining the line.

Arc radius

Only used when the type is "Arc defined by radius and point".

Mesh generation

The mesh generation parameters differ with mesh generation tool selected. The following parameters apply to [2D element mesh generation](#):

Generated nodal constraints

The constraints specified here will be applied to nodes generated along the line.

Mesh by

Choose from:

- Use region element size
- Element size
- Number of elements

Use region element size

When selected: Region's element size will be used to generate elements along the line.

Note: If the line is attached to several regions with differing element sizes, select 'Tie meshes in adjoining regions along this line' option to overcome conflicting element sizes along the line.

Element size (Element edge length)

When "Mesh by element size" is selected: specify the desired element edge length for elements generated along the line. The element edge lengths at end1 and end2 should be equal if the desired element edge length of the elements generated along the line is equal. Specify the appropriate element edge lengths at end1 and end2 to achieve the linear variation in element edge length along the line. Note that if the element size is specified (in the [Node Mesh Attribute Definition](#) dialog) for a node that is attached to either end of the line then the line element size at that node is overridden by the node element size.

If the selected mesh generation engine is quad-build (legacy option) linear variation of mesh sizes can be achieved by setting the region 'steps' parameter to 'linear'.

Number of elements

When 'Mesh by number of elements' is selected: specify the desired number of elements to be generated along the line.

If the selected mesh generation engine is quad-build(legacy option) specify the desired ratio of last element edge length to the first element edge length.

Tie meshes in adjoining regions along this line

When selected: during the mesh generation process, after the 2D elements have been generated, a [tied interface](#) is produced that ties the elements generated along the line for the region being considered with elements previously generated along this line for another region. In the determination of which region provides the master elements in the tied interface, an attempt is made to set the courser mesh as the master. However this is only done at the time the first tied interface is created for a line. If tied interfaces are required at a line that is common to more than two regions there is a chance that a course mesh is assigned as a *slave*, contrary to the recommendation that the courser mesh should be the master. The resulting tied interface should be checked by labelling tied interfaces, to ensure that adequate constraint is provided. Hint: there is an option on the right-click menu of the tied interfaces table to 'Switch Slave and Master'.

7.3.17 Area Definition

Name

The name is used only as a label for the area.

Type

Choose from:

- Void
- One way
- Two way
- Multi way

(In GSA 8.4: no use is made of the distinction between 'one way', 'two way' and 'multi way' spanning areas.)

Span angle

The span direction for one way spanning areas.

Group

The group number to be assigned to elements created for the area.

2D property

The 2D property number to be assigned to elements created for the area. Either choose a property from the list or type in a property number.

Lines

The lines making up the area. The lines must be specified in sequence around the area. The direction of the lines is not are specified is significant.

Mesh generation

No meshing parameters are required to fill in the Area Definition.

Quad-Build (Legacy Option)

The following parameters apply to [2D element mesh generation](#):

Refinement coefficient

In the transition from fine mesh to course mesh this coefficient influences the portion occupied by fine mesh: the higher the coefficient, the larger the area of fine mesh. If the mesh is of consistent density then this coefficient is not important and may be set to zero. The refinement coefficient must be set in the range 0.0 to 0.35 . If the area is bounded by an arc then a refinement coefficient of less than 0.15 will be taken as 0.15 . The refinement coefficient is ignored for void areas.

7.3.18 Region Definition

Name

The name is used only as a label for the region.

Grid plane

The grid plane in which the region is deemed to lie in. Exactly what this means depends on the context in which the region is used. Typically, where a planar region is required (e.g. for 2D element mesh generation), the components of the region are projected onto this grid plane. When the grid plane is set to 'Auto' the plane for the region is assumed to be the plane in which the largest area in the region lies.

Type

(Reserved for future use.)

Nodes, Lines, Areas

The nodes, lines and areas making up the region. The order they are specified is not significant. Lines referenced by included areas need not be specified and nodes referenced by included lines need not be specified.

Mesh generation

The following parameters apply to [2D element mesh generation](#):

Element type

The element type that is to be generated for the region. Choose from:

- Quad 4
- Quad 8

Element size

Specify the desired element edge length for elements generated along the lines attached to the region. Region's element size will only used when the line's Mesh by option 'Use region element size' is selected. Note that if the element size is specified (in the [Node Mesh Attribute Definition](#) dialog) for a node that is attached to the region then the region element size at that node is overridden by the node element size.

Split badly shaped quads into tri elements after meshing

When selected: Tri3 or Tri6 elements will be created by splitting the Quad elements whose minimum and maximum angles are not within the limits if any generated during the mesh generation process. This option is available only for Meshtools mesh generation engine.

Create tri elements during mesh generation

When selected: both quad and tri elements will be created during the mesh generation process. This option will be useful when the option 'Allow odd number of elements along a line' is switched on. This option is available only for Meshtools mesh generation engine.

Offset 2D elements downwards by half their thickness

When selected: during the mesh generation process, if the referenced property (i.e. the 2D property specified for the area) exists for a generated 2D element then the element is offset downwards, with respect to the plane of the region, by half the element thickness, to result in the top surface of the element being in the plane of the region.

Generate beam elements along aligned members

When selected: during the mesh generation process, after the 2D elements have been generated, beam elements are generated along all members aligned with lines in the region, connecting all the nodes along the member. A member is 'aligned' if a continuous string of lines in the region (whether internal or edge) spans the length of the member.

Create soil spring supports per generated 2D element

When selected: during the mesh generation process, after the 2D elements have been generated, a node is created at the centre of every 2D element generated for the region and ties that 'soil node' to the element by constraint equations. Raft interaction records are produced per 'soil node' generated, in case raft analysis is to be performed.

Advanced parameters:

The advanced parameters differ with the mesh generation engine.

MeshTools:

The following Meshtools parameters apply to [2D element mesh generation](#).

Internal element target size

The element size tends towards Internal element target size as they get away from the boundary or interior lines. This parameter can be used to reduce the total number of elements generated without losing much on element shape.

Maximum angle for Quads

This is the maximum allowable angle limit above which the quads split into triangles. Maximum angle of a quad is the maximum angle at all the corners. Note that the Quad elements will not split into Tri elements if the option 'Allow Tri elements to be generated' is not selected.

Minimum angle for Quads

This is the minimum allowable angle limit above which the quads split into triangles. Minimum angle of a quad is the minimum angle at all the corners. Note that Quad elements will not split into Tri elements if the option 'Allow Tri elements to be generated' is not selected.

Allow odd number of elements along a line

When selected: odd or even numbers of elements will be generated along each line, including just one element in the situation where the line length is equal to or shorter than the specified element size. If the flag is switched off Meshtools always generates even number of elements along a line.

Note that Meshtools will not be able to generate the mesh when the total number of element edges to be generated around an area is not an even number unless tri elements are allowed

to create during the mesh generation. It is suggested to use 'Create tri elements during mesh generation' in combination with this option. The following errors may be generated by Meshtools when this option is selected.

- At least one line has an odd number of edges. – Ensure that the total number of element edges around an area should be even number .
- At least one line is too irregular. – Try to enforce an even number of element edges on every line.

Mesh gradation

This parameter controls the elements size gradation from the size specified for lines and nodes to the Internal element target size at the inside the domain.

Mesh priority

It controls the extent to which the specified element size governs at the expense of the ideal element shape. Default value is having slight preference for shape over size.

Optimisation

Controls the trade-off between processing speed and quality of mesh

Quad-Build (legacy option):

The following parameters apply to [2D element mesh generation](#):

Steps

Determines the spacing of generated nodes along lines throughout the region. Choose from:

- Constant
- Linear

This setting is largely overridden by the Step Size / Number of Segments specified for the lines. Setting this parameter to Linear is usually preferred, though occasionally it is worth experimenting with Constant. Refer to the [Step By Step Guide, Generating 2D element meshes modelling tips](#) for more information on this parameter.

Density

The number of elements around the edges of areas is influenced by the parameters specified for the bounding lines. The density parameter determines the density of mesh in the transitional areas within the area. Choose from:

- Fine
- Dense
- Sparse

Fine and Dense result in a consistently fine mesh as determined by the parameters specified for the bounding lines. Where internal lines or nodes are specified, Fine results in a finer mesh than Dense; otherwise Fine and Dense should produce the same mesh. Sparse results in fewer, larger transitional elements, possibly at the expense of good element shape.

Rigour

Choose from:

- Fast
 - at the expense of quality of mesh.
- Rigorous (recommended)
 - up to 30% longer processing.

7.3.19 2D Polyline Definition

The 2D polyline definition dialog is used to define or modify a polyline. This dialog can be activated from the [2D Polyline table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Name

The name is used only as a label for the polyline.

Polyline definition

The vertices are entered in the table and the shape of the polyline is reflected in the graphical window. The vertices are represented graphically as circles and the initial point is as a larger circle. The line forming a closed polygon is represented as a broken line.

Note that vertices are defined by Cartesian x-y coordinates even when the polyline is to be used with a cylindrical grid plane.

A 2D polyline does not need to be closed to represent a polygon: closure will be inferred from the context.

Polyline checks

A badly formed polyline is one that has:

- coincident vertices, or
- overlapping or crossed lines, or
- vertices lying along lines.

Invalid lines and vertices are highlighted in the dialog and specific issues in the polyline are reported upon exiting the dialog.

(Invalid polylines are treated as errors in the pre-analysis data checks.)

7.3.20 Node Definition

The node definition dialog is used to define or modify a node. This dialog can be activated from [Nodes table view](#) using the wizard button or from a Graphic View by selecting a node and then “Edit Node” from the [right-click menu](#).

Name

The name is used only as a label for the node. (default: blank)

Grid node settings

Opens the [Node Grid Settings Definition dialog](#) to allow the node to be converted to grid node or standard node and to allow definition of the grid node settings.

Current grid plane

(Available only when the node is a 'standard node'.)

Opens the [Current Grid Definition dialog](#) to allow the coordinate reference axes for standard nodes (i.e. the 'current grid plane') to be set.

Coordinates

For standard nodes — the location of the node in current grid plane axes, in the current units. Refer to the [Grid axes and the current grid](#) documentation for information on defining coordinates for standard nodes in non-global directions.

For grid nodes — the location of the node in grid coordinates, in the current units.

Plane structures have only x and z coordinates; grid structures have only x and y coordinates. In these cases the coordinate that is not relevant is greyed.

Constraints

The [constraints](#) define if the node is fixed in particular directions. This is done by selecting an axis, normally global and selecting the directions to restrain. Directions that do not apply for that structure type are disabled. The restraint types Free, Pin and Encastre options provide a shortcut to setting the flags for the individual degrees of freedom. The symmetry options in the xy plane, yz plane and zx plane provide shortcuts for applying the boundary conditions corresponding to symmetry in these planes. The symmetry options do not unset other restraint conditions.

Stiffness

Opens the [Node Stiffness Definition dialog](#) to allow definition of the stiffnesses associated with the node.

Mesh attributes

Opens the [Node Mesh Attribute Definition dialog](#) to allow definition of the mesh generation attributes associated with the node.

7.3.21 Node Grid Settings Definition

Grid node

When checked, the node is defined as a 'grid node', and coordinates for the node are specified with respect to the specified datum on the specified grid plane. Otherwise the node is defined as a 'standard node'.

Other settings in this dialog are enabled only when this setting is checked (i.e. only for grid nodes).

Grid plane

Coordinates are specified with respect to this grid plane. (default: global)

Datum

Choose from:

- Grid plane origin. (default)
- Grid line intersection — the point of intersection of the specified grid lines projected onto the specified grid plane.

7.3.22 Node Stiffness Definition

The node stiffness is a stiffness that can be associated with a node in any of its directions. Most typically it is used to represent a soil stiffness. The units are displayed in the top right corner of the dialog.

Translational

This defines the translational stiffness in each of the constraint axis directions. Directions that do not apply for that structure type are greyed out.

Rotational

This defines the rotational stiffness about each of the constraint axis directions. Directions that do not apply for that structure type are greyed out.

7.3.23 Node Mesh Attribute Definition

The node mesh attributes are used in the [2D element mesh generation](#) process.

Tie node to mesh

When checked for an internal node of a region, the mesh generation process ignores the node but, as a post-generation step, the node is constrained by a tied interface to act as if connected to the generated 2D element in which it lies. Otherwise the mesh is generated such that the internal node is at the corner of a generated 2D element.

When **tie node to mesh** is checked, the **element edge length** and **radius of influence** are ignored.

Element size

The desired element edge length around the node. Note that when specified for a node that is attached to a line then this overrides the element size specified for the connected line (in the [Line Definition](#) dialog) but is ignored when 'mesh by number of elements' is specified for the line. For unattached nodes this setting is ignored when **tie node to mesh** is checked.

Radius of influence

This option is not active and is only available when quad-build (legacy option) mesh generation engine activated. The radius around an internal node that the element edge length has influence. (Ignored for nodes that are not internal nodes, and when **tie node to mesh** is checked.)

7.3.24 Element Wizard

This wizard can be activated from the Elements table view using the wizard button or from a Graphic View by selecting an element and then “Edit Element” from the right-click menu. The wizard takes the user through the following pages:

More:

[Element Wizard: Type Definition](#)

[Element Wizard: Properties](#)

7.3.24.1 Element Wizard: Type Definition

The first page allows the selection of the type of element.

Type

This defines the element type. The types available will depend on the structure type and the [preferences](#).

Group

This defines the group to which the element belongs. No further information is required to define a group.

Topology

This is where the nodes defining the element are specified. The order in which the nodes are specified matters for the definition of the [element axes](#).

Orientation

The specification of the [orientation](#) depends on the element type. In the most general case both the orientation **Node** and **Angle** can be specified. The angle is measured in degrees.

Dummy Element

When checked, the element is specified as a dummy element.

7.3.24.2 Element Wizard: Properties

This page of the wizard relates to element properties. Where an element may have releases or offsets these are reported here.

Beam Section or Property

Depending on the element type chosen this item will either be labelled “Beam section”, “Spring property”, “Mass property”, “2D property”, “Cable property”, “Link property”, or “Spacer property”. These are the properties that are used for analysis to determine the stiffness and mass matrices. Either choose a property from the list or type in a property number.

Releases

Opens up the [Beam Element Releases](#) (for beam elements) or [Element Releases](#) (for 2D elements) dialog to allow some of the degrees of freedom to be pinned rather than fixed. For element types where degrees of freedom cannot be released this will be greyed.

Offsets

This brings up the [Modify Element Offsets](#) dialog to allow offsets to be specified at the nodes. Offsets are normally used to offset beam from the centre of a column to the face.

Properties

Opens the appropriate property wizard for the selected element type and property number. Changing the property here will affect all other elements of the same type with the same property number.

7.3.25 Beam Element Releases

The Beam Element Releases dialog box is used to specify releases and stiffnesses on beam elements. It is opened from the [Element Wizard: Properties](#).

Translational and Rotational Releases

Each degree of freedom at each end of the beam element can be released. If the degree of freedom is released a stiffness can also be applied to connect the end of the beam to the node.

The buttons gives a shortcut to setting up common release options

Fully fixed – the end of the beam is not released in any direction

Release translations – the translation degrees of freedom are released, allowing the end of the element to translate freely, but the rotations are still fixed to the rest of the structure

Release moments – the moment degrees of freedom are released for the element, but translations are still fixed to the rest of the structure

Remove stiffness – the releases are retained but the stiffnesses are removed.

Note: care should be taken when applying releases that the resulting element does not become unstable. Various checks for invalid restraints are carried out to prevent singularities at a later stage in the analysis. It is invalid to:

- release a translational degree of freedom at both ends of a beam
- release the torsional degree of freedom at both ends of a beam
- release moment degree of freedom at both ends of beam and one of the corresponding translational releases

7.3.26 Element Releases

The Element Releases dialog box is used to specify releases on 2D elements. It is opened from the [Element Wizard: Properties](#).

Translational and Rotational Releases

When checked, the release condition will be set as specified for node n on the selected elements.

Modifications will only be applied to the nodes that are relevant for the element being modified. So, for example, the settings for only nodes 1 to 4 will be applied to quad 4 elements.

7.3.27 Member Wizard

This wizard can be activated from the Members table view using the wizard button or from a Graphic View by selecting a member and then “Edit Member” from the right-click menu. The wizard takes the user through the following pages:

More:

[Member Wizard: Definition](#)

[Member Wizard: Concrete Properties](#)

[Member Wizard: Steel Properties](#)

[Member Wizard: Releases and Offsets](#)

7.3.27.1 Member Wizard: Definition

The member wizard can be activated from the [Members](#) table view using the wizard button or from a Graphic View by selecting a member and then “Edit Member” from the [right-click menu](#).

Type

The type is where the member is identified as a beam column or other member and whether it is steel, concrete or another material. Other fields will depend on this choice.

Section

These are the properties that define the type, shape and dimensions of the section. Either choose a property from the list or type in a property number.

Group

This defines the group to which the member belongs. No further information is required to define a group.

Member Geometry

The geometry may be defined as Linear or Arc with third point on arc or Arc with radius and point or Explicit.

Orientation

The specification of the [orientation](#) depends on both the orientation **Node** and **Angle**. The angle is measured in degrees.

Topology

This is where the nodes defining the member are specified. The order in which the nodes are specified matters for the definition of the [member axes](#). The flip option allow the member to be reversed end for end.

Radius

Only used when the member type is 'Arc with radius and point'.

Internal Nodes

List of nodes defining the intermediate topologies, only used when the geometry is 'Explicit'. The list of nodes need not be ordered and may be entered as a GSA list (e.g. '11 to 20 step 2') but will be stored as a simple list.

The Internal Nodes should be an ordered list and only used when the geometry is Explicit.

Restraint Nodes

List of nodes defining additional restraint points along the member. The list of nodes need not be ordered and may be entered as a GSA list (e.g. '11 to 20 step 2') but will be stored as a simple list. The restraint nodes must lie along the member for the member to be valid.

7.3.27.2 Member Wizard: Concrete Properties

This page is available when the member type is a concrete beam, concrete column or concrete member to define concrete specific aspects of the member.

Design Property

The design property refers to the RC Beam Design Property and is used to define design specific parameters. Either choose a property from the list or type in a property number.

Bar Arrangement(s)

The bar arrangements are the specific arrangements of bars in the section. The option to initialize from the design property allows a bar arrangement to be initialized by selecting the first of the possible arrangements from the appropriate bar pattern.

Manage Bar Arrangements

Manage bar arrangements displays a list of the arrangements which exist for beams or columns, as appropriate. These can be edited or deleted as required.

7.3.27.3 Member Wizard: Steel Properties

This page is available when the member type is a steel beam, steel column or steel member to define steel specific aspects of the member.

Design Property

The design property refers to either a Steel or RC Beam Design Property and is used to define design specific parameters. Either choose a property from the list or type in a property number.

Restraint Property

The restraint property is used with steel members to define restraints along the member. Either choose a property from the list or type in a property number.

7.3.27.4 Member Wizard: Releases and Offsets

Where a member may have releases or offsets these are reported here.

Releases

Opens up the "[Member Releases](#)" dialog to allow some of the degrees of freedom to be pinned rather than fixed. For element types where degrees of freedom cannot be released this will be greyed.

Member releases are not used in current version of GSA. [Restraint properties](#) are used for checking the steel members.

Offsets

This brings up the "[Modify Member Offsets](#)" dialog to allow offsets to be specified at the nodes.

Offsets are normally used to offset beam from the centre of a column to the face.

7.3.28 Material Wizard

The material wizard is used to define material properties for both analysis and design. This wizard can be activated from the [Material table](#) using the wizard button or from a Graphic View by selecting an element and then “Edit Material” from the [right-click menu](#). The wizard takes the user through the following pages.

More:

[Material Wizard : Material Definition](#)

[Material Wizard : Elastic Properties](#)

[Material Wizard : Design and Non-linear properties](#)

[Material Wizard : Fabric Properties](#)

7.3.28.1 Material Wizard : Material Type

This page allows the user to select the material model and type.

Name

The name is used only as a label for the material. Where materials have to be selected this is the name that will be displayed.

Material Model

This is set to either “Isotropic” or “Elastic fabric” depending on whether this is a normal material or a fabric material. It is designed to allow for other material models to be specified in the future. Changing the material model to ‘Elastic Isotropic’, ‘Elastic Orthotropic’ or ‘Elastic-Plastic Isotropic’ preserves as many properties as possible and does not replace them with steel values.

Material Type

This is used for design and environmental impact purposes. At present a material can be described as one of “Steel”, “Concrete”, “Aluminium”, “Glass”, “Wood”. If none of these apply then the type should be set to “Undefined”. This is not applicable to fabric materials.

Environmental Impact

Unchecking **Use 'Environmental Impact Specification' values** enables environmental impact parameters to be specified for this material, overriding the global values. The **Environmental impact** button opens the [Environmental Impact Wizard](#). (Only enabled when the material type is defined.)

The **Reinforcement proportion (for environmental impact)** is used to combine the concrete values with the rebar values in determining values for reinforced concrete. (Only enabled when the material type is concrete.)

The current environmental values for the material, whether global or overridden, are reported.

Copy Standard Materials

This option is only available for user defined materials, and allows the material properties to be initialised to those of one of the standard materials.

Copying a standard material when ‘Elastic Orthotropic’ or ‘Elastic-Plastic Isotropic’ are selected copies the standard properties into the chosen model.

Trying to copy a standard material into Fabric or another model where copying the properties makes no sense reports an error message to the user.

7.3.28.2 Material Wizard : Elastic Properties

This page defines the properties used for elastic analysis. Fabric materials go to a separate page.

Elastic modulus

The elastic or Young's modulus of the material with dimensions of stress.

Poisson's ratio

The Poisson's ratio, which is dimensionless.

Shear modulus

The Shear modulus with dimensions of stress. For an isotropic material the shear modulus can be determined from the elastic modulus and the Poisson's ratio.

$$G = \frac{E}{2(1 + \nu)}$$

Density

The mass density of the material used to determine gravity loads and in dynamic analysis.

Coefficient of thermal expansion

The coefficient of thermal expansion is only used with thermal loading.

Damping Ratio

The damping ratio is used during a dynamic analysis to calculate an estimate of the modal damping ratio.

7.3.28.3 Material Wizard : Design and Non-linear properties

This page defines the material properties required for design and non-linear analysis. For linear analysis only this page can be ignored.

Yield stress

The stress at which the material first yields. For design this may be a value specified by a design code.

Ultimate stress

The stress at which the material will break. This is used mainly for design purposes.

Hardening modulus

Once the material has yielded it is assumed to follow a straight-line relationship between stress and strain. The slope of this line is the hardening modulus.

Hardening parameter

This is used in non-linear analysis to determine if the hardening model should be [isotropic or kinematic](#).

7.3.28.4 Material Wizard : Fabric Properties

This page defines the properties used for fabric analysis. Normal materials go to a separate page.

Fabric warp modulus

The elastic or Young's modulus of the material in the warp direction with dimensions of force per unit width.

Fabric weft modulus

The elastic or Young's modulus of the material in the weft direction with dimensions of force per unit width.

Shear modulus

The Shear modulus with dimensions of force per unit length. For an isotropic material the shear modulus can be determined from the elastic modulus and the Poisson's ratio.

$$G' = \frac{E'}{2(1 + \nu)}$$

Poisson's ratio (warp-weft)

The Poisson's ratio relating to the warp-weft direction, which is dimensionless.

Allow Compression

This box should be unchecked if the fabric is to allowed to behave like a true fabric and not allow compression.

7.3.28.5 Material Wizard : properties

You can enter material property values here. See [User-Defined Materials](#) or the OpenSees manual.

7.3.29 Section Wizard

The section wizard is where beam sections can be selected or defined. The basic choice is between selecting sections from a catalogue, defining the geometry of typical section shapes such as rectangles or defining sections explicitly.

This wizard can be activated from the [Beam Sections table](#) using the wizard button or from a Graphic View by selecting a beam or bar element and then "Edit Property" from the [right-click menu](#). The wizard takes the user through the following pages.

More:

[Section Wizard : Section type](#)

[Section Wizard : Catalogue Section](#)

[Section Wizard : Standard shapes](#)

[Section Wizard : Perimeter section definition](#)

[Section Wizard : Line segment section definition](#)

[Section Wizard : Explicit properties](#)

[Section Wizard : Section definition](#)

7.3.29.1 Section Wizard : Section type

The section type is where the basic method of selection is defined.

Name

The name is a label used to identify a particular section.

Material

The material will list all the material defined, and fill in with the material number where there are gaps in the numbering. If the required material has not yet been defined the number of that material can be entered directly.

Definition method

The definition method specifies how the material is to be defined. This breaks down into four main options.

- **Catalogue sections** are selected from online section catalogues.
- **Standard sections** are defined by a shape and its dimensions. The properties are then calculated from these.
- **Geometric sections** are defined either by a perimeter or by line segments. The properties are then calculated from these.
- **Explicit sections** are defined by entering the section values explicitly.

The page that follows this will depend on the selection made at this stage.

Section Definition Axis System

In the Section Wizard sections are defined in a conventional x/y axis system, referred to as the section definition axes. The resulting section is used in GSA such that the section appears the same as when defined in the wizard when the 1D element is viewed with the element x axis pointing into the page. To achieve this the section definition axis system used in the wizard is mapped to the element -y/z axes. More information on this is given in [Program Fundamentals — Beam Sections and Section Database](#).

7.3.29.2 Section Wizard : Catalogue Section

The catalogue sections are stored in a SQLite database, installed with GSA.

Catalogue

The catalogue is first level of selection for the section.

Type

The type is the type of section required e.g. Universal Beam or Equal Angles.

Section

The particular section of the type selected e.g. EA250x250x35. If the section is marked S/S that particular section has been superseded.

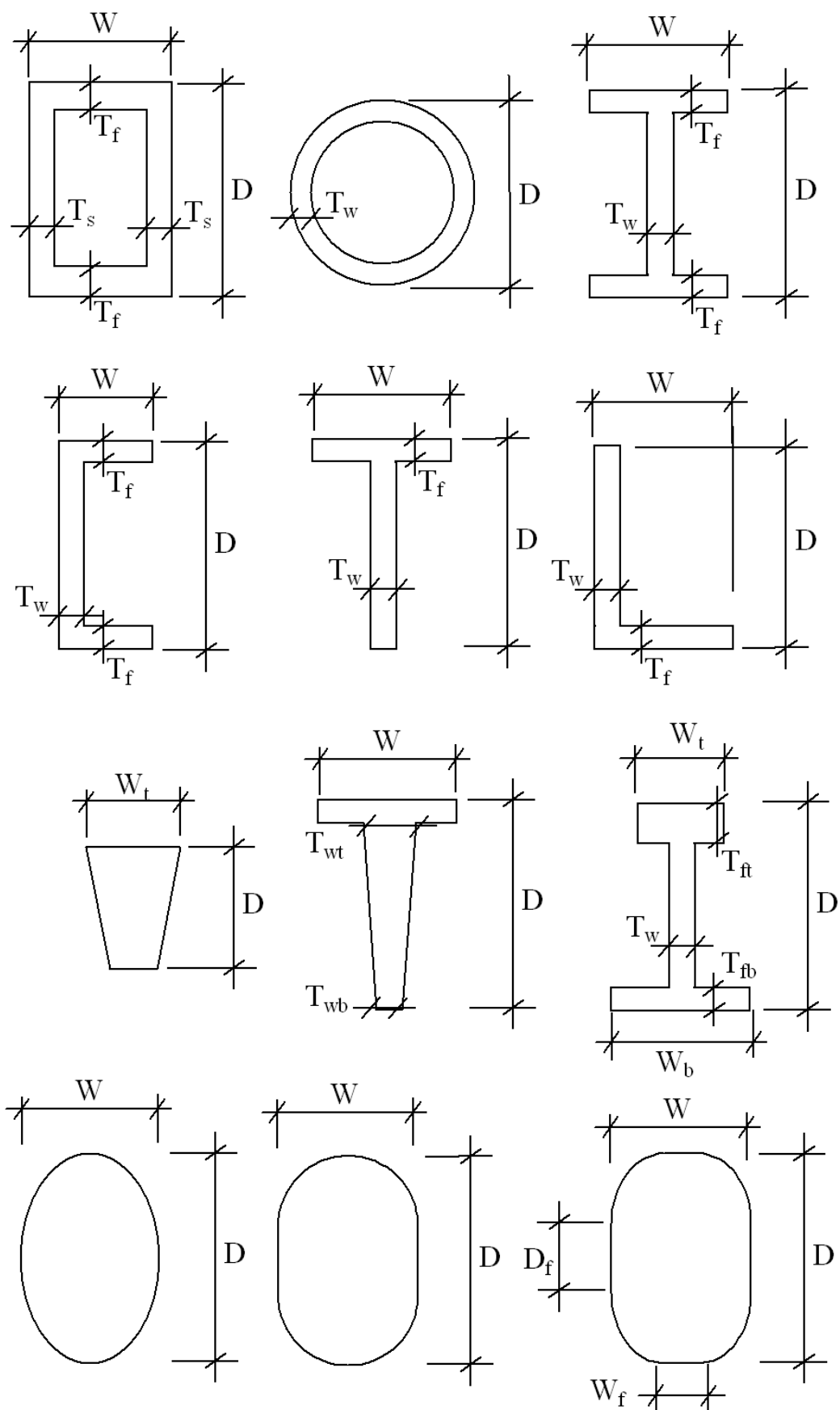
Include superseded sections

Superseded sections are to be displayed. If the model contains superseded sections this will be checked on entry.

7.3.29.3 Section Wizard : Standard shapes

Section dimensions

Standard sections can be rectangular, circular, I, channel, T or angles. Rectangular and circular sections can be either solid or hollow. Specify the overall external dimensions of the section and the thickness of component parts as shown below. Note that top and bottom flanges are always the same thickness, as are left and right walls of a rectangular hollow section.



Ellipse index (n)

This field is only available for the ellipse section type. It allows superellipses to be entered. A value of 1 corresponds to a diamond, 2 to an ellipse, and ∞ to a rectangle.

7.3.29.4 Section Wizard : Perimeter section definition

The outline of a perimeter section is defined by a series of coordinates describing a polyline. The polyline is automatically closed to form a polygon so an end point coincident with the start point need not be entered. Polyline segments may not intersect.

In addition, any number of voids may be defined in the section, again by a series of coordinates describing an unclosed polyline of non-intersecting segments. Voids may not intersect with each other or with the outline.

The section displayed in the wizard is as viewed from end 1 of the element towards end 2.

The centroid is calculated for the section and the section is assumed to lie centred at its centroid, — not at the datum coordinates.

The section can be imported or exported from a DXF file. For the import to work the DXF file should contain only LWPOLYLINE or POLYLINE entities that described the perimeter and void in the section. The export option allows the section to be exported as a series of LWPOLYLINE entities.

The bridge beam option is only enabled if the bridge beam database is available. This gives access to standard bridge beam sections.

The shear factors, K_y and K_z , are also set as zero. These may be modified using the [Section Modifiers dialog](#).

Calculate torsion constant 'J'

This option uses a purpose-built solver for the calculation of the torsion constant. The calculation will accept any arbitrarily defined perimeter section as input, as long as the perimeter is a valid section shape and the section area is sufficiently thick-walled (see below). No further input is necessary in setting up the analysis. Due to the increased complexity of this method, when selecting this option there may be a brief but noticeable delay when proceeding through to the next page of the Section Wizard as the Wizard constructs and solves the torsion problem.

The calculation of the torsion constant 'J' makes use of a numerical technique that is designed for use specifically on thick-walled sections - that is, section shapes with a sufficient area enclosed within the perimeter boundary relative to the perimeter length. Thin-walled sections are best considered as a separate concern and are typically solved using specific thin-walled theory. This theory is not currently implemented in this solver and hence it is not recommended to calculate the value of 'J' for thin-walled sections in this case. Section shapes that are considered to be too thin will prompt a warning message to the user, given at the point during the wizard when the value of J is calculated.

Internally, the solver makes use of the 'membrane analogy' to construct and solve a particular case of the Poisson problem. The differential equations along with the specific boundary conditions prescribed for the torsion problem are assembled and solved numerically by application of boundary element method (BEM) in a dedicated solver.

7.3.29.5 Section Wizard : Line segment section definition

Line segment sections are defined in terms of a series of lines of specified thickness. The section properties are calculated assuming the section is a thin-walled section, making no attempt to "tidy up" the section at segment junctions. Where a section may not be considered as thin-walled, a perimeter section should be used.

The section is described by a series of "Thickness", "Line to" and "Move to" instructions. An initial "Thickness" is specified; this is applied to all segments until another "Thickness" is specified. After the initial "Thickness" the start position is specified by a "Move to" instruction. Thereafter, "Line to"

and “Move to” instructions are given to complete the definition of the section.

The centroid is calculated for the section and the section is assumed to lie centred at its centroid, — not at the datum coordinates.

The shear factors, K_y and K_z , are not calculated for line segment sections and are set as zero. These may be modified using the [Section Modifiers dialog](#).

7.3.29.6 Section Wizard : Explicit properties

With this option the section properties are defined explicitly. Note: only an approximation to these sections can be drawn.

Area

The overall cross sectional area.

I_{yy}

The second moment of area of the section about the element yy axis through the centroid. Typically associated with major axis bending.

I_{zz}

The second moment of area of the section about the element zz axis through the centroid. Typically associated with minor axis bending.

J

The [torsional constant](#). This is not the polar moment of inertia except for circular sections. For circular section the following relationships hold for the second moments of area and the torsion constant:

$$I_{yy} = I_{zz} = I$$

$$J = 2 \times I$$

K_y

The [shear area factor](#) related to minor axis bending. If this value is set to zero the element is assumed to act as a simple beam, otherwise it is treated as a shear beam (i.e. the shear deformations are considered).

K_z

The shear area factor related to major axis bending.

7.3.29.7 Section Wizard : Section definition

This page summarises the section definition and allows extra data to be specified.

Convert to Perimeter

At times it may be useful to convert a section from a catalogue section or a standard shape to a perimeter (typically where the section may require some adjustment). When this option is selected the original section can no longer be modified.

Export

The export option allows the section shape to be exported to a DXF file. The section is exported as a series of LWPOLYLINE entities.

Modify Analysis Properties

The modify properties option allows the user to override the calculated section properties for

analysis purposes. This option should be used with care.

Properties

This shows the values of the section properties as used in analysis (i.e. with modification factors applied).

More displays a more comprehensive list of (unmodified) section property values.

Section type

The section type can be specified as rolled or welded. For sections other than steel sections the default of “not applicable” should be used.

Note that the section type affects the shear area calculated during the shear stress calculation (but does not affect the calculation of K_y and K_z). The section type is also used in steel design calculations.

Section cost

The cost of the section is specified as a cost per unit mass and is used to give an approximate cost for steelwork.

7.3.30 Section Modifiers

This is where the values of the section properties from catalogues or from calculation from the section shape can be modified for analysis. The modification only affects the stiffness properties, not the mass properties or gravity loads.

Method

The method by which the section properties can be either modified

- **By**—the actual value is modified by a modifier factor
- **To**—the actual value is changed to a modifier value.

Modifier

The modifier factor or modifier value for this property item, depending on the method selected.

Use modified properties in mass/weight calculation

The modifier section properties are always used in the calculation of the stiffnesses. If they are to be used in the calculation of weight (and gravity load) and mass (for dynamic analysis) the item should be set.

Ignore non-zero I_{yz} terms in stiffness calculation

When a section is non-symmetrical there is an I_{yz} term in addition to the I_{yy} and I_{zz} terms, which results in deflections in the plane orthogonal to the loading. Where other constraints on the section prevent this type of behaviour it is appropriate to ignore this effect.

Zero bending terms

If an element is to be considered as contributing only axial stiffness this will set all the bending and torsion terms to zero.

Simple beams

If an element is to be considered as a simple beam this will set the shear area factors to zero to flag this.

Multiple sections

If a section is to be used to represent more than one actual element or the element is on a symmetry plane, this option allows for [appropriate factors](#) to be set.

Calculate stresses

If a section is modified it may not be appropriate to calculate element stresses. If it is appropriate there is the option of calculating from the unmodified or modified properties.

The user is therefore given the options of calculating the section stresses:

- if unmodified
- using the unmodified properties
- using the modified properties

Reset

This button will reset all the factors to “By 1”.

7.3.31 Section - Multiple Sections

This is where the user can specify that the modification factors on the properties where the section is being used to represent multiple elements where only a single element has been specified. This modifies the section properties: area, I_{yy} , I_{zz} and J (K_y and K_z are unchanged) and the flag to use the modified properties in the mass/weight calculations is set.

Multiplier

The section properties: area will be factored by this number. No interaction between the elements represented by the section is assumed.

Set properties for symmetry plane

This allows for modifications for elements on a symmetry boundary. The area, I_{yy} , I_{zz} and J values are factored by 0.5. This factor can be applied with the number of elements in which case the factor is $0.5 \times (\text{number of elements})$.

7.3.32 Spring Property Wizard

This wizard can be activated from the [Spring Properties table](#) using the wizard button or from a Graphic View by selecting a spring element and then “Edit Property” from the [right-click menu](#).

The wizard takes the user through the following pages.

More:

[Spring Property Wizard : Type](#)

[Spring Property Wizard : Stiffnesses](#)

7.3.32.1 Spring Property Wizard : Type

The type is where the basic type of spring is selected. The units are displayed in the top right corner of the dialog.

Name

The name is a label used to identify a particular spring property.

Type

The type of spring which can be either:

- **Translational** — the spring is translational
- **Rotational** — the spring is rotational

- **Matrix** — the spring stiffness matrix is defined explicitly (only for ground springs)
- **Tension only** — the spring is translational and only stiff in tension
- **Compression only** — the spring is translational and only stiff in compression

7.3.32.2 Spring Property Wizard : Stiffnesses

This page will vary depending on the type of spring selected on the previous pages. The units are displayed in the top right corner of the dialog.

Axis

The axis set in which the spring acts. This will list local (based on the element topology) and global along with all the user-defined axes. Alternatively an axis number can be entered.

Non-linear spring curve

x/xx - non-linear spring curve reference in x/xx direction or "Linear" indicating the spring is linear

y/yy - non-linear spring curve reference in y/yy direction or "Linear" indicating the spring is linear

z/zz - non-linear spring curve reference in z/zz direction or "Linear" indicating the spring is linear

Stiffness

x/xx - x/xx direction stiffness if the spring is linear

y/yy - y/yy direction stiffness if the spring is linear

z/zz - z/zz direction stiffness if the spring is linear

Matrix

A reference to the matrix defining the stiffness

Damping ratio

The damping ratio is used during a dynamic analysis to calculate an estimate of the modal damping ratio.

7.3.33 Spring Curve Definition

Enter a table of force as a function of displacement or moment as a function of rotation for the spring. The load curve is then displayed as a graph.

7.3.34 Spring Matrix Definition

Name

The name is used to identify a particular matrix.

Matrix

Enter the upper triangle of the stiffness matrix. The units for each quadrant are different

- top left - force per unit length
- top right / bottom length - force
- bottom right - moment

7.3.35 Non-linear Spring Curve Definition

The non-linear spring curve definition dialog is used to define or modify a non-linear spring curve. This dialog can be activated from Non-linear Spring Curve table using the wizard button.

Name

The name is used only as a label for the non-linear spring curve.

Axis Value Type

This defines the dimension of the non-linear spring curve and the available types are:

- Force - Displacement: the spring is translational
- Moment - Rotation: the spring is rotational

Data Table

The data table is used to define the data point on the non-linear spring curve, it must be in ascending order.

Diagram

The diagram shows the curve of the non-linear spring defined in the data table.

7.3.36 Mass Definition

The mass definition dialog is used to define or modify a mass property. This dialog can be activated from [Mass Properties table view](#) using the wizard button or from a Graphic View by selecting a mass element and then "Edit Property" from the [right-click menu](#). The units are displayed in the top right corner of the dialog.

Name

The name is used only as a label for the mass property.

Axis

The axis in which the properties are defined.

Mass

The mass of the element.

Inertia

The inertia of the element is defined by a symmetric tensor. As the tensor is symmetric only one half needs to be defined. Care should be taken if off-diagonal terms are specified to ensure that the principal inertias are positive.

Modifiers

The modifiers allow different masses to be considered in different directions. These can be specified as a mass value or as proportion of the actual mass. This only applies to mass and in dynamic analysis.

7.3.37 2D Property Wizard

The 2D property wizard is used to define or modify a 2D element property. This wizard can be activated from the [2D Properties table view](#) using the wizard button or from a Graphic View by selecting a 2D element and then "Edit Property" from the [right-click menu](#).

More:

[2D Property Wizard : Property Type](#)

[2D Property Wizard : Properties](#)**7.3.37.1 2D Property Wizard : Property Type**

The type is where the basic type of property is selected. The units are displayed in the top right corner of the dialog.

Name

The name is used only as a label for the 2D element property.

Type

Depending on the structure type different options are available for the type.

For Plane Stress, Plane Strain and Axisymmetric structures the element type is defined by the structure type so there is no choice. In other cases choose the type that represents the required behaviour of the element. A plate element is one that has only bending stiffness while a shell element has both in-plane and bending stiffness. A wall element is a higher level 2D element that can be used to represent both in-plane and bending effects and can span over a storey without need for subdivision.

A load panel element is a non-structural element that is only used to apply load to frame type structures.

7.3.37.2 2D Property Wizard : Properties**Axis**

The axis in which the properties are defined. The axis set is [projected](#) on to the element as required.

Material

The material associated with this element property. All the materials that have been defined are given but the user may also enter a number. If the property type is fabric this must refer to a fabric material.

Thickness

The thickness is assumed constant over the element. For Plane Strain, Axisymmetric or Fabric property types this is not relevant so it is not enabled.

This is used when determining the element in-plane stiffness and mass.

Thickness factors

There are three stiffness factors - bending, in-plane and mass/weight. The bending and in-plane thicknesses are used to determine the stiffness of the element may be specified separate from the actual thickness of the element. The mass/weight thickness is used to determine the mass of the element of the weight in self-weight calculations. These allow for modelling situations such as cracked concrete slabs (reduced bending thickness to allow for cracking) or hollow slabs where the stiffness of the slab in bending and in-plane will be less than that for a solid slab.

The thickness can be specified as a factor on the actual thickness (expressed as a percentage) or as an actual thickness value. For Plane Strain, Axisymmetric or Fabric property types this is not relevant so it is not enabled. The bending thickness is also used for transverse shear stiffness.

The stresses and forces in 2D elements are based on the modified thickness.

Added Mass

In some cases it is useful to add some extra mass to the element. For Plane Strain or Axisymmetric property types this is not relevant and so it is not enabled. This is in addition to any mass from the element thickness and density. The value specified is the mass per unit area.

Modify for hollow slab

This allows simpler definition of effective thicknesses for hollow slabs based on the material concentrated in zones at the top and bottom surfaces.

7.3.37.3 2D Property Wizard : Properties (Load Panel)

Support type

The type of support provided for the load.

Reference edge

The reference edge along with the support type determines which of the edges will be loaded. The reference edge is not required if all edges are to be loaded.

7.3.38 Link Property Definition

The link property definition dialog is used to define or modify a link property. This dialog can be activated from [Link Properties table view](#) using the wizard button or from a Graphic View by selecting a link and then “Edit Property” from the [right-click menu](#).

Name

The name is used only as a label for the link property.

Type

This specifies how the link is to act.

- **All**—all directions are linked, rotations at slave nodes are linked to rotations of masters.
- **XY Plane**—the directions are linked to give rigidity in the xy plane but there is not constraint out of plane.
- **YZ Plane**—as XY Plane but for the yz Plane.
- **ZX Plane**—as XY Plane but for the zx Plane.
- **Pin All**—as All but the rotations of the slave nodes are not linked to the master.
- **Pin XY Plane**—as XY Plane but the rotations of the slave nodes are not linked to the master.
- **Pin YZ Plane**—as Pin XY Plane but for the yz Plane.
- **Pin ZX Plane**—as Pin XY Plane but for the zx Plane.
- **Tension**—the link can rotate and compress but not extend.
- **Compression**—the link can rotate and extend but not compress.
- **Bar**—the link can rotate but not extend or compress.
- **Custom**—the link acts like a joint in the specified coupled directions.

7.3.39 Cable Definition

The cable definition dialog is used to define or modify a cable property. This dialog can be activated from the [Cable Properties table view](#) using the wizard button or from a Graphic View by selecting a cable element and then “Edit Property” from the [right-click menu](#). The units are displayed in the top right corner of the dialog.

Name

The name is used only as a label for the cable property.

Stiffness

The stiffness of the cable. This is defined as

$$k = A \cdot E$$

Mass per unit length

The mass to associate with the cable for gravity load, specified as mass per unit length.

Temperature coefficient

The temperature coefficient of expansion. Used in the same as the temperature coefficient of expansion for beam and bar elements.

Damping Ratio

The damping ratio is used during a dynamic analysis to calculate an estimate of the modal damping ratio.

7.3.40 Spacer Definition

The spacer definition dialog is used to define or modify a spacer property. This dialog can be activated from the [Spacer Properties table view](#) using the wizard button or from a Graphic View by selecting a spacer element and then “Edit Property” from the right-click menu. Details of the data is in the “[Program Data](#)” section. The units are displayed in the top right corner of the dialog.

Name

The name is used only as a label for the spacer property.

Axis

The axis used when the “spacer leg length type” is “Projected ratio”.

Type

Spacer types can be Geodesic, Free or Bar.

Spacer leg length type

The spacer leg length types can be Proportional, Ratio, XY plane projected ratio, X axis projected ratio and XY plane projected length proportional original projected length.

Stiffness

The equivalent stiffness if the spacer is considered as a tie element.

Leg length ratio

Used for Ratio of Projected Ratio leg length types.

7.3.41 Steel Beam Design Property Definition

The steel beam design property definition dialog is used to define or modify the beam design properties for members. This dialog can be activated from [Beam Design Properties table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view. The units are displayed in the top right corner of the dialog.

Name

The name is used only as a label for the beam design property.

Effective lengths (L_{yy} , L_{zz} , L_{it})

These are the effective lengths to use in member checks. The effective lengths are calculated from the model data using the actual lengths and the restraint property information, but the user may choose to override them. There are values for bending about the major and minor axis axial buckling

(L_{yy} and L_{zz} respectively) and a value for lateral-torsional effects (L_{lt}). The values can be expressed either as a length or as a percentage of the member length.

These values can only be set if the Override Calculated Effective Lengths check box is set, otherwise, the member's effective lengths will be calculated from the member's restraint property information.

Maximum plastic:elastic ratio

This represents the maximum plastic bending capacity that can be used as a multiple of the elastic bending capacity ($p_y Z$). This limit is specified by some codes in order to limit the effects of plasticity at the working load (which could include excessive deflections, alternating plasticity, and incremental collapse).

Net : gross area ratio

The net : gross area ratio is used to define the reduction in area for tension due to holes in the member. For welded structures this value will be 1.

Shear lag factor (Beta)

Shear lag factor (Beta) is used to compute the design tensile strength of the member. This represents the shear lag effect of the member.

Temperature

The temperature is used in 'EN 1993-1-2:2005 Eurocode 3 Fire (UK)' design code only.

Exposure Type

The exposure type is used to define the adaptation factors defined 'EN 1993-1-2:2005 Eurocode 3 Fire (UK)' design code. The following exposure types can be defined.

- **All sides:** All four sides of the member are exposed to fire.
- **3 sides protected:** Protected member exposed on three sides.
- **3 Sides unprotected:** Unprotected member exposed on three sides.

7.3.42 Steel Restraint Property Definition

The steel restraint property definition dialog/wizard is used to define or modify a [Restraint Property](#). This dialog can be accessed from the Restraint properties table view using the wizard button, or right clicking on a member and selecting the "Edit | Restraint Property" pop-up menu option.

Name

The name is used only as a label for the beam design property.

Table of node and span restraints

This table lists the member restraints that are assigned to locations along a member. Click on a row in the left hand column to select the row, and copy the rows data down to the Edit section below. Double click instead to immediately edit the member restraint for that row using the [Member Restraint Dialog](#).

Edit

This section allows editing of a single row. You can select the position of a restraint using a number indicating the node or span position, "end1", "end2" or "all". The node or span radio buttons enable selection of either the specified point or span to be restrained. Press the **Edit** button to edit the Member Restraint, or type the mnemonic code in directly. An invalid entry will cause help on the correct syntax to be displayed. The changes you have made to are not incorporated until you press the **Update List** button.

Common Restraint Properties

This group of buttons allows the rapid set-up of frequently used restraint properties. Once used, the resulting settings can be customised.

7.3.43 Member Restraint

This dialog assists you in setting up [Member Restraints](#), which make up [Restraint properties](#).

Shortcuts

This group of buttons allow rapid set-up of commonly used member restraints. Once used, the resulting settings can be customized.

Preview

Pressing this button will display a message box that describes the current settings effects in more natural language.

Rotational Restraints, Translational Restraints

These groups of combo-boxes show the current restraint settings and allow you to change them.

Load Height

The load height can be set to indicate destabilizing or stabilizing loading conditions. A positive load height indicates destabilizing loading.

Connection

Some section shapes and design codes may need extra information about connections in order to have the restraint fully defined. See the documentation for the relevant code checker.

Desc

This box shows the mnemonic for the current restraint settings. Watch it change as you change the settings. It takes account of any implied restraint settings: for example, lateral restraint to both the top and bottom flanges implies torsional restraint. You cannot change the mnemonic directly.

Advanced

This check box enables/disables the [advanced settings](#) possible in member restraints.

7.3.44 RC Member Design Properties

The RC Member Design Properties dialog is used to define or modify the beam design properties for slabs. This dialog can be activated from the [RC Beam Design Properties table view](#) by giving the "Edit | Wizard" menu command. "Wizard" is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Note that RC Member Design Properties are currently only used in the options to Export to ADC and to Export to AdSec.

Basic Properties

Name

The name is used only as a label for the member design property.

Type

A property can apply to either a beam or column.

Materials

The material specification includes the **concrete grade** and the **reinforcement grade**, which are used in the design process. Also specified are the **link** (or tie or stirrup) **grade** and **link diameter** and the **aggregate** size as these affect the position of bars in the section.

Cover

The minimum cover to the reinforcement either specified as a uniform cover or a cover to each face. The final cover to the reinforcement will be a function of this cover, the links, the main reinforcement and the aggregate size.

Bar layout

These are the templates from which actual bar arrangements are derived.

7.3.45 RC Slab Design Properties

[RC Slab Design Properties: Basic Properties](#)

[RC Slab Design Properties: Concrete](#)

[RC Slab Design Properties: Reinforcement](#)

[RC Slab Design Properties: Analysis](#)

7.3.45.1 RC Slab Design Properties: Basic Properties

The Basic Properties page of the RC Slab design properties dialog is used to define or modify the basic design properties for slabs. The dialog can be activated from the [RC Slab Design Properties table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

The RC Slab design properties dialog contains two other property pages: [Concrete](#) and [Reinforcement](#).

Basic Properties**Name**

The name is used only as a label for the slab design property.

Slab Thickness

Enabled only when the **Override Analysis Thickness** flag is yes. In this circumstance, this thickness will be used as the slab thickness in the design.

Minimum Eccentricity

Minimum eccentricity to be used in the design, it is normally taken from the relevant design code, but it can be overridden and defined explicitly.

Additional Eccentricity

Additional eccentricity is used to consider the second order effect (increase) on the applied moment due to the slenderness of the slab. It is an user defined parameter to incorporate the second order moment if the slab is slender.

Material Grades – Concrete

The grade of concrete. The standard grades of concrete that are available for the specified RC Slab design code are available in this droplist. A “User Defined” option is also available. If this option is chosen, then direct specification of a non-standard concrete strength may be made in the [RC Slab Design Properties: Concrete property page](#).

If the 'Modify code parameters' option of the concrete properties has been checked, then the concrete grade will display 'User Defined'. Changing the grade from this setting to a standard grade will cause the 'Modify code parameters' option to become un-checked. The concrete parameters will then be derived from the concrete strength, as defined by the design code. A warning will be issued before this action is taken.

Material Grades – Reinforcement

The grade of reinforcement. The standard grades of reinforcement that are available for the specified RC Slab design code are available in this droplist. A “User Defined” option is also available. If this option is chosen, then direct specification of a non-standard reinforcement strength may be made in the [RC Slab Design Properties : Reinforcement property page](#).

If the 'Modify code parameters' option of the reinforcement properties has been checked, then the reinforcement grade will display 'User Defined'. Changing the grade from this setting to a standard grade will cause the 'Modify code parameters' option to become un-checked. The reinforcement parameters will then be derived from the reinforcement strength, as defined by the design code. A warning will be issued before this action is taken.

Direction of Reinforcement w.r.t. Local X Axes – A, B

The angle, in degrees, between reinforcements and the x direction of the element local axes.

Axis-to-Surface Distance of Reinforcement – A, B – Top, Bottom

The distance from the centre of the reinforcements to the surface of the concrete slab.

Minimum Areas of Reinforcement – A, B – Top, Bottom

The minimum area of reinforcement to be provided in the top or bottom face, and in directions A and B.

Strength Reduction Factors – Axial Compression, Axial Tension and Bending

Applied forces and moments are divided by the strength reduction factors to obtain design values for use within RC Slab. If the applied axial principal compressive stress exceeds 1% of the uncracked concrete strength, f_{cd1} , then the lower values are used.

The availability of these items is dependent on the chosen RC Slab design code.

Override Analysis Thickness

Flag to indicate whether to use the **2D Element Property Thickness** or to override this with the **Slab Thickness**.

Override Minimum Eccentricity

Flag to indicate whether to use code specified minimum eccentricity or to override this with an explicitly defined **Minimum Eccentricity**.

Override Minimum Compression Reinforcement

Flag to indicate whether to use code specified minimum compression reinforcement or to override this with an user defined **Minimum Compression Reinforcement**.

Modify Strength Reduction Factors

Flag to indicate whether to use the default Strength Reduction Factors as specified in the design code, or to override one or both of them.

The availability of this item is dependent on the chosen RC Slab design code.

7.3.45.2 RC Slab Design Properties: Concrete

The Concrete page of the RC Slab design properties dialog is used to define or modify the concrete design properties for slabs. The dialog can be activated from the [RC Slab Design Properties table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

The RC Slab design properties dialog contains two other property pages: [Basic Properties](#) and [Reinforcement](#).

Design Code

The chosen RC Slab design code.

Strength

The concrete strength. If a standard concrete grade has been chosen in the [Basic Properties page](#) of the RC Slab Design Properties dialog, then the concrete strength will be derived from this grade, and this edit box will be disabled.

Partial Safety Factor

The partial safety factor for concrete. If the **Modify Code Partial Safety Factor** check box is unchecked, then this value is as specified by the chosen RC Slab design code.

This availability of this item is dependent on the chosen RC Slab design code.

Modify Code Partial Safety Factor

Check this box to override the partial safety factor for concrete that is specified by the chosen RC Slab design code.

This availability of this item is dependent on the chosen RC Slab design code.

Modify Code Parameters

Check this box to set the concrete grade to 'user defined' and to override the following values for concrete that are otherwise derived by the chosen RC Slab design code:

- Uncracked Design Strength
- Cracked Design Strength
- Tensile Design Strength
- Compressive Plateau Strain
- Maximum Axial Compressive Strain
- Maximum Flexural Compressive Strain
- Minimum ratio of depth to neutral axis to effective depth in flexural situations
- Maximum ratio of depth to neutral axis to effective depth in flexural situations
- Proportion of depth to neutral axis over which constant stress acts

Checking this box will result in the concrete grade being shown as 'User Defined' in the Basic Properties page and in the RC Slab Table View.

7.3.45.3 RC Slab Design Properties: Reinforcement

The Reinforcement page of the RC Slab design properties dialog is used to define or modify the reinforcement design properties for slabs. The dialog can be activated from the [RC Slab Design Properties table view](#) by giving the "Edit | Wizard" menu command. "Wizard" is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

The RC Slab design properties dialog contains two other property pages: [Basic Properties](#) and [Concrete](#).

Design Code

The chosen RC Slab design code.

Strength

The reinforcement strength. If a standard reinforcement grade has been chosen in the [Basic Properties page](#) of the RC Slab Design Properties dialog, then the reinforcement strength will be derived from this grade, and this edit box will be disabled.

Partial Safety Factor

The partial safety factor for reinforcement. If the **Modify Code Partial Safety Factor** check box is

unchecked, then this value is as specified by the chosen RC Slab design code.

This availability of this item is dependent on the chosen RC Slab design code.

Modify Code Partial Safety Factor

Check this box to override the partial safety factor for reinforcement that is specified by the chosen RC Slab design code.

This availability of this item is dependent on the chosen RC Slab design code.

Modify Code Parameters

Check this box to override the following values for reinforcement that are otherwise derived by the chosen RC Slab design code:

- Design Strength in Tension
- Design Strength in Compression
- Elastic Modulus
- Maximum Linear Stress
- Yield Strain in Tension
- Yield Strain in Compression

Checking this box will result in the reinforcement grade being shown as 'User Defined' in the Basic Properties page and in the RC Slab Table View.

7.3.45.4 RC Slab Design Properties: Analysis

The information below presents a practical overview of the two methods. More detail on the [RC Slab analysis procedure](#) is available in the [RC Slab theory](#) section.

Two solution methods are available; a 'fast' method and a 'rigorous' method.

Fast method

The 'fast' method selects a faster non-iterative solution method wherever the loading is considered to be either predominately in-plane or predominately out-of-plane. For each element position where this be a reasonable assumption, this method will yield faster analysis times. When the loading is not considered to be predominately in-plane or out-of-plane, the method will fall-back to the slower iterative technique. The decision is made independently for each element position, on a position by position basis.

The user may also restrict use of the non-iterative technique to any case where the total reinforcement area is below a certain percentage of the total cross-sectional area. Should the total reinforcement calculated by the 'fast' method be above this threshold, the iterative technique is used.

Any use of the non-iterative technique will typically lead to slightly more conservative results. Note that by design, its use is limited to those cases that represent the most favourable trade-offs between analysis time and approximation accuracy.

More detail on the [non-iterative technique](#) is available in the RC Slab theory.

Rigorous method

The 'rigorous' method selects the iterative solution technique in all cases. This approach will always yield the most accurate result available, regardless of any opportunity for a performance/approximation trade-off.

A common work-flow scenario may be one that adopts the 'fast' method during the early stages of design, before later switching exclusively to the 'rigorous' approach.

Note that the 'rigorous' method is equivalent to the (sole) solution technique prior to GSA version 8.7.

7.3.46 RC Member Bar Layouts

[RC Member Bar Layouts: Definition](#)

[RC Member Bar Layouts: Beams](#)

[RC Member Bar Layouts: Rectangular Columns](#)

[RC Member Bar Layouts: Circular/Elliptical Columns](#)

[RC Member Bar Layouts: Perimeter](#)

7.3.46.1 RC Member Bar Layouts: Definition

The Definition page of the RC Member Bar Layouts dialog is used to define the type of member for which the bar layout is to be specified. The dialog can be activated from the [RC Member Bar Layouts table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Note that RC Member Bar Layouts are currently only used in the options to Export to ADC and to Export to AdSec.

7.3.46.2 RC Member Bar Layouts: Beams

The Beams page of the RC Member Bar Layouts dialog is used to define the bar layout for RC beams. The dialog can be activated from the [RC Member Bar Layouts table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Note that RC Member Bar Layouts are currently only used in the options to Export to ADC and to Export to AdSec.

Specify the reinforcement layout at the top and bottom of the beam in terms of bar sizes, number of layers and number of bars per layer.

7.3.46.3 RC Member Bar Layouts: Rectangular Columns

The Rectangular Columns page of the RC Member Bar Layouts dialog is used to define the bar layout for rectangular RC columns. The dialog can be activated from the [RC Member Bar Layouts table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Note that RC Member Bar Layouts are currently only used in the options to Export to ADC and to Export to AdSec.

Specify the reinforcement layout for the column in terms of bar size and layout.

7.3.46.4 RC Member Bar Layouts: Circular/Elliptical Columns

The Circular/Elliptical Columns page of the RC Member Bar Layouts dialog is used to define the bar layout for circular and elliptical RC columns. The dialog can be activated from the [RC Member Bar Layouts table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Note that RC Member Bar Layouts are currently only used in the options to Export to ADC and to Export to AdSec.

Specify the reinforcement layout for the column in terms of bar sizes, number of rings and number of bars per ring.

7.3.46.5 RC Member Bar Layouts: Perimeter

The Perimeter page of the RC Member Bar Layouts dialog is used to define the bar layout for RC members defined by perimeter. The dialog can be activated from the [RC Member Bar Layouts table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Note that RC Member Bar Layouts are currently only used in the options to Export to ADC and to Export to AdSec.

Specify the reinforcement layout around the section in terms of bar size, number of layers and either the number of bars per layer or the spacing of bars per layer.

7.3.47 Generalised Restraint Definition

The generalised restraint definition dialog is used to define or modify a generalised restraint. It has no effect on nodal restraints. This dialog can be activated from the [Generalised Restraints table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Name

The name is only used as a convenient way of identifying a restraint.

Stage

If the analysis stage feature is enabled the stage to which this generalised restraint applies can be selected from the list, or a numeric value entered directly.

Node List

The node list specifies the nodes that are to be constrained by this generalised restraint.

Constraints

The [constraints](#) define if the node is fixed in particular directions. Directions that do not apply for that structure type are disabled. The restraint types Free, Pin and Encastré options provide a shortcut to setting the flags for the individual degrees of freedom. The symmetry options in the xy plane, yz plane and zx plane provide shortcuts for applying the boundary conditions corresponding to symmetry in these planes. The symmetry options do not unset other restraint conditions.

7.3.48 Rigid Constraint Definition

The rigid constraint definition dialog is used to define or modify a rigid constraint. This dialog can be activated from the [Rigid Constraints table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Name

The name is only used as a convenient way of identifying a rigid constraint.

Stage

If the analysis stage feature is enabled the stage to which this rigid constraint applied can be selected from the list, or a numeric value entered directly.

Node List

The node list specifies the nodes that are to be constrained by this rigid constraint.

Master

The master node is the node that is retained in the solution and from which the behaviour of the slave nodes is calculated. The master retains all its degrees of freedom. This node may also be included in the node list.

Linkage Type

This should be set to "All" to make the constraint completely rigid. There are however many cases in which it is useful to make a constraint rigid in a plane (for example in modelling floor slabs) or a plate. The following planes can be specified:

Linkage	Linked degrees of freedom
all	x, y, z, xx, yy, zz
xy plane	x, y, zz
yz plane	y, z, xx
zx plane	x, y, zz
xy plate	z, xx, yy
yz plate	x, yy, zz
zx plate	y, xx, zz
custom	any

The "pin" linkage types do not include the rotational degrees of freedom at the slave node in the rigid body displacements. Note that the "pin" condition applies only to the slave node - the rotational degrees of freedom at the master are always retained. The degrees of freedom linked are always in the constraint axis of the nodes and all nodes must be assigned the same constraint axis.

Coupled directions

For most constraint types the couple directions are fixed, but for custom links they must be specified explicitly. This can lead to behaviour that is not consistent with a 'rigid' condition.

7.3.49 Joint Definition

The joint definition dialog is used to define or modify a joint. This dialog can be activated from the [Joints table view](#) by giving the "Edit | Wizard" menu command. "Wizard" is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Name

The name is only used as a convenient way of identifying a joint.

Stage

If the analysis stage feature is enabled the stage to which this joint applies can be selected from the list, or a numeric value entered directly.

Slave and Master

The slave node is tied to the master node in particular directions. Having no independent displacement or rotation. The master retains all its degrees of freedom.

Coupled directions and Special joints

These are the directions in which the slave node is coupled to the master node. Depending on which combination of these is selected a range of joint types can be specified. The special joint options allow a simple way of setting up pin, hinge, bearing and universal joints.

7.3.50 Constraint Equation Definition

The constraint equation definition dialog is used to define or modify a constraint equation. This dialog can be activated from the [Constraint equations table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Name

The name is only used as a convenient way of identifying a constraint equation.

Stage

If the analysis stage feature is enabled the stage this constraint equation applies to can be selected from the list, or a numeric value entered directly.

Slave node and Direction

The slave node is tied to the master nodes in particular directions. Having no independent displacement or rotation. The master retains all its degrees of freedom.

These are the directions in which the slave node is coupled to the master node. Depending on which combination of these is selected a range of joint types can be specified. The special joint options allow a simple way of setting up pin, hinge, bearing and universal joints.

Masters – Factor, Node and Direction

The displacement of the slave node is the summation of the factored displacements at the masters in the specified directions. Care should be taken when mixing translational and rotational freedom.

7.3.51 Tied Interfaces

The tied interface definition dialog is used to define or modify a [tied interface](#). This dialog can be activated from the [Tied interface table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Name

The name is only used as a convenient way of identifying a tied interface.

Stage

If the analysis stage feature is enabled the stage this tied interface applies to can be selected from the list, or a numeric value entered directly.

Slave type

The slave side of the tied interface can be defined either as nodes or elements. When elements are selected GSA will find the nodes on the slave side that need to be connected.

Slave list and master list

The nodes in the slave list are tied to elements in the master list to form a continuous entity. The master retains all its degrees of freedom. The master elements must be 2D elements, but the slave elements can be of any element type (although link elements should be avoided as these will give rise to conflicting constraints). So for example a beam element can be connected to the face of a 2D elements.

Type of Linkage

There are two options for the type of linkage if the slave type is “nodes”:

- **Auto** – the tied interface behaves as tied if the slave type is element and rigid if the slave type is node.
- **Tied** – the tied interface connects the node on the slave side to the element faces on the master side using constraint equations.

- **Rigid** – the tied interface take the node on the slave side as a master and creates a rigid constraint making the element on the master side rigidly connected with the node.

The third of these options is intended for situations where the node represent a line entity such as a column that is to be attached to a slab. In this case, if the beam element cross section is a similar size to the 2D element on the other face the effect of the connection will be to make the 2D element behave in a manner close to being rigid. This option will not work where there are nodes to connect to the same element face or adjacent element faces.

The rigid linkage cannot be used when the slave type is element as this would constraint conflict.

Surface tolerance

This is used to limit the nodes that are connected to only those along the edge. Nodes outside this tolerance are excluded from the tied interface.

7.3.52 Node Loading Definition

This dialog is used to define or modify a node load, applied displacement or settlement, depending on the context in which it is opened. This dialog can be activated from the [Node Loads table view](#), [Applied Displacements table view](#) or [Settlements table view](#), respectively, by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

This dialog box is also used in the [sculpt commands that create node loading](#). In this circumstance the operation of the dialog box differs from when it is used as a definition dialog box. These differences are identified below.

Name

The name is only used as a convenient way of identifying a load.

Node list

The list of nodes to which the load is to be applied.

In sculpt: The node list is set to the current selection set. The node list cannot be edited in the dialog.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Load title

The loading is stored by case number. However it is good practice to give load cases names. The load title gives access to the Load Case Titles wizard to simplify this procedure. The wizard is opened for the currently displayed load case.

Direction

The direction in which the load is applied, with respect to the specified axis.

Axis

The direction of the load is with respect to this axis set. Settlements are applied at restrained nodes and so these are always in the node local axis system.

Value

The magnitude of the load.

7.3.53 Beam Load Definition

This dialog is used to define or modify a beam load. This dialog can be activated from the [Beam Loads table view](#), by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

This dialog box is also used in the [sculpt commands that create beam loading](#). In this circumstance the operation of the dialog box differs from when it is used as a definition dialog box. These differences are identified below.

Name

The name is only used as a convenient way of identifying a load.

Beam list

The list of elements to which the load is to be applied.

In sculpt: The beam list is set to the beam and bar elements in the current selection set. The beam list cannot be edited in the dialog.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Load title

The loading is stored by case number. However it is good practice to give load cases names. The load title gives access to the Load Case Titles wizard to simplify this procedure. The wizard is opened for the currently displayed load case.

Load type

The user has a choice of the type of loading applied to the elements.

- **Point** – single load at position along the element
- **Uniform** – load applied along the length of the element
- **Linear** – load which varies linearly along the length of the element
- **Patch** – a load applied to a section of the element with linear variation between the end points
- **Tri-linear** – similar to a patch load but the load tapers off to zero outside the patch

Direction

The direction in which the load is applied, with respect to the specified axis.

Axis

The direction of the load is with respect to this axis set. Setting this to Local implies element local axes and Deformed local implies a element local axis that is embedded in the element as it deforms (GsRelax only).

Projected

When set, the load is projected from onto the element to result in the magnitude of the load actually applied to the element being the specified magnitude factored by an amount dependent on the inclination of the element to the load direction. (E.g. for snow loading.)

Value

The magnitude of the load.

Position

The position associated with the load (in the case of point, patch and tri-linear loads). This can be specified as an absolute position from end 1 of the element (e.g. 1.25) or a relative position by specifying the value as a percentage (e.g. 20%).

In sculpt: When the position of a patch load is specified, in absolute terms, to extend beyond the end of the element(s) the load is automatically “broken up” and applied to elements positioned to take the load. A warning is given if such elements cannot be found.

7.3.54 Beam Pre-stress Definition

This dialog is used to define or modify a beam prestress. This dialog can be activated from the [Beam Prestress table view](#), by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

This dialog box is also used in the [sculpt commands that create beam loading](#). In this circumstance the operation of the dialog box differs from when it is used as a definition dialog box. These differences are identified below.

Name

The name is only used as a convenient way of identifying a load.

Beam list

The list of elements to which the load is to be applied.

In sculpt: The beam list is set to the beam and bar elements in the current selection set. The beam list cannot be edited in the dialog.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Load title

The loading is stored by case number. However it is good practice to give load cases names. The load title gives access to the Load Case Titles wizard to simplify this procedure. The wizard is opened for the currently displayed load case.

Load type

The user has a choice of the type of loading applied to the elements.

- **Pre-stress force** – an axial force applied to the element
- **Initial strain** – an initial axial strain is specified in the element
- **Lack of fit** – this is similar to an initial strain but the change in length of the element is specified
- **Tendon prestress** – this is similar to the pre-stress force but the loading is assumed to come from a pre-stressing tendon rather than a stress locked into the element

Value

The magnitude of the prestress is specified as appropriate for the load type.

Offset

Where the prestress is due to a tendon this is normally located remote from the centroid of the section. The offset allows GSA to take account of the moment effects as well as the axial load effects.

7.3.55 Beam Distortion Definition

This dialog is used to define or modify a beam distortion. This dialog can be activated from the [Beam Distortion table view](#), by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

This dialog box is also used in the [sculpt commands that create beam loading](#). In this circumstance

the operation of the dialog box differs from when it is used as a definition dialog box. These differences are identified below.

Name

The name is only used as a convenient way of identifying a load.

Beam list

The list of elements to which the load is to be applied.

In sculpt: The beam list is set to the beam and bar elements in the current selection set. The beam list cannot be edited in the dialog.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Load title

The loading is stored by case number. However it is good practice to give load cases names. The load title gives access to the Load Case Titles wizard to simplify this procedure. The wizard is opened for the currently displayed load case.

Direction

The direction in which the load is applied, with respect to the specified axis.

Distortion

The magnitude of the distortion applied to the element – either a displacement or rotation.

Position

The position at which the distortion is applied. This can be specified as an absolute position from end 1 of the element (e.g. 1.25) or a relative position by specifying the value as a percentage (e.g. 20%).

7.3.56 Beam Thermal Load Definition

This dialog is used to define or modify a beam thermal load. This dialog can be activated from the [Beam Thermal Loads table view](#), by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

This dialog box is also used in the [sculpt commands that create beam loading](#). In this circumstance the operation of the dialog box differs from when it is used as a definition dialog box. These differences are identified below.

Name

The name is only used as a convenient way of identifying a load.

Beam list

The list of elements to which the load is to be applied.

In sculpt: The beam list is set to the beam and bar elements in the current selection set. The beam list cannot be edited in the dialog.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Load title

The loading is stored by case number. However it is good practice to give load cases names. The load title gives access to the Load Case Titles wizard to simplify this procedure. The wizard is

opened for the currently displayed load case.

Load type

The user has a choice of the type of loading applied to the elements.

- **Uniform** – a constant temperature rise applied to the whole element
- **Gradient in y** – a temperature gradient is applied in the element y direction
- **Gradient in z** – a temperature gradient is applied in the element z direction

Uniform temperature change

The change in temperature applied to the whole element.

Location of surface and surface temperature

A temperature gradient is applied to the element by specifying two locations and the temperature rise at these locations. From these two value pairs GSA calculates a temperature gradient and the average temperature rise in the element as a whole.

7.3.57 2D Element Face Load Definition

This dialog is used to define or modify a 2D element face load. This dialog can be activated from the [2D Element Face Loads table view](#), by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

This dialog box is also used in the [sculpt commands that create 2D element loading](#). In this circumstance the operation of the dialog box differs from when it is used as a definition dialog box. These differences are identified below.

Name

The name is only used as a convenient way of identifying a load.

2D element list

The list of elements to which the load is to be applied.

In sculpt: The 2D element list is set to the 2D elements in the current selection set. The 2D element list cannot be edited in the dialog.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Load title

The loading is stored by case number. However it is good practice to give load cases names. The load title gives access to the Load Case Titles wizard to simplify this procedure. The wizard is opened for the currently displayed load case.

Load type

The user has a choice of the type of loading applied to the elements.

- **Uniform** – uniform load applied over the whole face of the element
- **Variable** – load which varies across the face of the elements interpolating between values specified at the corner nodes

If the elements to which this relates are load panels then the load type must be uniform.

Direction

The direction in which the load is applied, with respect to the specified axis.

Axis

The direction of the load is with respect to this axis set. Setting this to Local implies element local axes.

Projected

When set, the load is projected from onto the element to result in the magnitude of the load actually applied to the element being the specified magnitude factored by an amount dependent on the inclination of the element to the load direction. (E.g. for snow loading.)

Value

The magnitude of the load specified at the corner nodes of the element.

7.3.58 2D Element Edge Load Definition

This dialog is used to define or modify a 2D element edge load. This dialog can be activated from the [2D Element Edge Loads table view](#), by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

This dialog box is also used in the [sculpt commands that create 2D element loading](#). In this circumstance the operation of the dialog box differs from when it is used as a definition dialog box. These differences are identified below.

Name

The name is only used as a convenient way of identifying a load.

2D element list

The list of elements to which the load is to be applied.

In sculpt: The 2D element list is set to the 2D elements in the current selection set. The 2D element list cannot be edited in the dialog.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Load title

The loading is stored by case number. However it is good practice to give load cases names. The load title gives access to the Load Case Titles wizard to simplify this procedure. The wizard is opened for the currently displayed load case.

Edge

This allows the edge to be loaded to be selected. The edges are identified in the same order as the corner nodes of the elements.

Direction

The direction in which the load is applied, with respect to the specified axis.

Axis

The direction of the load is with respect to this axis set. Setting this to Local implies element local axes.

Value

The magnitude of the load specified at the ends of the edge of the element. Depending on the edge load option set in the Analysis Specification this can be either a pressure or a force per unit length.

7.3.59 2D Element Pre-stress Load Definition

This dialog is used to define or modify a 2D element pre-stress load. This dialog can be activated from the [2D Element Pre-stress Loads table view](#), by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

This dialog box is also used in the [sculpt commands that create 2D element loading](#). In this circumstance the operation of the dialog box differs from when it is used as a definition dialog box. These differences are identified below.

Name

The name is only used as a convenient way of identifying a load.

2D element list

The list of elements to which the load is to be applied.

In sculpt: The 2D element list is set to the 2D elements in the current selection set. The 2D element list cannot be edited in the dialog.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Load title

The loading is stored by case number. However it is good practice to give load cases names. The load title gives access to the Load Case Titles wizard to simplify this procedure. The wizard is opened for the currently displayed load case.

Direction

The direction in which the load is applied. This is either x, y or both x and y directions where x and y are element local directions.

Load type

The user has a choice of the type of loading applied to the elements.

- **Pre-stress force** – an in-plane force applied to the element
- **Initial strain** – an initial in-plane strain is specified in the element
- **Tendon pre-stress** – an in-plane force is applied to the element at an offset from the neutral axis.

Value

The magnitude of the pre-stress or strain is specified as appropriate for the load type.

Offset

The prestress can be located remote from the neutral axis of the section. The offset allows GSA to take account of the moment effects as well as the in-plane load effects.

7.3.60 2D Element Thermal Load Definition

This dialog is used to define or modify a 2D element thermal load. This dialog can be activated from the [2D Element Thermal Loads table view](#), by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

This dialog box is also used in the [sculpt commands that create 2D element loading](#). In this circumstance the operation of the dialog box differs from when it is used as a definition dialog box. These differences are identified below.

Name

The name is only used as a convenient way of identifying a load.

2D element list

The list of elements to which the load is to be applied.

In sculpt: The 2D element list is set to the 2D elements in the current selection set. The 2D element list cannot be edited in the dialog.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Load title

The loading is stored by case number. However it is good practice to give load cases names. The load title gives access to the Load Case Titles wizard to simplify this procedure. The wizard is opened for the currently displayed load case.

Load type

The user has a choice of the type of loading applied to the elements.

- **Uniform** – uniform load applied over the whole of the element
- **Gradient in local z** – load which varies through the thickness of the elements, but not across the surface of the element
- **General** – load which varies over the whole element

Temperature

The magnitude of the temperature change as required by the load type.

7.3.61 Grid Loading Definition

This one dialog box is used to define or modify a grid point load, grid line load or grid area load, depending on the context in which it is opened. This dialog can be activated from the [Grid Point Loads table view](#), [Grid Line Loads table view](#) or [Grid Area Loads table view](#), respectively, by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

This dialog box is also used in the [sculpt commands that create grid loading](#). In this circumstance the operation of the dialog box differs from when it is used as a definition dialog box. These differences are identified below.

Name

The name is only used as a convenient way of identifying a load.

Grid plane

The grid plane to which the grid load is to be applied.

In sculpt: The grid plane is set to the current grid. The grid plane cannot be edited in the dialog.

Grid coordinates

Grid Point Load definition only: The coordinates of the grid point load on the grid plane.

In sculpt: A grid point load will be defined at each point on the current polyline. The grid coordinates cannot be edited in the dialog.

2D Polyline

Grid Line Load definition only: A reference to a 2D Polyline that defines the line of the load on the grid plane or an explicit polyline definition.

Grid Area Load definition only: A reference to a 2D Polyline that bounds the area of the load on the grid plane or an explicit polyline definition. The polyline is assumed to be closed.

In sculpt: An existing 2D Polyline identical to the current polyline is referenced or the current polyline is saved as a 2D Polyline and referenced. The 2D Polyline cannot be edited in the dialog.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Load title

The loading is stored by case number. However it is good practice to give load cases names. The load title gives access to the Load Case Titles wizard to simplify this procedure. The wizard is opened for the currently displayed load case.

Axis

The direction of the load is with respect to this axis set. Setting this to Local implies panel local axes.

Direction

The direction in which the load is applied, with respect to the specified axis.

Projected

Grid Line Load and Grid Area Load definition only: When set, the load is projected from the grid plane onto the panel to result in the magnitude of the load actually applied to the panel being the specified magnitude factored by an amount dependent on the inclination of the panel with respect to the grid plane. (E.g. for snow loading.)

Value

The magnitude of the load.

7.3.62 Gravity Load Definition

This dialog is used to define or modify a gravity load. This dialog can be activated from the [Gravity Loads table view](#), by giving the "Edit | Wizard" menu command. "Wizard" is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

This dialog box is also used in the [sculpt commands that create gravity loading](#).

Name

The name is only used as a convenient way of identifying a load.

Element list

The list of elements to which the load is to be applied.

In sculpt: The element list is set to the elements in the current selection set. The element list cannot be edited in the dialog.

Load case

This is the load case in which the load applies. The load case gives a way of grouping load effects together.

Load title

The loading is stored by case number. However it is good practice to give load cases names. The load title gives access to the Load Case Titles wizard to simplify this procedure. The wizard is opened for the currently displayed load case.

Gravity factors

The gravity loads can be considered as an acceleration applied to the whole model. So the weight of

the structure corresponds to $-1g$. These accelerations can be applied in any direction as specified by the gravity factors.

7.3.63 Response Spectrum Wizard

This wizard can be activated from the [Response Spectra table](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Response spectrum wizard

The wizard takes the user through the following steps.

- Select spectrum type

For code spectra

- Code specific data to define spectrum
- Derived code specific values
- Damping associated with the spectrum
- Spectrum values

For user defined spectra

- Spectrum values

Code defined spectra

The following code defined spectra are

- ASCE 7-05 (USA)
- ASCE 7-10 (USA)
- Eurocode 8 : 1994 (Europe)
- Eurocode 8 : 2003 (draft) (Europe)
- Eurocode 8 : 2004 (Europe)
- FEMA 356 (USA)
- GB 50011-2001 (China)
- GB 50011-2010 (China)
- IBC 2000 (USA)
- IBC 2006 (USA)
- IBC 2009 (USA)
- IS 1893 (Part 1) : 2002 (India)
- Ordinanza PCM 3274 (Italy)
- UBC 1994 (USA)
- UBC 1997 (USA)

Damping

CQC and other double sum combination methods require a damping value to establish the coupling coefficients between modal responses. The default option is constant 5% damping, but this can be changed to code specified damping, modal damping values or a table of damping values.

- No adjustment from

code

- Newmark-Hall : 50th percentile
- Newmark-Hall : 84th percentile
- ISO 19901-2 $\ln(100/\eta)/\ln(20)$
- Eurocode 8 $\sqrt{10/(5+\eta)}$

More:

[Response Spectrum Wizard : Spectrum Type](#)

[Response Spectrum Wizard : Tabulated Spectrum](#)

[Response Spectrum Wizard : UBC 1994 Spectrum](#)

[Response Spectrum Wizard : UBC 1997 Spectrum](#)

[Response Spectrum Wizard : IBC 2000 Spectrum](#)

[Response Spectrum Wizard : ASCE 7-05 / IBC 2006 / IBC 2009 Spectrum](#)

[Response Spectrum Wizard : FEMA 356 Spectrum](#)

[Response Spectrum Wizard : Eurocode 8 : 1994 Spectrum](#)

[Response Spectrum Wizard : Eurocode 8 : 2003 DRAFT Spectrum](#)

[Response Spectrum Wizard : Ordinanza PCM 3274 Spectrum](#)

[Response Spectrum Wizard : IS 1893 \(Part 1\) : 2002 Spectrum](#)

[Response Spectrum Wizard : GB 50011 2001 Spectrum](#)

7.3.63.1 Response Spectrum Wizard : Spectrum Type

The type is where the basic type of response spectrum is selected. The following page will depend on the choice at this stage.

Name

The name is used only as a label for the response spectrum.

Spectrum definition method

The user has a choice of the type of spectrum to use. This can be either code based: one of

- ASCE 7-05
- ASCE 7-10
- Eurocode 8 : 1994
- Eurocode 8 : 2003 DRAFT
- Eurocode 8: 2004
- FEMA 356
- GB 50011-2001
- IBC 2000
- IBC 2006 (ASCE 7-05)

- IBC 2009 (ASCE 7-05)
 - IBC 2015 (ASCE 7-10)
 - IS 1893
 - Ordinanza PCM 3274
 - UBC 1994
 - UBC 1997
- or
- User defined

7.3.63.2 Response Spectrum Wizard : ASCE 7-05 / IBC 2006 / IBC 2009

The ASCE 7-05 / ASCE 7-10 spectrum is defined by a set of parameters defined in the code. The IBC 2006 and IBC 2009 codes refer directly to the ASCE 7-05 code. The IBC 2015 code refer directly to the ASCE 7-10 code.

Site class

Sites are classed as one of A, B, C, D or E depending on the soil. Type F soil requires specialist geotechnical advice and is not included as an option.

S_s and S_1

These are the short period response acceleration parameter and the 1s response acceleration parameter as defined in section 11.4.3. Seismic Hazard maps for the USA are available from the U.S. Geological Survey web site by following the USGS hyperlink.

Occupancy category

The occupancy category from Table 1-1 is used to define the seismic design category (Tables 11.6-1 and 11.6-2) and the importance factor (I) (Table 11.5-1).

Response modification and deflection amplification factor

The response modification factor (R) and deflection amplification factor (Cd) are determined from Table 12.2-1 and section 12.9.2.

Frame type and Building period coefficient

The approximate fundamental period of the building (section 12.8.2.1) is calculated from equation (12.8-7)

$$T = C_t (h_n)^x$$

The building period coefficient, C_t , depends on the type of building frame. The frame types are:

- Steel moment-resisting
- Reinforced concrete moment-resisting
- Eccentrically braced
- Other

And h_n is the height of the building (in feet).

Ignore short period force reduction

It may be unconservative to reduce spectral values for low periods. This option ignores the reduction and extends the plateau to cover the low periods.

7.3.63.3 Response Spectrum Wizard : Eurocode 8 : 2004 Spectrum

The Eurocode 8 : 2004 spectrum is defined by a set of parameters defined in the code.

Subsoil class

Sites are classed depending on the subsoil as A, B, C, D or E as described in Part 1-1 Table 3.1.

Ratio of ground acceleration to gravity (a_{gR})

The parameter a_{gR} is used to characterise the seismicity of the site.

Importance factor (γ_1)

The importance factor allow an amplification for topographical effects depending on the type of structure.

Spectrum

The spectrum can be either an elastic spectrum or a design spectrum.

Structure damping ratio (ξ)

The spectrum values to use depend on the level of damping in the structure as expressed in the damping ratio. Elastic spectra only.

Behaviour (q , q_d)

A behaviour factor is defined which adjusts the level of the spectral values. A separate value, q_d , can be applied to scale the displacements. If q_d is not specified the value is q is used instead. Design Spectra only.

Lower bound limit (β)

A lower bound factor sets a minimum level below which the spectrum cannot fall. Design Spectra only.

7.3.63.4 Response Spectrum Wizard : Eurocode 8 : 2003 Spectrum

The Eurocode 8 : 2003 spectrum is defined by a set of parameters defined in the code.

Subsoil class

Sites are classed depending on the subsoil as A, B, C, D or E as described in Part 1-1 Table 3.1.

Ratio of ground acceleration to gravity (a_{gR})

The parameter a_{gR} is used to characterise the seismicity of the site.

Importance factor (γ_1)

The importance factor allow an amplification for topographical effects depending on the type of structure.

Spectrum

The spectrum can be either an elastic spectrum or a design spectrum.

Structure damping ratio (ξ)

The spectrum values to use depend on the level of damping in the structure as expressed in the damping ratio. Elastic spectra only.

Behaviour (q)

A behaviour factor is defined which adjust the level of the spectral values. Design Spectra only.

7.3.63.5 Response Spectrum Wizard : Eurocode 8 : 1994 Spectrum

The Eurocode 8 : 1994 spectrum is defined by a set of parameters defined in the code.

Subsoil class

Sites are classed depending on the subsoil as A, B, or C as described in Part 1-1 Section 3.

Ratio of ground acceleration to gravity (α)

The parameter α is used to characterise the seismicity of the site.

Spectrum

The spectrum can be either an elastic spectrum or a design spectrum.

Structure damping ratio (ξ)

The spectrum values to use depend on the level of damping in the structure as expressed in the damping ratio. Elastic spectra only.

Behaviour (q)

A behaviour factor is defined which adjust the level of the spectral values. Design Spectra only.

7.3.63.6 Response Spectrum Wizard : FEMA 356 Spectrum

The FEMA 356 spectrum is defined by a set of parameters defined in the code.

Site class

Sites are classed as one of A, B, C, D or E for the soil type as described in Table 1615.1.1. Type F soil requires specialist geotechnical advice and is not included as an option.

S_s and S_1

These are the short period response acceleration parameter and the 1s response acceleration parameter as defined in Section 1.6.1.1. Seismic Hazard maps for the USA are available from the U.S. Geological Survey web site by following the USGS hyperlink.

Damping ratio (β)

The damping ratio should be determined according to FEMA 356 1.6.1.5.3

Frame type and Building period coefficient / exponent

The empirical period of the building (Section 3.3.1.2.2) is calculated from equation (3-7)

$$T = C_t (h_n)^\beta$$

The building period coefficient, C_t , and the building period exponent, β , depend on the type of building frame. The frame types are:

- Steel moment-resisting
- Reinforced concrete moment-resisting
- Eccentrically braced
- Wood
- Other

And h_n is the height of the building (in feet).

Factor for vertical excitation

The spectral values for vertical excitation should be $2/3$ of that used for horizontal excitation. (FEMA 356 1.6.1.5.2)

Ignore short period force reduction

It may be unconservative to reduce spectral values for low periods. This option ignores the reduction and extends the plateau to cover the low periods. FEMA 356 1.6.1.5.1 states that "Use of spectral response accelerations calculated using Equation (1-8) in the extreme short period range ($T < T_0$) shall only be permitted in dynamic analysis procedures and only for modes other than the fundamental mode".

7.3.63.7 Response Spectrum Wizard : GB50011-2001 Spectrum

The GB 50011-2001 spectrum is defined by a set of parameters defined in the code.

Seismic design intensity

Seismic analysis is required when the seismic intensity is 6 or greater. If the value is greater than 9 more detailed analysis than response spectrum analysis is required. For design intensities 7 and 8 there are lower and higher values of design ground accelerations. Reference should be made to the code for the appropriate intensity for a particular site.

Site class

Sites are classed depending on the soil type and shear wave velocity. from Type I (for firm soil) to Type IV (for weak, soft soil).

Building category

Buildings are categorized as one of 4 types.

Type	Structure
A	Important structures; severe hazardous consequences
B	Structures whose functionality needs to be maintained
C	Structures other than types A, B or D
D	Less important structures

the importance factor is 1 but this can increase to 1.5. Determine the importance factor (I) from Table 6.

Seismic group

Three different seismic design groups (1, 2 and 3) are defined in the code. The characteristic period is a function of this design group

Seismic recurrence

Two different scenarios are considered: frequent earthquakes with a smaller ground acceleration and rare earthquakes with a larger ground acceleration. The frequent earthquake is one with a probability of exceedance of 63% within a design period of 50 years. The rare earthquake is one with a probability of exceedance of 2-3% within a design period of 50 years.

Structure damping ratio (ζ)

The spectrum values to use depend on the level of damping in the structure as expressed in the damping ratio. This value will adjust the spectral accelerations.

7.3.63.8 Response Spectrum Wizard : IBC 2000 Spectrum

The IBC 2000 spectrum is defined by a set of parameters defined in the code.

Site class

Sites are classed as one of A, B, C, D or E for the soil type as described in Table 1615.1.1. Type F

soil requires specialist geotechnical advice and is not included as an option.

S_s and S₁

These are the short period response acceleration parameter and the 1s response acceleration parameter as defined in Section 1615. Seismic Hazard maps for the USA are available from the U.S. Geological Survey web site by following the USGS hyperlink.

Importance factor

Determine the importance factor (I_E) from Section 1616.2 and Table 1604.5.

Response modification factor

The structural system factor (R) is determined from Table 1617.6.

Frame type and Building period coefficient

The approximate fundamental period of the building (Section 1617.4.2.1) is calculated from equation (16-39)

$$T = C_t(h_n)^{3/4}$$

The building period coefficient, C_t , depends on the type of building frame. The frame types are:

- Steel moment-resisting
- Reinforced concrete moment-resisting
- Eccentrically braced
- Other

And h_n is the height of the building (in feet).

Factor for vertical excitation

It is common to reduce the spectral values for vertical excitation to $2/3$ of that used for horizontal excitation.

Ignore short period force reduction

It may be unconservative to reduce spectral values for low periods. This option ignores the reduction and extends the plateau to cover the low periods.

7.3.63.9 Response Spectrum Wizard : IS 1893 (Part 1) : 2002 Spectrum

The IS 1893 (Part 1) : 2002 spectrum is defined by a set of parameters defined in the code.

Seismic zone

The seismic zones are defined as II, III, IV or V (See Fig. 1 and Annex E).

Soil type

Sites are classed depending on the soil type as one of Type I (rock or hard soil), Type II (medium soil) or Type III (soft soil).

Importance factor

For most buildings the importance factor is 1 but this can increase to 1.5. Determine the importance factor (I) from Table 6.

Response reduction factor

Depending on the structure type a different response reduction factor can be used. Determine the response reduction factor (R) from Table 7.

Structure damping ratio (ξ)

The spectrum values to use depend on the level of damping in the structure as expressed in the damping ratio. (See Table 3). This value will normally be adjusted during an analysis.

Factor for vertical excitation

Clause 6.4.5 recommends that the spectral values for vertical excitation be $2/3$ of that used for horizontal excitation. Checking this option applied the $2/3$ factor for the vertical direction.

Ignore short period force reduction

It may be unconservative to reduce spectral values for low periods. This option ignores the reduction and extends the plateau to cover the low periods.

7.3.63.10 Response Spectrum Wizard : Ordinanza PCM 3274 Spectrum

The Ordinanza PCM 3274 (Italian) spectrum is defined by a set of parameters defined in the code.

Seismic Zone

The seismic zones are defined as 1, 2, 3 or 4.

Subsoil class

Sites are classed depending on the subsoil as A, B, C, D or E.

Ratio of ground acceleration to gravity (a_g)

The parameter a_g is used to characterise the seismicity of the site.

Spectrum

The spectrum can be either an elastic spectrum or a design spectrum.

Structure damping ratio (ξ)

The spectrum values to use depend on the level of damping in the structure as expressed in the damping ratio. Elastic spectra only.

Behaviour (q)

A behaviour factor is defined which adjust the level of the spectral values. Design Spectra only.

7.3.63.11 Response Spectrum Wizard : UBC 1994 Spectrum

The UBC 1994 spectrum is defined by a set of parameters defined in the code.

Zone number

This is the Zone in which the structure is to be located and is one of 1, 2A, 2B, 3, 4 as defined in Table 23-I).

Soil Type

Enter S1, S2 or S3 for the soil type as described in Table 23-J. Type S4 soil requires specialist geotechnical advice.

Importance factor

Determine the importance factor (I) from Table 23-L.

Structural system factor

The structural system factor (R_w) is determined from Table 23-O (building structures) or 23-Q (non-building structures).

Empirical building period

Enter the value of the empirical building period derived from the code. (Clause 2334(b) 2A.)

Target base shear

Select from 90% or 100% as determined from Clause 2335(e)3.

Factor for vertical excitation

It is common to reduce the spectral values for vertical excitation to $2/3$ of that used for horizontal excitation.

Ignore short period force reduction

It may be unconservative to reduce spectral values for low periods. This option ignores the reduction and extends the plateau to cover the low periods.

7.3.63.12 Response Spectrum Wizard : UBC 1997 Spectrum

The UBC 1997 spectrum is defined by a set of parameters defined in the code.

Zone number

This is the Zone in which the structure is to be located and is one of 1, 2A, 2B, 3, 4 as defined in Table 23-I).

Seismic source type

For Zone 4 the seismic source type is categorised by types A, B or C determined from Table 16-U.

Closest distance to known seismic source

For Zone 4 the spectrum will depend on the distance to a known seismic source in the range 2km to 15km. See Table 16.

Soil Type

Enter SA, SB, SC, SD or SE for the soil type as described in Section 1636. Type SF soil requires specialist geotechnical advice.

Importance factor

Determine the importance factor (I) from Table 16-K.

Structural system factor

The structural system factor (R) is determined from Table 16-N (building structures) or 16-P (non-building structures).

Empirical building period

Enter the value of the empirical building period derived from the code (Section 1630.2.2) or set to zero for automatic calculation based on frame type.

Frame type and Building period coefficient

The empirical period of the building (Section 1630.2.2 – Method A) is calculated from equation (30-8)

$$T = C_t(h_n)^{3/4}$$

The building period coefficient, C_t , depends on the type of building frame. The frame types are:

- Steel moment-resisting
- Reinforced concrete moment-resisting
- Eccentrically braced

- Other

And h_n is the height of the building (in feet).

This is ignored if the building period is specified directly.

Target base shear

Select from 90% or 100% as determined from Section 1631.5.4.

Factor for vertical excitation

It is common to reduce the spectral values for vertical excitation to $2/3$ of that used for horizontal excitation.

Ignore short period force reduction

It may be unconservative to reduce spectral values for low periods. This option ignores the reduction and extends the plateau to cover the low periods.

7.3.63.13 Response Spectrum Wizard : User Defined Spectrum

The user-defined spectrum must be defined externally and can be imported in this dialog.

Import

The user is given a file selection dialog to select the file to import. The files supported are:

- [Text - tab delimited](#) (.txt)
- [Text - comma delimited](#) (.csv)
- [T/HIS curve file](#) (.cur)
- GFA binary file (.gfa)

The files must contain values of displacement / velocity / acceleration as a function of period / frequency.

The types for the abscissa (x-axis) should be one of:

- **FREQUENCY**
- **PERIOD**

The types for the ordinate (y-axis) should be one of:

- **DISP**
- **VELOCITY**
- **ACCELERATION**

Export

The user is given the option of exporting a file. This allows a file to be modified if required and then re-imported into GSA. The files supported are:

- [Text - tab delimited](#) (.txt)
- [Text - comma delimited](#) (.csv)
- [T/HIS curve file](#) (.cur)
- HTML files file (.htm,.html)

Modify

In many cases the imported file contains the correct data but the data needs some modification, such as scaling so that the units are correct. See [Modify Curve](#).

7.3.63.14 Response Spectrum Wizard : Damping

Damping

In general the seismic code assumes a value of damping (typically 5%). In general the code specified value should be used, but GSA allows the user to select the way in which damping is to be applied in the response spectrum analysis. Most of these options are only relevant if modal damping values have been calculated. The options are:

- **Code defined damping** — the value set up in the response spectrum module (set to 5% and not adjustable for a number of codes).
- **Constant** — the user specifies a constant damping ratio
- **Damping table** — the damping is specified in a separate table of damping for each mode
- **Modal damping** — the values calculated in the modal dynamic analysis as used.

Scale response spectrum

When a constant or modal damping option has been selected the user is given the option of scaling the spectrum to allow for different levels of damping.

- **Code defined scaling only** — the spectrum is modified if the code allows for this, otherwise it is left unchanged.
- **Scale by $\ln(100-\eta)/\ln(20)$ [ISO19901-2]** — this applies a scaling of $\ln(100/\eta)/\ln(20)$ as outlined in ISO 19901-2 over the whole spectrum
- **Newmark-Hall : 50th percentile** — the median acceleration modifiers from Newmark and Hall (1982) are applied to the calculated spectral values
- **Newmark-Hall : 84th percentile** — the acceleration modifiers at one standard deviation from Newmark and Hall (1982) are applied to the calculated spectral values

7.3.64 Variable UDL Wizard

A Variable UDL (VUDL) is a uniformly distributed load that has an intensity (Force/Length/path) which can depend on the loaded length L .

Each segment of the VUDL curve between transition lengths has intensity per unit length per path given by the equation

$$VUDL = A \times \left(\frac{1}{L}\right)^n$$

Up to three segments of VUDL can be specified. The user needs to ensure the intensity matches either side of the transition length.

A constant UDL can be entered as one segment with the intensity as Factor(A) and Index(n) = 0, or obtained by factoring the UDL(1kN) in the Path Loading module.

Details of Variable UDLs are given in the [Program Data section](#).

More:

[Variable UDL Wizard : Definition](#)

[Variable UDL Wizard : Graph](#)

7.3.64.1 Variable UDL Wizard : Definition

Name

The name is used to refer to a particular VUDL.

Number of segments

The number of segments used to define the complete VUDL (with a maximum of three segments).

Factor (A) / Index (n)

The intensity of the VUDL is specified in terms of a factor and index as follows:

$$\text{VUDL} = A \times \left(\frac{1}{L} \right)^n$$

Transition point (L)

The load length up to which the previous factor and index apply and beyond which the next factor and index apply.

7.3.64.2 Variable UDL Wizard : Graph

This provides a view of the load intensity as a function of load length. This is provided as visual confirmation of the VUDL definition.

7.3.65 Path Loading

Path Loading can be used with the results of an influence line analysis to derive optimized bridge loading load cases.

Details of the path loading is described in the [Program data](#).

Path

The path to which this loading is related.

Load type

The type of load allows the selection of variable UDL, knife edge and vehicle loads.

Variable UDL record / Factor

The variable UDL and the scaling factor that applies to this loading.

Knife edge load per path

The value of the knife edge load for that path. Note this is a load and not a load intensity.

Vehicle / Factor / Exclusion length from mid-point of vehicle

The particular [vehicle](#) can be selected here. The factor can be used to modify the intensity of standard Vehicles. The exclusion length is measured from the midpoint of the vehicle to the point where the VUDL starts on either side of the vehicle (so an exclusion length of 0 gives an uninterrupted VUDL).

7.3.66 Load Case Titles

The load case titles dialog can be activated from [Cases table view — Load page](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view.

Name

The name for a load case where the loads are defined elsewhere.

Type

Defines the type of load case. The options offered are specific to the selected load design code, but include dead, live, snow, seismic, etc.

Source

Used to differentiate different load sets.

Category

Specified the category of loading. The options offered are specific to the selected load design code, but include domestic office, storage, roof, etc.

Direction

For loading such as seismic the direction in which it is applied

Include

Specified if adverse and/or beneficial effects should be included.

7.3.67 Combination Cases

The combination case dialog can be activated from [Cases table view — Combination page](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view. Combinations can be either simple combinations of analysis cases or envelope cases.

Name

The name is used only as a label for the combination case.

Description

The [description](#) is where the combination case is defined in terms of analysis cases.

- **Simple combinations** are of the form “1.6A1 + 1.4A2”.
- **Enveloping cases** are of the form “(–1.0 or 1.6)A1 + (–1.0 or 1.4)A2”.

Append to description

To assist in creating the case description the append options allow the user to select analysis cases, other combinations cases or lists (of cases) or a list of cases to envelope and add to the description. The cases selected are then appended to the description.

Expand

A combination case may refer to another combination case that is defined as a simple combination. If this is the case the expand to analysis cases will convert the description to one in terms of analysis cases only.

7.3.68 Case Descriptions

The combination case dialog can be activated from [Case Descriptions table view](#) by giving the “Edit | Wizard” menu command. “Wizard” is also available on the [Data Options toolbar](#) and on the right-click menu for the table view. Case descriptions can be descriptions of load cases, analysis cases or combinations cases. Case descriptions are used to store commonly used case descriptions. For example, the dead load may have contributions from several load cases. The combination of the loads can be entered as a case description and this can then be used when a case is required in an analysis task.

Name

The name is used only as a label for the combination case.

Description

This can be any valid case description, so will typically be of the form "1.6A1 + 1.4A2".

7.3.69 Append Analysis Case, Append Combination Case, Append List Case

The Append Analysis Case, Append Combination Case and Append List Case dialogs are all similar and offer the user a list of cases from which to select. The selected case is then appended to the combination case description.

Factor

The factor is the factor that multiplies the selected case. This may be a single number or an envelope factor of the form "(-1.0 or 1.6)". The parentheses are optional.

Special

The special option is only available in the Append Combination Case dialog.

The following special operators may be applied to enveloping combination cases:

- **max** – maximum values of the envelope
- **min** – minimum values of the envelope
- **abs** – the greater of the absolute maximum value and the absolute minimum value of the envelope
- **signabs** – as "abs" but keeping the original sign

These should only be applied to enveloping combination cases.

Reference by

The selected case can be referred to by number (e.g. A4) or by name ("Dead load").

7.3.70 Append Envelope

The Append Envelope dialog allows a set of cases to be selected that will be interpreted as an envelope. The selected cases are assembled into a description, which is then added to the case description.

Select the cases that are to be enveloped by:

- dragging the mouse over a range of cases or
- by Shift-clicking on a case to select the end of a range or
- by Ctrl-clicking on a case to toggle the selection state of that case

7.3.71 Assemblies

Assemblies are a way of grouping elements for further processing such as cut section forces. These give a convenient way of working with parts of the model.

Name

The name is used as a convenient way of referring to assemblies.

Element list

This defines the elements that make up the assembly

Assembly definition

There are two ways in which assemblies can be defined: by node or by axis. When the node option

is selected the definition uses topology (nodes) to define the assembly. When the axis option is selected the definition uses an axis and offsets to define the assembly.

Topology

This defines the nodes at the ends of the element.

Orientation node

The plane formed by the topology items and the orientation node is the xy plane of the assembly. The orientation node cannot be colinear with the topology nodes

Internal topology

This is a list of nodes that define the curve of the assembly. The order of the nodes is not important as the nodes are ordered along a vector between ends one and two.

Curve fit

For a curved element there is a choice of how the curve is fitted to the points. The default is to use Lagrange interpolation to fit a curve through all the intermediate points. The other option which is only valid if there is just a single internal topology item is for a circular arc from end one passing through the internal node to end two.

Axis

This defines the local axis system of the assembly, so for a vertical assembly the x axis should be aligned with the global z axis.

Extents

The extents of the assembly can be specified directly as end 1 and end 2 or GSA can determine this from the elements that define the assembly.

Transverse offset

This is used to locate the centre line of the assembly. 'Set from element list' sets the transverse offset to be at the average extents of the assembly in the y and z directions of the axis.

Point definition, points, spacing and offsets

This is how the assembly is defined internally (as used for assembly forces and moments). There are three options

- **Number of points** – a number of equally spaced points along the axis direction of the assembly
- **Spacing of points** – the assembly has points spaced along its length
- **By storey** – the assembly is divided by storey (the assembly local x axis must be aligned with global z for this to be valid).
- **Explicit** – the assembly has points at the explicitly defined positions.

These points can be offset from the ends of the assembly. The sign of the offset is ignored and the offset always acts to reduce the active length of the assembly for calculation of intermediate points.

7.4 Analysis Dialogs and Wizards

[Analysis Case Definition](#)

[Analysis Wizard](#)

[Advanced Solver Settings](#)

[GsRaft Analysis Progress](#)

[GsRaft Damping Coefficient](#)

[GsRaft Displacement Residual](#)

[View List](#)

7.4.1 Analysis Case Definition

The case description definition dialog is made available when an analysis case definition is required, it is not invoked directly.

Name

This gives the name to identify the analysis case.

Description

This is where the description of the analysis case is entered. See [Analysis Case Descriptions](#). The dialog lists all the existing load cases by name, where the name exists, or by number, and all the [case descriptions](#). Double clicking on any of these entries adds the expanded load description to the description field.

7.4.2 Analysis Wizard

The analysis wizard is available from the [Cases table view — Analysis page](#) or from the “Analysis | New Analysis Task” menu command. This wizard handles the setting up of analysis cases based on particular analysis requirements, and for the management of analysis case data.

More:

[Analysis Wizard : Solver Option](#)

[Analysis Wizard : Static Analysis Cases](#)

[Analysis Wizard : Gss Static P-delta](#)

[Analysis Wizard : Gss Modal Dynamic Parameters](#)

[Analysis Wizard : Gss Ritz Analysis Parameters](#)

[Analysis Wizard : Gss Buckling Parameters](#)

[Analysis Wizard : Gss Mass Calculation](#)

[Analysis Wizard : GsRelax Non-linear Static Options](#)

[Analysis Wizard : GsRelax Form Finding Options](#)

[Analysis Wizard : GsRelax Analysis Task Control](#)

[Analysis Wizard : GsRelax Member Identification](#)

[Analysis Wizard : GsRelax Case Control](#)

[Analysis Wizard : GsRelax Analysis Progress Information](#)

[Analysis Wizard : Response Spectrum Parameters](#)

[Analysis Wizard : Harmonic Analysis Parameters](#)

[Analysis Wizard : Linear Time History Analysis Parameters](#)

[Analysis Wizard : GsSpec Periodic Load Analysis](#)

[Analysis Wizard : GsSpec Footfall Induced Vibration Analysis](#)

[Analysis Wizard : GsRaft Control](#)

[Analysis Wizard : GsRaft Progress](#)

[Analysis Wizard : GsBridge Control](#)

[Analysis Wizard : Envelopes](#)

[Analysis Wizard : Cases Set Up](#)

7.4.2.1 Analysis Wizard : Solver Option

The analysis wizard solver option is where the user must choose which type of analysis is required.

Task name

This is a name to associate with this task. This is used as the basis of the analysis case name, so that if there is more than one modal analysis the particular modes can be clearly identified.

Solution type

The solution type breaks down into a number of main categories:

- **Static** – [linear static](#), [static P-delta](#) and [non-linear static](#) analysis.
- **Dynamic** – [modal](#), [modal P-delta](#) and [Ritz](#) analysis.
- **Dynamic response** – [response spectrum](#), pseudo response spectrum, [harmonic analysis](#), [linear time history analysis](#), [footfall analysis](#) and [periodic load analysis](#)
- **Buckling** – [modal](#) or member buckling analysis.
- **Soil-structure** – [raft](#) analysis.
- **Bridge** – bridge optimisation analysis (pre-analysis and no analysis case should exist).
- **Miscellaneous** – mass analysis.
- **Lightweight structures** – [form finding](#) analysis.
- **Post-processing** – [analysis envelopes](#).
- **Diagnostic** – [model stability](#) analysis

Solver and solution option

The solver and solution option depend on the solution type and are updated depending on the solution type selected.

Analysis stage

Select the analysis stage that is to be analysed. This option is only available when the “Analysis stages” option on the “Advanced Features” page of the [Preferences](#) dialog is enabled.

7.4.2.2 Analysis Wizard : Static Analysis Cases

This is where the loading associated with the analysis case is specified. In the simplest case each analysis case will have a single, unfactored load case (e.g. case A1, description L1). For a P-delta or other non-linear analysis results should not be combined post analysis so the factored and combined loads must be specified for the analysis case (e.g. case A1, description 1.4L1+1.6L2 or 1.5A3 or 1.6C1).

For GsRelax a blank entry is allowed for form-finding analysis. In this case the analysis will find the equilibrium position under form-finding forces only.

The details of the loading associated with each analysis case are displayed in a list box:

- **Name**—case name
- **Case**—case identifier
- **Description**—a description of the case
- **Status**—the status of the case
- **Solver**—the solver that will be used.

On entry this will be empty and as new cases are defined they are added to the list. In the list box the items can be selected by clicking on the Name field. Normal rules for shift+click and control +click apply to this control.

Add

Allows the user to add new cases. A dialog allows the user to enter the name and description.

Delete

Allows the selected analysis cases to be deleted.

Modify

Allows the user to modify the name and description of the analysis case.

Create default cases

Sets up a set of default analysis cases, one for each load case.

7.4.2.3 Analysis Wizard : Gss Static P-delta

P-delta case

For a static P-delta analysis the user must specify the basis for the geometric stiffness. The options are:

- Each analysis case defining its own geometric stiffness
- Load case definition
- Analysis case

The basis on which the geometric stiffness is calculated. The first option requires a separate solution for each analysis case as a new geometric (or differential) stiffness matrix is required for each analysis case, based on the load in that case. The second option allows a load definition (e.g. 1.2L1 + 1.2L2) to be specified. In this case GSA first analyses the structure under this load and generates a geometric stiffness from the resulting forces/moments. In the third an analysis case is specified and the geometric stiffness is constructed from the results in this analysis case.

7.4.2.4 Analysis Wizard : Gss Modal Dynamic Parameters

For both a "modal dynamic" and "modal dynamic p-delta" analysis the user is brought to this page.

Number of modes

The number of modes required from the eigenanalysis.

Start mode

Occasionally it is only higher modes that are of interest. This allows the lower modes to be excluded.

Maximum no. iterations

All eigensolvers are iterative. This is used to set up the maximum number of iterations that will be allowed. It is rare that this value will need to be increased. The maximum number of iterations can also be set in the [Gss Analysis Settings dialog](#).

Additional restraint

For a large class of structure the horizontal and vertical modes are essentially uncoupled. In these situations it may be better to do separate modal analysis for horizontal and vertical modes. The unwanted degrees of freedom can be suppressed using the additional restraints; for example, setting 'horizontal' additional restraint will produce vertical modes only.

Frequency shift & cut-offs

The frequency shift specifies the frequency around which the modes are calculated. The modes

closest to this frequency are retained. This options allows for frequencies outside the range of interest to be discarded. These natural frequencies will still be calculated but the results are discarded rather than stored.

Mass option

This specifies how the mass of the elements is to be included in a [dynamic analysis](#). A scale factor can be applied to modify the element masses. This is typically used to increase the mass to allow for connections.

Mass derived from loads

In addition to the mass of the elements, extra mass can be inferred from loads (Case definition) applied in a particular Direction, with a Scale factor if required. This is used where “dead loads” have been applied to the structure which represent mass in the real structure. Only externally applied forces are included; the effect of gravity (i.e. self weight) is taken into account by the mass option.

P-delta case

The basis on which the geometric stiffness is calculated. This can either be a load description (e.g. 1.2L1 + 1.2L2) or an analysis or combination case (e.g. A3 or C4). When a load description is specified, GSA first analyses the structure under this load and generates a geometric stiffness from the resulting forces/moments. When an analysis or combination case is specified the geometric stiffness is constructed from the results in this analysis or combination case. Combination cases can only be simple combination cases (i.e. of the form $a.A_a + b.A_b + \dots$)

This option is only present if a “modal dynamic p-delta” analysis has been requested.

7.4.2.5 Analysis Wizard : Gss Ritz Analysis Parameters

For a Ritz analysis the user is brought to this page.

Number of Ritz vectors

The number of Ritz vectors and thus the number of modes required from the eigenanalysis.

Direction of Ritz vectors

This is used to specify the (global) direction of interest for this analysis.

Maximum no. iterations

All eigensolvers are iterative. This is used to set up the maximum number of iterations that will be allowed. It is rare that this value will need to be increased. The maximum number of iterations can also be set in the [Gss Analysis Settings dialog](#).

Mass option

This specifies how the mass of the elements is to be included in a [dynamic analysis](#). A scale factor can be applied to modify the element masses. This is typically used to increase the mass to allow for connections.

Mass derived from loads

In addition to the mass of the elements, extra mass can be inferred from loads (Case definition) applied in a particular Direction, with a Scale factor if required. This is used where “dead loads” have been applied to the structure which represent mass in the real structure.

P-delta case

The basis on which the geometric stiffness is calculated. This can either be a load description (e.g. 1.2L1 + 1.2L2) or an analysis or combination case (e.g. A3 or C4). When a load description is specified, GSA first analyses the structure under this load and generates a geometric stiffness from the resulting forces/moments. When an analysis or combination case is specified the geometric stiffness is constructed from the results in this analysis or combination case. Combination cases

can only be simple combination cases (i.e. of the form $a.A_a + b.A_b + \dots$)

This option is only present if a “Ritz p-delta” analysis has been requested.

7.4.2.6 Analysis Wizard : Gss Buckling Parameters

For a “buckling” analysis the user is brought to this page.

Number of modes

The number of modes required from the eigenanalysis.

Maximum no. iterations

All eigensolvers are iterative. This is used to set up the maximum number of iterations that will be allowed. For some buckling analyses this value may need to be increased. The maximum number of iterations can also be set in the [Gss Advanced Settings](#) dialog.

Load factor shift & cut-offs

The load factor shift specifies the load factor around which the modes are calculated. The modes closest to this frequency are retained. The cut-offs allows for load factors outside the range of interest to be discarded. These buckling modes will still be calculated but the results are discarded rather than stored. A lower cutoff of zero will ensure that negative modes are ignored.

P-delta case

The basis on which the geometric stiffness is calculated.. This can either be a load definition (e.g. 1.2L1 + 1.2L2) or an analysis or combination case (e.g. A3 or C4). When a load description is specified, GSA first analyses the structure under this load and generates a geometric stiffness from the resulting forces/moments. When an analysis or combination case is specified the geometric stiffness is constructed from the results in this analysis or combination case. Combination cases can only be simple combination cases (i.e. of the form $a.A_a + b.A_b + \dots$)

7.4.2.7 Analysis Wizard : Gss Mass Calculation

For a “mass” analysis the user is brought to this page.

Task name

This is a name to associate with this task. This is used as the basis of the analysis case name, so that if there is more than one mass analysis the particular cases can be clearly identified.

Mass option

This specifies how the mass of the elements is to be included. A scale factor can be applied to modify the element masses. This is typically used to increase the mass to allow for connections.

Mass derived from loads

In addition to the mass of the elements, extra mass can be inferred from loads (Case definition) applied in a particular Direction, with a Scale factor if required.

7.4.2.8 Analysis Wizard : GsRelax Non-linear Static Options

For a “non-linear static” analysis the user is brought to this page.

Element slenderness included

When checked, slenderness effects on individual beam elements are included by the use of stability functions. These obviate the need to include intermediate nodes in beam elements to consider their stability.

Automatic load increment

When checked, the magnitude of each load increment (except the first increment) will be chosen automatically by the program. The **load applied in the first increment** is specified as a percentage of the total imposed loads. During the analysis, the loading is increased until the specified limits are exceeded. These limits can be expressed in terms of nodal displacement (translation or rotation) or load factor. They are defined in the [GsRelax Analysis Task Control](#) page of the Analysis Wizard. The magnitude of the automatic load increment is chosen by the program according to the likelihood of the convergence of the analysis in the previous load increment: the easier the previous load increment was to converge, the larger of the current load increment will be, and vice versa. If the analysis cannot converge for the current load increment, the load increment is reduced until the analysis does converge or the specified minimum load increment is reached. If the specified minimum load increment is reached, the analysis will stop; the current load level is probably the ultimate capacity of the model. The records of the incremental load factors and corresponding displacements are available after the analysis as “Global Results | Load Factor / Displacement Relationship” output.

7.4.2.9 Analysis Wizard : GsRelax Form Finding Options

Ignoring form-finding properties

The analysis proceeds as for the single increment option except that no results are stored. Instead, the resulting deflected coordinates and equilibrium element distortions overwrite the original data to create a modified data file.

Soap-film form-finding

The analysis proceeds as for the single increment option except [soap-film properties](#) are used for elements where these have been defined. The resulting deflected coordinates and equilibrium element forces overwrite the original data to create a modified data file.

Force-density form finding

The analysis proceeds as for the single increment option except [force density properties](#) are used for elements where these have been defined. The resulting deflected coordinates and equilibrium element forces overwrite the original data to create a modified data file.

7.4.2.10 Analysis Wizard : GsRelax Analysis Task Control

This option is required when “automatic load increment analysis” has been selected. The limits set here control the automatic load increment option and can be used to define the “capacity” of the model.

Displacement termination

Set the maximum permitted deflection or rotation at a node. The analysis terminates when this condition is reached.

Load factor termination

Set the max load factor at which the program should cease to search for the model capacity. Use this to avoid unnecessary analysis.

Automatic increment limits

The program will successively increase the load factor. The size of each increment can be limited here.

7.4.2.11 Analysis Wizard : GsRelax Member Identification

This information is required when the “[Individual member buckling](#)” option has been selected.

In this option, the effective length of a chain of compressive Beam elements in a structure can be estimated. Elements specified should not be Bars or Struts.

The axial load in these elements will be increased, while the rest of the model is subjected to the defined analysis case. Thus the effective restraint offered to the member by the surrounding structure can be investigated.

The program analyses the structure with 100% imposed loads. The program stores the equilibrium element forces and adds a small disturbance to the member under investigation. The next step is to reanalyse with the resultant element forces and imposed loads from the initial analysis but with a factored axial force in the member under consideration. This is repeated until the member in question buckles.

Elements

Input a list of elements to be investigated for individual member buckling. These must be adjacent beam elements.

7.4.2.12 Analysis Wizard : GsRelax Analysis Progress Information

The progress of an analysis can be viewed in a progress report and Graphic View that are opened at the start of the analysis. The Graphic View shows the deflection of the model as the analysis progresses.

Updating the graphic and progress report slows the analysis down. Therefore it is advisable to update these views more frequently during the initial runs of the model, reducing the frequency or disabling the views as the performance of the model becomes familiar.

If the option to save residuals to file is selected then the residual forces and moments at the last iteration calculated are saved in [user modules](#) in a GWA file. This GWA file can then be imported into the model using the “File | Import | Text (GWA file)” command to make the residuals available for tabular and contoured display.

7.4.2.13 Analysis Wizard : Response Spectrum Parameters

The basic response cases are set up in the data. This page confirms which output is required. If no modal results are present the user is warned: “There are no modal results present. This analysis will terminate”.

Modal analysis task

If more than one set of modal dynamic results exist GsSpec needs to know which set of results are to be used. In this situation the task selection is enabled with a list of the available analysis tasks.

Combination method

This determines how the modes are to be combined. The combination methods offered are

- Absolute sum
- Complete quadratic combination (CQC)
- Modified Rosenbluth
- Square root sum of squares (SRSS)

The CQC combination method requires damping value which depends on the particular analysis options.

Axis

In general the calculation is carried out in global exes, however a different axis can be specified

provided that the z axis is aligned with the global z axis.

Source

This allows for this analysis to be included as a source for combining results.

Direction, spectrum and modes

Each of the x, y and z directions can be selected for inclusion in the analysis. The spectra that have been defined are offered as options or the user may enter a spectrum number directly. The user may select particular modes to be included in the response, although in most cases the default of "all" will be most appropriate.

Rigorous or fast

This option is only relevant for This gives a choice between a rigorous calculation only applies for combination method which couple the responses in different modes (CQC and modified Rosenbluth). A rigorous calculation calculates the coupling between all modes. A fast calculation ignores the coupling between modes where the coupling coefficient is less than 1%.

GsSpec output option

The user can select the nodal output from GsSpec. The options are:

- **Displacement** — these are always output
- **Velocity**
- **Acceleration**

Storey drifts

Storey drifts can be specified at various locations. A storey drift is identified by a name, a location in the global xy plane and a list of storeys that are included. By specifying storeys it is possible to carry out drift calculations where storey levels may be different in different parts of the structure.

Relative nodal displacements

In a response spectrum analysis relative displacements cannot be calculated from the difference in the response displacements at the nodes but need to be calculated from the relative modal displacements. The relative nodal displacement is specified by a name, a pair of nodes and an axis for the output.

7.4.2.14 Analysis Wizard : Response Spectrum Dynamic Details

A pseudo response spectrum analysis is based on mode shapes arrived at by static analysis. In order to combine these for a response spectrum analysis the modal information must be supplied.

Dynamic details

The dynamic details must be supplied for each pseudo mode to be considered. These are the frequency, modal mass and effective mass. Only the pseudo modes for which values are not specified will be included in the analysis

7.4.2.15 Analysis Wizard : Harmonic Analysis Parameters

This page defines the analysis parameters and the output requirements. If no modal dynamic analysis results are present, this option is disabled and this page cannot be opened.

GsSpec harmonic analysis parameters

- **Modal analysis task**

If more than one set of modal dynamic analysis results exist it needs to know which set of results are to be used. If only one modal dynamic analysis task is available, it will be used by the analysis.

- **Load description** — define the harmonic load, the syntax of the description is the same as other analyses, e.g. 1.4L1+1.6L2.
- **Excitation frequencies** — define the lowest, highest frequencies and the frequency increment. These will be used to generate the actual frequencies to be used in the harmonic analysis. The number of harmonic analysis cases will be equal to the number of input harmonic frequencies determined by the lowest, highest frequencies and frequency increment.
- **Acceleration**

Damping ratio

This defines the critical damping ratios use for the analysis. The options are:

- **Constant for all modes** — the user specifies a constant damping ratio which will be used for all modes.
- **Damping from modal analysis** — use the damping ratios calculated from modal dynamic analysis. If the damping ratio is not available from modal dynamic analysis, the analysis cannot be carried out.
- **Damping frequency relationship table** — query the selected damping table to determine the damping ratio for those modes defined in the table. For all the other modes that damping ratios have not been defined in the damping table, the damping ratio defined below will be used.

GsSpec harmonic analysis output options

The user can select the nodal output from and the options are:

- **Displacement** — these are always output
- **Velocity**
- **Acceleration**
- **Output nodes** — the node list that the output will be generated
- **Output elements** — the element list that the output will be generated

If not all nodes and elements are interested, it is recommended to define the interested output nodes and output elements to increase analysis speed and reduce the file size.

7.4.2.16 Analysis Wizard : Linear Time History Analysis Parameters

This page defines the analysis parameters for linear time history analysis. If no modal dynamic analysis results are present, this option is disabled and this page cannot be opened.

GsSpec linear time history analysis parameters

- **Modal analysis task**

If more than one set of modal dynamic analysis results exist it needs to know which set of results are to be used. If only one modal dynamic analysis task is available, it will be used by the analysis.

Excitation

- **Force excitation** — define the magnitude and locations of the dynamic loads, the syntax of the description of the loads is the same as that for static analysis, e.g. 1.4L1+1.6L2.
- **Base excitation** — The direction of the base acceleration is defined by giving X, Y and Z components of the direction vector. This vector only defines the direction of the base acceleration and its length is not used in the analysis. The peak value of the base acceleration should also be defined here.
- **Load curve** — Choose a load curve from the combo box that will define the variation of the dynamic loads against time. The load curve will be normalized before use.

Damping ratio

This defines the critical damping ratios use for the analysis. The options are:

- **Constant for all modes** — the user specifies a constant damping ratio which will be used for all modes.
- **Damping from modal analysis** — use the damping ratios calculated from modal dynamic analysis. If the damping ratio is not available from modal dynamic analysis, the analysis cannot be carried out.
- **Damping frequency relationship table** — query the selected damping table to determine the damping ratio for those modes defined in the table. For all the other modes that damping ratios have not been defined in the damping table, the damping ratio defined below will be used.

Output options

- **Output time interval** — define the time interval that the analysis results will be saved.
- **Stop time** — define the time that the analysis will stop.
- **Displacement** — this will always be checked.
- **Velocity** — check to make the velocity output available.
- **Acceleration** — check to make the acceleration output available.

7.4.2.17 Analysis Wizard : Periodic Load Analysis Parameters

This page defines the analysis parameters for periodic load analysis. If no modal dynamic analysis results are present, this option is disabled and this page cannot be opened.

Modal analysis task

If more than one set of modal dynamic analysis results exists then GSA needs to know which set of results is to be used. If only one modal dynamic analysis task is available, it will be used by the analysis.

Load description

This defines the magnitude and spatial distribution of the periodic loads, the syntax of the description of the loads is the same as that for other analyses, e.g. 1.4L1+1.6L2

Dynamic load factor curve

This defines the dynamic load factor curves to be used for this analysis. Dynamic load factor curves are defined in Dynamic load factor table.

Excitation frequencies

This defines the range of the periodic load frequencies to be considered in the analysis as well as the increment of the periodic load frequency. Smaller increment of the frequencies will give more accurate results, but will result in a longer analysis time and vice versa.

Damping ratio

This defines the critical damping ratios use for the analysis. The options are:

- **Constant for all modes** — the user specifies a constant damping ratio which will be used for all modes
- **Damping from modal analysis** — use the damping ratios calculated from modal dynamic analysis. If the damping ratio is not available from modal dynamic analysis, the analysis cannot be carried out.
- **Damping frequency relationship table** — query the selected damping table to determine the damping ratio for those modes defined in the table. For all the other modes that damping ratios

have not been defined in the damping table, the damping ratio defined below will be used.

7.4.2.18 Analysis Wizard : Footfall Analysis Parameters

This page defines the analysis parameters. If no modal dynamic analysis results are present, this option is disabled and this page cannot be opened.

GsSpec footfall induced vibration analysis option

- **Modal analysis task**

If more than one set of modal dynamic analysis results exist it needs to know which set of results are to be used. If only one modal dynamic analysis task is available, it will be used by the analysis.

- **Excitation method**

1. Self excitation - the excitation node will be the same as response node, i.e. only excite the response node itself for obtaining its response (maximum response may be missed)
2. Full excitation - Rigorous - excite all the specified excitation nodes for each of the response nodes. For a large model, this will take very long time for the analysis to complete.
3. Full excitation - Rigorous (exclude response node) - excite all the specified excitation nodes for each of the response nodes but exclude the response node. For a large model, this will take very long time for the analysis to complete.
4. Full excitation - Fast - use search technique to determine the nodes to be excited for each of the response nodes in order to reduce the computing time (maximum response may be missed, but better than self excitation)

- **Response nodes** — Node list to define the nodes that the responses will be calculated in the analysis. Responses will not be available if the nodes are not included in the response node list.

- **Excitation nodes** — Node list to define the nodes that will be excited for evaluation of the response of the response nodes. If search technique is used, only a selection of the nodes, that will generate large response at the response nodes according to search technique, will be excited to evaluate the responses of the response nodes. If self excitation method is used, the definition of excitation nodes is not required, so it will be greyed out.

Damping ratio

This defines the critical damping ratios use for the analysis. The options are:

- **Constant for all modes** — the user specifies a constant damping ratio which will be used for all modes.
- **Damping from modal analysis** — use the damping ratios calculated from modal dynamic analysis. If the damping ratio is not available from modal dynamic analysis, the analysis cannot be carried out.
- **Damping frequency relationship table** — query the selected damping table to determine the damping ratio for those modes defined in the table. For all the other modes that damping ratios have not been defined in the damping table, the damping ratio defined below will be used.
- **Number of footfalls** — The number of footfalls to be considered in the analysis.
- **Mass of the walker** — The mass of the person who will walk on the floor or give the weight of the person.
- **Response direction** — The direction of response in GSA global axis direction. It is the response direction in GSA global axis and it is not the Basicentric axes of the human body described in BS6472 and BS6841. It can be in Z (vertical), X, Y or XY plane (horizontal).
- **Frequency weighting curve** - Frequency weighting curve (FWC) to be used in calculating response factors, standard and user defined frequency weighting curve can be used.

According to BS6841, the standard FWC W_b is to be applied to vertical vibration with respect to its effect on health, comfort and perception; standard FWC W_d is to be applied to horizontal vibration with respect to perception etc, and applied to vertical vibrations for people lying horizontally; standard FWC W_g is to be applied to vertical vibration with respect to its effect on hand control and vision. See BS6482 and BS6841 for more detailed description of their uses.

Excitation forces (DLFs)

This defines the way of the structure to be excited (the dynamic Load Factor to be used). The options are:

- **Walking on floor (CCIP-016)** — Walking on floor; Arup method is used.
- **Walking on floor (SCI P354)** — Walking on floor; SCI P354 - "Design of Floors for Vibration: A New Approach" method is used.
- **Walking on stairs (Arup)** — Walking on stairs; Arup method is used.
- **Walking on stairs (SCI P354)** — Walking on stairs; SCI P354 - "Design of Floors for Vibration: A New Approach" method is used.
- **Walking on floor (AISC SDGS11)** — Walking on floor; AISC Steel Design Guide Series 11, Floor Vibration Due to Human Activity.
- **Walking on floor (AISC SDG11 2nd ed)** — Walking on floor; AISC Steel Design Guide 11, second edition, Vibrations of Steel-Framed Structural Systems Due to Human Activity.
- **Walking on stair (AISC SDG11 2nd ed)** — walking on stairs; AISC Steel Design Guide 11, second edition, Vibrations of Steel-Framed Structural Systems Due to Human Activity.
- **Running on floor (AISC SDG11 2nd ed)** — Running on floor; AISC Steel Design Guide 11, second edition, Vibrations of Steel-Framed Structural Systems Due to Human Activity.
- **User defined Dynamic Load Factor** — If they are defined in 'Dynamic Load Factor' table

Walking frequency

Define the range (minimum & maximum) of walking frequencies to be considered in the analysis.

7.4.2.19 Analysis Wizard : GsRaft Control

The GsRaft analysis is a static analysis of a single analysis case including soil structure interaction.

Analysis options: Case description

The description of the analysis case that is being considered.

Analysis options: Maximum number of iterations

The soil structure interaction is an iterative procedure so that maximum number of iterations should be specified. If convergence has not been reached by this stage the user is prompted whether to continue or abandon the analysis.

Analysis options: Pdisp analysis

For the Pdisp part of the analysis the user should choose which model is used for the soil. This can be

- **Mindlin**
- **Boussinesq**

The user should refer to the [Pdisp](#) manual for more information about these soil models.

Displacement convergence control

The solution is converged when the convergence criteria are met. The convergence can be based on either or both of the relative displacement or absolute displacement change from one iteration to the next.

Force convergence control

This is only used for piled-raft analysis and it is the tolerance of the pile-soil interaction force.

Damping and stiffness

The damping and stiffness option allows access to advanced features that may help achieve more rapid convergence.

7.4.2.20 Analysis Wizard : GsRaft Progress

Progress information

These options are to allow the user to view the progress on the solution. The options are

- **Progress report**—a dialog reports convergence as the solution progresses.
- **Show stiffness contour**—a Graphic View with contours of the stiffness of the “soil springs”.
- **Show deformed shape**—show the deformation of the structure as the solution progresses. With this option the deformation magnification should also be specified.

Post-process options: Write Pdisp file

Pdisp input data files may be written every specified number of iterations. This includes the pressure exerted by the raft on the soil at that iteration.

Post-process options: Save soil stiffness

If checked, the nodal support stiffnesses calculated by GsRaft to represent the soil stiffness are preserved in the input data at the end of the analysis. Note that this option modifies the input data. Subsequent GsRaft analysis of any analysis case will use the preserved support stiffnesses as initial soil stiffnesses, as usual.

This option is disabled if any non-GsRaft results exist.

7.4.2.21 Analysis Wizard : GsBridge Control

Two possible methods of performing a the GsBridge Analysis are available, as described in the [Step-by-Step Guide](#), and the Dialog deals with these differently:

Method A – Full carriageway optimisation for UK/HK loading

The code to be used has been specified in the [Bridge Specification](#). It is normal to carry out all stages of this analysis in one operation, in which case all the analysis options should be on (the default). If the user wishes to review and modify the Static Bridge Loads before carrying out the static analysis, then the last two options can be deselected (the only other option available). Note that combination cases will not be generated automatically if this staged option is chosen.

Method B – Lane by lane optimisation

The code is Undefined in the Bridge Specification Table. It is normal to modify the static bridge loads before expanding them, so the default is to do the Influence Analysis only, but some or all of the later steps can be selected as well to give a batch analysis.

In either case the subsequent steps can be carried out using the Tools: [Optimise Bridge Loading](#) and [Expand Bridge Loading](#), and the [Analysis Wizard](#) for the final static analysis.

Retain results at influence points only

Bridge analysis can result in very large post-analysis files. This option trims the results so that only those at the influence points are retained, resulting in significantly reduced file sizes.

7.4.2.22 Analysis Wizard : Envelopes

The use of analysis envelopes is described in the [Step-by-Step Guide](#).

Description

Enter the envelope description here, using the [syntax of combination case descriptions](#).

1D element envelope cases

Select how many analysis cases to generate for 1D element results. Choose from:

- **Worst along element**—generate one pair of analysis cases for each result type, corresponding to the position (as specified in 1D element results) with the greatest result value.
- **Per node**—generate two pairs of analysis cases for each result type, corresponding to node positions 1 and 2.

1D element results

Opens the [1D Element Results dialog box](#) for the specification of the positions along 1D elements at which displacements and forces are to be calculated.

2D element envelope cases

Select how many analysis cases to generate for 2D element results. Choose from:

- **Worst in element**—generate one pair of analysis cases for each result type, corresponding to the position (node or centre) with the greatest result value.
- **Centre only**—generate one pair of analysis cases for each result type, corresponding to the result value at the centre of each element.
- **Per node and centre**—generate up to nine pairs of analysis cases for each result type, corresponding to each node position and the centre of each element.

Tree control

Select which results to generate envelopes for by ticking the boxes. Each nodal result type generates one pair of analysis cases; the number of cases for each element result type depends on the element envelope case settings.

7.4.2.23 Analysis Wizard : Cases Set Up

This page confirms that all the data required for the analysis is present.

Advanced

For most analyses all the information required to specify the analysis is now ready. However, advanced user may wish to adjust some of the solver parameters. The details of what can be adjusted will depend on the model and the analysis options chosen.

Analyse

- **Now**—the analysis is started immediately.
- **Later**—the analysis cases are set up but the analysis is not started. It can be started later with the "Analysis | Analyse" command.

7.4.2.24 Analysis Wizard : Model Stability

A model stability analysis is set up using the Analysis Wizard from the "Analysis | New Analysis Task" menu command. Select the "Model Stability" option under the "Diagnostic" section. The next page allow the user to specify the number of modes, the maximum number of iterations and the shift to be employed.

In most cases a handful of modes (say 5) should suffice. The "correct" number of modes will

depend on how ill conditioned the model is and how many errors there are. As a rule of thumb, if the result from running the analysis returns all eigenvalues as small, double the number of modes requested.

The shift option allows highly ill conditioned models to be solved successfully. If the default value does not work, the shift can be increased by moving the slider to the right.

See [stability analysis](#) for details about the Model stability analysis.

7.4.3 Advanced Solver Settings

The advanced settings are accessed from the “Advanced” button in the [Analysis Wizard](#). It is generally best to use default settings for analyses, however there are times when it is useful to be able to adjust these. This should be done carefully as it is possible to change parameters resulting in inaccurate solutions. The values set here remain for the duration of the current session, but are not saved.

Reset

This resets all the values to the default settings.

More:

[Advanced Solver Settings : Stiffness Solution](#)

[Advanced Solver Settings : Eigensolution](#)

[Advanced Solver Settings : Ritz Analysis](#)

[Advanced Solver Settings : 2D Element Analysis](#)

[Advanced Solver Settings : Non-linear](#)

[Advanced Solver Settings : Convergence Control](#)

[Advanced Solver Settings : Case Control](#)

[Advanced Solver Settings : Time Step](#)

[Advanced Solver Settings : Results](#)

[Advanced Solver Settings : Pre-load](#)

[Advanced Solver Settings : Geometric Stiffness](#)

[Advanced Solver Settings : Raft](#)

7.4.3.1 Advanced Solver Settings : Stiffness Solution

Solution details

There are four matrix solvers in GSA, each with different characteristics:

- **Active Column**
- **Conjugate Gradient**
- **Sparse Direct**
- **Sparse Parallel direct**

The original GSA solver is the active column solver. This is a robust solver for both static and modal solutions, but has largely been superseded by the sparse direct and sparse parallel direct solvers. The default solver is the sparse parallel direct solver - this is the best solver, in most cases, for medium to large modes and can be used for both static and modal solutions (but cannot be used where the model consists of more than one discrete structure). Generally the sparse parallel direct solver is significantly faster than the sparse direct solver since it can take advantage of multiple cores in modern processors.

The conjugate gradient solver is also a sparse solver but is iterative. This solver can handle the largest models, but as it is iterative it can be difficult to know if the solution has converged properly and is generally slower than the sparse direct solver. This solver can only be used for static analysis.

More information is given in the [GSS Theory, Matrix Solver](#) section.

The Conjugate Gradient solver uses a pre-conditioner to improve the rate of convergence: of those available the **Line Jacobi** pre-conditioner is recommended. As the solution is iterative both the maximum number of iterations and convergence tolerance must be specified. Theoretically the solution should converge in no more iterations than the number of degrees of freedom.

Degree of freedom ordering

The way in which the degrees of freedom are ordered can have a significant impact on the size of the stiffness matrix. The default option in GSA is **Geometric** where the degrees of freedom are ordered along the "length" of the structure. In most cases this works well enough, but for some structure types the **Reverse Cuthill-McKee** (or **Cuthill-McKee**) method (a bandwidth minimisation technique) may be more effective.

If the Geometric option is selected either GSA can select the "length" vector (the Automatic determination of front order option) or it can be used to define the vector explicitly.

The sparse direct solver works only with the approximate minimum degree (AMD) algorithm.

Stiffness matrix reduction

- **Limiting minimum:maximum stiffness ratio**—this is the limit where the solution will trigger a warning about possible ill-conditioning problems.
- **Minimum stiffness distinguished from zero**—this is the limit at which a stiffness is considered. Any stiffness value less than this value is considered to be zero.

Calculate error norm / condition number of stiffness matrix

Where there are reservations about the integrity of the model it is useful to get feedback on the solution. The error norm gives a measure of the error in the solution and the condition number gives feedback on the best accuracy that can be expected. These options allow for forcing the calculation of the [error norm](#) / [condition number](#) of the matrix.

Save system files

The stiffness and mass/geometric stiffness matrices can be stored in matrix market format. This can then be loaded into programs such as MATLAB for further analysis.

7.4.3.2 Advanced Solver Settings : Eigensolution

Eigensolver

The Subspace Iteration [Jacobi] is the original eigensolver. The Subspace Iteration with Shifting + Locking is a new solver that uses a faster algorithm and is more robust. Use Subspace Iteration [Jacobi] only if the improved solver fails. Note that while using Subspace Iteration with Shifting + Locking, the only parallel direct solver is the only valid 'Solution method'.

Maximum number of iterations

The maximum number of iterations can be set here and the associated convergence tolerance. The actual maximum number of iterations is the larger of the number of iterations set here and the number specified in the [Analysis Wizard](#).

Jacobi Iteration

When the Subspace Iteration [Jacobi] is selected there is an internal iteration loop (using Jacobi iteration). The maximum number of iterations and the convergence tolerance for the Jacobi iteration can be set here. These rarely need to be adjusted.

Number of subspace vectors

The number of subspace vectors (always higher than the number of modes requested) has an impact on the progress of the iteration. Select 'auto' to use GSA defaults but in case the convergence is slow, try a number between 1.1 times to 2 times the number of modes requested. This is only available for the Subspace Iteration with Shifting + Locking option.

Shift strategy

The shift strategy accelerates the solution in the case of dynamic analysis (but not currently in buckling) so it must be set to 'None' when doing a buckling analysis. For dynamic analysis, select 'Aggressive' for fast convergence and use 'Conservative' or 'None' only if there are issues converging with aggressive. This is only available for the Subspace Iteration with Shifting + Locking option.

Normalisation

The results of a modal analysis are mode shapes which can be scaled arbitrarily. By default modes shapes are normalised so that the maximum displacement is 1m, but different normalisation can be set by selecting an appropriate length unit.

When the analysis is dynamic there is an alternative which is to normalise the modes shape based on the modal mass. This option scales the mode shapes so that the modal mass is 1kg, by default, or to the mass units selected.

Check for missing eigenvalues

The eigensolver determines the eigenvalues and eigenvectors but there is no guarantee that these are the eigenvalues found are the lowest ones. As a final stage in the eigensolution GSA does a Sturm sequence check which is used to check that no eigenvalues have been missed. At times this check fails and the solution fails. This option allows this check to be skipped.

7.4.3.3 Advanced Solver Settings : Ritz Analysis

Randomize the load vector

A Ritz analysis sets up load vectors for each of the required directions which correspond to a unit gravity load. This option applies a randomization to the results node loads. This can be useful to seed torsional modes in symmetric structures.

7.4.3.4 Advanced Solver Settings : 2D Element Analysis

Linear element formulation

Available separately for both in-plane and out-of-plane problems, GSA provides a separate additional method for the formulation of the element's stiffness. In each case there is an original formulation kept for consistency with existing models, and an improved formulation based on refinements to the standard 2D element isoparametric theory. More information is provided below on the differences between the different options.

In general, both later formulations are driven by a desire to upgrade the performance and/or stability of GSA's Linear TRI / QUAD elements, admitting their use in a wider spread of applications where a preference for linear elements is of practical concern. Both make use of changes to the element formulation that help alleviate common problems of shear locking that is symptomatic of standard linear TRI and QUAD elements. As a general rule, the improved formulations perform much better in bending-type problems than their original formulations. Note however that the formulations remain as elements based on linear approximation theory and so while improved, do not match the bending performance of parabolic elements (but at greater computational cost).

For out of-of-plane behaviour, the formulations are named:

- **Mindlin**—Original formulation consistent with versions of GSA v8.4 and previous. Based on Mindlin-Reissner plate theory.

- **MITC**—An improved formulation that both removes the inherent problems of out-of-plane shear locking with the traditional Mindlin formulation while not reducing the number of stiff modes and the consequential problems of [hourglassing](#).

For in-plane behaviour, the formulations are named:

- **Bilinear**—Original formulation consistent with versions of GSA v8.5 and previous. Uses a linear interpolation for the dependent variable in both dimensions.
- **Allman-Cook**—An improved formulation that makes use of the local 'zz' 'drilling' degree of freedom to add four additional stiff modes of deformation. While not required by the basic formulation, these modes both help alleviate in-plane shear locking while also gaining the practical significance of providing access to stiff 'zz' degrees of freedom.

Use the combination box on the 2D Element tab of the Advanced Solver Settings dialog to set the formulations individually for the current task. Alternatively, a preference is available to set the default formulation for all future tasks. By default both options are set to the recommended, later formulations. This preference setting can be found in the miscellaneous tab of the GSA Preferences.

See the section on [GSS Theory](#) for more information.

Considerations when using the Allman-Cook formulation

The Bilinear formulation associates no stiffness with the local 'zz' degrees of freedom in linear 2D elements. In this case, these rotational 'drilling' degrees of freedom are not activated in the solution (or are [suppressed](#)) such that any topological connection to these local freedoms go unrestrained in the solution.

Choosing the Allman-Cook formulation activates the local 'zz' degrees of freedom through refinements to the stiffness formulation that connect the local rotational 'zz' freedoms to the in-plane translational stiffness of the element. The results provides a means to connect a moment applied in the 'zz' direction to translational strain of the element. While this has the clear advantage in practical problems, consideration must be made to allow for a suitably stable connection. Specifically, 'zz' moment connections to 2D elements made over a single node are not recommended as these can cause unwanted local deformations around the individual node and possible imbalances in load transfer. A warning from the solver is provided in this case. Either use additional in-plane beams to distribute the load connection over a local finite area or use a rigid constraint to achieve a similar effect. As an example, if tying a column to a slab, the area under the column itself may be a suitable candidate area to use for the connection.

Suppression of non-stiff degrees of freedom

This governs the method by which the non-stiff degrees of freedom on 2D elements are handled. This problem arises because the 2D elements do not have any zz stiffness. The options are

- **Geometry based automatic constraints, Flatness parameter**—This approach removes the degrees of freedom with no stiffness based on a [geometric or pseudo stiffness criterion](#). The surface is planar and the degree of freedom removed if the minimum principal pseudo stiffness is less than the flatness parameter. This is the recommended option.
- **Stiffness based automatic constraints, Zero stiffness value**—This approach removes the degrees of freedom with no stiffness based on the directions of principal stiffness at the nodes. The degree of freedom surface is removed if the minimum principal stiffness is less than the zero stiffness value.
- **Artificial zz stiffness in shells**—This approach retains the degrees of freedom but assigned a small artificial stiffness to shells in the zz direction. This option should be used with caution.

Geometry checks

Normally when the element geometry is poor the results are likely to be less reliable. The default behaviour of **Treat geometry check failures as Errors** will cause the analysis to abort when poor geometry is found. Selecting **Severe warnings** allows the analysis to continue despite the presence of badly shaped elements.

This option affects geometry checks on

- large internal angles in elements
- warping of flat shell elements

7.4.3.5 Advanced Solver Settings : Non-linear

Convergence control

- Maximum number of iterations — this is maximum number of iterations that are used in a Gss analysis with tie or strut elements.
- Convergence tolerance — this is the limit at which the solution is considered to have converged.

Residual output

A non-linear analysis iterates until the convergence criteria are met or the maximum number of iterations is reached. GSA can create a file with details of the convergence.

- Do not save — a file is never created.
- Save if analysis not converged — this only creates a file if the solution has not converged in the maximum number of iterations.
- Save always — this will always create a file with the convergence details.

7.4.3.6 Advanced Solver Settings : Convergence Control

Accessed by selecting “Advanced” in the [Cases Set Up](#) box in the “Analysis Wizard”. The parameters on this dialog control the iterative process. The following parameters can be adjusted if convergence problems are experienced, or to speed up analysis.

Control Parameters

- **Number of cycles to check residuals**
- **Dummy mass update frequency**
 - Automatic update - dummy mass will be updated once the peak kinetic energy is passed
 - User defined cycle to update
- **Dummy mass and dummy inertia**
 - Automatic
 - User defined
 - dummy mass and dummy inertia factors - to modify the dummy mass and inertia linearly
 - dummy mass and dummy inertia power - to modify the dummy mass and inertia exponentially
- **Damping (percentage of damping)**
 - Choice of using viscous damping or artificial damping
 - viscous damping - damping forces are proportional to nodal velocities
 - artificial damping - a proportional of last nodal displacement is used for the new nodal displacement
 - Adjust damping automatically
 - Use user defined percentage of damping

7.4.3.7 Advanced Solver Settings : Case Control

The analysis will run for a maximum number of cycles or length of time, checking periodically to see if the out of balance or "residual" forces are smaller than the limits set. If they are the model is said to have converged and the analysis stops.

Termination control

Set the maximum number of cycles or the length of time for which the analysis should run before terminating.

Residual control

Set the maximum acceptable out-of-balance conditions.

The convergence of the analysis can be adjusted by selecting the "Advanced" button. This will allow you to adjust the damping and mass, and the frequency at which the dynamic relaxation process is paused to check residuals, and recalculate the dummy mass.

7.4.3.8 Advanced Solver Settings : Time Step

GSA will calculate a time step for the numerical integration if auto is selected. Only override this when a particular time step is required in the time history analysis. No check is carried out to ensure that this will be convergent.

7.4.3.9 Advanced Solver Settings : Results

Nodal results

Allows the user to select which nodal results are to be retained following the calculation. Some results will not be available depending on the solver option requested.

Element results

Allows the user to select which element results are to be retained following the calculation. Some results will not be available depending on the solver option requested.

2D element forces and stresses at centre only

When checked 2D element forces and stresses are stored at centre positions only, not at nodal positions. Tabular output and diagrams will be reported at element centres only; contouring will use the centre value at each nodal position to result in a constant contour value across each element (unless results are averaged at nodes). When combining centre only results with results from tasks that do store results at nodal positions the centre value will be used as the nodal value.

2D elements forces and stresses at centre only

This option means that force/moment and stress results are stored for only the centre of the element. This means that significantly less memory is required to hold the results, with only a minimal loss of data in most models.

7.4.3.10 Advanced Solver Settings : Pre-load

Element pre-load

Allows the user to specify a pre-load to be applied to the elements. This pre-load must refer only to pre-stress beam or 2D loads. The internal loads are used to calculate a geometric stiffness (K_g matrix) for the element which is added into the normal element stiffness K . By providing an "out-of-plane" stiffness this allows pre-tensioned bars to be analysed in the linear solver.

Use of pre-load in a modal P-delta analysis

Note that for a modal P-delta analysis, the geometric stiffness K_g is calculated as a result of solved displacements including the effect of the pre-load. It follows that the derivation of the K_g matrix is to

include the effect of the pre-load as well as the supplied P-delta force.

7.4.3.11 Advanced Solver Settings : Geometric Stiffness

Geometric stiffness options for beam elements

This allows certain terms to be excluded from the geometric stiffness matrix. The options are

- Include all terms in the Kg matrix - this is the default option
- Suppress torsional buckling terms in the Kg matrix - this can be useful in situations where a low torsion constant can lead to spurious torsional modes that would be suppressed by the warping restraint that is not included in the calculation if it is known that the end connection will act to suppress the torsional modes.
- Suppress torsional and lateral-torsional buckling terms in the Kg matrix - similar to previous option but removing more of the destabilizing terms.
- Treat the beam element as a bar for Kg terms - includes only the Euler buckling.

Elements excluded from geometric stiffness

At times it can be useful to exclude some elements from the geometric stiffness matrix and this option allows elements to be specified which should not contribute to the geometric stiffness matrix. If all elements are excluded this reduces a P-Δ analysis to a linear analysis, so it should be used with care.

7.4.3.12 Advanced Solver Settings : Raft

There are times when it may be difficult to achieve convergence, particularly where the soil stiffness varies significantly over the site. Adjusting the damping and stiffness parameters may help achieve convergence.

Damping coefficient

The damping coefficient relates to damping of the numeric solution, and is where the proportion of the old spring stiffness that is to be retained is specified. For a damping coefficient d the new stiffness will be

$$k_{i+1} = d \cdot k_i + (1 - d) \frac{f}{x}$$

Initial support stiffness

GsRaft needs an initial stiffness for the soil. The closer this is to the actual soil stiffness the more rapid the convergence is likely to be. At the start of the analysis GsRaft assigns this initial support stiffness to the support stiffness of all soil interaction nodes that do not have support stiffness explicitly defined.

7.4.4 GsRaft Analysis Progress

This dialog is opened when a GsRaft analysis is started and keeps the user informed of the solution progress in a GsRaft analysis.

Damping

This allows the user to adjust the solution damping parameter.

Residual

This allows the user to adjust the displacement residual.

Pause / Start

This allows the user to pause and restart the GsRaft analysis.

Stop

This will terminate the GsRaft analysis.

7.4.5 GsRaft Damping Coefficient

This dialog is displayed if the user selects Damping from the “GsRaft Analysis Progress” dialog. This allows adjustments to be made to the solution damping, which supersede the values set for the GsRaft analysis in the [Raft page of the Advanced Solver Settings dialog](#).

7.4.6 GsRaft Displacement Residual

This dialog is displayed if the user selects Residual from the “GsRaft Analysis Progress” dialog. This allows adjustments to be made to the convergence criteria that supersede the values set for the GsRaft analysis in the [Analysis Wizard](#).

7.4.7 View List

The View List dialog is used for creating a new [view list](#) and for editing an existing view list. It is displayed by selecting “Edit “<view list name>” ” or “New View List” on the right-click menu having right-clicked on a view list on the [Views tab in the Gateway](#).

View list name

The name by which the view list is referred. Giving the same name as another view list will result in that view list being overwritten by this. It is preferable to give a name that is distinct from any saved or preferred views since in some circumstances action is taken on the first view or view list with a given name.

Available Views

This tree control displays the names of all saved and preferred Graphic Views and Output Views. Views that are already represented in the view list are marked with a tick. Double clicking on a view that is not already in the view list will add the view to the bottom of the list. Double clicking on a view that is in the view list removes all occurrences of that view from the list.

View List (in print order)

The view list.

Add

The selected view in the Available Views tree control is added to the bottom of the View List

Remove

The selected view in the View List is removed from the view list.

Move Up

The selected view in the View List is moved up one position.

Move Down

The selected view in the View List is moved down one position.

Print on OK

The views in the View List are printed in the order in which they appear in the list, upon clicking OK. The printer to which the views are sent is set in 'File | Print Setup'.

Save to file on OK

The views in the View List are saved to file in the order in which they appear in the list, upon clicking OK. The file format is determined by the 'Batch output file formats' settings on the [Miscellaneous page](#) of the [Preferences dialog](#).

Part VIII

8 Other Dialogs

[Graphic View Dialog Boxes](#)

[Sculpt Dialog Boxes](#)

[Output View Dialog Boxes](#)

[Tools dialogs](#)

[Miscellaneous Dialogs](#)

[Preference Dialogs](#)

8.1 Graphic View Dialogs

[Wizard: Graphic Settings](#)

[Labels and Display Methods](#)

[Deformation Settings](#)

[Contour Settings](#)

[Diagram Settings](#)

[Bridge Options](#)

[Further Options](#)

[Animation Settings](#)

[Orientation Settings](#)

[Graphic Fonts and Styles](#)

8.1.1 Wizard: Graphic Settings

The Wizard: Graphic Settings dialog box allows the parameters that govern the format and content of Graphic Views to be specified. All Graphic View settings can be specified either in this Graphic Settings dialog box or in dialog boxes accessible from this.

The default Graphic View settings are stored as preferences. Refer to [Working with the Program: – Default View Settings](#) for details.

Open this dialog box by giving the “Graphics | Graphic Settings” (Ctrl+W) menu command or the Wizard button on the [Data Options toolbar](#) when the current view is a Graphic View.

View name

When specified and when the legend is displayed, this text appears at the top of the legend, both on the screen and on printed output.

This is also the [saved view name](#).

Structure scale, Scale

Specifies the scale at which the structure is to be drawn. This is disabled when “Do scale to fit” is checked.

Structure scale, Lock Scale

When checked the specified scale is not affected by changing the window size or sending the image to a printer. This setting is overridden by any scale to fit or zoom operation.

Structure scale, Do scale to fit

When checked a [scale to fit](#) is done. This operation modifies the scale, eye to object distance,

object point and mid-point.

The “Scale to fit” option on the [Orientation toolbar](#) may also be used to do this.

Structure scale, Engineering scale

When checked any scaling to fit produces an engineering scale.

Structure scale, Printed image

When the printed image scale method is set to **Auto**, the scale used for printing is based on the current scale on the screen and whether this was scaled to fit, zoomed or explicitly defined. When the printed image scale is specified then this is used for printed output regardless of the current scale on the screen. Refer to [“Scaling for printed output \(and changed window sizes\)”](#) for details.

Shrink elements

When checked the size of drawn elements is reduced by the [preferred shrink factor](#). Details are given in [Working with the Program: – Shrinking elements](#).

This may also be set using the “Graphics | Display | Shrink” menu command or via the [Display toolbar](#).

Shade surfaces

When checked the shade of colour filled surfaces is adjusted according to the current lighting settings. Details are given in [Working with the Program: – Shading surfaces](#).

This may also be set via the [Display toolbar](#).

Highlight edges

When checked, intended and unintended edges of the structure are highlighted. Details are given in [Working with the Program: – Highlighting element edges](#).

This may also be set using the “Graphics | Display | Highlight Edges” menu command or via the [Display toolbar](#).

Case list

Specifies the list of cases, nodes, elements or members to be represented in diagrams, contours and the deformed shape. This is ignored for data that is case independent.

Note that, while typically a single case will be specified, a list of cases can be specified to result in diagrams and deformations being superimposed for all cases. Contours are only displayed for one of the cases in the list.

Refer to the [list syntax](#) for details.

The cases may also be specified on the [Lists toolbar](#) associated with the view.

Print by case

When checked, diagrams and contours for each case specified in the case list are printed separately. See [Working with the Program: – Printing from Graphic Views](#). This also affects the saving of the image to file in some file formats; see [Working with the Program: – Saving the graphic image to file](#).

Note that when “Recalculate extents of displayed data on change of case” in the [Graphics Preferences](#) is checked and the data scale is not locked in the [Further Options dialog box](#) for the contour or diagram the data scale will be recalculated for each case.

Entities, Node list | Element list | Member list

Specifies the list of nodes, elements or members that are to be drawn.

When the Analysis layer is being displayed only elements that are connected to nodes in the node list and included in the element list are drawn.

When the Design layer is being displayed only members that are connected to nodes in the node list and included in the member list are drawn.

Refer to the [list syntax](#) for details.

The entities may also be specified on the [Lists toolbar](#) associated with the view. Selecting All on this toolbar resets all entity lists to “All” (and deletes any specified volumes).

Entities, Clear all volumes

When checked, any specified volumes are deleted. A description of the use of volumes is given in [Working with the Program: – Volume clipping](#).

Note that selecting All on the [Lists toolbar](#) associated with the view also has the effect of deleting any specified volumes (and resets all entity lists to “All”).

Entities, Display excluded faint

When checked, entities excluded from display, whether by exclusion by volume or display list, are drawn faint, rather than not at all. (Auto-scaling ignores excluded entities regardless of this setting.)

Layer, Analysis (elements) | Design (members)

Select Analysis to display the analysis model and Design to display the design model.

Refer to [Working with the Program: – Analysis and Design Layers](#) for details.

Stage

Select the analysis stage to be displayed. This option is only enabled when the analysis stages [advanced feature preference](#) is enabled.

Refer to [Step by Step Guide: – Analysis Stages](#) for a discussion on this topic.

Legend, Show legend panel

When checked, the legend is displayed.

This may also be set using the “Graphics | Display | Legend Display” (Ctrl+Alt+L) menu command.

Note that the same information may be displayed in a message box by giving the “Graphics | View Legend” menu command. In this circumstance the contour colour panel is excluded.

Legend, Width of legend panel

The width of the legend panel is set to this size in mm on the picture.

Legend, Picture area to exclude legend panel

When checked, the picture area used for the legend is used exclusively for the legend and that used for the display of the image is reduced by the width of the legend panel. Otherwise the legend is superimposed on the image.

Automatic scaling is based on the picture area, – excluding the legend panel when this option is checked.

Apply

Applies the current settings to the image without exiting the dialog box.

Copy

Copy all of the current view settings to the clipboard in text format, GWA syntax).

Paste

Paste the view settings from the clipboard. The view settings must be in text format, GWA syntax.

Reset display

Resets all settings except the data type to the [default view settings](#).

Refer to [Working with the Program: – Resetting the display](#) for details.

This may also be set using the “Graphics | Display | Reset Display” menu command or via the [Display toolbar](#).

Other settings

All of the options in this group open the respective settings dialog box. Other ways of accessing these settings are as follows.

[Labels](#) – “Graphics | Display | Labels and Display Methods” menu command or via the [Display toolbar](#). Commonly used labelling options are available on the [Display Favourites toolbar](#)

[Deformation](#) – “Graphics | Display | Settings | Deformation Settings” menu command. The deformed image may be switched on or off via the [Display toolbar](#).

[Contour](#) – “Graphics | Display | Settings | Contour Settings” menu command or via the [Display toolbar](#).

[Diagram](#) – “Graphics | Display | Settings | Diagram Settings” menu command or via the [Display toolbar](#). Commonly used labelling options are available on the [Display Favourites toolbar](#)

[Animation](#) – “Graphics | Display | Settings | Animation Settings” menu command.

[Orientation](#) – “Graphics | Orientation | Orientation Settings” menu command.

[Translucency](#) – “Graphics | Orientation | Translucency Settings” menu command.

[Lighting](#) – “Graphics | Orientation | Lighting Settings” menu command.

[Unwrap](#) – “Graphics | Orientation | Unwrap Settings” menu command.

See also:

[Working with the Program — Working with Graphic Views](#)

[Working with the Program — Working with Saved Views and Preferred Views](#)

[Program Fundamentals — Sets and Lists](#)

8.1.2 Labels and Display Methods

The Labels and Display Methods dialog box allows labels to be switched on and off and the method by which entities are represented to be adjusted.

Some of the more frequently used labelling options may also be switched from the [Display Favourites toolbar](#).

Open this dialog box by giving the “Graphics | Display | Labels and Display Methods” menu command or from the [Graphic Display toolbar](#) when the current view is a Graphic View. This dialog box may also be opened from the [Graphic Settings dialog box](#).

See also:

[Graphic Settings dialog box](#)

More:

[Labels and Display Methods: Axes](#)

[Labels and Display Methods: On Nodes](#)

[Labels and Display Methods: On Elements](#)

[Labels and Display Methods: On Geometric Entities](#)

[Labels and Display Methods: Display Methods](#)

8.1.2.1 Labels and Display Methods: Axes

The Axes page of the Labels and Display Methods dialog box is where various types of axes may be switched on for display in the current Graphic View. All axis sets are displayed as colour coded axes triads.

Options are as follows.

- **Global axes**
- **Output axes**—Display all axis sets referred to by all diagram and contour displays in the current Graphic View.
- **User axes**—Display all user axes for which the origin lies within all currently defined [volumes](#).
- **Grid plane axes**—Display all grid plane axes at the grid elevation.
- **Storey references**
- **Due north**—Display a north arrow in the legend axes triad.

Reset Page

Resets the label settings on this page to the [default view settings](#).

Apply

Applies the current settings to the image without exiting the dialog box.

See also:

[Graphic Settings dialog box](#)

8.1.2.2 Labels and Display Methods: On Nodes

The On Nodes page of the Labels and Display Methods dialog box is where various types of node related labels may be switched on for display in the current Graphic View.

Node label options are as follows.

- **Dots at nodes**
- **Node references**
- **Constraint axis ref.s** are labelled in the format “a” followed by the axis set number, only at nodes for which the constraint axes are not global.
- **Restraints and Supports**—Restraints are represented either by symbol or by text label, depending on the Graphic Data Display Preference. Spring supports are labelled in the format “SprSpt.” followed by the direction codes.
- **Joints**—Master nodes are labelled in the format “Jt” followed by the direction codes. A line is drawn from each slave node to its master.
- **Rigid constraints**—Master nodes are labelled in the format “Rgd” followed by the direction codes. A line is drawn from each slave node to its master.
- **Constraint equations**—Slave nodes are labelled in the format “CEq” followed by the direction codes. A line is drawn from each master node to its slave.

Data references, such as constraint axis references, may be labelled either by number, by name or by name and number. Except for node references if no name is given then the reference is by number, regardless of this setting.

The **Apply to** group of options determines which nodes in the current Graphic View are to have the node labels applied. They are as follows.

- **Apply to, All**

- **Apply to, Selection**—The labels are to be applied to the currently selected nodes. This is only enabled when the cursor mode is “Select Nodes” and some nodes are selected.
- **Apply to, Current set**—The labels are to be applied to the same set of nodes to which labels are currently applied, regardless of the current [cursor mode](#) and selection set. This is only enabled when node labels were previously applied to a selection set.

Reset Page

Resets the label settings on this page to the [default view settings](#).

Apply

Applies the current settings to the image without exiting the dialog box.

See also:

[Graphic Settings dialog box](#)

8.1.2.3 Labels and Display Methods: On Elements

The On Elements page of the Labels and Display Methods dialog box is where various types of element related labels may be switched on for display in the current Graphic View.

Element label options are as follows.

- **Element references**
- **Property references** are labelled in the format “pb” (etc. depending on element type) followed by the number.
- **Design prop. references** are labelled in the format “d” followed by the design property number. (Members only.)
- **Restraint prop. references** are labelled in the format “r” followed by the number. (Members only.)
- **Group numbers** are labelled in the format “g” followed by the number.
- **Material references** are labelled with the material name. User material names are preceded by “m” followed by the number.
- **Section descriptions**—Beam section descriptions are labelled for all beams and bars, etc.
- **Section shapes**—Outlines of cross-sections are drawn for all beams and bars, etc.
- **Orientation node dots**—Node dots are drawn at all nodes that are referenced as orientation nodes by the currently displayed elements.
- **Orientation nodes** are labelled in the format “n” followed by the number.
- **Orientation angle (total)**—The angles are labelled in the format “o” followed by the angle in degrees. The angle is the orientation angle required to achieve the orientation without the influence of an orientation node or property axis set.
- **2D element edges**—An arrow aligned with edge one is drawn from the first node to the second node of each 2D element.
- **Initial stage ref.s** are labelled in the format “s” followed by the number of the first analysis stage in which the element appears.
- **Element lengths** are labelled for 1D elements only.
- **Element releases**—Lines are drawn for each release to represent the release axis at the position the release is specified.
- **Element x axis**—Arrow-heads pointing in the direction of the element x direction are drawn at the centre of each element.

- **Element axes**—Axes triads representing the element axes are drawn at the centre of each element.
- **Principal axes**—Axes triads representing the element principal axes are drawn at the centre of each element for which the principal axes are not aligned with the element axes.
- **Load panels**—On 2D load elements the reference edge is identified and lines are drawn to indicate the load pattern.

Data references, such as property references, may be labelled either by number, by name or by name and number. Except for element references if no name is given then the reference is by number, regardless of this setting.

The **Apply to** group of options determines which elements in the current Graphic View are to have the element labels applied. They are as follows.

- **Apply to, All**
- **Apply to, Selection**—The labels are to be applied to the currently selected elements. This is only enabled when the [cursor mode](#) is “Select Elements” and some elements are selected.
- **Apply to, Current set**—The labels are to be applied to the same set of elements to which labels are currently applied, regardless of the current cursor mode and selection set. This is only enabled when element labels were previously applied to a selection set.

Reset Page

Resets the label settings on this page to the [default view settings](#).

Apply

Applies the current settings to the image without exiting the dialog box.

See also:

[Graphic Settings dialog box](#)

8.1.2.4 Labels and Display Methods: On Geometric Entities

The On Geometric Entities page of the Labels and Display Methods dialog box is where various types of geometric entity related labels may be switched on for display in the current Graphic View.

Line label options are as follows.

- **Dots along lines**—displays dots at the approximate positions at which element corner nodes will be generated.
- **Line references**
- **Constraint axis ref.s**
- **Restraints and Spts.**—Restraints are represented either by symbol or by text label, depending on the Graphic Data Display Preference. Spring supports are labelled in the format “SprSpt.” followed by the direction codes.

Area label options are:

- **Area references**
- **Area types**
- **Property ref.s** are labelled in the format “p” followed by the number.
- **Group numbers** are labelled in the format “g” followed by the number.

Region label options are:

- **Region references**

- **Region types**

Assembly label options are:

- **Assembly ref.s**—displays both numbers and names.
- **Assembly x axis**—displays a line along the assembly x axis
- **Assembly axes**—Axes triads representing the assembly axes are drawn at the assembly axis origin for each assembly.
- **Assembly shapes**—Outlines of assemblies are represented by centre-lines of each element flattened into a cross-section per assembly.

Data references, such as constraint axis references, may be labelled either by number, by name or by name and number. Except for entity references if no name is given then the reference is by number, regardless of this setting.

The **Apply to** group of options determines which entities in the current Graphic View are to have the entity labels applied. They are as follows.

- **Apply to, All**
- **Apply to, Selection**—The labels are to be applied to the currently selected entities. This is only enabled when the [cursor mode](#) is “Select Lines”, “Select Areas” or “Select Regions” and some entities are selected.
- **Apply to, Current set**—The labels are to be applied to the same set of geometric entities to which labels are currently applied, regardless of the current cursor mode and selection set. This is only enabled when entity labels were previously applied to a selection set.

Reset Page

Resets the label settings on this page to the [default view settings](#).

Apply

Applies the current settings to the image without exiting the dialog box.

See also:

[Graphic Settings dialog box](#)

8.1.2.5 Labels and Display Methods: Display Methods

The Display Methods page of the Labels and Display Methods dialog box is where the form and colour of the various element types may be specified and where the drawing of unattached nodes may be specified.

Beams and Bars may be drawn as follows.

- **Line**
- **Section Outline**—The edges of the section are drawn. The hidden lines are not removed.
- **Section Filled**—The faces of the section are drawn.
- **Outline & Filled**—The edges and faces of the section are drawn; only the faces are coloured according to the Colour Elements setting.
- **Outline & Filled Bkgnd.**—The edges of the section are drawn. The faces are drawn in the background colour to simulate the removal of the hidden lines.
- **Off**—Beams and bars are not drawn.

An equivalent solid rectangular section is calculated for elements that have explicitly defined beam section properties.

In this context bar refers to all generic bar elements.

1D Springs may be drawn as follows.

- **Line**
- **Coil**
- **Off**—1D springs are not drawn.

0D and Support Springs may be drawn as follows.

- **Symbol**
- **Coil**
- **Off**—0D and support springs are not drawn.

Note that this applies to both Ground Spring elements and nodes with support stiffness.

Masses may be drawn as follows.

- **Symbol**
- **Weight**
- **Off**—Mass elements are not drawn.

2D Elements may be drawn as follows.

- **Outline**
- **Outline & Filled**
- **Outline, Filled & Thickness**
- **Filled**
- **Filled & Thickness**
- **Off**—2D elements are not drawn.

When the elements are drawn “filled” only the faces are coloured according to the Colour Elements setting.

When the thickness is drawn the top and bottom surfaces are drawn offset from the plane of the element by half the thickness.

Links, Cables and Spacers may be drawn as follows.

- **On**
- **Off**

Soil zones and profiles may be drawn as follows.

- **Outline**
- **Outline & Filled**
- **Filled**
- **Off**—Soil zones and profiles are not drawn.

The **Colour Elements** options are as follows.

- **By type**—Elements are drawn in their standard colour. Note that 2D elements are coloured differently on the top (positive z) and bottom (negative z) surfaces.
- **By property**
- **By group**
- **By material**
- **By initial stage**

When colouring by property, group or material the program assigns colours by dividing a colour spectrum into as many parts as there are properties, groups or materials in the model.

When colouring by initial stage the program assigns colours by dividing a colour spectrum into as many parts as there are analysis stages in the model. An element is coloured according to the first analysis stage it appears in.

The **Draw inactive elements** option draws, as wire-frame, elements that are flagged as being dummy (non-analysed) elements and elements that are not included in the current stage. Otherwise such inactive elements are not drawn.

The **Draw current grid panels** option draws panels on the [current grid](#). A panel is an area bounded by beam elements. In the process of expanding grid loading valid panels are identified for the grid plane to which the load is applied. Invalid panels are displayed in red.

The **Draw load panel edges** option highlights beam elements that are identified as forming an edge to a load panel element. For loading to be transferred from a load panel to the surrounding beam elements the load panel must be completely bounded by beam elements.

The **Draw offset marks** option draws lines between the node and the flexible part of the element for elements that are offset, to represent the rigid arm of the offset.

The **Draw nodes at shrunk position** shifts the position at which nodes are displayed to the average of half-way towards the shrunk end of attached elements. This is useful for identifying coincident nodes and their attachments (e.g. at joints or tied interfaces).

The **Draw unattached nodes** group of options is as follows.

- **Unused**—Draw dots at nodes that have not been referenced.
- **Orientation**—Draw dots at nodes that have been referenced as orientation nodes (but not as an element topology).
- **Constrained** —Draw dots at nodes that have been referenced by a constraint, e.g. a Rigid Constraint, Joint, Constraint Equation or Tied Interface (but not as an element topology).

Reset Page

Resets the display method settings to the [default view settings](#).

Apply

Applies the current settings to the image without exiting the dialog box.

See also:

[Graphic Settings dialog box](#)

8.1.3 Deformation Settings

The Deformation Settings dialog box allows the parameters that govern the drawing of the deformed image to be specified.

Open this dialog box by giving the “Graphics | Display | Settings | Deformation Settings” menu command when the current view is a Graphic View. This dialog box may also be opened from the [Graphic Settings dialog box](#).

Draw deformed shape

When checked, the deformed shape is drawn for each case specified in the Cases list on the [Lists toolbar](#).

The deformed shape may also be switched on and off from the [Graphic Display toolbar](#).

Draw initial state (undeformed)

When checked, the undeformed shape is drawn faint whenever the deformed shape is drawn. Otherwise the undeformed shape is not drawn when the deformed shape is drawn.

Scaling, Magnification

Specifies the amount by which the displacements are magnified in the deformed image (but see also “[Factor by](#)” below). This is disabled when “Do auto-scale” is checked.

Scaling, Factor by

The calculated or specified magnification value is factored by this amount to arrive at the total magnification factor used when magnifying the displacements for the deformed image.

The “Double size of diagrams” and “Halve size of diagrams” options on the [Graphic Display toolbar](#) adjust this value.

Scaling, Lock scale

When checked the specified scale is not affected by any auto-scaling.

Scaling, Do auto-scale

When checked, an auto-scale is done. See [Scaling of diagrams, contours and deformations](#) for details.

The “Rescale data” option on the [Graphic Display toolbar](#) may also be used to do this.

Scaling, Engineering scale

When checked any auto-scaling produces an engineering scale. (Defaults to the preference setting for each new Graphic View.)

1D Elem Results

Opens the [1D Element Results dialog box](#) for the specification of the positions along 1D elements at which displacements and forces are to be calculated.

Note that as well as specifying the number of points at which the deformed shape is calculated, the 1D Elem Results settings offered here are also the default settings for all new contour and diagram settings for the current Graphic View.

Apply

Applies the current settings to the image without exiting the dialog box.

See also:

[Working with the Program — Working with Graphic Views — Deformed image](#)

[Working with the Program — Working with Graphic Views — Scaling of diagrams, contours and deformations](#)

8.1.4 Contour Settings

The Contour Settings dialog box allows the parameters that govern the display of contours to be specified.

Open this dialog box by giving the “Graphics | Display | Settings | Contour Settings” menu command or from the [Graphic Display toolbar](#) when the current view is a Graphic View. This dialog box may also be opened from the [Graphic Settings dialog box](#).

Data (tree control)

The selected data will be contoured for the first case only in the Cases list on the [Lists toolbar](#).

Reset

Resets all settings except the data type to the [default view settings](#).

Clear

Clears the current contour settings to produce no contouring.

Envelope method

When the case being contoured is an enveloping combination case, this setting determines what values of the envelope are output. Select whether to contour the Minimum, Maximum, Absolute (i.e. the greater of the absolute Min and absolute Max) or Signed Absolute (i.e. as Absolute but keeping the original sign) values.

Output of 2D Element Results, Stress layer

Select whether top, middle, bottom or bending stresses are to be contoured.

This is enabled only when a 2D element result is selected.

Output of 2D Element Results, Centre values only

When checked, the centre value is applied to the nodal positions per element to result in constant contouring values for each element.

This is enabled only when a 2D element result is selected.

Output of 2D Element Results, Average stresses and forces at nodes

When checked, results are averaged across elements at common nodes. (See [Output Options — Stress averaging in 2D elements](#) for details.)

This is enabled only when a 2D element result is selected and when 'Centre values only' is not selected.

Number of contours

Specifies the number of contours. The number of contour intervals is one less than this number.

The specified number of contours is used only as a guide when “Automatic contour values | Contours based on rounded range” is set.

This field is disabled when the contour values are specified in terms of a “constant interval”.

Automatic Contour Values

- **Contours based on rounded range** — contours are dispersed at rounded intervals between the minimum and maximum extents of data values being contoured, rounded down and up, respectively.
 - **Set middle contour to zero** — the middle contour is set at zero and contours are assigned at different rounded intervals above and below zero to cover the rounded range. Where all data being contoured is +ve the lowest contour is set at zero (and vice versa for all -ve data).
 - **Equal +ve and -ve contour intervals** — the middle contour is set at zero and contours are assigned at the same rounded intervals above and below zero to cover the rounded range.

For example, if the range being contoured is -25.2 to 14.1:

- “Contours based on rounded range” produces contours from -30 to +15 in intervals of 5.
- “... - Set middle contour to zero” produces contours at -30 -20 -10 zero +5 +10 +15.
- “... - Equal +ve and -ve contour intervals” produces contours from -30 to +30 in intervals of 5.

When contours are based on a rounded range the specified number of contours is used only as a guide.

- **Contours based on actual values** — the specified number of contours is evenly dispersed between the minimum and maximum extents of data values being contoured.
- **Specify constant interval** — contours are spaced at the specified interval, based at zero or a multiple of ten if zero is not within the range being contoured.

Note that these options use the automatically calculated data extents. These may or may not be recalculated upon a change of case, depending on the current preferences but will be recalculated

upon a “Rescale data”. (See [Working with Graphic Views — Scaling of diagrams, contours and deformations](#) for details.)

See the section on 'Graded by value' for the behaviour of 'Automatic Contour Values' in that circumstance.

Specify contour values and colours

When “Specify contour values and colours” is selected clicking the adjacent “Specify” button opens the Contour Values dialog. In this, contour values (or percentages) and colours may be specified for some or all contours. The values of contours for which no value is specified are interpolated between known contour values, where the lowest contour defaults to the minimum extent and the highest to the maximum extent. The colours of contours for which no colour is specified are also interpolated. For filled contours the colour relates to the contour interval between “this” contour and the next. The lowest contour is numbered zero.

Labels may also be specified in the Contour Values dialog. These are displayed in the legend against the respective contour interval.

Constant size on nodes, and 0D and 1D elements

When set, the size of contour blobs at nodes and 0D elements and the thickness of the displayed contour barrels is determined by the “Nodes, and 0D and 1D element contour size” setting in the [“Graphic Fonts and Styles”](#) dialog and not scaled according to the value being contoured. The scaling factor is applied.

Don't paint white contours

When set, contour intervals that would have been painted white are left unpainted. This includes out-of-range values and intervals for which the colour has been manually set to white.

Contours: Line | Filled | Graded by value

Contours may be displayed as lines along the contours or as filled intervals.

The 'Graded by value' option contours only data at the specified contour values; data at other values are treated as out of range. When contour values are calculated automatically for this option a contour value is set for every discrete data value encountered in the part of the structure being contoured, and colours are assigned automatically. 'Graded by value' is only available for some data types, e.g. properties, grid loading, ...

Value | Percent

When “Percent” is selected the contour values specified are interpreted as a percentage of the extents of the data being contoured. Otherwise the values are interpreted as being data values in the current contour setting units.

This field is disabled when “Contours based on rounded range” or “Contours based on actual values” is set.

Magnitude | Signed

When “Magnitude” is selected the sign (+ or –) of the data values being contoured is ignored and the contour range is positive. Otherwise the sign is taken into account.

1D Elem Results

Opens the [1D Element Results dialog box](#) for the specification of the positions along 1D elements at which displacements and forces are to be calculated.

Units

Opens the [Units dialog box](#) for the specification of the units in which the contour values are to be expressed.

Numeric Format

Opens the [Numeric Format dialog box](#) for the specification of the numeric format in which the

contour values are to be expressed.

Axes

Opens the [Axis dialog box](#) for the specification of the axes in which the contour values are to be transformed.

Further Options

Opens the [Further Options dialog box](#).

See also:

[Working with the Program — Working with Graphic Views — Contours](#)

[Working with the Program — Working with Graphic Views — Scaling of diagrams, contours and deformations](#)

[Output Options](#)

8.1.5 Diagram Settings

The Diagram Settings dialog box allows the parameters that govern the display of diagrams to be specified.

This is a multi-paged dialog box. Each page holds the settings for one diagram. On entry, each currently specified diagram appears on a separate page and the last page is named “New Diagram”. The “New Diagram” page may be used to specify another diagram in addition to what is already displayed. Alternatively existing diagrams may be re-specified.

Some of the more frequently used diagrams may also be switched from the [Display Favourites toolbar](#).

Open this dialog box by giving the “Graphics | Display | Settings | Diagram Settings” menu command or from the [Graphic Display toolbar](#) when the current view is a Graphic View. This dialog box may also be opened from the [Graphic Settings dialog box](#).

Data (tree control)

The selected data will be represented as a diagram for all cases in the Cases list on the [Lists toolbar](#).

Reset

Resets all settings except the data type to the [default view settings](#).

Clear

Clears the current diagram.

Draw loads as components in output axes directions

When checked, loading diagrams are drawn as three components in output axes directions. Otherwise loading is shown in the direction in which it is applied. (Beam loading is always broken down into components in output axes directions.)

Output of 2D Element Results, Stress layer

Select whether diagrams are to be of top, middle, bottom or bending stresses.

This is enabled only when a 2D element result is selected.

Output of 2D Element Results, Centre values only

When checked, diagrams are drawn at the centre position on the element only and not the nodal positions.

This is enabled only when a 2D element result is selected.

Output of 2D Element Results, Average stresses and forces at nodes

When checked, results are averaged across elements at common nodes. (See [Output Options — Stress averaging in 2D elements](#) for details.)

This is enabled only when a 2D element result is selected and when 'Centre values only' is not selected.

Data Extents, Activate extents

When checked, a range of interest, or extents, can be specified. Portions of diagrams that do not lie within the specified extents are drawn faint (but see [Hide excluded diagrams](#), below).

Data Extents, Min. and Max.

These specify the minimum and maximum extents.

Data Extents, Value | Percent

When "Percent" is selected the extents specified are interpreted as a percentage of the extents of the diagram data. Otherwise the values are interpreted as being data values in the current diagram units.

Data Extents, Magnitude | Signed

When "Magnitude" is selected the sign (+ or -) of the diagram data values is ignored. Otherwise the sign is taken into account.

When "Exclusive" is selected values **within** the extents are deemed to be **out** of range. Otherwise values within extents are within range.

Data Extents, Hide excluded diagrams

When checked the out of range portions of diagrams are not displayed. Otherwise the out of range portions of diagrams are drawn faint.

Annotate envelope

When annotating envelope diagrams select whether to annotate the Minimum and Maximum, Minimum only or Maximum only values.

Draw diagrams in plane of picture

When checked, all diagrams are drawn in the plane of the picture regardless of the direction associated with the data being represented. This allows diagrams to be displayed when the orientation of the image would otherwise cause the diagram to be invisible; for example, when drawing vertical (global Z direction) loads on a plan view.

Draw simple diagrams

When checked, diagrams are drawn in their simplest state. Normally diagrams along 1D elements are drawn with bars at points along the element at which values have been calculated. Simple diagrams exclude these bars.

Bridge Options

Opens the [Bridge Options dialog box](#). This button is visible only when the bridge analysis [advanced feature preference](#) is enabled. The button is enabled only when a relevant data option is selected.

1D Elem Results

Opens the [1D Element Results dialog box](#) for the specification of the positions along 1D elements at which displacements and forces are to be calculated.

Units

Opens the [Units dialog box](#) for the specification of the units in which the diagram values are to be expressed.

Numeric Format

Opens the [Numeric Format dialog box](#) for the specification of the numeric format in which the

diagram values are to be expressed.

Axes

Opens the [Axis dialog box](#) for the specification of the axes in which the diagram values are to be transformed.

Further Options

Opens the [Further Options dialog box](#).

Apply

Applies the current settings to the image without exiting the dialog box.

See also:

[Working with the Program — Working with Graphic Views — Diagrams](#)

[Working with the Program — Working with Graphic Views — Scaling of diagrams, contours and deformations](#)

8.1.6 Bridge Options

The Bridge Options dialog box allows parameters that relate specifically to bridge analysis diagrams to be specified.

Open this dialog box from the [Diagram Settings dialog box](#). It is only accessible when a relevant diagram data option is selected.

Alignments

Specify the list of alignments that are to be included. This option is disabled when it is not relevant to the current diagram.

Paths

Specify the list of paths that are to be included.

Influence effect

Specify the Node Influence Effect or Beam Influence Effect that is to be considered. This option is disabled when it is not relevant to the current diagram.

Draw centre line

When checked the centre lines of paths are drawn.

Draw edges

When checked the edges of paths are drawn. This option is disabled when it is not relevant to the current diagram.

Label centre lines

When checked alignment and path centre lines are labelled.

8.1.7 Further Options

The Further Options dialog box allows less frequently used parameters that govern the display of contours and diagrams to be specified.

Open this dialog box from the [Contour Settings](#) or [Diagram Settings](#) dialog box.

Scaling, Scale

Specifies the scale at which the contour or diagram is to be drawn (but see also "[Factor by](#)" below). This is disabled when "Do auto-scale" is checked.

When contouring the scale is used in the contouring of nodal data, 0D element data for the diameter of the balloons and 1D element data for the diameter of the barrels. Scaling does not come into the contouring of 2D element data.

Scaling, Factor by

The calculated or specified scale is factored by this amount to arrive at the total scale used.

The “Double size of diagrams” and “Halve size of diagrams” options on the [Graphic Display toolbar](#) adjust this value.

Scaling, Lock scale

When checked the specified scale is not affected by any auto-scaling.

Scaling, Do auto-scale

When checked, an auto-scale is done. See [Scaling of diagrams, contours and deformations](#) for details.

The “Rescale data” option on the [Graphic Display toolbar](#) may also be used to do this.

Scaling, Engineering scale

When checked any auto-scaling produces an engineering scale. (Defaults to the preference setting for each new contour or diagram specified.)

Annotation

Specify for which entities this contour or diagram is to be annotated, as follows.

- **Annotation, All**—This contour or diagram is annotated for all entities regardless of whether they are selected in the “Select for annotation” cursor mode.
- **Annotation, By Selection**—This contour or diagram is annotated for entities selected in the [“Select for annotation”](#) cursor mode.
- **Annotation, None**—This contour or diagram is not annotated for any entities regardless of whether they are selected in the “Select for annotation” cursor mode.

Usually it is convenient to leave the annotation setting set at “by selection” and use the “Select for annotation” cursor mode to interactively interrogate the image. When working with contouring and multiple diagrams it is sometime useful to specify which are to be annotated.

Annotation, Full annotation

For 1D element diagrams, when checked diagrams are annotated at each point along the element at which values have been calculated. Otherwise annotation appears at element ends and where the gradient of the diagram changes sign or at the middle of the element if the diagram is constant along the element.

For 2D element contours, when checked annotation is displayed at the element centre and at each node. Otherwise annotation is displayed at the element centre and, if space permits, at each node.

This setting does not switch annotation on. It does affect how annotation is displayed when it is switched on. Refer to [Annotating diagrams and contours](#) for more details.

Apply to

The **Apply to** group of options determines which entities in the current Graphic View are to have this contour or diagram applied. They are as follows.

- **Apply to, All**
- **Apply to, Selection**—This contour or diagram is to be applied to the currently selected entities. This is only enabled when the [cursor mode](#) is “Select Nodes” when the data is nodal or “Select Elements” when the data is per element, and some entities are selected.
- **Apply to, Current set**—This contour or diagram is to be applied to the same set of entities as it is currently, regardless of the current cursor mode and selection set. This is only enabled when

this contour or diagram was previously applied to a selection set.

8.1.8 Animation Settings

The Animation Settings dialog box allows the parameters that govern animation to be specified.

Open this dialog box by giving the “Graphics | Display | Settings | Animation Settings” menu command. This dialog box may also be opened from the [Graphic Settings dialog box](#).

(Animation may be switched on by giving the “Graphics | Display | Animation” menu command or from the [Orientation toolbar](#).)

Period of animation

Specifies the approximate time to be taken to animate from the undeformed position through to maximum deformation and back to undeformed. Mode shapes are animated from positive deformation to negative deformation; in this case the period is for a half animation cycle.

When animation is by stepping through the cases in the cases list the period of animation specifies the approximate time to step through all of the cases.

When **Override with modal period if animating mode shape** is checked and a mode shape is being animated the period of animation is automatically set to the modal period and the specified value is ignored. However the specified period of animation is used if ($0.1s < \text{modal period} < 60s$).

Animate diagrams and contours with deformation

When checked the scales of diagrams and contours are reduced with the deformation.

This assumes a linear relationship between diagram / contour data and deformation, which may not be appropriate. This feature is only offered for visualisation purposes.

Rotation, Period of complete rotation about model

Specifies the approximate time to be taken to rotate the model through 360° . Rotation only occurs when “Do rotate when animating” is set.

Rotation mode

- **Don't rotate**—The model is not rotated.
- **Do rotate**—The model is rotated about the object point when animated.
- **Rotate if undeformed**—The model is rotated about the object point when animated if the image is undeformed. No rotation occurs if the image is deformed.

See also:

[Working with the Program — Working with Graphic Views — Animation](#)

8.1.9 Orientation Settings

The Orientation Settings dialog box allows the orientation, projection and associated data to be explicitly specified. There are many [higher level operations](#) that modify these settings indirectly.

Open this dialog box by giving the “Graphics | Orientation | Orientation Settings” menu command. This dialog box may also be opened from the [Graphic Settings dialog box](#).

Projection, Perspective view

When checked the image is drawn in a perspective projection. Otherwise the image is drawn orthographically.

Projection, Eye to object distance

Specifies the distance from the object point to the point at which the model is viewed. The shorter this distance is the greater the effect of a perspective projection.

Direction of view, Longitude

Specifies the longitudinal position of the eye point, about a vertical axis through the object point. By default, zero longitude produces a view looking in the positive Y direction. 90° longitude produces a view looking in the negative X direction.

Direction of view, Latitude

Specifies the latitude of the eye point, about a horizontal axis through the object point in the picture plane. By default, zero latitude produces an elevation. 90° latitude produces a plan.

Object point, X, Y, Z

Specifies the object point in global axes and metres.

Mid-point, X, Y, Z

Specifies the mid-point in picture axes and metres.

Picture rotation

Specifies the rotation of the display about the line of sight in degrees. Clockwise rotation is specified by a negative value.

Apply

Applies the current settings to the image without exiting the dialog box.

See also:

[Working with the Program — Working with Graphic Views — The orientation of the image](#)

8.1.10 Graphic Fonts and Styles

The Fonts and Styles dialog box allows the fonts and styles of Graphic View entities to be adjusted.

Open this dialog box by giving the “Window | Settings | Fonts and Styles” menu command when the current view is a Graphic View.

In this context “Print” applies to WMF output as well as printed output and print previews; “Screen” applies to bitmap output and the screen image.

Colour overrides:

Over-ride the default colours to produce **Grey in-fill**, **Black lines**, **Black text**, **Black symbols**, **White background** or **Dark Background** on **Print** or **Screen**.

Print settings also apply to saved images, e.g. PNG or JPeg.

Dark background inverts the colour of the background to black or almost black, renders black objects as white and renders other objects a bit lighter. The **Dark Background** on **Print** setting only applies to saved images; it is ignored in print operations.

Scaling factors:

Factor the default **Font height** or **Symbol size** on **Print** or **Screen**.

Print settings also apply to saved images, e.g. PNG or JPeg.

Feature size:

Specify the **Spring coil radius size** in mm as drawn. This is used when drawing spring elements.

Specify the **Nodes**, and **0D and 1D element contour size** in mm as drawn. This is used as the contour blob size and barrel width when “Constant thickness size on nodes, and 0D and 1D elements” is set in the “[Contour Settings](#)” dialog and when values of “Infinity” or “no value” are being contoured.

8.2 Sculpt Dialogs

[Create User Axes](#)

[Modify Nodes](#)

[Connect 1D Elements](#)

[Split 1D Elements](#)

[Split Quad Elements](#)

[Refine Quad Elements](#)

[Split Tri Elements](#)

[Generate 2D Elements](#)

[Modify Elements](#)

[Modify Element Releases](#)

[Modify Element Offsets](#)

[Modify Members](#)

[Disconnect Elements](#)

[Move Nodes / Move Elements / Copy Nodes / Copy Elements](#)

[Extrude](#)

[Flex](#)

8.2.1 Create User Axes

The Create User Axes dialog box is used to create a user axis set which may either overwrite an existing axis set or become a new one.

Some nodes or grid points must be [selected](#) prior to opening this dialog box. These nodes or grid points are used to define the origin, x vector and xy vector of the axis set. The xy vector need only be defined for space structures so three nodes or grid points must be selected for space structures; otherwise two.

Open this dialog box by giving the “Sculpt | Create User Axes” menu command or from the [Sculpt toolbar](#) when the current view is a Graphic View.

New axis set | Overwrite set

- **New axis set**—The defined axis set will be appended to the current module of user axis sets.
- **Overwrite set**—The defined axis set will overwrite the existing set specified here.

Name

The name of this axis set.

Type

Whether the axis set is Cartesian, Cylindrical or Spherical.

Axes Definition

Use the spin control to select the required axes from the various axes that can be defined by the selected nodes or grid points. The origin, x vector and xy vector identified by the current selection are displayed in the dialog box and the axes are displayed on the Graphic View.

See also:

[Working with the Program — Working with Graphic Views — Selection sets](#)

8.2.2 Modify Nodes

The Modify Nodes dialog box is used to modify the attributes of existing nodes.

The nodes that are to be modified must be [selected](#) prior to opening this dialog box.

Open this dialog box by giving the “Sculpt | Modify Nodes” menu command or from the [Sculpt toolbar](#) when the current view is a Graphic View.

Modify constraint axis

When checked, the constraint axis of the selected nodes will be modified to the axis set specified by **Constraint Axes**. Otherwise the constraint axis will not be modified.

Support condition

The support condition may be specified for each direction, as follows:

- **Don’t modify**—The restraint condition in this direction will not be modified.
- **Restrain**—This direction will be restrained for all selected nodes.
- **Free**—This direction will be unrestrained for all selected nodes. In this case the **Support Stiffness** will be specified set as the support stiffness for all selected nodes.

The restraint types Free, Pin and Encastre options provide a shortcut to setting the flags for the individual degrees of freedom. The symmetry options in the xy plane, yz plane and zx plane provide shortcuts for applying the boundary conditions corresponding to symmetry in these planes. The symmetry options do not unset other restraint conditions.

See also:

[Working with the Program — Working with Graphic Views — Selection sets](#)

[Working with the Program — Sculpting — Modifying nodes graphically](#)

8.2.3 Connect 1D Elements / Connect Lines

This one dialog box is used to connect 1D elements or lines, depending on the context in which it is opened.

The Connect 1D Elements dialog box is used to connect beams, bars etc. that intersect, maintaining element releases and offsets as originally defined. The standard case is where two elements that cross are connected. Sculpt creates a new node at the point of intersection and replaces the two original elements with four elements meeting at the new node.

The elements that are to be connected must be [selected](#) prior to opening this dialog box. Two or more elements must be selected. Non-1D elements will be ignored.

Open this dialog box by giving the “Sculpt | 1D Element Operations | Connect 1D Elements” menu command when the current view is a Graphic View.

Offset tolerance

The offset tolerance defines the maximum separation of two elements at the point at which they “cross” for the elements to be deemed to be intersecting.

Trim tolerance

If an element crosses another within “trim tolerance” of the element end then the element is trimmed to end at the crossing point.

Coincident nodes, Use existing nodes where present

When checked, wherever a new node is required during the execution of this operation the program

searches existing nodes for one that lies within the **Coincidence tolerance** of the required position, and uses the first node found instead of creating a new node. Otherwise new nodes are always created.

Split loads and replace references to old elements with references to new

When checked, loads applied to original elements are mapped onto new elements and, other than in loading records, references to the original elements in element lists are replaced by references to the new elements. Note that the splitting of loads does not occur where a load is applied to an element because that element is included in a saved list or grid plane that is referred to by the loading record rather than being referred to directly in the loading record list. See [Connecting 1D elements graphically](#) for more details.

See also:

[Working with the Program — Working with Graphic Views — Selection sets](#)

[Working with the Program — Sculpting — Connecting 1D elements graphically](#)

8.2.4 Split 1D Elements / Split Lines

This one dialog box is used to split 1D elements or lines, depending on the context in which it is opened.

The Split 1D Elements dialog box is used to split beams, bars etc. into a number of parts maintaining element releases and offsets as originally defined.

The elements that are to be split must be [selected](#) prior to opening this dialog box. Non-1D elements will be ignored.

Open this dialog box by giving the “Sculpt | 1D Element Operations | Split 1D Elements” menu command or from the [Sculpt toolbar](#) when the current view is a Graphic View.

The split may be specified in one of the following ways:

- **Split in n equal parts**—The selected elements will be split into this many equal parts.
- **Split at n [length]**—The selected elements will be split into two parts at this many model units of length from the end of the elements. The length can be specified **from end 1** or **from end 2**.
- **Split at n %**—The selected elements will be split into two parts at this percentage of the way along from end one of the elements.
- **Split at existing nodes**—The selected elements will be split at every node found within the specified coincidence tolerance along its length. If the **align nodes** option is checked then found nodes are moved onto the line of the original element. ('Arc' lines are not handled by this option.)
- **Split at plane**—The selected elements will be split where they intersect the specified plane. The plane may be specified by a grid plane or by three nodes. To define the plane by three nodes, first select the three nodes that define the plane; then select the elements that are to be split; then open this dialog box. Selected elements that do not intersect the plane will be ignored.

Coincident nodes, Use existing nodes where present

When checked, wherever a new node is required during the execution of this operation the program searches existing nodes for one that lies within the **Coincidence tolerance** of the required position, and uses the first node found instead of creating a new node. Otherwise new nodes are always created.

Split loads and replace references to old elements with references to new

When checked, loads applied to original elements are mapped onto new elements and, other than in loading records, references to the original elements in element lists are replaced by references to the new elements. Note that the splitting of loads does not occur where a load is applied to an element because that element is included in a saved list or grid plane that is referred to by the loading record rather than being referred to directly in the loading record list. See [Splitting elements](#)

[graphically](#) for more details.

Preview

Display the consequences of this operation in the current Graphic View without committing to applying the operation. Exiting this dialog box by “Cancel” will “undo” the preview.

See also:

[Working with the Program — Working with Graphic Views — Selection sets](#)

[Working with the Program — Sculpting — Splitting elements graphically](#)

8.2.5 Split Quad Elements

The Split Quad Elements dialog box is used to split Quad4 and Quad8 elements into a number of elements of the same type maintaining element releases and offsets as originally defined. The split is specified in terms of an equal number of elements along edges 1 and 3 and an equal number along edges 2 and 4.

The original elements are deemed to be of a shape as defined by the shape function of the elements. So splitting a non-planar Quad8 element, which is a parabolic element, will result in a mesh fitting a paraboloid surface.

The Quad elements that are to be split must be [selected](#) prior to opening this dialog box. Non-Quad elements will be ignored.

Open this dialog box by giving the “Sculpt | 2D Element Operations | Split Quad Elements” menu command when the current view is a Graphic View.

Split along edge

Specify the split along edges **1 and 3** and **2 and 4**.

Coincident nodes, Use existing nodes where present

When checked, wherever a new node is required during the execution of this operation the program searches existing nodes for one that lies within the **Coincidence tolerance** of the required position, and uses the first node found instead of creating a new node. Otherwise new nodes are always created.

Replace references to old elements with references to new

When checked, references to the original elements in element lists are replaced by references to the new elements so, for example, a loading record that applied to the original element will apply to each of the new elements after the split. Note: This will have a bizarre effect if, say, a variable face load is applied to the original element: the same variable face load will be applied to each new element to produce a rasp-like load pattern.

Preview

Display the consequences of this operation in the current Graphic View without committing to applying the operation. Exiting this dialog box by “Cancel” will “undo” the preview.

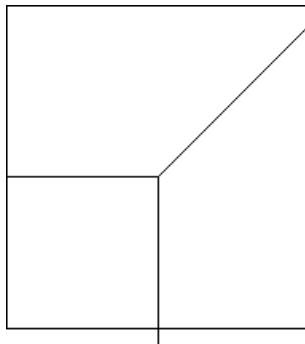
See also:

[Working with the Program — Working with Graphic Views — Selection sets](#)

[Working with the Program — Sculpting — Splitting elements graphically](#)

8.2.6 Refine Quad Elements

The Refine Quad Elements dialog box is used to refine Quad4 and Quad8 elements at a specified corner by splitting the elements in the following manner:



New elements of the same type are created and the original element releases and offsets are maintained.

The original elements are deemed to be of a shape as defined by the shape function of the elements. So refining a non-planar Quad8 element, which is a parabolic element, will result in a mesh fitting a paraboloid surface.

The Quad elements that are to be refined must be [selected](#) prior to opening this dialog box. Non-Quad elements will be ignored.

Open this dialog box by giving the “Sculpt | 2D Element Operations | Refine Quad Elements” menu command when the current view is a Graphic View.

Refined corner

Select the corner at which the refinement is to occur.

Coincident nodes, Use existing nodes where present

When checked, wherever a new node is required during the execution of this operation the program searches existing nodes for one that lies within the **Coincidence tolerance** of the required position, and uses the first node found instead of creating a new node. Otherwise new nodes are always created.

Replace references to old elements with references to new

When checked, references to the original elements in element lists are replaced by references to the original elements in element lists are replaced by references to the new elements so, for example, a loading record that applied to the original element will apply to each of the new elements after the split. Note: This will have a bizarre effect if, say, a variable face load is applied to the original element: the same variable face load will be applied to each new element to produce a rasp-like load pattern.

Preview

Display the consequences of this operation in the current Graphic View without committing to applying the operation. Exiting this dialog box by “Cancel” will “undo” the preview.

See also:

[Working with the Program — Working with Graphic Views — Selection sets](#)

[Working with the Program — Sculpting — Splitting elements graphically](#)

8.2.7 Split Tri Elements

The Split Tri Elements dialog box is used to split Tri3 and Tri6 elements into either Tri or Quad elements maintaining element releases and offsets as originally defined.

The original elements are deemed to be of a shape as defined by the shape function of the elements. So splitting a non-planar Tri6 element, which is a parabolic element, will result in a mesh fitting a paraboloid surface.

The Tri elements that are to be split must be [selected](#) prior to opening this dialog box. Non-Tri elements will be ignored.

Open this dialog box by giving the “Sculpt | 2D Element Operations | Split Tri Elements” menu command when the current view is a Graphic View.

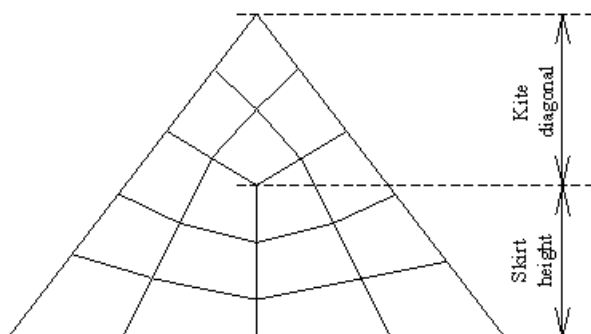
The split may be specified in one of the following ways:

- **Split into Tri elements**—The selected elements will be split into **Split along each edge** Tri elements per edge.
- **Split into Quad elements**—The Split along edge is specified for each edge, abiding by the following rule:

When splitting Tri elements into Quads there must be an equal split along two of the edges. The split along the other edge must be an even number and not greater than the equal split plus 2.

If the two equal splits are into 2 then the split along the other edge must also be 2.

The split is carried out in the following manner:



The **Kite diagonal : Skirt height** ratio may be specified to adjust the mesh.

Coincident nodes, Use existing nodes where present

When checked, wherever a new node is required during the execution of this operation the program searches existing nodes for one that lies within the **Coincidence tolerance** of the required position, and uses the first node found instead of creating a new node. Otherwise new nodes are always created.

Replace references to old elements with references to new

When checked, references to the original elements in element lists are replaced by references to the original elements in element lists are replaced by references to the new elements so, for example, a loading record that applied to the original element will apply to each of the new elements after the split. Note: This will have a bizarre effect if, say, a variable face load is applied to the original element: the same variable face load will be applied to each new element to produce a rasp-like load pattern.

Preview

Display the consequences of this operation in the current Graphic View without committing to applying the operation. Exiting this dialog box by “Cancel” will “undo” the preview.

See also:

[Working with the Program — Working with Graphic Views — Selection sets](#)

[Working with the Program — Sculpting — Splitting elements graphically](#)

8.2.8 Modify Elements

The Modify Elements dialog box is used to modify the attributes of existing elements, including, to some degree, the type of the element. Modifying 2D elements from linear to parabolic also causes new nodes to be created at the mid-sides.

The elements that are to be modified must be [selected](#) prior to opening this dialog box.

Open this dialog box by giving the “Sculpt | Modify Elements” menu command or from the [Sculpt toolbar](#) when the current view is a Graphic View.

Modify type of 0D elements

When checked, selected 0D elements will be changed to the specified 0D type.

Modify type of 1D elements

When checked, selected 1D elements will be changed to the specified 1D type. This is useful for changing beam elements to bars, for example.

Modify type of 2D elements

When checked, selected 2D elements will be changed to the specified 2D type. Select “Linear” to convert selected Quad8 and Tri6 elements to Quad4 and Tri3 elements respectively. Select “Parabolic” to convert the other way, in which case new nodes are created where necessary.

Modify property

When checked, the property number of all selected elements will be changed to the specified property number.

Modify group

When checked, the group number of all selected elements will be changed to the specified group number.

Orientation, Modify orientation node

When checked, the orientation node of all selected elements will be changed to the specified orientation node.

Orientation, Modify orientation angle

When checked, the orientation angle of all selected elements will be changed to the specified orientation angle.

Modify dummy status

When checked, the dummy element status is set accordingly.

Beam Releases

Open the [Modify Beam Element Releases](#) dialog box to specify the modifications to the releases for all selected elements.

2D Releases

Open the [Modify Element Releases](#) dialog box to specify the modifications to the releases for all selected elements.

Offsets

Open the [Modify Element Offsets](#) dialog box to specify the modifications to the offsets for all selected elements.

See also:

[Working with the Program — Working with Graphic Views — Selection sets](#)

[Working with the Program — Sculpting — Modifying elements graphically](#)

8.2.9 Modify Beam Element Releases

The Modify Element Releases dialog box is used to specify modifications to element releases and stiffnesses of existing beam elements. It is opened from the [Modify Elements dialog box](#).

Modify node n

When checked, the release condition will be set as specified for node n on the selected elements.

Translational and Rotational Releases

Each degree of freedom at each end of the beam element can be released. If the degree of freedom is released a stiffness can also be applied to connect the end of the beam to the node.

The buttons gives a shortcut to setting up common release options

Fully fixed – the end of the beam is not released in any direction

Release translations – the translation degrees of freedom are released, allowing the end of the element to translate freely, but the rotations are still fixed to the rest of the structure

Release moments – the moment degrees of freedom are released for the element, but translations are still fixed to the rest of the structure

Remove stiffness – the releases are retained but the stiffnesses are removed.

Note: care should be taken when applying releases that the resulting element does not become unstable. Various checks for invalid restraints are carried out to prevent singularities at a later stage in the analysis. It is invalid to:

- release a translational degree of freedom at both ends of a beam
- release the torsional degree of freedom at both ends of a beam
- release moment degree of freedom at both ends of beam and one of the corresponding translational releases

8.2.10 Modify Element Releases

The Modify Element Releases dialog box is used to specify modifications to element releases of existing elements. It is opened from the [Modify Elements dialog box](#).

Modify node n

When checked, the release condition will be set as specified for node n on the selected elements.

Modifications will only be applied to the nodes that are relevant for the element being modified. So, for example, the settings for only nodes 1 and 2 will be applied to beam elements.

8.2.11 Modify Element Offsets

The Modify Element Offsets dialog box is used to specify modifications to element offsets of existing elements. It is opened from the [Modify Elements dialog box](#).

Modify node n

When checked, the offset will be set as specified for node n on the selected elements.

Modifications will only be applied to the nodes that are relevant for the element being modified. So, for example, the settings for only nodes 1 and 2 will be applied to beam elements.

Offset definition axis

All offsets are stored in global axis direction. However it is often convenient to be able to specify the offsets with respect to the element local axes assuming that no offsets currently exist. If this option is chosen the offsets are converted to global axes before being stored.

8.2.12 Modify Members

The Modify Members dialog box is used to modify the attributes of existing members.

The members that are to be modified must be [selected](#) prior to opening this dialog box.

Open this dialog box by giving the “Sculpt | Modify Members” menu command or from the [Sculpt toolbar](#) when the current view is a Graphic View.

Modify section property

When checked, the property number of all selected members will be changed to the specified property number.

Modify design property

When checked, the design property number of all selected members will be changed to the specified design property number.

Modify restraint property

When checked, the restraint property number of all selected members will be changed to the specified restraint property number.

Modify group

When checked, the group number of all selected members will be changed to the specified group number.

Orientation, Modify orientation node

When checked, the orientation node of all selected members will be changed to the specified orientation node.

Orientation, Modify orientation angle

When checked, the orientation angle of all selected members will be changed to the specified orientation angle.

Modify dummy status

When checked, the dummy member status is set accordingly.

See also:

[Working with the Program — Working with Graphic Views — Selection sets](#)

[Working with the Program — Sculpting — Modifying elements graphically](#)

8.2.13 Disconnect Elements

The Disconnect Elements dialog box is used to disconnect elements from the existing structure at a specified nodal position. New nodes are created at the original positions and the selected elements are then attached to the new nodes. Joints may be created to join the original nodes with the new.

The elements that are to be disconnected must be [selected](#) prior to opening this dialog box.

Where several elements are to be disconnected from a particular node it may be necessary to [flip](#) some of the elements to result in the particular node appearing at the same nodal position for all selected elements, prior to opening this dialog box.

Open this dialog box by giving the “Sculpt | Disconnect Elements” menu command when the current view is a Graphic View.

Topology

The selected elements will be disconnected at the specified nodal position.

Connect new node to existing by Joint

When checked, a joint is created for every new node, joining it with the existing node. Default joint directions are applied to the new joints.

See also:

[Working with the Program — Working with Graphic Views — Selection sets](#)

[Working with the Program — Sculpting — Disconnecting elements graphically](#)

[Working with the Program — Sculpting — Flipping elements graphically](#)

8.2.14 Move Entities / Copy Entities

This dialog box is presented in several guises depending on the context in which it is opened; it is used to move or copy nodes, elements, members, lines, areas, or regions.

The entities that are to be moved or copied must be [selected](#) prior to opening this dialog box.

In describing the operation of the move or copy, below, the term “subject node” is used. When moving nodes the “subject nodes” are the selected nodes. When moving other entities the “subject nodes” are the nodes attached to the selected entities. When copying nodes the initial set of “subject nodes” is the selected nodes; when making multiple copies “subject nodes” of subsequent sets is the previous copy set. When copying, say, elements the initial set of “subject nodes” is the nodes attached to selected elements, etc.

Open this dialog box by giving the “Sculpt | Move Selection” menu command or the “Sculpt | Copy Selection” menu command when the current view is a Graphic View.

Number of copies

Specifies the number of copies to be made. This is disabled when moving entities or when copying by reflecting.

Transformation axes

The move or copy performs coordinate transformations with respect to the transformation axes. Axes default to the axis set defined for the current grid (not adjusted by the grid plane elevation).

The move or copy may be specified in one of the following ways:

- **Shift**—The specified coordinates are transformed from the transformation axis set to global. The shift 'by' coordinates are added to the respective coordinates of the “subject” nodes; the shift 'to' coordinates overwrite the respective coordinates of the “subject” nodes.
 Shift 'to' is only available when 'moving, not 'copying'. Shift 'to' x, y, and z (all three) is only available when moving nodes.
 The default 'shift by' vector is the vector from 1st current [polyline](#) point to the 2nd point, if specified.
 The default 'shift to' coordinates are the coordinates of the first selected entity.
- **Rotate**—The coordinates of the subject nodes are rotated about the specified axis of the transformation axes by the specified number of degrees. A positive rotation is as defined by the “right hand screw” rule.
- **Reflect**—The coordinates of the subject nodes are reflected through the specified plane of the

transformation axes.

Node number increment

When checked, the new node numbers are the “subject node” number plus the specified increment. Otherwise new nodes are appended to the nodes module with no attempt to form a constant increment. This is disabled when moving nodes or elements.

Entity number increment

When checked, the new element, member or line numbers are the “subject entity” number plus the specified increment. Otherwise new entities are appended to the respective module with no attempt to form a constant increment. When copying areas or regions it is the copied lines that have the increment applied. This is disabled when moving entities or copying nodes.

Coincident nodes, Use existing nodes where present

When checked, wherever a new node is required during the execution of this operation the program searches existing nodes for one that lies within the **Coincidence tolerance** of the required position. Of the nodes found within the tolerance the closest is used instead of creating a new node; the restraint condition of the existing node is retained. Otherwise new nodes are always created.

Copy references

When checked, where copied nodes/elements are referred to in data modules references to the new nodes/elements are added.

The references are copied in the Lists module first, if **in lists** is checked, and are not repeated in subsequent modules if the copy has been done in the Lists module to a list that is referred to in the subsequent module. However, in the case where **in lists** is not checked and, say, **in loading** is checked and a load is applied by reference to a list in the Lists module then the load will be “copied” by appending to the list field in the loading module.

Where both nodes in a Joint are copied, the Joint is copied.

Where at least two nodes in a Rigid Constraint are copied a new Rigid Constraint is created for the new nodes.

Where any nodes in a Constraint Equation are copied, the Constraint Equation is copied. Note that no attempt is made to rotate the copied constraint equation; this will need to be done manually for copies 'by rotation'.

Node references in Generalised Restraints are not copied.

This option is disabled when moving nodes or elements.

Preview

Display the consequences of this operation in the current Graphic View without committing to applying the operation. Exiting this dialog box by “Cancel” will “undo” the preview.

See also:

[Working with the Program — Working with Graphic Views — Selection sets](#)

[Working with the Program — Sculpting — Moving and copying entities graphically](#)

[Program Fundamentals — Axes — Definition Axes](#)

8.2.15 Extrude

Either a polyline must be [formed](#) or a set of nodes or elements [selected](#) prior to opening this dialog box, to serve as the basis of the extrusion.

The extrude operation copies a set of nodes by a specified **number of increments** and **increment length** either along a specified **Axis** of a specified axis set or along a specified **Alignment**, if any [Alignments](#) are defined. The option is given to **Include Beam elements along extrusion** and to

Include Quad4 elements along extrusion during the extrusion process.

If the selected elements (or members) have the same local axes then selecting the 'local' axis option specifies the local element axis set as the axis for the extrude; otherwise, similarly, if the selected nodes have the same constraint axes then these will be the local axis directions. Note that the entity type used for local axis interpretation is independent of the type of entities being extruded. E.g. When extruding nodes, it is possible first to select the element that is to define the local axis directions, then to select the nodes that are to be extruded before opening the Extrude dialog.

Assignment of Property and Group numbers to new elements

For extrusions that are based on a selection of elements, the copies of the selected elements at each increment are assigned the same property and group number as the original element. Otherwise the default property is assigned as it is assumed that the property numbers will have to be adjusted manually. To facilitate selection for the purpose of the assignment of property numbers, unique group numbers are assigned to the group of new longitudinal beam elements, to the group of new 2D elements and to the group of new transverse beam elements, respectively.

Extruding a polyline

If the extrusion is based on a polyline then nodes are created at the vertices of the polyline at each increment. The option is given to **Include the closing line segment** of the polyline in the element generation process.

The option to **Include transverse Beam elements** produces Beam elements along the polyline at each increment.

Extruding a set of nodes

If the extrusion is based on a set of nodes and the **Include Quad4 elements along extrusion** option is selected then the nodes are ordered geometrically for the purpose of connecting nodes with Quad4 elements along the extrusion. This can lead to bizarre element generation if the nodes cannot be ordered sensibly; this can be overcome by carrying out the extrusion in several steps.

Extruding a set of elements

If the extrusion is based on a set of elements then copies of the selected elements are included at each increment. If the **Include Quad4 elements along extrusion** option is selected a Quad4 element is created between each increment for each 1D element selected.

Coincident nodes

The option is given to check for an **existing node** within the **coincidence tolerance** of the position of a required node before creating a new node.

The extrusion can be **previewed** before committing to the creation of nodes and elements.

See also:

[Working with the Program — Working with Graphic Views — Polylines in Graphic Views](#)

[Working with the Program — Working with Graphic Views — Selection sets](#)

[Working with the Program — Sculpting — Extruding nodes and elements graphically](#)

8.2.16 Flex

The nodes that are to be flexed must be [selected](#) prior to opening this dialog box.

The flex operation moves a set of nodes, that are initially approximately in a line, into a specified shape. The shape can be linear, circular, elliptical or parabolic. The new position of one of the nodes is specified; this node is referred to as the shift node. A more complete description of the flex operation is given in the [“Flexing lines of nodes graphically”](#) section of the Working With GSA chapter.

Open this dialog box by giving the “Sculpt | Flex Selection” menu command when the current view is

a Graphic View. “Flex Selection” is also available on the [right-click menu](#) that is displayed when the [cursor mode](#) is set to Select Nodes. Invoking Flex via the “Sculpt | Flex Selection” menu command assumes the middle node as the node to be explicitly shifted; Flex via the right-click menu requires that a node has been right-clicked and this node is assumed as the shift node. Either way the shift node can be re-selected from within the Flex dialog box.

Axes

Opens the [Axes dialog box](#). The shift coordinates are expressed with respect to the current axes. Axes default to the axis set defined for the current grid (not adjusted by the grid plane elevation).

Shift node

The node that is shifted to the shift coordinates. The shift node must be one of the current selection of nodes. Nodes at the ends of the line of nodes may be selected as the shift node only when flexing linearly.

Note that the current shift node is highlighted on the Graphic View.

Shift coordinates

The shift coordinates specify the new position of the shift node. Note that **Shift By vector** specifies a shift relative to the initial position and **Shift To position** is an absolute position. In both cases the coordinates are with respect to the defined axes for the flex. Note also the special case of a circular flex with specified radius, described below.

Flex shape

The flex may be specified in one of the following ways:

- **Linear**—The nodes are arranged linearly between the end nodes and the shift node. The shift node may be an end node. Shifting an internal node results in two lines of nodes meeting at the shift node. This option may be used to align or evenly distribute a set of nodes.
- **Circular**—The nodes are arranged on a circular arc that goes through the end points and the new position of the shift node.
- **Elliptical**—The nodes are arranged on an ellipse or a sheared-ellipse that goes through the end points and the new position of the shift node. If the shift node is shifted normal to the line between the end nodes then a true ellipse is formed. If the shift node is also shifted parallel to the line then the ellipse is sheared to result in the shift node being positioned at its shift coordinates.
- **Parabolic**—The nodes are arranged on a parabola or a sheared-parabola that goes through the end points and the new position of the shift node. If the shift node is shifted normal to the line between the end nodes then a true parabola is formed. If the shift node is also shifted parallel to the line then the parabola is sheared to result in the shift node being positioned at its shift coordinates.

When the flex shape is set to circular the **Circle radius** may be specified. Note that when the circle radius is specified the shifted position of the shift node is used only to indicate the general direction of the flex; the shift node is shifted onto the circle defined by the end points and radius (which is not necessarily the specified shift position).

Spacing of nodes

The spacing of nodes can be one of:

- **Space evenly**—The nodes are spaced evenly between the end nodes and the shift node.
- **Space proportionally**—The spacing of the nodes between the end nodes and the shift node remain at the same proportion as their initial spacing along the line of nodes.

This spacing is only true for linear and circular flexes. Elliptical and parabolic flexes use this spacing to distribute the nodes along the line of nodes before projecting them onto the ellipse or parabola. So, for elliptical and parabolic flexes, the spacing is a basis for the final positioning of the nodes, not an actual spacing.

Coincident nodes, Use existing nodes where present

When checked, wherever a flexed node is shifted to within the **Coincidence tolerance** of an existing node the flexed node is replaced by the existing node. Otherwise the flexed node is always retained even when it is coincident with another node.

Preview

Display the consequences of this operation in the current Graphic View without committing to applying the operation. Exiting this dialog box by “Cancel” will “undo” the preview.

See also:

[Working with the Program — Working with Graphic Views — Selection sets](#)

[Working with the Program — Sculpting — Flexing lines of nodes graphically](#)

[Program Fundamentals — Axes — Definition Axes](#)

8.3 Output View Dialogs

[Wizard: Output Settings](#)

[Output Wizard: Further Options](#)

8.3.1 Wizard: Output Settings

The Wizard: Output Settings dialog box allows the parameters that govern the format and content of Output Views to be specified. All Output View settings can be specified either in this Output Settings dialog box or in dialog boxes accessible from this.

The default Output View settings are stored as preferences. Refer to [Working with the Program: – Default View Settings](#) for details.

Open this dialog box by giving the “Output | Output Settings” (Ctrl+W) menu command or the Wizard button on the [Data Options toolbar](#) when the current view is an Output View.

Data (tree control)

The selected data will be output. Many of the items are tables containing several components.

Multiple selection can be made by holding down the Ctrl or Shift key while selecting.

Output of 2D Element Results, Stress layer

Select whether top, middle, bottom or bending stresses are to be contoured.

This is enabled only when a 2D element result is selected.

Output of 2D Element Results, Centre values only

When checked, results are output for the centre position on the element only and not the nodal positions.

This is enabled only when a 2D element result is selected.

Output of 2D Element Results, Average stresses and forces at nodes

When checked, results are averaged across elements at common nodes. (See [Output Options — Stress averaging in 2D elements](#) for details.)

This is enabled only when a 2D element result is selected and when 'Centre values only' is not selected.

By Case

When checked, data that is output in table format and per case is output in several tables, each table being the output for one case in the cases list. Otherwise each case is output on consecutive

lines per entity.

By Property

When checked, data that is output in table format and per element or member is output in several tables, each table being the output for one property represented in the element or member list. Otherwise all entities in the corresponding list are output in one table.

In separating the elements into properties heed is taken of the property type, so beam elements with property one will appear in a different table to spring elements with property one.

By Group

When checked, data that is output in table format and per element or member is output in several tables, each table being the output for one group represented in the element or member list. Otherwise all entities in the corresponding list are output in one table.

Output Table | Output Summary

The table is the table of data specified. The [summary](#) is a short table of maxima and minima encountered per component of data in the table and, in some cases, the total accumulated in the table. Checking “Output Table” and “Output Summary” switches the output of the table and summary on.

Case list | Node list | Element list | Member list

Specifies the list of cases, nodes, elements or members to be represented in the output. This is ignored for data that is case, node, element or member independent.

Refer to the [list syntax](#) for details.

The lists may also be specified on the [Lists toolbar](#) associated with the view.

Apply

Applies the current settings to the image without exiting the dialog box.

Reset

Resets all settings except the data type to the [default view settings](#).

Copy

Copy all of the current view settings to the clipboard in text format, GWA syntax).

Paste

Paste the view settings from the clipboard. The view settings must be in text format, GWA syntax.

Export

Write the currently specified output to file in tab delimited, comma delimited or HTML format. The output need not have been displayed previously in the Output View and neither does exporting the output to file cause the output to be displayed to the Output View.

Copy Output

Write the currently specified output to the clipboard in tab delimited format. The output need not have been displayed previously in the Output View and neither does copying the output to the clipboard cause the output to be displayed to the Output View.

Gen. User Mod.

Generate [user modules](#) for each case of each component of currently specified output. User modules are only generated for data that are per node or per element. E.g. not properties or modal frequencies.

1D Elem Results

Opens the [1D Element Results dialog box](#) for the specification of the positions along 1D elements at which displacements and forces are to be output for this view.

Units

Opens the [Units dialog box](#) for the specification of the units in which the data are to be expressed for this view.

Numeric Format

Opens the [Numeric Format dialog box](#) for the specification of the numeric format in which the data are to be expressed for this view.

Axes

Opens the [Axis dialog box](#) for the specification of the axes in which the data are to be transformed for this view.

Further Options

Opens the [Output Wizard: Further Options dialog box](#).

See also:

[Working with the Program — Working with Output Views](#)

[Working with the Program – Working with Saved Views and Preferred Views](#)

[Program Fundamentals — Sets and Lists](#)

8.3.2 Output Wizard: Further Options

The Output Wizard: Further Options dialog box allows less frequently used parameters that govern the output to be specified.

Open this dialog box from the [Wizard: Output Settings dialog box](#).

Subject Component (tree control)

The selected component is identified as being the “subject component”. For this to be useful it must be a component of the data that has been selected for output in the Wizard: Output Settings dialog box.

The “subject component” is used in the “Data Extents of ‘Subject Component’” and “Envelope ‘Subject Component’ only” options described below.

Data Extents of “Subject Component”, Activate extents

When checked, a range of interest, or extents, can be specified. Records in the output in which the “subject component” does not lie within the specified extents are disregarded, both in the table and in assessing the maxima and minima for the summary.

See [“subject component”](#), above, for details on specifying what this is.

This is ignored for non-tabular output.

Data Extents of “Subject Component”, Min. and Max.

These specify the minimum and maximum extents.

Data Extents of “Subject Component”, Magnitude | Signed

When “Magnitude” is selected the sign (+ or –) of the data values is ignored. Otherwise the sign is taken into account.

Data Extents of “Subject Component”, Inclusive | Exclusive

When “Exclusive” is selected values **within** the extents are deemed to be **out** of range. Otherwise values within extents are within range.

Envelope “Subject Component” only

When checked and when the case being output is an enveloping combination case only the “subject

component" is enveloped; coexistent values are output for other components.

When checked and no relevant "subject component" is selected, output is produced for the envelope of every component, together with co-existent values.

When not checked, each component is enveloped.

See "[subject component](#)", above, for details on specifying what this is.

Summary, Summarise worst n values

Specifies the number of records of maxima and minima per component to be output in the summary.

Spreadsheet formats, Fully populate fields

When checked the output displayed in a format that is convenient for using spreadsheet filtering options (e.g. MS Excel 'Filter' and 'PivotTable'). Column header fields and row lead fields are fully populated and the column header row is written immediately above the table.

Spreadsheet formats, Output all data per entity on a single row

When checked output for a given entity is concatenated into one line.

Data references

Data references, such as constraint axis references, may be reported either by number, by name or by name and number. If no name is given then the reference is by number, regardless of this setting.

Include entity names in results output

When checked, node, element and member names are reported in results output alongside the number, where specified.

See also:

[Program Fundamentals — Enveloping in GSA](#)

8.4 Chart View Dialogs

Chart views are the plots showing the relationship as a graph. The values on the ordinate (y) are a function of the values on the abscissa (x) of Y versus X, e.g. Nodal response factor versus Walking frequency, Nodal velocity versus Time and Nodal acceleration versus Harmonic load frequency etc. Most of the chart options depend on some previous analysis being carried out.

The following chart views are available:

[Non-linear analysis](#)

[Modal Analysis Details](#)

[Harmonic analysis](#)

[Periodic load analysis](#)

[Linear time history analysis](#)

[Footfall analysis](#)

[Storey displacements and forces](#)

[Forces on 2D element cut](#)

[Cut section forces](#)

8.4.1 Non-linear Analysis Chart

Non-linear analysis chart views are for the following two analyses:

1. Automatic load increment analysis. For this analysis, the chart view will plot the curve of overall load factor versus nodal displacement (rotation) used in the analysis.
2. Member buckling analysis. For this analysis, the chart view will plot the curve of axial force of the member versus the nodal displacement (rotation)

After opening the non-linear analysis chart view setting dialog box, just select the analysis cases from the list box and click OK, the chart views will be presented.

8.4.2 Modal Analysis Details

This dialog is accessible from the "View | New Chart View | Modal Analysis Details" menu command. This allows the user to plot various modal properties as a function of mode number.

Task

This identifies the task from which the results are to be extracted. This must be either a modal or Ritz analysis or a response spectrum analysis.

Output option

The output from this is a bar chart with the result as a function of mode. The options that can be output are selected from:

- Participation factor
- Rotational participation factor
- Effective mass
- Effective inertia

And if the task is a response spectrum task:

- Modal multiplier
- Spectral acceleration
- Elastic base shear

Participation factors and effective mass/inertia are plotted for all three directions (X, Y and Z). The modal multiplier, spectral acceleration and elastic base shear are plotted for each response case.

8.4.3 Harmonic Analysis Chart

Harmonic analysis chart view shows the plots of nodal responses versus harmonic load frequencies for the chosen nodes. The settings include:

Items to be plotted (tree control)

The selected items will be plotted in the chart views. Multiple selection can be made and it will result in multiple chart views to be opened.

Harmonic analysis task

If more than one set of Harmonic analysis results (task) exist it needs to know which set of results are to be used. If only one harmonic analysis task is available, it will be used in generating the chart views.

Output nodes

Node list to define the output nodes in the chart views.

8.4.4 Periodic Load Analysis Chart

Periodic load analysis chart view shows the plots of nodal responses versus periodic load frequencies for the chosen nodes. The settings include:

Items to be plotted (tree control)

The selected items will be plotted in the chart views. Multiple selection can be made and it will result in multiple chart views to be opened.

Periodic load analysis task

If more than one Periodic load analysis task exist, choose the one to be used in generating the chart views. If only one harmonic analysis task is available, it will be used in generating the chart views.

Output nodes

Node list to define the output nodes in the chart views.

8.4.5 Linear Time-history Analysis Chart

Time history analysis chart view shows the plots of nodal responses (displacements, velocities and accelerations) versus time for the chosen nodes. The settings include:

Items to be plotted (tree control)

The selected items will be plotted in the chart views. Multiple selection can be made and it will result in multiple chart views to be opened.

Linear time history analysis task

If more than one linear time history analysis tasks exist, choose the one to be used in generating the chart views. If only one linear time history analysis task is available, it will be used in generating the chart views.

Output nodes

Node list to define the output nodes in the chart views.

Modal contributions

Check this box if the contributions from a particular mode or a number of modes need to be plotted. If this box is Unchecked, the total responses will to be plotted.

Mode list

List of the modes to be plotted for their contributions to total responses, it is only available if "Modal contributions" box is checked.

8.4.6 Footfall Analysis Chart

Footfall induced vibration analysis chart view shows the plots of responses factor versus Walking frequency, Velocity versus Time and Participation factor versus modes etc for the selected response node and excitation node. The settings include:

Items to be plotted (tree control)

The selected items will be plotted in the opened chart views. Multiple selection can be made and it will result in multiple chart views being opened.

Footfall analysis task

If more than one set of Footfall analysis results (tasks) exist it needs to know which set of results are to be used. If only one footfall analysis task is available, it will be used in generating the chart views.

Response node

The response node number used for the plot. It can be typed in or selected from the list. It must be a single node number.

Excitation node

The excitation node number used for the plot. It can be typed in or selected from the list. It must be a single node number.

Criteria of response factor

This is used for comparison with the calculated responses of "Acceleration v. Harmonic forcing frequency" and "Velocity response v. Walking frequency" in the chart views. Two criteria can be selected which can be response factors or selected from standard criteria. The selected criteria will be represented by two curves on the chart views. The two criteria (reference) curves can be used to tell how far/close of the calculated responses from the criteria (reference) curves. It will not be used for other plots, so it is disabled if these two plots are not selected on the tree control.

Walking frequency

The walking frequency is used only in "Velocity v. Time" plot. It will not be used for other plots, so it is disabled if "Velocity v. Time" plot is not selected on the tree control.

8.4.7 Assembly Displacements and Forces

This dialog is accessible from the "View | New Chart View | Assembly Displacements and Forces" menu command. This allows charts to be produced for the assemblies.

The displacements and drifts are calculated by averaging across the cut surface on the assembly. For 1D elements the forces and moments are calculated where the cutting plane bisects the element. For 2D elements the forces and moments are calculated by integrating the stresses where the cutting plane slices the element. As the results for 2D elements are based on the element stresses they will be less accurate than those for 1D elements.

Link elements, rigid constraints, joints and constraint equations are ignored in the calculation.

Assembly

Select from the assemblies in the model or <new> to define a new assembly.

Case list

The results can be either analysis cases or combination cases, provided the combination case has only a single permutation or a single permutation of an envelope case is selected.

Assembly results

Select the results for which output is required. The options are:

- Displacements and rotations
- Drift
- Drift index (drift divided by storey height)
- Forces and moments

If the Output is either Report or File then the accompanying results will also be output (even if not selected).

Use averaged 2D stresses

When this is ticked, use averaged 2D stresses to calculate storey forces, otherwise unaveraged stresses are used.

Output axes

The assembly results are calculated relative to the origin of the assembly axis. The results can be transformed into any axis system. The moments are still reported relative to the origin of the cutting plane.

Output as

The output can be to a chart (the default option). In addition the results can be reported in the property view or exported to a comma separated (CSV) file.

8.4.8 Storey Displacements and Forces Chart

This dialog is accessible from the “Tools | Storey Displacements and Forces” and “View | New Chart View | Storey Displacements and Forces” menu commands. When accessed from the Tools menu tabular output of storey displacements, drifts or forces is produced; when accessed from the View menu, storey output is displayed on charts. Output is in global axis directions.

Storey Displacements

This option calculates the displacements of the each storey and reports in-plane displacements and rotation about the global z axis.

Storey Drifts

This option calculates the relative displacements of the each storey and reports in-plane displacements and rotation about the global z axis.

Storey Forces

The storey forces are calculated at mid-storey height. The structure is cut at this level and the force summed on the cutting plane. Either averaged or unaveraged 2D stresses may be used in the calculation.

Force Calculation

To provide more control over the force calculation the elements to be considered can be specified (for example to select a single core in a building). By default the force/moment calculations are about the global origin, but this can be offset if this is convenient. Where the model contains 2D elements there is an option to use averaged 2D stresses to calculate storey forces, otherwise unaveraged stresses are used.

Result case

This allows any analysis or combination case (provided it has a single permutation) to be selected for which the displacements of forces will be calculated.

Storey drift and displacement calculations are calculated from the average of all the selected nodes on that storey. Rotations are about the centre of mass of the storey.

If the selected case is a response spectrum cases, the displacement/drift will be calculated for each mode individually and these results combined using the appropriate combination method for that case.

8.4.9 Forces on 2D Element Cut Chart

This dialog is accessible from the “View | New Chart View | Forces on 2D Element Cut” menu command. This allows the user to calculate the force and moment (per unit length) along the cut. Output is in the local axis defined by the grid plane and cut.

Grid plane

This identifies the grid plane on which the element lie that are to be cut.

Line defining cut

The line defining the cut can be either a polyline or defined explicitly by a start and end point. If the polyline is associated with a grid plane the elements on this grid plane will be cut. If the polyline is a 2D polyline or if a line is specified the current grid plane will be used.

Output selection

The output from this is a chart, or set of charts, with the variation of force/moment per unit length along the cut. These results are defined in a local axis for each segment of the polyline where z is the grid plane z axis, y is the direction along the cut (for the current segment) and x is in the outward normal direction from the cut. Thus the in-plane forces on the cut are N_x , and the bending moment about the cut are M_x , and the through thickness shear forces are Q_x . Stresses may be averaged or unaveraged.

When output "By Case" is selected a new chart is created for each case. When "By Component" is selected a new chart is created for each selected component. If both "By Case" and "By Component" are selected then there is a new chart for each case and component.

Result case

This allows any analysis or combination case (provided it has a single permutation) to be selected for which the displacements of forces will be calculated.

8.4.10 Cut Section Forces

This dialog is accessible from the "Tools | Legacy Options | Cut Section Forces" menu command. This allows the user to specify cutting planes. GSA will then calculate the force that result from cutting selected elements in the structure on this cutting plane for a selected results case.

For 1D elements the forces and moments are calculated where the cutting plane bisects the element. For 2D elements the forces and moments are calculated by integrating the stresses where the cutting plane slices the element. As the results for 2D elements are based on the element stresses they will be less accurate than those for 1D elements.

Link elements, rigid constraints, joints and constraint equations are ignored in the calculation.

Cut

Cut sections are calculated on assemblies. These can either be defined explicitly or by reference to an existing assembly. The cuts are spaced along the x axis of the assembly.

Case list

The results can be either an analysis case or combination case, provided the combination case has only a single permutation.

Report

This allows either forces and/or moments to be reported.

Use averaged 2D stresses

When this is ticked, use averaged 2D stresses to calculate storey forces, otherwise unaveraged stresses are used.

Output axes

The cut section forces and moments are calculated relative to the origin of the assembly axis. The results can be transformed into any axis system. The moments are still reported relative to the origin of the assembly axis.

Display

Display the cut section forces for the specified cutting planes in the Object Viewer.

Export

Export the cut section forces for the specified cutting planes to file. The file format options are:

- Tab separated (.TXT) file.
- Comma separated (.CSV) file.
- Sigraph graphics (.SIG) file, for viewing in Oasys Sigraph.

Save assembly

This saves the currently defined assembly.

Assembly definition

If an assembly is selected this will be populated from the [assembly definition](#). For an explicit cut select a name, axis, a list of elements, and specify the cut(s). The cuts can be

- **number of positions** – specify the number of positions along the cutting axis
- **specify spacing** – specify the distance between cuts
- **by storey** – a cutting plane will be set at the storey height (cutting the elements below the cutting plane) except for the first and last storeys which are offset by 0.1% from the ends to avoid problems at the base and the top of the structure.
- **explicit** – an explicit definition of the points where the cuts are located: either a simple list or one using the keywords to and step (eq 2 to 8 step 2 12)

8.5 Tools

[Create Members from Elements](#)

[Create Regions per Grid Plane](#)

[Create New Model from Deformed Geometry](#)

[Rationalize Lines](#)

[Split Warped 2D Elements](#)

[Create Mass from Loads](#)

[Expand Grid Loading](#)

[Storey Displacements and Forces](#)

[Create a New Raft Model](#)

[Generate Static Vehicle Load](#)

[Optimise Path Loading](#)

[Expand Bridge Loading](#)

[Batch Bridge Analysis](#)

[Wheel Patch Definition](#)

[CEAP Modal Damping](#)

[Equivalent Static / Accidental Torsion Load](#)

[Storey Masses](#)

[Cut Section Forces](#)

[Footfall Response Data](#)

[Manage Data](#)

[Manage Nodal User Modules / Manage Element User Modules](#)

[Generate 2D Mesh for Polylines \(legacy option\)](#)

8.5.1 Create Members from Elements

This dialog can be accessed from the "Tools | Manipulate Model | Create Members from Elements" menu command.

The Create Members from Elements command generates members from 1D elements.

The options are:

- **One element = one member**
- **Many elements = one member**

Element List

You can select which elements are available for "conversion" using the combo-box at the top of the dialog box. You can either use a predefined list by using the pull down part of the box, use the current selection set by pressing the button below, or type in your own list of elements. The default list is All.

Elements already covered by a member will not be used even if they are in the element list. This means that you can create members from elements in several stages (possibly with different settings each time) without worrying that you are going to end up with coincident members.

Restrictions on member construction

This group of settings allows you finer control over how members are generated:

Setting	Description
Divide members at change of elements' group	If this checkbox is set, then all the elements in a member must have the same group, otherwise the member construction tool ignores elements' groups.
Divide members at intersections with columns	You select which direction "columns" run in using the x,y,z list box. This causes members to be broken at columns. Tip: This feature can be used to differentiate secondary beams from primary beams by setting the "column" direction to the direction of the primary beams. The secondary beams will be broken (discontinuous) where they meet the primary beams, even if they are continuous in alignment.
Member straightness tolerance	All steel members are assumed in their check calculations to be straight, however, the analysis model may not be exactly aligned. Use this setting to allow definition of members from elements that aren't exactly in line with each other (though they must all be connected together). Be careful not to be too generous with this setting, because although the members will be created without complaint, the resulting calculations may be invalid.

8.5.2 Create Regions per Grid Plane

This dialog can be accessed from the "Tools | Manipulate Model | Create Regions per Grid Plane" menu command.

The Create Regions per Grid Plane command creates lines, areas and a region per specified grid plane. The internal procedure is as follows:

1. for each grid plane,
2. search for panels,
3. create lines to enclose the panel, if not already present,
4. create an area per panel,
5. create a region that references all the new areas,
6. add to the line list for the region all loose lines on the grid plane within the outer boundary of the region
7. add to the node list for the region all loose nodes on the grid plane within the outer boundary of the region.

Note:

- New lines, areas and regions adopt the current default attributes,
 - except the default line type is ignored and lines are created as straight lines, not arcs.
- All referenced nodes are converted to grid nodes.
- New areas and regions are created without checking whether one already exists in the required location.

(Lines are only created where required.)

Grid plane List

A region will be created for each region specified in this list.

Panel Identification

Panels are enclosed areas on a grid plane. Panels can be identified by a closed loop of **lines**, **members** or **1D elements**. The panels will be detected automatically.

8.5.3 Create New Model from Deformed Geometry

This dialog can be accessed from the “Tools | Manipulate Model | Create New Model for Deformed Geometry” menu command.

The Create New Model for Deformed Geometry command creates a new model from the existing model by taking the geometry and updating it with displacements from an analysis or combination case.

Before entering this dialog the user is warned that a new model will be created and offers the user the opportunity of saving the current model before generating a new model.

Case

The analysis case or combination case on which the deformation is to be based.

Scale displacements by

The displacements are scaled by the given factor before the coordinates are updated.

Maximum displacement

The displacements are scaled to give the maximum displacement specified before the coordinates are updated.

Adjust orientation angles for rotational deformation

When set, beam element orientation angles are adjusted to account for the mid-span rotation of the element; orientation angles are also adjusted to account for elements that have moved in or out of being recognised as 'vertical'; orientation nodes are reset to 'not specified' in this process. When not set, orientation angles and nodes are not adjusted.

8.5.4 Generate 3D Orthogonal Frame

The information supplied on these pages provides a description of an portal or orthogonal frame, which is subsequently used to generate the structural model. The units of length are reported on the dialog, alongside a sketch of the structural form.

Generation

The user can choose to create analysis and/or design models with

- **Generate elements**
- **Generate members**

Bay width, Number of bays

The width of the structure is defined by the number of bays of equal bay width.

Storey height, Number of storeys

The height of the structure is defined by the number of storeys of equal storey height.

Column

Opens the [Section Wizard](#) to allow the column section to be selected. This is marked as unspecified until a section is assigned when the actual section will be displayed.

Beam

Opens the Section Wizard to allow the beam section to be selected. This is marked as unspecified until a section is assigned, when the actual section will be displayed. On the Orthogonal Frame page, there are separate buttons for the beams spanning in the x direction and for the beams spanning in the y direction.

Location

This is used to locate the new frame with respect to the existing structure. This is done by identifying the corner of the new structure that relates to a specific position or node on the existing structure.

Include supports

Encastre supports are created at the base of the columns. By default no restraints are applied to the model.

Merge with existing nodes

The generation tool initially creates a new set of nodes and elements. This option allows new nodes co-located with existing nodes to be removed and the new structure to connect with the existing.

8.5.5 Rationalize Lines

This dialog can be accessed from the "Tools | Manipulate Model | Rationalize Lines" menu command.

The Rationalize Lines command iterates through all the lines in the model and converts co-linear lines into a string of continuous but discrete lines. The original lines are deleted and the corresponding references in areas are replaced by references to resultant lines.

8.5.6 Split Warped 2D Elements

This dialog can be accessed from the "Tools | Manipulate Model | Split Warped 2D Elements" menu command.

The Split Warped 2D Elements command splits each non-planar Quad element that is currently

selected into two Tri elements.

8.5.7 Check Entity Lists for Completeness

This gives a check across multiple lists to see if they give a complete description of the model.

Check

This gives a selection of the entities to be checked, such as nodes or elements.

Lists

Normally multiple lists will be selected and checked. If any entities are missing or duplicated these are added to new lists.

8.5.8 Expand Grid Loading

This dialog is accessible from the “Tools | Expand Grid Loading” menu command. This expands grid loads and maps them to beam loads. Expanding grid loads is not necessary for grid loads to be used in the solution.

This operation does not report any errors or failure to identify elements on which to place the load, however a check can be made using the “Analysis | Check Data” menu command before carrying out this operation.

Grid loading case

The load case that is to be expanded. All grid loads in this case are expanded.

Overwrite grid loading with expanded loads

The grid loads are expanded in the same load case. The expanded loads are retained and the grid loads deleted.

Create new load case for expanded loads

The grid loads are expanded and the expanded grid loads are saved in a new load case. The grid loads are retained.

New load case

The load case for expanded grid loads, if the grid loads do not overwrite the grid loads.

See also:

[Tolerances Specification](#)

8.5.9 Expand Gravity Loading

This dialog is accessible from the “Tools | Expand Gravity Loading” menu command. This converts gravity loads to equivalent element or node loads for elements that have mass; i.e. elements that reference section, cable, mass or 2D element properties.

This operation does not report any errors or failure to identify elements on which to place the load, however a check can be made using the “Analysis | Check Data” menu command before carrying out this operation.

Gravity loading case

The load case that is to be expanded. All gravity loads in this case are expanded.

Overwrite gravity loading with expanded loads

The gravity loads are expanded in the same load case and the gravity loads are deleted.

Create new load case for expanded loads

The expanded gravity loads saved in a new load case. The gravity loads are retained.

New load case

The load case for expanded gravity loads, if the option to overwrite is not selected.

8.5.10 Storey Displacements and Forces

This dialog is accessible from the “Tools | Storey Displacements and Forces” and “View | New Chart View | Storey Displacements and Forces” menu commands. When accessed from the Tools menu tabular output of storey displacements, drifts or forces is produced; when accessed from the View menu, storey output is displayed on charts. Output is in global axis directions.

Storey Displacements

This option calculates the displacements of the each storey and reports in-plane displacements and rotation about the global z axis.

Storey Drifts

This option calculates the relative displacements of the each storey and reports in-plane displacements and rotation about the global z axis. At least two storeys must be specified in the storey list

Storey Forces

The storey forces are calculated at mid-storey height. The structure is cut at this level and the force summed on the cutting plane.

Storey List

Normally all the storeys will be included in the calculation. However there are situation where different parts of the structure have different storeys. The storey list allows the appropriate storeys to be selected. The drift calculations assume that the selected storeys in order determine the storeys for relative calculations. For example a storey list '3 to 7 step 2' assume drift calculation for storey 5 (relative to storey 3) and storey 7 (relative to storey 5).

Displacement/Drift Calculation

To provide more control over the displacement and calculations the nodes to be considered can be specified (for example to select a single core in a building).

The displacement of the storey is calculated by summing over the selected nodes in the storey

$$u_{storey} = \frac{\sum_{i=1}^{n_{storey}} u_i}{n_{storey}}$$

the storey rotation is then calculated from the nodal displacements relative to the translational part and the coordinates relative to the centre of mass

$$\begin{aligned}\tilde{u}_i &= u_i - u_{storey} \\ \tilde{c}_i &= c_i - c_{mass}\end{aligned}$$

giving

$$\theta_{storey} = \frac{\sum_{i=1}^{n_{storey}} \left(\frac{\tilde{u}_{i,y} \tilde{c}_{i,x} - \tilde{u}_{i,x} \tilde{c}_{i,y}}{|\tilde{c}_i|^2} \right)}{n_{storey}}$$

Force Calculation

To provide more control over the force calculation the elements to be considered can be specified (for example to select a single core in a building). By default the force/moment calculations are about the global origin, but this can be offset if this is convenient. Where the model contains 2D elements there is an option to use averaged 2D stresses to calculate storey forces, otherwise unaveraged stresses are used.

Result case

This allows any analysis or combination case (provided it has a single permutation) to be selected for which the displacements of forces will be calculated.

8.5.11 Create a New Raft Model

This dialog can be accessed from the “Tools | Raft Analysis | Create New Raft Model” menu command. This command creates a new model from the existing model by taking the geometry and updating it with nodal support stiffnesses based on the soil stiffnesses calculated for the specified GsRaft analysis case.

GsRaft analysis case

The GsRaft analysis case upon which the new support stiffnesses are to be based.

8.5.12 Generate Static Vehicle Load

This dialog is accessible from the “Tools | Bridge Analysis | Generate Static Vehicle Load” menu command. This allows the user to generate a set of loads for a statically positioned vehicle. The origin of the vehicle (normally the centre of the back axle) is positioned in this dialog.

Vehicle

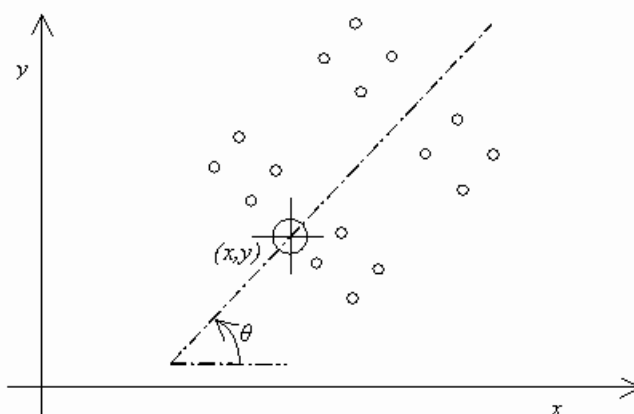
The vehicle which contributes the load. This can be either a standard vehicle or a [user defined vehicle](#).

Load case

The load case in which the vehicle load will be generated.

In space

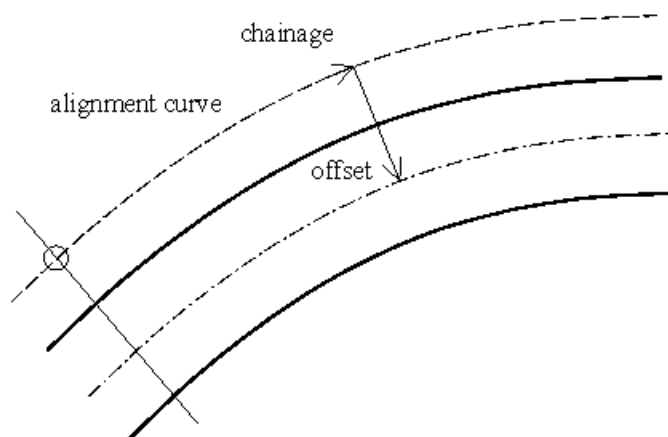
The vehicle is positioned in space. The vehicle is located at coordinates $(x, y, 0)$ in the axis set and the vehicle is rotated by the angle about the z axis. Loads are dispersed onto elements included in the element list; other elements are ignored in the load expansion process.



By alignment

The vehicle is positioned at a chainage and offset relative to an alignment curve. An alignment curve

is defined by a set of chainage and curvature values in the Alignment and Alignment Curve tables. Loads are dispersed onto elements included in the element list; other elements are ignored in the load expansion process.



By path

Vehicles can also be positioned in Paths. In this case the offset is from the centre of the path instead of from the alignment, and will normally be zero.

Advanced

Normally each wheel load is represented by a single grid point load. The advanced button gives access to the [Wheel Patch Definition dialog](#) which allows the user to specify that each vehicle wheel load should be represented by a series of grid point loads in a rectangular patch.

8.5.13 Optimise Path Loading

This dialog is available from the “Tools | Bridge analysis | Optimise path loading” menu command. This allows the user to generate a series of [Static Bridge Loads](#), based on the results of an [Influence Analysis](#) and the data in the [Path Loading](#) module. For each [Node Influence Effect](#) and [Beam Influence Effect](#) and each entry in the Path Loading table, two groups of Static Bridge Loads are generated, to give the most positive and most negative Effect (force/displacement/reaction). Each of these groups can then be used to generate a load case comprising [Grid Point Loads](#) using the [Expand Bridge Loading](#) Tool.

Optimisation options

The Static Bridge Loads module contains two types of load, optimised loads and user defined loads. The user can edit optimised loads, which converts them to user defined loads. Three options are given for what should happen to existing Static Bridge Loads when a new optimisation is carried out:

- Delete all existing static bridge loads,
- Delete existing optimised bridge loads only (user defined loads retained)
- Append to existing static bridge loads (all existing optimised loads converted to user defined loads).

Start Group Number

The first group number to be generated. If existing Static Bridge Loads are to be retained this needs to be greater than the largest existing group number.

8.5.14 Expand Bridge Loading

This dialog is accessible from the “Tools | Bridge Analysis | Expand Bridge Loading” menu command. This allows the user to generate a series of grid point load equivalent to the static bridge loads and/or moving bridge loads defined in [Static Bridge Load](#) and [Moving Bridge Load](#) tables. Load cases are created in the order of the Bridge Loads appear in the table. For Static Bridge Loads, the grid point loads generated from each group will have the same load case number. For Moving Bridge Loads, the [grid point loads](#) generated by each increment of the moving bridge load will have a new load case number. Note that changes to the bridge loading table, or to the vehicles and alignments referred to in it will not change the Grid Point Loads already generated. The load case title for the newly generated grid point loads are the same as the name of Static/Moving Bridge Loads but with a suffix of “(i)” where i equals the order of this operation, i.e. first expansion is 1 and second is 2 and so on. Ideally, Grid Point Loads already generated should be deleted manually before generating new ones through Expand Bridge Loading.

Expand bridge loading:

- **Static bridge loads**, if checked, static bridge loads will be expanded.
- **Moving bridge loads**, if checked, moving bridge loads will be expanded.

Initial load case

The first load case number to be generated. This must always be greater than the largest existing load case.

8.5.15 Batch Bridge Analysis

This carries out the following operations:

- Influence analysis
- Bridge load optimization
- Expand static bridge load
- GSS static analysis for every static load case

It is not recommended to carry out a batch bridge analysis until any problems have been identified and resolved in a step by step analysis. In many situations the user will benefit from modifying the Static Bridge Loads before expansion; batch analysis is not appropriate for these files.

Delete existing grid point loads

Normally it will be appropriate to delete all existing Grid Point Loads since they will be recreated by the batch bridge analysis process.

Delete existing static bridge loads

Normally it will be appropriate to delete all existing Static Bridge Loads since they will be recreated by the batch bridge analysis process.

If the user chooses not to delete either of these then additional Grid Point Loads will be generated for all existing and all new Static Bridge Loads. This is not recommended.

Analysis stage

The analysis stage to be analysed.

8.5.16 Wheel Patch Definition

The wheel patch definition dialog is accessed from the “[Generate Static Vehicle Load](#)” dialog and allows the user to specify that each vehicle wheel load should be represented by a series of grid point loads in a rectangular patch. This will only affect the analysis if the patch size is large compared to the panel size of the grillage. The patch size will generally be determined from loading code requirements for the contact area of the wheel with the road surface and the spread of load down to the neutral axis.

Axle number

The user selects the axle number to define.

Contact area length

The length of the contact area along the length of the vehicle.

Contact area width

The width of the contact area across the width of the vehicle.

Spread to neutral axis

The amount by which the surface patch has spread before the load reaches the neutral axis of the beam elements. This forms a uniform width border around the contact patch, giving a patch length of contact area length plus twice spread and a patch width of contact area width plus twice spread.

Number of point loads

This allows the user to influence the number of point loads to be used to represent the wheel patch. Unless the patch will cover several panels of the grillage, requesting more than 4 point loads across the width of a patch is unlikely to affect the results.

8.5.17 Create CEAP Modal Damping File

This dialog can be accessed from the “Tools | Non-linear Dynamic Analysis | Create CEAP Modal Damping File” menu command. This command creates a modal damping file for use in Oasys CEAP (Civil Engineering Application Package).

Node list

This is the list of nodes whose modal responses are written for use by CEAP.

Mode list

The list of modes that are exported to the modal damping file.

Damping

This specifies the percentage damping that is to be specified. This value is output for all modes. Any variation of damping from one mode to another must be made by editing the modal damping file.

8.5.18 Effective Mass by Part

This allows for a breakdown of the [effective mass](#) of the structure by part where the parts are defined by element lists.

The effective mass for mode i in direction j is defined as

$$\tilde{m}_{ij} = \frac{(\{\phi_i^T\}[m]\{u_j\})^2}{\hat{m}}$$

This gives information about the mass mobilised in a particular direction for a particular mode, but gives no indication as to which part of the structure this is associated with. The effective mass by

part modifies this equation to work for the elements, k , in the list so that

$$\tilde{m}_{ij,part} = \frac{\left(\sum_k \left(\phi_{ik}^T [m_k] \{u_{jk}\} \right) \right)^2}{\hat{m}}$$

Normally we are interested in the summation over all the modes so we arrive at this value by summing the modal participation factors to report

$$\tilde{m}_{j,part-sum} = \sum_i \tilde{m}_{ij,part}$$

Parts

These are the lists of elements that are to be considered that are to be considered.

Analysis task

This is the dynamic analysis (modal or Ritz) for which the effective masses are to be calculated.

Target value

Most seismic code require that a target percentage of the mass is recovered. Values that do not meet the target will be highlighted. This can be compared with either total mass or unrestrained mass.

Include modal details

By default only the summation of the effective masses over the modes is reported but mode by mode results can be provided if required.

8.5.19 Equivalent Static / Accidental Torsion Load

This dialog can be accessed from the “Tools | Seismic Analysis | Equivalent Static Load | Accidental Torsion” menu command. This command creates a load case corresponding to either an equivalent static load or an accidental torsion load to UBC 1997, IBC 2000 or FEMA 356.

Load type

The same dialog is used for both Lateral load (equivalent static load) and Accidental torsion.

Direction

The loading will be generated for either the X or Y direction. Z is assumed to be vertical

Mass offset

This is the percentage of the building dimension used in the accidental torsion case. By default this is 5% to satisfy the codes, but can be adjusted if required. GSA determines the building dimension by looking at the nodes that comprise the storey.

Method of base shear calculation

The base shear can be calculated from the procedures laid down in the UBC 1997, IBC 2000 and FEMA 356 codes or based on the results of a response spectrum analysis.

Dynamic response analysis – a list of the dynamic response cases allows the particular case to be selected. The base shear calculated in this calculation will then be used.

Seismic code procedure – the seismic details for the calculation will be as specified in the selected design code and spectrum. The supported seismic codes are:

- ASCE7-05
- Eurocode 9
- FEMA 356

- IBC 2000, 2006, 2009
- IS 1893
- UBC 1997

Equivalent lateral force procedure

Seismic Code

The code used to determine the equivalent static/accidental torsion loads.

Response details from Spectrum

The seismic details for the calculation are defined along with the other details for the response spectrum analysis. The response spectra for the selected code are listed and the relevant one should be selected. If none or no appropriate spectra are defined the "New spectrum" allows the definition of a new spectrum.

Fundamental period

By default the calculation uses the empirical period defined in the code. This can be overridden if a better can be provided.

Additional mass from loads

Where the static procedures are used the program has to calculate the weight (or mass) of the structure. In most cases the dead load should be included in the seismic weight along with a component of live load. This uses the normal load syntax eg 1.4L1 +0.25L2.

Leave blank if none.

Output

The output from this is a set of node loads. These are created for the load case specified. A title can be used to identify this case.

8.5.20 Spectrum Scaling of Modes

When a modal dynamic analysis is carried out the results are normalised to either a unit displacement or unit modal mass depending on the selection made by the user when creating the task. For response spectrum analysis these mode shapes are scaled to predict displacement values but the CQC combination procedure then combines these results across all the modes. It is useful to be able to recover directly the scaled modal values. This dialog creates a set of combination cases that correspond to the modal case scaled by the modal multiplier.

$$\alpha_i = f_{part_i} \frac{a_{spectral}}{(2\pi f)^2}$$

So that for each mode there is a combination case of the form $\alpha_i Am$.

Response spectrum

The response spectrum is used to define the spectral acceleration. This may be different for horizontal and vertical excitation, and will be affected by the participation factor, so the direction must also be specified.

Modal task

Any dynamic modal (or Ritz) task with results is made available. The process will then generate a set of combination cases which scale the modal values according to the response spectrum and direction.

8.5.21 Storey Masses

In seismic analysis there is often a need to extract mass and (torsional) inertia data for each storey of a structure. This option looks at the structure, storey by storey, summing the mass and I_{zz} inertia across all the nodes in that storey. The inertia of each storey is relative to the centre of mass of that storey. The results are output to a csv file which can be opened in Excel.

This dialog can be accessed from the “Tools | Seismic Analysis | Storey Masses” menu command. In order to use this option the model must contain storeys and there must be at least one dynamic analysis task.

Analysis task

A list of dynamic analysis tasks are displayed. The mass calculation is based on the mass specification for the modal analysis (including additional mass).

SI or User units

Output can either be in SI or the current model units.

8.5.22 Footfall Response Data

This dialog can be accessed from the “Export | Footfall Response” menu command. This allows control of the data written for analysis in the Footfall Response spreadsheet.

Nodes to export

This is the list of nodes whose modal responses are written for use by the footfall response spreadsheet. The program offers “all”, “<current selection>” if any and the stored node lists.

Excitation node set

This is the sets of nodes that comprise a path. These are defined by GSA lists. The user may select none or select one or more of the node lists. Any nodes defined in these lists are automatically added to the nodes exports in the node list.

Modal analysis task

Where there is more than one modal analysis task, the user must select the task to output.

Direction of mode shape output (Z if it is for footfall analysis)

Modal output is generated in the specified axis. Enter Z if the response is to be treated as footfall.

8.5.23 Manage Data

This dialog is accessible from the “Data | Manage Data” menu command. It is only enabled when results are not present.

Offered in this dialog are options to manipulate the data for the model as a whole. Any data management operation that causes nodes or elements to be renumbered also renumbers references to these items in other modules. (This can take a long time for large models.)

Data management of nodes, elements and load cases can be selected for execution in one pass.

Delete blank nodes

Deletes all blank nodes, renumbering subsequent nodes and the references to these in other modules. References to blank nodes are also deleted.

Delete references to blank nodes

Deletes any references to blank nodes. E.g. node loads applied to blank nodes.

Renumber nodes incrementally

Renumbers all nodes in a specified range by a constant increment. The increment can be negative but cannot be such that a node will be renumbered to less than one. If the specified renumbering results in a node being renumbered to the number of an existing node the renumbered node will overwrite the existing. References to renumbered nodes in other modules are also renumbered.

Renumber list of nodes

Renumbers a specified list of nodes. The nodes that are to be renumbered can be specified as:

- an ordered list of nodes — the 'to' field must also be an ordered list of the same number of items. E.g. from "11 13 15" to "21 3 25" renumbers node 11 to node 21, 13 to 3 and 15 to 25.
- a GSA node list — the 'to' field must be the new number of the first item in the node list; other node numbers are incremented by the same amount, thereby preserving gaps. E.g. from "2 to 6 step 2" to "102" renumbers node 2 to node 102, 4 to 104 and 6 to 106.

Renumber nodes along vector

Renumbers all nodes progressively along the direction of a specified vector. The vector is specified in global axes. References to renumbered nodes in other modules are also renumbered.

Delete list of nodes

Deletes nodes in the specified list. Nodes that are referenced by elements or members are not deleted. References to deleted nodes in other modules are also deleted.

Delete blank elements

Deletes all blank elements, renumbering subsequent elements and the references to these in other modules. References to blank elements are also deleted.

Delete references to blank elements

Deletes any references to blank elements. E.g. beam loads applied to blank elements.

Renumber elements incrementally

Renumbers all elements in a specified range by a constant increment. The increment can be negative but cannot be such that an element will be renumbered to less than one. If the specified renumbering results in an element being renumbered to the number of an existing element the renumbered element will overwrite the existing. References to renumbered elements in other modules are also renumbered.

Renumber list of elements

Renumbers a specified list of elements. The elements that are to be renumbered can be specified as:

- an ordered list of elements — the 'to' field must also be an ordered list of the same number of items. E.g. from "11 13 15" to "21 3 25" renumbers element 11 to element 21, 13 to 3 and 15 to 25.
- a GSA element list — the 'to' field must be the new number of the first item in the element list; other element numbers are incremented by the same amount, thereby preserving gaps. E.g. from "PB1" to "102" renumbers the lowest numbered element that references beam section 1 to element 102 and other elements that reference beam section 1 are incremented by the same amount.

Renumber elements along vector

Renumbers all elements progressively along the direction of a specified vector. The vector is specified in global axes. References to renumbered elements in other modules are also renumbered.

Delete list of elements

Deletes elements in the specified list. References to deleted elements in other modules are also deleted.

Renumber list of load cases

Renumbers a specified list of load cases. The load cases that are to be renumbered can be specified as:

- an ordered list of load cases — the 'to' field must also be an ordered list of the same number of items. E.g. from "11 13 15" to "21 3 25" renumbers load case 11 to load case 21, 13 to 3 and 15 to 25.
- a GSA case list — the 'to' field must be the new number of the first item in the case list; other load case numbers are incremented by the same amount, thereby preserving gaps. E.g. from "2 to 6 step 2" to "102" renumbers load case 2 to load case 102, 4 to 104 and 6 to 106.

Delete loads for a list of load cases

Deletes all loads applied in any of the load cases specified in the list of load cases.

8.5.24 Manage Nodal User Modules / Manage Element User Modules

These dialogs can be accessed from the "Data | Manage User Modules" menu command and from the right-click menu displayed by right-clicking on "Nodal User Modules" or "Element User Modules" on the Output tab of the Gateway.

User modules can be:

- **renamed**
- **deleted**
- **exported**
- **generated**

These operations may be carried out for selected user modules.

Modify

Open the Modify User Module dialog for the selected user module to allow that module to be deleted, undeleted or renamed.

Delete

Flag the selected user module for deletion. If already flagged for deletion then this button switches off that flag.

Export

Export the selected user modules to GWA file.

Generate

Create a new user module by applying the one of the following functions to the values in the selected user modules.

- min()
- max()
- abs()
- signabs()
- sum()
- srss()

Export All

Export all user modules to GWA file..

Reset All

Resets settings for all (nodal / element) user modules to the state on entry to this dialog box.

Delete All

Flags all (nodal / element) user modules for deletion.

8.5.25 Generate 2D Mesh for Polylines (legacy option)

2D element meshes can be generated for areas enclosed by one or more planar polygons. The option is given to use either the current polyline or a list of references to the 2D Polylines module.

Note that this method of 2D element mesh generation has been replaced by the method described in the Step By Step Guide section on [Generating 2D element meshes](#).

When a list of 2D Polylines is used to define the region meshes may be generated for several regions at a time. Where the edges of two regions are connected by coincident polyline segments, the mesh is generated as a connected set of elements. For this purpose a pair of coincident segments is defined as two lines with coincident end points, though the end of one line may be the beginning of the other and vice versa.

Polylines are assumed to be closed. Closing the polyline by setting the last point coincident with the first will confuse the mesh generator.

One of the parameters used to guide the mesh generation is the number of elements along each line in the polygon. Since this is specified as a single value for all lines it will be apparent that using a series of short lines will produce a finer mesh than a single long line. This can be used to persuade the mesh generator to produce areas of fine mesh. (Though in fact the mesh generator uses the number of elements along each line as a minimum rather than as an exact value on which to base the element generation.)

The mesh generator is a proprietary tool that has been plugged into GSA. In fact the tool only generates planes of 2D elements. In order to generate elements in 3D space GSA effectively unfolds the regions onto a surface and presents these to the generator. The generated elements are then folded back up into the original 3D space. Problems may arise if mesh generation is attempted on regions that cannot be unfolded onto a surface without splitting a seam.

2D element mesh generation is invoked using the “Tools | Legacy Options | Generate 2D Mesh for Polylines” menu command. The procedure is as follows.

1. The node and element attributes assigned to new nodes and elements are taken from the data defaults as set in the [Data Defaults dialog box](#).
2. Set the [cursor mode](#) to Polyline.
3. [Form a polyline](#) to define the region to be filled with 2D elements.
4. If several regions are to be generated then give the “Edit | Save Selection as 2D Polyline” command and repeat for each region.
5. Give the “Tools | Legacy Options | Generate 2D Mesh for Polylines” command.
6. Specify the mesh generation control parameters in the Generate 2D Elements dialog box, as follows:

The Generate 2D Elements dialog box is used to generate a mesh of Quad4 elements.

Note that the mesh generator employed in this operation uses the parameters specified in the dialog box as guidance only. It is always advisable to use the “Preview” command before committing to the generated mesh.

Current line / 2D polyline list

Either use the current polyline, as displayed in the Graphic View, or a list of saved 2D Polylines to define the regions in which 2D elements are to be generated. The list may be typed in directly, e.g “1 3”, or may refer to a saved list, in which case the saved list should be type “Unspecified”.

Num. Elements per segment (minimum)

Specifies the minimum number of elements formed along each segment of the region.

Generate layer of boundary elements

When checked, a layer of elements is formed around the boundary of the regions acting as a barrier between the boundary and the more complex inner parts of the mesh. It is generally better to leave this option checked.

Mesh refinement, Refinement coefficient

Specifies the extent of influence of an area of refined mesh. Enter a number between 0 and 0.35. The closer the coefficient is to zero the smaller will be the area of fine mesh.

Coincident nodes, Use existing nodes where present

When checked, wherever a new node is required during the execution of this operation the program searches existing nodes for one that lies within the **Coincidence tolerance** of the required position, and uses the first node found instead of creating a new node. Otherwise new nodes are always created.

Preview

Display the consequences of this operation in the current Graphic View without committing to applying the operation. Exiting this dialog box by "Cancel" will "undo" the preview.

The resulting mesh is made up of linear 2D elements. These may be modified to parabolic elements using the Sculpt "[Modify Elements](#)" command.

Note that more explicit 2D element mesh generation can also be achieved by [splitting elements](#).

See also:

[Working with the Program — Working with Graphic Views — Polylines in Graphic Views](#)

[Step By Step Guide — Constructing a GSA model — Generating 2D element meshes](#)

8.5.26 Hollow Slab Definition

This dialog is accessed from the 2D element property wizard and is to simplify the specification of slabs that may contain voids.

Slab thickness

The actual thickness of the slab

Thickness of top/bottom zone

The top and bottom zones are assumed to have the same thickness and the material is assumed to be concentrated in these zones.

Proportion of material in void zone

Typically the central part of the slab will have some material so a percentage can be specified here.

Effective depth

These are the effective depths for bending and in-plane stiffness calculations, based on an overall thickness, D , top and bottom zones of thickness, t , and proportion of material in void, v

For in-plane effects the effective thickness is:

$$\hat{D} = vD + (1 - v)2t$$

For bending effects the effective thickness that gives an equivalent second moment of area is:

$$\tilde{D} = \nu D + (1 - \nu) \sqrt[3]{6D^2t - 12Dt^2 + 8t^3}$$

8.5.27 Create Load Panels on Surface

This tool gives a quick route to creating load panels on a surface. An element list is used to define the surface - either the current selection, a saved list or a user defined list of beam elements. Properties are assigned for simple load panels (triangular and rectangular panels) and compound load panels (general polygonal panels) - typically the support conditions will be different for the two. A new list can be created to make it easier to identify the load panels.

The [Create Facade List](#) tool can be useful in identifying the nodes on facades.

8.5.28 Create Facade List

There are a set of tools to manipulate the model and create facades. The simpler tools allow creation of facades on the north, south east and west extremes of the structure. This is useful for simple rectangular structures but does not cope with more complex geometries.

The more general tool creates facades on the whole model, with out as a list of 'Exterior Nodes' and 'Exterior Elements'.

These are normally used in conjunction with the [Create Load Panels on Surface](#) to skin a structure.

8.5.29 Create Patterned Loads

In most design codes there is a need to consider different load factors on different parts of the structure. This dialog assists in creating the loading that needs to be considered. Details of this methodology are covered in the technical note [Patterned Load Analysis](#)

Elements considered

Only these elements will be considered for pattern loading. This allows the technique to be applied to one part of the model, for example a single floor.

Dead and imposed load factors

This specifies the adverse and beneficial factors ($\gamma_{G,sup}$ and $\gamma_{G,inf}$) for dead and the adverse and beneficial factors ($\gamma_{Q,sup}$ and $\gamma_{Q,inf}$) imposed loads. These are used to set up mean and difference loads

$$f_{mean} = \frac{\gamma_{G,sup} + \gamma_{G,inf}}{2} f_G + \frac{\gamma_{Q,sup} + \gamma_{Q,inf}}{2} f_Q$$

$$\Delta f_i = \frac{\gamma_{G,sup} - \gamma_{G,inf}}{2} f_{G,i} + \frac{\gamma_{Q,sup} - \gamma_{Q,inf}}{2} f_{Q,i}$$

Case descriptions for dead/imposed loads

These are the descriptions of the loads cases that specify the dead and imposed load patterns. This allows for the specification of the spatial distribution of these loads.

Include loads

Determines which loads should be included in the load patterns. Only beam force/moment loads, face loads on load panels and gravity loads are considered.

The loads specified in the relevant load cases are factored according to the equations above. One load case for the mean loads and one load case for each element considered for pattern loading. A

set of load case titles are set up for these with the title "Mean load" for the means load and "Difference load L#" for the difference loads when # is the element number. These new load cases should then be included in the analysis task.

See also [Create Patterned Load Combinations](#).

8.5.30 Create Patterned Load Combinations

In most design codes there is a need to consider different load factors on different parts of the structure. This dialog assists in creating the combination cases that needs to be considered. Details of this methodology are covered in the technical note [Patterned Load Analysis](#)

Analysis case for mean loads

This is the analysis case that contains the mean loads (f_{mean})

Analysis cases for load differences

These are the analysis case that contains the difference loads (Δf_i). These are assumed to be contiguous.

Description of other cases

This allows any other cases to be added to the description.

The combination cases to be considered are:

$$f_{Amax} = f_{mean} + \sum_i |\Delta f_i|$$

$$f_{Amin} = f_{mean} - \sum_i |\Delta f_i|$$

This will set up a series of combination cases that correspond to the Δf_i analysis cases. The absolute value of these are then combined with the analysis case corresponding to the mean loads to give the maximum and minimum cases. So for example if the mean case is A3 and the difference cases are A4 to A8. Then assuming no other cases the combination cases will be defined as follows:

C1	A4
C2	A5
C3	A6
C4	A7
C5	A8
C6	A3 + (C1abs + C2abs + C3abs + C4abs + C5abs)
C7	A3 - (C1abs + C2abs + C3abs + C4abs + C5abs)

See also [Create Patterned Load](#).

8.5.31 Create Mass from Loads

This dialog can be accessed from the "Tools | Manipulate Model | Create Mass from Loads" menu command.

This command creates a set of mass elements that give a mass corresponding to a specified load in a given direction. This is typically used to generate masses for exporting to programs that do not have the facility to include the effects of dead load as mass for dynamic analysis.

If the tool is run before properties have been assigned to 2d elements, the additional mass is assumed to be zero.

Case definition

The load definition that is to be used to generate equivalent masses.

Direction

This specifies the direction of load to be used in the conversion of the force (a vector) to mass (a scalar). This will in most cases be the z direction.

Scale factor

The loads are scaled by this factor before being assigned mass.

Element group

This specifies the group number for the generated mass elements. This should be chosen so that these elements can be accessed as a unique group.

Analysis stage

This specifies which analysis stage is to be considered in the creation of mass, defaulting to the whole model. This option is only available if the Analysis Stages option is enabled in the [Preferences | Advanced Features](#).

8.5.32 Create Mass from Grid Loads

There are cases where dead loads representing mass have been defined as loads. In some instances, particularly for dynamic analysis, where it is useful to be able to convert these to masses at nodes. The additional mass option allows for this, but in some instances, for example staged analysis, where it is more convenient to create a set of mass elements to represent this mass.

The case definition allow the loading to be specified – either a single analysis case (eg A2) or a combined case (eg 1.4A1 + A2). The overall case description can also be scaled by the scaling factor. There is choice of expansion of grid point loads, grid line loads and grid area loads. Any loads that do not map on to the structure are ignored. Only the vertical component of load is considered and loads in the positive z direction will result in negative masses.

To make this work for stages it is possible to define a list of elements that will be excluded. This can be used to ensure that secondaries are not considered when determining where mass can be placed.

At each point where a load is to be applied a new mass element is created and a property defined that allocates the correct mass.

8.5.33 Extract a Sub-model

This feature allows a sub-model to be extracted from GSA, so that the sub-model can be investigated in more detail. There are two aspects to the sub-model: the elements that form the sub-model and the tasks that are to be associated with the sub-model.

The nodes that form the sub-model and the nodes that form the boundary between the sub-model and the remainder of the model are identified. A new model is then created for the sub-model. Nodes on the boundary are fully restrained. Properties are copied directly. Constraints and loading are updated to include only those associated with the sub-model. Tasks are updated based on the tasks selected for the sub-model.

Static analysis tasks

For static analysis tasks the displacements at the boundary nodes are extracted for each analysis case. These are then applied in a new load case as settlements, and the analysis cases updated to include these settlements.

Modal dynamic and Ritz analysis cases

For dynamic tasks there is the option of Local or Global response.

Local

If the mode is predominantly local it may be more appropriate to consider the boundary of the sub-model as fixed and to carry out a modal analysis of the sub-model. In this case the model extraction is straightforward. In this case the analysis task is copied directly

Global

It is not possible to carry out a sub-model model analysis when the mode is global. In this case the modal results are used to create a set of static loads

The modal analysis of the full model gives us eigenvalues and eigenvectors which satisfy

$$K\varphi_i - \lambda_i M\varphi_i = 0$$

This can be rearranged in the form

$$K\varphi_i = \lambda_i M\varphi_i$$

Which we can consider as a static (pseudo modal) analysis of

$$K\varphi_i = p_i$$

where

$$p_i = \lambda_i M\varphi_i$$

When extracting the sub-model the frequency (eigenvalue) and mode shape (eigenvector) can be used to create a set of node loads:

$$^n p_i = \lambda_i^n M^n \varphi_i$$

$$^n p_i = (2\pi f_i)^2 \cdot ^n M^n \varphi_i$$

The modal task in the full model is mapped to a static task in the sub-model. As for the static analysis the displacements at the boundary nodes are used to determine settlements at the boundary nodes and the inertia loads are saved as node loads. The static analysis of these loads will then recover the mode shapes of the original model.

Response spectrum analysis

Response spectrum analysis is a combination of modal results scaled to match a given response spectrum. Extracting the modal results as static load cases means that the dynamic details are lost to the sub-model. To overcome this problem the response spectrum tasks are mapped to pseudo response spectrum tasks. A pseudo response spectrum task assumes that a static analysis case represents a mode shape. The dynamic details are supplied directly to the task in the form of frequency, modal mass and effective masses. When a response spectrum task is extracted from the full model these details are recovered from the modal analysis and included in the pseudo response spectrum analysis task.

Given a shape from a pseudo modal analysis the scaling for each mode i , for excitation in the j direction is:

$$u_i = \Gamma_{i,j} \frac{a_{spectral}}{(2\pi f_i)_2} \varphi$$

Where

$$\Gamma_{i,j} = \sqrt{\frac{\mu_{i,j}}{\tilde{m}}}$$

If the pseudo modal analysis is carried out using the static procedure above, and the frequency and participation factor are known from the master model, then the modal contribution to the response spectrum analysis can be estimated. Thereafter the combination is just as for any normal modal analysis. Enter topic text here.

8.6 Preference Dialogs

[Preferences](#)

[Saved Image Settings](#)

[Preferred Units](#)

[Preferred Section](#)

[Preferred Folders](#)

8.6.1 Initial Setup Wizard

The Initial Setup wizard is displayed on entry to GSA to encourage users who are new to GSA to set some basic preference settings to suit their needs, or, at least, to make them aware of the existence of these preferences. All preferences offered in the Initial Setup wizard, and more, can also be edited in the [Preferences dialog](#).

The Initial Setup wizard displays the [General Settings](#) and [Advanced Features](#) pages.

Check the '**Don't show this setup wizard again**' option to stop showing the Initial Setup wizard on entry to GSA.

8.6.2 Initial Setup: General Settings

This page of the [Initial Setup wizard](#) offers various preference settings that may also be edited on the respective page of the [Preferences dialog](#). Refer to the documentation for the respective pages, as follows:

General settings — [Preferences: Miscellaneous](#) and [Preferences: Input](#)

Data references — [Preferences: Input](#)

Tables — [Preferences: Input](#)

Graphic settings — [Preferences: Graphics](#)

Startup — [Preferences: Miscellaneous](#)

8.6.3 Preferences

This is accessible from the "Tools | Preferences" (Ctrl+F7) menu command. Preferred settings are stored so that they can be used across sessions.

Further to the preference settings included in the preference dialogs:

- Toolbar positions and the GSA window position and size are automatically stored as preferences.
- [Default view settings](#) for Graphic Views and Output Views are stored as preferences.

Refer to [Program Fundamentals: — User Preferences](#) for a general discussion on this topic.

More:

[Preferences: Graphics](#)

[Preferences: Sculpting](#)

[Preferences: Input Data](#)

[Preferences: Results](#)

[Preferences: Miscellaneous](#)

[Preferences: Advanced Features](#)

8.6.3.1 Preferences: Graphics

This page handles preferences that relate to Graphic Views.

Scaling, Auto-scale margin

Scaling to fit produces a scale that will result in the image fitting within this margin within the picture area. The margin is expressed as a percentage of the picture dimension.

Scaling, Recalculate displayed data scales on change of case

When checked, an auto-scale is calculated for each diagram and contour specified whenever the Cases list is changed for a Graphic View.

Scaling, Recalculate extents of displayed data on change of case

When checked, the maximum and minimum values for each diagram and contour specified are calculated whenever the Cases list is changed for a Graphic View. This has impact on contours when default contour values are used or when contours are specified in terms of percentage of extents and on diagrams when diagram extents are expressed in terms of percentage of extents.

Regenerate graphics on standard view

When checked, the graphics is regenerated when any of the standard view commands (e.g. 'Plan') is invoked. Note that regenerating the graphics can be slow when working with large models.

Mouse operation, Mouse sensitivity when dragging

The sensitivity of the mouse can be adjusted in terms of the affect that 1mm of mouse movement has on the operation in question. The operations that may be tuned in this way are:

- **Horizontal and vertical rotational increment**—the number of degrees of rotation per mm of mouse movement in the [Rotate cursor mode](#).
- **Zoom increment**—the amount the scale is factored per mm of mouse movement in the [Rotate cursor mode](#).
- **Eye distance increment**—the amount the eye distance is factored per mm of mouse movement in the [Rotate cursor mode](#).
- **Pan increment**—the amount the image is panned in picture mm per mm of mouse movement in the [Rotate cursor mode](#).

Mouse operation, Limit before ghost image drawn when dragging

When using the mouse in the [Rotate cursor mode](#) to rotate, zoom, pan or adjust the eye distance the image is drawn as a ghost image if the number of elements being drawn is larger than the number set here. Note that the “number of elements being drawn” is affected by the current [Volume](#) and [entity list](#).

Keyboard operation, Rotational increment per arrow key stroke

When using the arrow keys to rotate the image rotate by this many degrees per arrow key stroke.

Keyboard operation, Zoom to engineering scale

When checked, the “Graphics | Orientation | Zoom In” (Ctrl+Up) and “... | Zoom Out” (Ctrl+Dn)

menu commands change the scale up or down to the next engineering scale instead of factoring the scale by the zoom factor.

Keyboard operation, Zoom factor

When “zoom to engineering scale” is not set, the “Graphics | Orientation | Zoom In” (Ctrl+Up) and “... | Zoom Out” (Ctrl+Dn) menu commands factor the scale by this amount.

Shrink, Factor

When [shrink elements](#) is set reduce the size of drawn elements by this amount.

Labels, Restraint symbols

When not checked, restraint labels are displayed as a text string of the form “yzxx”. When checked, pinned and encastred supports are represented by symbols representing these restraint conditions, and other restraint conditions are displayed as text.

This setting does not switch restraint labels on. It does affect how the labels are displayed when they are switched on. Refer to [Labels](#) for more details.

Labels, Release symbols

When not checked, element release labels are displayed as a text string of the form “FFFFRR”. When checked, element releases are represented by symbols that indicate the direction of release.

This setting does not switch release labels on. It does affect how the labels are displayed when they are switched on. Refer to [Labels](#) for more details.

Labels, Rigid link dots

When checked, in addition to the standard representation when labelling rigid links, dots are drawn at the master and each slave node, to highlight the position of the rigid link.

This setting does not switch rigid link labels on. It does affect how rigid link labels are displayed when they are switched on. Refer to [Labels](#) for more details.

Property tips: Off | Brief | Full

Property tips can be displayed alongside the cursor when the cursor hovers over an entity (node, element, member, ...). “Brief” displays the reference of the entity plus some other key properties (e.g. topology of elements). “Full” displays the full properties for that entity (i.e. the same properties as are displayed on the [Properties tab of the Object Viewer](#) when an entity is clicked on).

Saved Image Settings

Set the preferred saved image settings that govern the output of graphic images. This option opens the [Saved Image Settings dialog box](#).

8.6.3.2 Preferences: Sculpting

This page handles preferences that relate to sculpting and to entity selection in Graphic Views.

Reference point of selection items

This setting defines the reference point of entities for [entity selection](#) and for determining whether an entity is within a [volume](#). It does not affect the identification of point entities, i.e. nodes, OD elements and grid points.

- **Any part**—If any topological point or the centre of the entity is in the selection area then the entity is selected. Click on any part to select a single entity.
- **All**—If all topological points and the centre of the entity are in the selection area then the entity is selected. Click on any part to select a single entity.
- **Drag to right — all; drag to left — any part**—If the selection area has been formed by dragging the mouse from left to right then the selection is as “All” otherwise it is as “Any part”.

- **At centre**—If the centre of the entity is in the selection area then the entity is selected. Click on the centre to select a single entity.

Append to selection when click is unmodified

When checked, previous selections are not emptied by a plain click or drag, and Ctrl+click or Ctrl+drag de-selects identified items. Otherwise <Shft> must be pressed while selecting, to append to the selection, and pressing <Ctrl> while selecting inverts the selection state of the identified items.

Highlight selected elements by line

When checked, selected 1D and 2D elements and members are highlighted by a line along or around the entity. Otherwise highlighting is done by a symbol at the entity centre.

Snap to grid points

When checked, the position of the cursor on the [construction grid](#) is deemed to be at the nearest grid point. Otherwise the position is the actual position of the cursor, to an accuracy dependent on the structure scale and the screen resolution: the larger the scale and the finer the resolution, the greater is the accuracy of the position.

The coordinates of this position (the grid coordinates) are displayed in the status bar.

Copy lists expanded

When checked, the 'Copy Selection as List' operation copies to the clip-board an explicit list of entity references, without condensing ranges.

E.g. when checked: "11 12 13 14"; when not checked: "11 to 14".

Pick tolerance

When selecting entities by clicking, an entity is selected if it is found within the pick tolerance of the cursor position. The pick tolerance is expressed in mm as measured on the screen, regardless of the structure scale.

Tolerance for coincidence

Two entities are deemed to be coincident if they lie within this tolerance of each other. The tolerance for coincidence is expressed in m as measured on the model.

Various operations, especially sculpt operations, use this setting as a default and then allow this default to be adjusted before the operation is carried out. Other operations use this setting directly.

Straightness tolerance

When selecting entities that must form a straight line the straightness tolerance is used to check the angle between pairs of entities to decide if they lie on a straight line. The straightness tolerance is expressed in °.

8.6.3.3 Preferences: Input

This page handles preferences that relate to the input data.

Defaults – Tolerances

The default tolerances for new models should be set in the [Tolerances dialog box](#).

New models adopt these tolerances. Tolerances for existing models may be reset to these from within the specification for the model.

Defaults – Units

The units that are used most frequently should be set up in the [Preferred Units dialog box](#).

New models adopt these units. Units for existing models may be reset to these from within the specification for the model.

Defaults – Section

The default section catalogue should be set up in the [Preferred Section dialog box](#).

Defaults – Design

Set the default design codes.

Data references

Data references, such as constraint axis references or load cases, may be offered either by number, by name or by name and number. If no name is given then the reference is by number, regardless of this setting.

Precision of generated node coordinates

Set the precision of coordinates stored for generated nodes, in terms of number of significant figures.

Tables – Limit on number of points in property tables

A number of tables require a set of points or pairs of values to be defined. The number of points that can be entered in the table can be set here. This does not affect existing tables, only tables opened after the setting has changed. The tables affected are:

- **Property tables**

Tables – Show drop-down list in selections

When a table requires a selection from a list of alternatives the user can choose to type in the items or alternatively get a drop down list showing all the possible options for which to select.

Tables – Show names in loads tables

Name fields are displayed in loads tables when this option is selected. (Names can always be edited in the load definition dialogs.)

Combination case tables – Number of analysis case factors

For simple combinations of analysis cases it can be convenient to specify analysis case factors directly in a spreadsheet style, but the number of fields depends on the number of analysis cases in the model. Set the number of analysis cases to be accommodated in the combination cases table, here.

Undo – Number of undo steps

The limit to the number of operations that are preserved in the undo and redo buffers. Refer to [Working with the Program — Undo limit](#) for more details.

Undo analyses

Enables the option to undo analyses. Refer to [Working with the Program — Undoing analyses](#) for more details.

8.6.3.4 Preferences: Output

This page affects the way output is presented.

Numeric Format

The numeric format governs the way that numbers are output. The settings in the preferences form the default for output and apply to any new output views or diagram. Existing output views and diagrams are not updated but the numeric format can be adjusted for each diagram or output view.

The options are

- **Engineering** — numbers are output to the specified number of significant figures with exponents which are multiples of 3
- **Decimal** — a number of decimal places is specified

- **Scientific** — numbers are output to the specified number of significant figures in exponent form.

2D Element Postprocessing – Facet Angle

When results for 2D elements are averaged some checks have to be made that stresses should be averaged across elements. When the boundary between elements is not flat but an edge the averaging should not take place. This is determined by the “Facet angle”.

GWA Numeric Precision

Select whether GWA format floating point data is output to:

- **Default Precision** — ~6 sig. fig.
- **High Precision** — ~8 sig. fig. for single precision data and ~18 sig. fig. for double precision data

Note that the COM GwaCommand function ignores this setting and always operates in High Precision.

LS-DYNA Plot Files

D3PLOT version — GSA can export selected results to LS-DYNA plot files. More recent versions of D3PLOT provide better support for files from GSA, but these features will be regarded as a corrupt database by earlier versions.

Autoscale displacements — LS-DYNA plot files store displaced coordinates rather than displacements. For linear static analysis these can be small compared with the coordinates so this option allows the displacements to be autoscaled to provide more accuracy and better visuals in D3PLOT.

8.6.3.5 Preferences: Miscellaneous

This page handles miscellaneous preferences that do not fit clearly into any other category.

Error Messages, Warning messages and notes

It is often useful to be able to limit the number of error messages that are displayed. Limiting the number of messages is achieved by setting the number of messages. The number of messages determines the number of items each individual error or warning is reported for, sequentially or in a list.

File handling, GWB read/write via memory file

When checked, data is assembled in memory on its way to or from disk; otherwise reading and writing is directly to/from disk. The preferred size of memory file buffer increment may be set; the larger the increment the faster it is to read and write large models, but more memory is used in the process.

Reading and writing via memory file is significantly faster than otherwise but huge models can choke the memory file; in this circumstance GSA offers the option of switching off this setting and continuing.

File handling, Timed backup interval

GSA will write a backup file either on demand (a manual backup) or at a timed interval (a timed backup) The timed backup interval set here is used to determine the frequency of the timed backups. Setting the timed backup interval to zero disables the timed backup mechanism.

File handling, Include results in timed backup

When checked, results are included in the timed backup file; otherwise not.

File handling, Prompt for history note when saving

When checked, when the model is saved the History dialog is displayed so as to prompt for the addition of a history note.

File handling, Save file after batch analysis

When checked, the data file is automatically saved to disk at the end of every batch analysis; otherwise not.

Startup, Enable version checking

The version of each executable file being run is compared with the currently recognised versions listed on the Oasys web server. A version status is reported in the [Welcome to GSA dialog](#). Details of checks are reported in 'Help | About'.

Startup, Show Welcome to GSA

The [Welcome to GSA dialog](#) is displayed on startup.

Startup, Initial Views

Graphic Views and Output Views open at the time of saving a model, are displayed when the model is opened, depending on the **Initial Graphic Views** and **Initial Output Views** settings. (At the time of saving the model all open Graphic Views and Output Views are always saved with the model, regardless of these settings.) Setting **Initial Graphic Views** to 'empty' results in the element and member lists for the initial Graphic View being set to 'None'.

Batch output file formats

Batch 'save to file' options for **Saved Graphic Views** and **Saved Output Views** will be saved in the format specified here.

Solver options – GSS Solution

The default solution method for GSS should be set up in the Preferred Solution dialog box. Use this to set the default matrix solution method, the default plate formulation for linear 2D elements and eigensolver defaults.

Advanced

Opens the [Advanced Preferences dialog](#) to specify the preferred behaviour of various view types.

Company Info

Opens the Company Information dialog to set up your company name and logo that appear on printed output.

File Locations

Opens the [Preferred Folders dialog](#) to determine where the database of catalogue section is stored and backup files are created.

Page Setup

Opens the Page Setup dialog allowing the style of output for printed text and graphics to be selected.

If "Calculation Sheet Layout" is selected the page is formatted as a calculation sheet with details inserted in the page header. If "Logo" is selected the company logo is inserted in the top left corner of the page. If "Border" is selected this gives a border but no header information. If "Clipped" is selected the output is clipped leaving a space for the logo. This has no effect on text output.

8.6.3.6 Preferences: Advanced Features

The advanced features is where the user can selected to simplify the interface by switching off options or allow access to more advanced features.

If the feature is greyed then it has not been licensed.

Analysis

This switches on various features associated with analysis options:

- [Modal dynamic analysis](#)
- [P-delta](#) and [buckling](#) analysis

- [Dynamic response analysis](#)—requires Modal dynamic analysis
- [Non-linear analysis](#)
- [Fabric analysis](#)—requires non-linear analysis
- [Form finding analysis](#)—requires non-linear analysis
- [Raft analysis](#)
- [Bridge analysis](#)
- [Analysis stages](#)

Design

This allows for design options to be enabled:

- [Steel design](#)
- [Concrete design](#)

8.6.4 Saved Image Settings

The Saved Image Settings dialog box allows the parameters that govern the output of graphic images to be specified.

Open this dialog box from the [Graphics page](#) of the [Preferences dialog box](#).

PNG and JPEG, Screen resolution / Print resolution

PNG and JPEG, Image size: % of window size / specified size

The total size of PNG and JPEG images, in pixels, is determined by: resolution x size

JPEG Quality factor

Specifies the quality of the JPEG image. The lower the quality setting, the smaller the file size and the poorer the picture quality. For images containing large colour-filled areas a lower quality factor is adequate (~75%). Images of intricate linework require a higher quality factor to avoid colour bleeding (~90%).

AVI Frame rate

Specifies the number of frames per second at which the recording is to be played. A higher frame rate results in a faster playback of the recording.

AVI Freeze time

Specifies the number of seconds the image is to be frozen each time the Freeze recording command is given during a recording. The image is displayed as a still image for this many seconds during playback.

See also:

[Working with the Program — Working with Graphic Views — Saving the graphic image to file](#)

[Working with the Program — Working with Graphic Views — Recording a moving graphic image](#)

8.6.5 Preferred Units

The Preferred Units dialog is available from the [Input page](#) of the Preferences. This is where the user can select his preferred set of units. These are the units that are used by default for a new model.

Force

The user can select preferred force units from the list or define a new force unit and give the

conversion factor to N.

Length (large)

The user can select preferred “large” length units from the list or define a new length unit and give the conversion factor to m. “Large” length units are used for data that are typically large such as coordinates.

Length (small)

The user can select preferred “small” length units from the list or define a new displacement unit and give the conversion factor to m. “Small” length units are used for data that are typically small such as displacements and cross-sectional areas. “Small” length units are considered as distinct from “large” length units.

Length (sections)

The user can select preferred “section” length units from the list or define a new unit and give the conversion factor to m. “Section” length units are used for beam section properties such as I values. “Section” length units are considered as distinct from “small” and “large” length units.

Mass

The user can select preferred mass units from the list or define a new mass unit and give the conversion factor to kg.

Time

The user can select preferred time units from the list or define a new time unit and give the conversion factor to s.

Temperature

The user can select preferred temperature units from the list or define a new temperature unit and give the conversion factor to K.

Stress

The user can select preferred stress units from the list or define a new stress unit and give the conversion factor to Pa.

Acceleration

The user can select preferred acceleration units from the list or define a new acceleration unit and give the conversion factor to m/s².

Energy

The user can select preferred energy units from the list or define a new energy unit and give the conversion factor to J.

Gravity

The gravity value is reported based on the current length and time units.

8.6.6 Preferred Section

The Preferred Section dialog is available from the [Input Data page](#) of the Preferences. This is where the user can select his preferred section catalogue.

8.6.7 Preferred Folders

The preferred folders dialog is available from the [Miscellaneous page](#) of the Preferences. This is where the user can select locations for files.

Section database

Pathname for the database where characteristics of [catalogue sections](#) are stored. The user may choose to specify and use a particular location for this file, or may choose to use the default location. The default location is that of the installed section database file in the program directory.

The choice of default or specified location is stored in the system registry for the current user, together with the pathname of the specified location. Hence these options may vary depending on which user is running the program.

Temporary files

Folder for manual and timed backup files, and for temporary files used during analysis. The user may choose to specify and use a particular folder, or may choose to use the default folder. The default folder is the system temporary folder.

The choice of default or specified folder is stored in the system registry for the current machine, along with the pathname of the specified location. These are picked up whichever user is logged in there.

Nationally Determined Parameters files

Pathname for the file where nationally determined parameters are stored. The user may choose to specify and use a particular location for this file, or may choose to use the default location. The default location is that of the installed nationally determined parameters file in the program directory.

The choice of default or specified location is stored in the system registry for the current user, together with the pathname of the specified location. Hence these options may vary depending on which user is running the program.

8.6.8 Advanced Preferences

The Advanced Preferences dialog is available from the [Miscellaneous page](#) of the Preferences. This is where various low level settings can be set including those relating to threading and the behaviour of various view types.

Graphic View Options

Render Graphic Views in a separate thread – when checked, Graphic Views are rendered in a separate thread allowing other GSA activity to continue during graphics rendering, and the graphics rendering to be interrupted. Note that animation is disabled when this option is not set.

Crude graphics rendering in Graphic Views – when checked, Graphic Views are displayed using the relatively crude GDI graphics library; otherwise OpenGL is used. Some graphics options, such as translucency and animation, are not available when crude graphics rendering is set.

GDI overlay in Graphic Views – when checked, the legend, selection marks and rubber-banding are drawn using GDI. Otherwise these items are drawn using OpenGL or GDI depending on the 'Crude graphics rendering in Graphic Views' setting.

Output View Options

Render Output Views in a separate thread – when checked, the content of Output Views is prepared in a separate thread allowing other GSA activity to continue during output preparation, and the output preparation to be interrupted.

Analysis Options

Perform analysis in a separate thread – when checked, analysis is performed in a separate thread allowing other GSA activity to continue during analysis, and analysis to be interrupted.

Number of threads for parallel solver – specifies the number of cores/processors that are to be used by the parallel solver. When set to 'auto' all cores/processors available will be used.

Debug Diagnostics

These should normally be unchecked, but can be useful for tracking down problems in the GSA code.

General Options

Enable undo for edits – when checked, undo is enabled for edits carried out on the model. Refer to [Working with the Program — Undoing Edits](#) for more details.

Use whole model undo logic – changes the way that data is saved to support undoing sculpting and bulk tool edits.

Legacy options

Mesh generator – select the default mesh generation engine.

8.7 Miscellaneous Dialogs

[Find](#)

[Replace](#)

[Modify](#)

[Go To](#)

[Modify Curve](#)

[Curve Data Selection](#)

[GWA Import Options](#)

[CAD Export Options](#)

[CAD Import Options](#)

[DXF Export Options \(legacy option\)](#)

[DXF Import Options \(legacy option\)](#)

[NASTRAN Export Options](#)

[OpenSees Export Options](#)

[ADC AdBeam Export](#)

[Export Member Input Data to CSV](#)

[1D Element Results](#)

[Numeric Format](#)

[Axes](#)

[History](#)

8.7.1 Find

The find dialog is available from the “Edit | Find” (Ctrl+F) menu command and from the Find option on the [Data options toolbar](#). This is used to find values in tables.

Find what

This is the string that is to be searched for in the table.

Match case

Check this box if the match is to be case sensitive.

Find in

The search can either be in the selected text or the whole file.

Direction

The search can be either up or down through the table.

Find Next

This will continue the search for the string.

Close

This finishes the search and closes the dialog.

8.7.2 Replace

The replace dialog is available from the “Edit | Replace” (Ctrl+H) menu command and from the Replace option on the [Data options toolbar](#). This is used to find values in tables and replace with another value.

Find what

The string that is to be searched for in the table.

Replace with

The string that will replace the found string in the table.

Match case

The match is to be case sensitive. By default the match is case insensitive.

Replace in

The search can either be in the selected text or the whole file.

Find Next

Continue the search for the string.

Replace

Confirms that you want to replace the found string with the new string.

Replace All

Confirms that you want to replace all occurrences of the old string with the new string.

8.7.3 Modify

The modify dialog is available from the “Edit | Modify” (Ctrl+M) menu command and from the Modify option on the [Data options toolbar](#). This is used to modify numeric values in tables. It does not apply to text or list fields.

Modify by

This is the numeric value that applies to the modification.

Add

The value will be added to the existing table values.

Factor

The existing table values will be factored by the numeric value.

Power

The existing table values will be raised to the power of the specified value.

Absolute

The existing table values are all made positive.

Modify in

The modification can either be in the selected text or the whole file.

8.7.4 Go To

The go to dialog is available from the “Edit | Go To” (Ctrl+G) menu command and from the Go To option on the [Data options toolbar](#). This is used to go to a particular line in a table.

Record number

Specifies the record to go to in the table.

8.7.5 Modify Curve

This dialog is made available when a curve is selected for modification. The units that apply are displayed.

Factor curve by

The user can choose to apply a factor to the curve to scale the values.

Normalise curve to

The user can choose to scale the curve so that the maximum value is as specified.

8.7.6 Curve Data Selection

When reading some file formats it is possible to have more than one set of data in a file. The curve data selection allows the user to choose the values to be read.

Abscissa

Select the value to be used for the abscissa (x axis).

Ordinate

Select the value to be used for the ordinate (y axis).

8.7.7 GWA Import Options

This dialog is accessed from the “File | Import | Text (GWA File)” menu command or from the Gateway Paste Special command and specifies the options to be applied to the GWA import. Data is imported either from file or from the clipboard in text format, GWA syntax into the existing model. For details of the GWA syntax see [GSA Text \(ASCII\) File](#). Options that would invalidate existing analysis results are disabled.

The default behaviour for the importing of GWA data is that data records that are identified by record number (and often referred to by that number) such as nodes, elements and properties overwrite existing data; other data, such as constraints and loading, append to existing data.

Data options

Data records relating to data options that are not checked will be ignored by the import.

Import mode, Overwrite

When overwrite is selected, all imported model data overwrite the original data of the same number; imported loads are imported into their respective loading modules, with no adjustment to the load case numbers. Note that saved views are always appended.

Import mode, Append

When append is selected, imported data in the following modules are appended to the original data.

- Axes
- Grid planes
- Polylines
- Nodes
- Elements
- Members
- Materials
- Properties (all types)
- Load case specifications (and load case references in loading)
- Analysis tasks and cases
- Combination cases
- User modules
- Saved views (see 'Import mode, Overwrite' above)

The numbering of the imported data is adjusted to follow on from the original data, leaving a small break in the numbering to give a visual break between the two data sets; the break in the numbering is adjusted so as to result in the renumbered data being incremented by a multiple of 10. Gaps in item numbering in both the original and imported model are preserved. For example, model A has two nodes: 2 4, as does model B; model B is appended to model A to result in model A having 4 nodes: 2 4 12 14.

References to renumbered data in the imported data are renumbered.

Note the following :

- Where properties of non-specific property type (e.g. P1, as opposed to PB1) are referenced in a list then the property reference is not renumbered.
- Combination cases are overwritten and analysis case references in combination case descriptions are not renumbered, even when the import mode is set to Append

Paste Append command

Giving the Gateway Paste Append command has the same effect as selecting all data options and selecting the Append import mode in the GWA Import Options dialog box.

8.7.8 CAD Export Options

AutoCAD version

This allows the user to select the target version of AutoCAD. The model can be exported to AutoCAD 2.5 through AutoCAD 2004.

Units of exported data

All data in GSA is stored internally as SI. However, this may not be the most useful option for AutoCAD, so the user can specify the units in which the dimensions are to be output.

Entity colours

Entity colours in the CAD file can be assigned in 3 different ways.

- **Colour by layer:** Elements will be exported to layers based on their respective property numbers. In this case, the entities are coloured by layer, and so inherit the layer colour, which in turn is based on the element property.

- **Apply GSA element colours to CAD entities:** Elements will be exported to the same layer in the CAD file, with colours corresponding to the element type in the GSA model.
- **Apply colours based on GSA element groups:** Elements will be exported to the same layer and their colours will be based on the GSA element group numbers.

Export

The user can choose to export the “Analysis” layer or the “Design” layer, exporting respectively elements and members.

Export 3D sections

If this is checked 3D representations of beams will be exported as well as a wireframe representation. The 3D sections lie on a separate layer, whose name is based on the element property name by prefixing “3D_”.

Export element offsets

If this is checked then element offsets will be considered when exporting the 3D sections.

Transform node coordinates

If this is checked and a grid plane is specified then coordinates will be transformed so as to be with respect to the grid plane axes.

8.7.9 CAD Import Options

[CAD Import: Select Layers to Import](#)

[CAD Import: Map layers to properties](#)

[CAD Import: Import Options](#)

[CAD Import: Post import options](#)

8.7.9.1 CAD Import: Select Layers to Import

This page enables selection of layers in the drawing that need to be imported.

Select layers to import

This will list all the layers in the AutoCAD drawing, and the user can select the ones that have to be imported.

Base element property names on layer names

Names of the element properties will be based on names of the layers in AutoCAD.

Edit relationship layers and element properties

Enables the property page where the relationship between layers and property names/property numbers can be edited.

Base element group numbers on entity colours

Element group numbers will be based on entity colours.

8.7.9.2 CAD Import: Map layers to properties

This page is enabled only when the “Edit relationship between layers and element properties” option in the previous page is checked. The table on this page specifies the relationships between the selected layers and the property names and numbers they are assigned, in addition to the section description, if applicable.

Tip: To automatically assign sections to the given property, the section description must be enclosed

within curly braces { and } as a part of the layer name.

Tip: The default attributes of all the properties will correspond to the defaults in the property tables of the current model.

Note: The property numbering will start from the highest section property number in the model. Changing the property number to a value already present in the property tables will overwrite it without any warning.

8.7.9.3 CAD Import: Import Options

The data can be imported to the

- **Analysis layer** —creating elements.
- **Design layer** —creating members.

Point

Specifies how AutoCAD points are to be treated in GSA.

Line

Specifies how AutoCAD lines are to be treated in GSA.

Arc / Circle / Ellipse

Specifies how AutoCAD arc, circle and ellipse entities are to be treated in GSA. The user provides either a minimum arc angle for splitting the arcs or a number of equal segments.

Poly Line

Specifies how AutoCAD polylines (LWPolyLine, 2DPolyLine and 3DPolyLine entities) are to be treated in GSA. The user can specify if curve fitting is to be used for curved polylines.

Polygon Mesh

Specifies how AutoCAD polygon meshes are to be treated in GSA. This allows the user to specify two entities, so for example the mesh may be modelled in GSA as quad elements stiffened by a beam mesh.

3D Face / Poly Face Mesh

Specifies how AutoCAD 3D Faces and Poly Face meshes are to be treated in GSA.

8.7.9.4 CAD Import: Post import options

Units of length in CAD model

Extents of the CAD model: Specifies the geometrical bounding box extents of the AutoCAD model. There are no units for the extents, and this is provided merely to provide a cue to what the model units *could* be.

Suggested units of length

This is the unit of length guessed by GSA.

Units of length and Position of the model

Import units of length: Specifies the unit of length chosen for importing the drawing.

Note: If the units selected do not correspond to the units in which the drawing was made, import may not behave as expected. For example, if a CAD drawing of a roof truss is in meters, and it is imported in mm, the element sizes could be too small in the GSA model and/or the nodes can get collapsed since they would be too close to each other.

Shift the origin of the model to...: Specifies the coordinates of the new origin (in metres, with respect to the CAD origin) which all model coordinates are transformed to. When the origin is

shifted a grid plane is created at the CAD origin, enabling coordinates to be viewed with respect to the CAD origin, despite the shift in origin.

Importing CAD models created with coordinates that are offset a long distance away from global origin results in nodal coordinates with high order of magnitude. This can lead to rounding and precision errors. To avoid such a situation, the coordinates can be transformed to a point closer to the model. (In determining whether the model is offset away from the global origin, GSA only uses entities from the layers selected on the 'Select Layers to Import' page.)

Rounding Tolerance

Same as [rounding tolerance](#).

Connect 1D elements

Same as [Connect 1D elements](#).

Auto Offset

Sets element offsets for horizontal beam elements automatically.

Coincident nodes

Sets tolerance for collapsing coincident nodes. See [collapse coincident nodes](#).

8.7.10 DXF Export Options (legacy option)

This dialog is accessed from the "Tools | Legacy Options | Export AutoCAD (DXF file)" menu command. Note that this is a legacy option. For more details on DXF export see [DXF Export](#).

AutoCAD Version

This allows the user to select the target version of AutoCAD.

Units of dimensions

All data in GSA is stored internally as SI, however, this may not be the most useful option for AutoCAD, so the user can specify the units in which the dimensions are to be output.

Export

The user can choose to export the "Analysis" layer or the "Design" layer, exporting respectively elements and members.

Export 3D sections

If this is checked 3D representations of beams will be exported as well as a wireframe representation.

8.7.11 DXF Import Options (legacy option)

This dialog is accessed from the Tools | Legacy Options | Import AutoCAD (DXF file)" menu command. Note that this is a legacy option. For more details on DXF import see [DXF Import](#).

Layer Option

The data can be imported to the

- **Analysis layer** —creating elements.
- **Design layer** —creating members.

Point

Specifies how AutoCAD points are to be treated in GSA.

Line

Specifies how AutoCAD lines are to be treated in GSA.

Arc / Circle / Ellipse

Specifies how AutoCAD arc, circle and ellipse entities are to be treated in GSA. The user specified either a minimum arc angle for splitting the arcs or a number of equal segments.

Poly Line

Specifies how AutoCAD polylines (LWPolyLine, 2DPolyLine and 3DPolyLine entities) are to be treated in GSA. The user can specify if curve fitting is to be used for curved polylines.

Polygon Mesh

Specifies how AutoCAD polygon meshed are to be treated in GSA. This allow the user to specify two entities, so for example the mesh may be modelled in GSA and quad elements on a cable mesh.

3D Face / Poly Face Mesh

Specifies how AutoCAD 3D Faces and Poly Face meshes are to be treated in GSA.

Units of dimensions

All data in GSA is stored internally as SI, however, this may not correspond to the units used in AutoCAD, so the user can specify the units of the AutoCAD model.

8.7.12 NASTRAN Export Options

This dialog is accessed from the “File | Export | NASTRAN (Bulk data file)” menu command. For more details on NASTRAN export see [NASTRAN Export](#).

Bulk data format: Small field, Large field

NASTRAN data can be written in either small or large fields. In general large field format is preferable.

Solution method / Number of modes

This determines if items such as loading should be written (static) or the user asked for the number of modes (dynamic or buckling).

Analysis stage

The model corresponding to the particular analysis stage will be exported.

Export section shape information

Some versions of NASTRAN (e.g. MSC/NASTRAN) support section shape information. If this option is selected the section dimensions are exported otherwise only the section properties (area, second moments of area etc) are exported.

Export modified section properties

If this is ticked [section modifiers](#) are taken into account when writing section properties, otherwise they are ignored.

8.7.13 OpenSees Export Options

This dialog is accessed from the “File | Export | OpenSees (.tcl file)” menu command. For more details on OpenSees export see [OpenSees Export](#).

Analysis stage

The model corresponding to the particular analysis stage will be exported.

Cases

Choose which load case to export loads for.

Export bars as:

Choose whether to export GSA bars as OpenSees Truss or Corotational Truss elements.

Export steel beams as:**Export concrete beams as:**

Choose whether to export GSA beams as OpenSees elasticBeamColumn, nonlinearBeamColumn or dispBeamColumn elements. You can choose different element models for beams with steel and concrete materials.

Export beam sections as:

Choose whether to export GSA beam sections as OpenSees Elastic or Fiber sections.

Rectangular fiber sections

Input the number of subdivisions in the Y and Z directions of rectangular sections when Fiber sections are selected.

Circular fiber sections

Input the number of subdivisions around the circumference and in the radial direction of rectangular sections when Fiber sections are selected.

General fiber sections

Input the number of subdivisions along and across the flanges and webs of I, T, channel, double channel, angle, double angle, taper, general I, taper T, taper angle and taper I sections when Fiber sections are selected.

8.7.14 ADC AdBeam Export

This dialog is accessed from the “File | Export | ADC (AdBeam file)” menu command. For more details on ADC export see [Concrete Design](#) in the “Step by Step Guide” and [Export ADC](#) in “[Interaction with Other Software](#)”.

Members

Select the members to be exported. GSA will try to assemble these into continuous beams for AdBeam.

RC Bar Limits

Opens the RC Bar Limits dialog to specify the maximum and minimum bar sizes in the AdBeam model.

Loading and results

The possible analysis cases for dead, imposed and wind loads are extracted and the user can select the cases for each load type. These should be analysis cases for unfactored loads.

Open AdBeam on completion

If this option is checked GSA will try to open AdBeam with the first file exported.

8.7.15 AdSec Export

This dialog is accessed from the “File | Export | AdSec Section (ADS file)” menu command. This allows a section from the model to be exported with optionally the forces and moments to AdSec. The concrete design code in GSA will be used in the export.

There may, at times, be a mismatch in the design codes supported by GSA and those supported by AdSec, so it may be that AdSec is not able to open the file generated by GSA.

Members/Assembly

This gives a choice of exporting a list of members or a single assembly. If the member option is chosen the data relating to that member must be consistent in order for the output file to be generated. If an assembly is selected then the position or positions must be specified. One section will be generated for each of the positions.

Cases

Analysis or combination results can be exported as AdSec loads. These are selected as a case list. If a combination case is selected it must contain only a single permutation.

Analysis properties

If an assembly is selected there is not enough information in the assembly to create a useful AdSec model. The analysis properties allow these parameters to be defined. The concrete, steel and reinforcement grades as the relevant grades for the selected design code. The cover and link size are used to locate the bars in the section. The bar size and maximum bar spacing are used to add bars to the section based on spacing bars around the perimeter.

Open AdSec on completion

If this option is checked GSA will try to open AdSec with the first file exported.

8.7.16 Export Member Input Data to CSV

This dialog allows export of comma separated variable (CSV) data that describes the information conveyed to the code checking DLLs. It can be used with the companion program SteelDesign.exe. The data in the csv file can be manipulated with a text editor or within SteelDesign.

The format of the file is not guaranteed to be stable, so do not rely on older versions of this data to be compatible with future editions of SteelDesign.

Members

Specify any valid list of members (including named lists)

Cases

Specify any valid case or list of cases (including named) lists.

The exported file contains a member definition for each member/case combination specified. Each member is titled so as to identify the GSA model, member number and case.

8.7.17 1D Element Results

The 1D Element Results dialog is accessible from the [“Analysis Wizard : Envelopes”](#), [“Diagram Settings”](#), [“Contour Settings”](#) and [“Output Settings”](#) dialogs.

During analysis, displacements and forces are calculated for 1D elements at the element ends. Intermediate results can then be calculated during post-processing, as needed. These results can be calculated at any position along the elements. The positions at which 1D element intermediate results are calculated depend on these settings.

Note that these settings are not exclusive; intermediate results can be calculated at both interesting points and a number of equidistant points.

At interesting points

When checked: intermediate results are calculated at positions that are interesting, based on the result being output and the applied loading in the case being considered.

For example:

An interesting point for displacements is where there is a displacement discontinuity.

An interesting point for shear forces is immediately to either side of a point load.

An interesting point for moments is at point loads, and along UDLs. (The number of interesting points along a UDL is set arbitrarily by the program, depending on context.)

Number of equidistant points

Intermediate results are calculated at the specified number of points evenly spaced along elements.

Include intermediate results on bars

When checked: intermediate results, including displacements, forces and results derived from these are calculated for bar elements. Otherwise only end values are reported for bars.

Wall stick to be used for equivalent beam results

Select which of the equivalent beams (wall stick) of wall element is to be viewed, it can be in either primary or secondary direction. See [Interpreting Data and Results — 2D element results — 2D wall element results](#) for more details. (This setting is ignored in analysis envelopes.)

8.7.18 Numeric Format

The Numeric Format dialog is accessible from the “[Diagram Settings](#)”, “[Contour Settings](#)” and “[Wizard: Output Settings](#)” dialog boxes, and the Numeric format button on the “[Data Options](#)” toolbar.

The numeric format allows the user to adjust the format of numeric output. The default settings are set in the [Miscellaneous page](#) of the Preferences.

The options are

- **Engineering**—numbers are output to the specified number of significant figures with exponents that are multiples of 3.
- **Decimal**—a number of decimal places is specified.
- **Scientific**—numbers are output to the specified number of significant figures in exponent form.

See also:

[Program Fundamentals — Numeric Formats](#)

8.7.19 Axes

The Axes dialog is accessible from the “[Diagram Settings](#)”, “[Contour Settings](#)” and “[Wizard: Output Settings](#)” dialogs: various [Sculpt dialog boxes](#), and the axis button on the “[Data Options](#)” toolbar.

This allows the user to choose the appropriate [axis system](#) from:

- **Global**
- **X-elevation**
- **Y-elevation**
- **Local**
- **User**—the user can then select a user defined axis from the list.
- **Default for data**

Not all of the axis options may be enabled depending on the context.

See also:

[Program Fundamentals — Axes](#)

8.7.20 History

History functionality

The History is a list of history records, each made up of a date, time, user name and an optional note, to serve as a history of changes to the model.

History records are automatically appended to the History as follows:

- When the model is first created, and each time the model is opened.
- When the model is imported; the import source is noted.
- When the model is saved in GWB format with a new name; the new name is noted.

The History is reported in the 'Titles and Model Statistics' output.

The History is saved with the model

The History is especially useful for tracking changes to a model when several people are working on the same model. In this circumstance, all participants should be encouraged to record their changes as notes in the History.

History dialog

The History dialog is accessed from the "Data | History" menu command.

In the History dialog:

- **Notes** may be edited for any history record.
- The **New** button appends a new history record with the current date and time.
- The **Delete** button deletes the currently selected history record.
- The **Delete All** button deletes the all history records.

Prompt for history note when saving

Set the 'Prompt for history note when saving' preference on the [Miscellaneous page](#) of the Preferences to have the History dialog displayed upon saving.

Part IX

9 Output Options

This chapter describes the options offered by GSA for displaying input data and results.

Most output options are presented in GSA either in tables via the Output View or as diagrams, contours or labels via the Graphic View. The following table summarises the output options available in GSA.

Subsequent sections in this chapter deal with the interpretation of this output.

9.1 Model Data

Data Type	Components ("all" unless noted)	Default axes (n/a unless noted)	Axes may be transformed	Output Views	Diagrams	Contours	Labels
Titles and Model Statistics				✓			
Analysis Task Details				✓			
Combination Case and Envelope Details				✓			
Axes				✓			
	Global Axes						✓
	Output Axes						✓
	User Axes						✓
Nodes		Global	✓	✓			
	Dots						✓
	Node Numbers						✓
	Constraint Axis Numbers						✓
Restraints and Supports				✓			✓
Joints				✓			✓
Rigid Constraints				✓			✓
Constraint Equations				✓			✓
Tied Interfaces				✓			✓
Elements				✓			
	Element Numbers						✓

	Property Numbers				✓
	Group Numbers				✓
	Section Descriptions				✓
	Section Shapes				✓
	Orientation Node Dots				✓
	Orientation Node Numbers				✓
	Orientation Angle (total)				✓
	Element Lengths				✓
	Element Releases				✓
	Element x axis				✓
	Element Axes				✓
	Element Principal Axes				✓
	Element Direction Cosines	Global	✓	✓	
Members				✓	
	Member Numbers				✓
	Property Numbers				✓
	Design Property Numbers				✓
	Restraint Property Numbers				✓
	Group Numbers				✓
	Section Descriptions				✓
	Section Shapes				✓
	Orientation Node Dots				✓

	Orientation Node Numbers				✓
	Orientation Angle (total)				✓
	Member Lengths				✓
	Member Restraints				✓
	Member x axis				✓
	Member Axes				✓
Element Geometry Shape Checks				✓	
2D Element Projected Axes		✓			
Nodal Stiffness (including Grounded Spring Stiffness)				✓	
RC Beams		✓			
Grid Planes		✓			
	Grid Plane Axes				✓
Standard Materials		✓		✓	✓
User Defined Materials		✓		✓	✓
Beam Sections		✓			
	Area		✓	✓	
	Moment of Inertia, I_{yy} Local		✓	✓	
	Moment of Inertia, I_{zz} Local		✓	✓	
	Torsion Constant, J		✓	✓	
	Shear Factor, K_y Local		✓	✓	
	Shear Factor, K_z Local		✓	✓	

	$A \times K_y$	Local		✓	✓	
	$A \times K_z$	Local		✓	✓	
	$E \times A$			✓	✓	
	$E \times I_{yy}$	Local		✓	✓	
	$E \times I_{zz}$	Local		✓	✓	
	$G \times J$			✓	✓	
	$G \times A \times K_y$	Local		✓	✓	
	$G \times A \times K_z$	Local		✓	✓	
	Section Descriptions					✓
	Section Shape					✓
Beam Section Summary			✓			
Beam Section Extended Properties			✓			
Beam Design Properties			✓			
Spring Properties			✓			
Non-linear Spring Properties			✓			
Mass Properties			✓			
	Mass			✓	✓	
2D Element Properties			✓			
	Thickness				✓	
	Bending Thickness				✓	
	In-plane Thickness				✓	
	Load Panel (Support Pattern & Ref. Edge)					✓
	Additional Mass				✓	
Link Properties			✓			
Cable Properties			✓			

Spacer Properties		✓	
RC Bar Limits		✓	
RC Beam Design Properties		✓	
Response Spectra		✓	
Basic Responses		✓	
Raft & Pile Interaction	Numbers		✓
	Areas		✓
Lists		✓	

9.2 Load Data

Data Type	Components ("all" unless noted)	Default axes (n/a unless noted)	Axes may be transformed	Output Views	Diagrams	Contours	Labels
Grid Point Loads	Force	Global	✓	✓	✓		
Grid Line Loads	Force	Global	✓	✓	✓		
Grid Area Loads	Force	Global	✓	✓	✓	✓	
Node Loads				✓			
	Force	Global	✓		✓		
	Moment	Global	✓		✓		
Applied Displacements	Translation	Global	✓	✓	✓		
	Rotation	Global	✓		✓		
Settlements				✓			
	Translation	Global	✓		✓		
	Rotation	Global	✓		✓		
Beam Loads				✓			
	Point Force	Global	✓		✓		
	Point Moment	Global	✓		✓		
	Patch Force	Global	✓		✓		
	Patch Moment	Global	✓		✓		

Member				✓			
Distortions	Translation	Global	✓		✓		
	Rotation	Global	✓		✓		
Pre-stress				✓			
Loads	Force	Global	✓		✓		
	Moment	Global	✓		✓		
Beam				✓			
Thermal	Thermal				✓		
Loads	Load				✓		
	Thermal				✓		
	Gradient						
2D Element				✓	✓		✓
Face Loads							
2D Element				✓	✓		✓
Edge Loads							
2D Element				✓			
Pre-stress	Force	Global	✓		✓		✓
Loads	Moment	Global	✓		✓		✓
2D Element				✓			
Thermal	Thermal				✓		✓
Loads	Load				✓		✓
	Thermal				✓		✓
	Gradient						
Gravity				✓	✓		
Loading							

9.3 Results

Data Type	Components ("all" unless noted)	Default axes (n/a unless noted)	Axes may be transformed	Output Views	Diagrams	Contours	Labels
Mass Details				✓			
Dynamic Details				✓			
Dynamic Summary				✓			
Buckling Details				✓			
Response Spectrum Details				✓			
Steel Design Summary				✓			
Load Factor				✓			

/					
Displacement Relationship					
Axial Force / Displacement Relationship			✓		
Total Loads and Reactions			✓		
Deformed Elevation					✓
Nodal Displacements	Global	✓	✓	✓	✓
Nodal Velocities	Global	✓	✓ (on nodes & elements)	✓	✓ (on nodes & elements)
Nodal Accelerations	Global	✓	✓ (on nodes & elements)	✓	✓ (on nodes & elements)
Reactions	Global	✓	✓	✓	✓
Rigid Constraint and Joint Forces and Moments	Global	✓	✓	✓	✓
Nodal Forces and Moments	Global	✓	✓		
Nodal Mass			✓		✓
Soil Nodal Results			✓ (on nodes & elements)		✓ (on nodes & elements)
Footfall Induced Vibration Results	Global		✓ (on nodes & elements)		✓ (on nodes & elements)
Grounded Spring & Mass Displacements	Global	✓	✓	✓	✓
Grounded Spring Forces and Moments	Global	✓	✓	✓	✓
Beam and Spring Displacements	Local	✓	✓	✓	✓

Beam End Relative Rotations	Local	✓		✓
Beam and Spring Forces and Moments	Local	✓	✓	✓
Beam Stresses	Local	✓	✓	✓
Beam Derived Stresses	Local	✓	✓	✓
Beam Strains	Local	✓		
Beam Strain Energy Density	n/a	✓	✓	✓
Average Beam Strain Energy Density	n/a	✓	✓	✓
Steel Utilisation		✓	✓	✓
Force Lines			✓	
2D Element Displacements		✓	✓	✓
2D Element Principal 2D Derived Moments			✓	
Forces and Moments	Principal 2D Forces	✓		
	Maximum 2D Moment, M_{max}	✓		✓
	Minimum 2D Moment, M_{min}	✓		✓
	Angle to M_{max}	✓		
	Maximum 2D Force, N_{max}	✓		✓
	Minimum 2D Force, N_{min}	✓		✓
	Angle to N_{max}	✓		
2D Element 2D Moment		✓	✓	

Projected Moments	2D Moment, M_x		✓	✓		✓
	2D Moment, M_y		✓	✓		✓
	2D Moment, M_{xy}		✓	✓		✓
	2D Moment, $ M_x + M_{xy} $		✓	✓		✓
	2D Moment, $ M_y + M_{yx} $		✓	✓		✓
2D Element Projected Forces	2D Force		✓		✓	
	2D Through-Thickness Shear				✓	
	2D Force, N_x		✓	✓		✓
	2D Force, N_y		✓	✓		✓
	2D Force, N_{xy}		✓	✓		✓
	2D Through-Thickness Shear, Q_x			✓		✓
	2D Through-Thickness Shear, Q_y			✓		✓
	2D Stress, Principal				✓	
2D Element Derived Stresses	2D Stress, Max			✓		✓
	2D Stress, Min			✓		✓
	2D Stress, Global Angle to Max	Global	✓	✓		
	2D Stress, Max Shear			✓		✓
	2D Stress, Von Mises			✓		✓
	2D Stress, Average			✓		✓
	2D	Global	✓	✓	✓	✓

Projected Stresses	Projected Stress, xx					
	2D Projected Stress, yy	Global	✓	✓	✓	✓
	2D Projected Stress, zz	Global	✓	✓	✓	✓
	2D Projected Stress, xy	Global	✓	✓	✓	✓
	2D Projected Stress, yz	Global	✓	✓	✓	✓
	2D Projected Stress, zx	Global	✓	✓	✓	✓
2D Wall Element Forces and Moments - wall element forces and moments are output as the equivalent beam forces and moments. There are two equivalent beams for each wall element, one along x direction (primary direction) and one along y direction (secondary direction)	Axial Force of the equivalent beam, F_x	Local		✓		✓
	Shear Force in y direction of the equivalent beam, F_y	Local		✓		✓
	Shear Force in z direction of the equivalent beam, F_z	Local		✓		✓
	Torsional moment of the equivalent beam, M_{xx}	Local		✓		✓
	Moment about y axis of the equivalent beam, M_{yy}	Local		✓		✓
	Moment about z axis of the equivalent beam, M_{zz}	Local		✓		✓
	2D Element Stress Error			✓		✓

Soil 2D Element Contact Bearing Pressure	✓		✓
RC Slab Reinforcem ent	✓	✓	✓
Element Strain Energy	✓		✓
Assembly Displaceme nts	✓	✓	✓
Assembly Forces and Moments	✓	✓	✓

9.4 Bridge Data

Data Type	Components ("all" unless noted)	Default axes (n/a unless noted)	Axes may be transformed	Output Views	Diagrams	Contours	Labels
Alignments				✓	✓		
Paths				✓	✓		
Standard Vehicles				✓			
User Vehicles				✓			
Standard Bridge VUDL				✓			
User Bridge VUDL				✓			
Node Influence Effects				✓	✓		
Beam Influence Effects				✓	✓		
Path Loading				✓			
Static Bridge Load				✓			
Moving Bridge Load				✓			

9.5 Bridge Results

Data Type	Components ("all" unless noted)	Default axes (n/a unless noted)	Axes may be transformed	Output Views	Diagrams	Contours	Labels
Node Influence Results				✓			
Beam Influence Results				✓			
Node Influence Lobe Details				✓			
Beam Influence Lobe Details				✓			
Influence Lines					✓		

9.6 Analysis Stage Data

Data Type	Components ("all" unless noted)	Default axes (n/a unless noted)	Axes may be transformed	Output Views	Diagrams	Contours	Labels
Stage Definition				✓			
Stage Properties				✓			

9.7 User Modules

Data Type	Components ("all" unless noted)	Default axes (n/a unless noted)	Axes may be transformed	Output Views	Diagrams	Contours	Labels
Nodal User Modules				✓		✓	
Nodal User Modules on Elements				✓		✓	
Element User Modules				✓		✓	

9.8 Analysis Diagnostics

[Error norm](#)

9.8.1 Error norm

Static analysis

In a static analysis the error norm is calculated only when ill-conditioning is suspected. The calculation is as follows:

Calculate the residual forces and solve for residual displacements

$$\begin{aligned}\{f_R\} &= \{f\} - [K]\{u\} \\ \{u_R\} &= [K]^{-1}\{f_R\}\end{aligned}$$

and then the error norm is

$$e = \frac{\|u_R\|}{\|u\|}$$

where

$$\|u\| = \sqrt{\sum u_i^2}$$

These error norms are reported in the Analysis Details output.

Modal analysis

In a modal analysis the error norm is always calculated. The calculation for a dynamic analysis is as follows:

$$e = \frac{\|[K]\{u\} - \lambda[M]\{u\}\|}{\|[K]\{u\}\|}$$

λ is the square of circular frequency.

For a buckling analysis:

$$e = \frac{\|[K]\{u\} + \lambda[K_g]\{u\}\|}{\|[K]\{u\}\|}$$

λ is the load factor.

These are reported in the Analysis Details output.

9.9 Results

[Static analysis results](#)

[Modal analysis results](#)

[Displacements](#)

[Reactions](#)

[Soil Contact Bearing Pressure](#)

[Beam element results](#)

[2D element results](#)

[Stress averaging in 2D elements](#)

9.9.1 Static analysis results

The basic results of an analysis are the displacements and rotations at nodes, generally referred to as displacements. All other results are derived from the displacements. The forces or stresses in elements depend on the loading on the elements and the displacements at nodes. The force and stress results available will depend on the element types. The other results at nodes are the reactions. Reactions at nodes are calculated by looking at all the element forces and the load applied to the node. For an unrestrained node these should be in equilibrium but for a restrained node the out-of-balance force must be balanced by the reaction force.

9.9.2 Modal analysis results

In a static analysis there is a unique set of displacement results. By contrast, in a modal analysis the displacements are mode shapes and so can be scaled by any constant and still represent a solution of the eigenproblem. Some method is required to decide on the scaling of the results and in GSA the results are scaled so that the maximum translational displacement at any node in the mode is 1m. (If there are no translation the results are scaled so that the maximum rotation at any node in 1 rad.)

Once the scaling for the modes is fixed there are a number of other results specific to modal analysis. These are described below.

Dynamic Analysis

The motivation for a modal dynamic analysis is to characterise the dynamic response of a structure in discrete modes. Along with the mode shape (the eigenvectors) are the load factors (the natural frequencies). The “Global Results — Buckling Details” reports the following data:

- **Frequency**
- **Modal mass**
- **Modal stiffness**
- **Total mass and inertia of structure**
- **Centre of mass**
- **Effective mass and inertia**
- **Participation factors**

These terms are defined in the [Gss Theory](#) section.

The natural frequencies, modal stiffness and modal mass are related as follows:

$$f = \frac{1}{2\pi} \sqrt{\frac{[K]}{[M]}}$$

The effective masses can be used to check that sufficient modes have been chosen to capture the dynamic behaviour of the structure. In the limit the sum of the effective masses for each of the global direction should approach the total mass of the structure.

Buckling Analysis

The motivation for a buckling analysis is to characterise the response of a structure, to the given

loading, in discrete modes. Along with the mode shape (the eigenvectors) are the load factors (the eigenvalues). The “Global Results — Buckling Details” reports the following data:

- **Load factor**
- **Modal stiffness**
- **Modal geometric stiffness**

These terms are defined in the [Gss Theory](#) section.

The load factor, modal stiffness and modal geometric stiffness are related as follows:

$$\gamma = \frac{[K]}{[K_g]}$$

Note that load factors may be positive or negative. Negative load factors are not a cause for concern, however if the loads leading to a negative load factor can be reversed the factor will become positive.

9.9.3 Displacements

Displacements (or mode shapes) are calculated at the nodes. When viewing results graphically it is useful to be able to see deflections at intermediate points on elements. The displacements at these intermediate points are not stored but calculated as required, from the end conditions on the element and any load on the element. The number of intermediate points at which these are calculated can be selected by the user, but GSA will add points that are significant e.g. points at which distortion loads are applied.

The magnitude of the displacements in a mode shape is not significant, only the relationship between the displacements and the modal details.

Displacements can be viewed in a number of ways. The most direct way is to display the deformed geometry using the “Deformed image” command on the [Graphical Display toolbar](#), but the displacements can also be displayed as [diagrams](#) or [contours](#).

9.9.4 Reactions

Reactions are calculated at constrained nodes. The constraint can be due to a restraint or a spring support.

Total Loads & Reactions

‘Total loads’ and ‘Total reactions’ represent a sum of all loads and reactions observed in a model. Moments are reported about the global origin. It is a useful check to monitor that the total loads reported in a model are approximately in equilibrium with the total reactions. If it is judged that the total loads and reactions differ significantly, this may in general be a result of two types of error:

1. A load is applied to a degree of freedom that is not active in the model. For example, from applying a moment to the end of a bar element. In this case the load is reported as being applied, but is ignored by the solver.
2. The imbalance is as a result of the numerical solution itself. This may be evidence of [ill-conditioning](#).

To help differentiate the difference within a model, an entry ‘Ignored loads’ is reported that accounts for the first type of error above.

9.9.5 Soil Contact Bearing Pressure

The soil contact bearing pressure is available for output during or after a raft or piled-raft analysis. It is calculated as the reaction at a interaction node divided by the interaction area as specified in the Raft and Pile Interaction tables.

The pressure is reported as zero if the node is restrained in the vertical direction. After the analysis this situation will only arise if the node has been restrained explicitly. Note, however, that GsRaft may temporarily apply a restraint to a node in order to impose a settlement during the iterative process, to result in the bearing pressure being reported as zero at that stage in the analysis.

9.9.6 Beam element results

[Displacements](#)

[Rotations](#)

[Forces](#)

[Intermediate forces](#)

[Force sign convention](#)

[Beam stresses](#)

[Strain energy density](#)

[Torque lines](#)

9.9.6.1 Displacements

Displacements reported are element displacements; end values may differ from nodal displacements when the element has end releases or is offset.

9.9.6.2 Rotations

Element rotations are calculated by interpolation from the element end rotations. The end rotations for an element are those of the node unless the element end has degrees of freedom that are released. In these situations the rotation at the end of the element has to be calculated

The basic equation relating force and displacement for the element is

$$\{f\} = [K]\{u\}$$

This can be partitioned depending on whether a degree of freedom is released or not.

$$\begin{Bmatrix} f_r \\ f_f \end{Bmatrix} = \begin{bmatrix} K_{rr} & K_{rf} \\ K_{fr} & K_{ff} \end{bmatrix} \begin{Bmatrix} u_r \\ u_f \end{Bmatrix}$$

where the subscripts r and f refer to released and fixed respectively. Knowing the forces that result from the loads on the element and the displacements at the fixed degrees of freedom we can extract the following equation for the displacements at the released degrees of freedom.

$$\{u_r\} = [K]^{-1}(\{f_r\} - [K_{rf}]\{u_f\})$$

9.9.6.3 Forces

Forces are calculated at the end of beam, bar, tie, strut, cable and spring elements, from the displacements at the nodes.

Beam element forces may only be output in Local axis directions.

The most commonly used beam force results: F_x axial force diagram, F_z shear force diagram and M_{yy} bending moment diagram are available from the [Display Favourites toolbar](#).

9.9.6.4 Intermediate forces

Forces, and other force based results, along elements are calculated from the end conditions on the element and any load on the element. The positions at which intermediate forces are reported can be specified either in terms of a number of equidistant points or by selecting intermediate forces at 'interesting points'.

Interesting points are determined from the applied loading, as follows.

For moments: an interesting point is located at every point load on the beam¹ and at the ends of every distributed load. It also places points at a number of equidistant points along distributed loads. In Graphic Views the number of equidistant points along distributed loads is dependent on the number of elements currently being displayed: if more than 50 elements are displayed then the number is 3, if less than 5 elements are being displayed (even if there are many more in the whole model) then the number is 19. In Output Views the number is always 3.

For shear forces: an interesting point is located immediately to either side of every point load on the beam and to either side of each end of every distributed load.

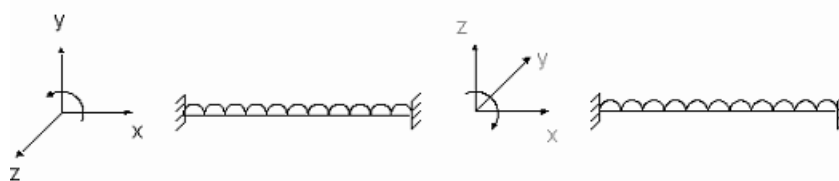
¹ Note that grid loading is not considered when interesting points are being determined. This includes the case where, say, a grid point load happens to be applied directly to a beam.

9.9.6.5 Force sign convention

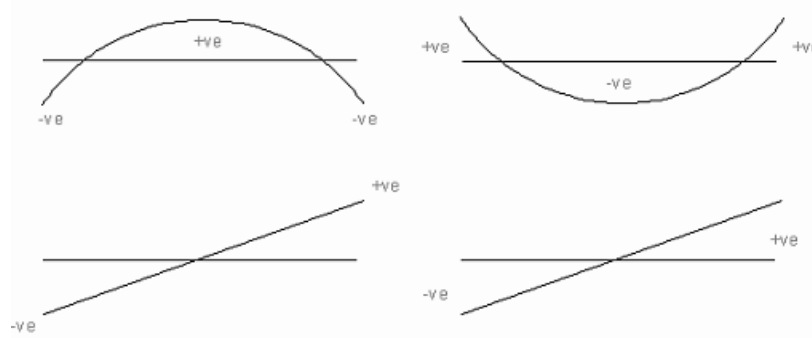
The force in the beam at any point is the force required to maintain equilibrium if the beam is cut at that point and the end 2 part of the beam is discarded. Thus:—

- Axial forces and stresses are positive when tensile.
- For a beam viewed horizontally with its Z-axis pointing vertically **up**, a positive M_{yy} is hogging.
- For a beam viewed horizontally with its Y-axis pointing vertically **down**, a positive M_{zz} is hogging.

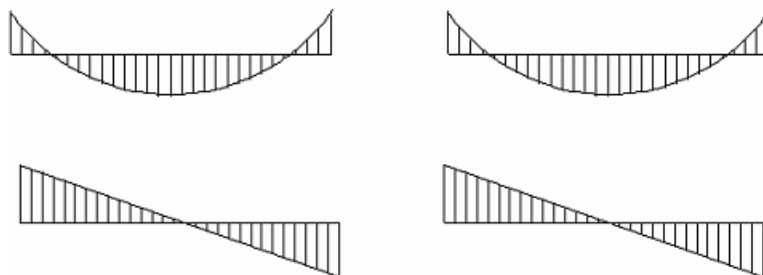
This is demonstrated for the following two beams:—



Drawing a graph of the numerical output of moment and shear force gives:-



Drawing bending moment and shear force diagrams in graphics gives:



Sign Convention Definitions

- Moment: positive/negative depending on direction of moment relative to axis direction (rotational direction).
- Force: positive/negative depending on direction of force relative to axis direction.
- Bending moment diagram: drawn on tension side
- Shear force diagram: positive corresponds to negative slope of bending moment diagram where a bending moment diagram reads from left to right (as viewed) or bottom to top, if vertical.

9.9.6.6 Beam stresses

Beam stresses are calculated from the forces, moments and the shape of the section. The assumption is that the stresses are linear through the thickness, so the stresses are incorrect for elements in which yield has taken place.

Where beam section properties have been modified from the base values the properties used in the stress calculation depend on the "Stress Calc. Basis" setting for the property. The "Stress Calc. Basis" determines whether the modified or unmodified properties are used and can also be set to result in the stresses not being calculated.

The beam stresses are as follows:

- **A**—Axial stress
- **B**—Bending stress
- **C**—Combined stress
- **S**—Shear stress

Only axial stresses are output for sections that have properties explicitly defined.

The axial stress is:

$$A = \frac{F_x}{\text{area}}$$

For symmetric sections the bending stresses are:

$$B_y = \frac{M_{yy}}{I_{yy}} D_z, \quad B_z = -\frac{M_{zz}}{I_{zz}} D_y$$

For non-symmetric sections there is an interaction between the two directions. So a bending stress about one axis will depend on the moments about both axes. In this case the bending stresses are:

$$B_y = \frac{1}{I_{yy}I_{zz} - I_{yz}^2} (I_{zz}M_{yy} + I_{yz}M_{zz}) D_z,$$

$$B_z = \frac{-1}{I_{yy}I_{zz} - I_{yz}^2} (I_{yz}M_{yy} + I_{yy}M_{zz}) D_y$$

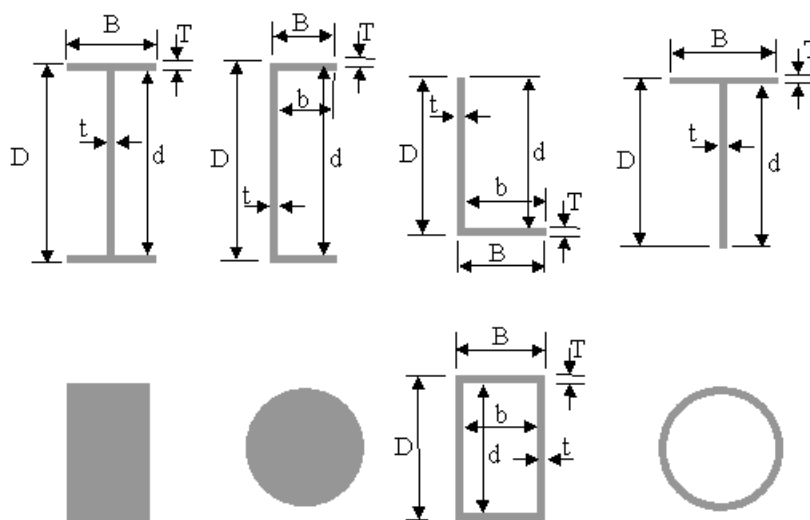
The combined stresses are:

$$C = A + B(y, z)$$

The shear stresses are:

$$S_y = \frac{F_y}{area_y}, \quad S_z = \frac{F_z}{area_z}$$

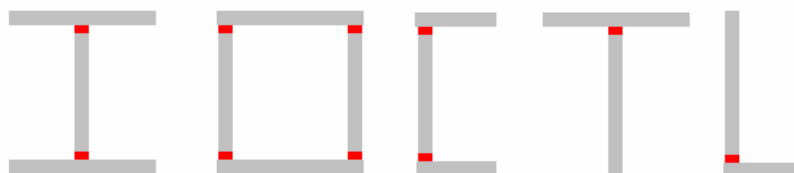
The shear stresses are of most use for steel structures so the shear stresses are calculated based on the shear area ($area_y$ and $area_z$) in BS 5950-1:2000 cl.4.2.3, and they depend on the section shape and if the section is rolled or welded. The section dimensions are shown below:



and $area$ is the area of the section. The shear areas are calculated as follows:

	Concrete		Steel: welded		Steel: rolled, formed, etc.	
	z	y	z	y	z	y
I section	$0.8 \times D \times t$	$1.6 \times B \times T$	$d \times t$	$1.8 \times B \times T$	$D \times t$	$1.8 \times B \times T$
Ch	$0.8 \times D \times t$	$1.6 \times B \times T$	$d \times t$	$1.8 \times B \times T$	$D \times t$	$1.8 \times B \times T$

anel						
Re	$1.6 \times D \times t$	$1.6 \times B \times T$	$2 \times d \times t$	$1.8 \times B \times T$	$\frac{area \times D}{(D + B)}$	$\frac{area \times B}{(D + B)}$
cta						
ng						
ula						
r						
hol						
low						
Re	$0.8 \times area$	$0.8 \times area$	$0.9 \times area$	$0.9 \times area$	$0.9 \times area$	$0.9 \times area$
cta						
ng						
ula						
r						
solid						
An	$0.8 \times D \times t$	$0.8 \times B \times T$	$0.9 \times d \times t$	$0.9 \times B \times T$	$0.9 \times D \times t$	$0.9 \times B \times T$
gle						
Te	$0.8 \times D \times t$	$0.8 \times B \times T$	$d \times t$	$0.9 \times B \times T$	$D \times t$	$0.9 \times B \times T$
e						
Cir -		-	$0.6 \times area$	$0.6 \times area$	$0.6 \times area$	$0.6 \times area$
cul						
ar						
hol						
low						
Cir	$0.7 \times area$	$0.7 \times area$	$0.9 \times area$	$0.9 \times area$	$0.9 \times area$	$0.9 \times area$
cul						
ar						
solid						
d						



Welded sections are assumed to have weld positions as shown above.

Sections not specified as steel or concrete

For sections not specified as steel or concrete the areas are

$$area_y = area \times k_y, \quad area_z = area \times k_z$$

For doubly symmetrical sections the maximum and minimum combined stresses are:

$$C_1 = A + |B_y| + |B_z|, \quad C_2 = A - |B_y| - |B_z|$$

Stress measures in beams

The derived stress values output from GSA for beam elements are:

- maximum torsional stress
- maximum shear stress (y and z)
- von Mises stress

for the following section shapes:

- rectangular
- RHS
- circular
- CHS
- I
- Tee
- channel
- angle
- oval

A new property – C_T (or W_T) the torsion modulus – is loaded or calculated for these sections.

Torsional stress

Torsional stress is calculated ignoring warping of the section by using the torsion modulus C_T (or W_T). We then get the maximum torsional stress from

$$S_T = \frac{M_{xx}}{C_T}$$

The values for C_T are calculated from Bautabellen für Ingenieure, Klaus-Jürgen Schneider, (pp 4.30 & 4.31) and Formulas for Stress and Strain, Roark and Young, McGraw Hill (Table 20)

Shear Stress

The calculation of the shear stresses S_{Ey} and S_{Ez} are calculated as described in the Technical Note: [Calculation of Shear Areas](#) with the exception of ovals which are calculated from the equation:

$$K = \frac{2}{3 + \left(\frac{a}{b}\right)^n}$$

where K is the shear area factor, $n = 1.75$ for $a < b$ and 1.5 for $a > b$, with load being applied in the b direction. The maximum shear stress = shear force / [K(stress) × total area]. The expression gives a that tends to $\frac{2}{3}$ (as for rectangles) for tall, narrow ellipses where $b \gg a$.

von Mises Stress

A precise calculation of the von Mises stress is difficult so a number of simplifications have been made. The von Mises stress depends on the axial forces, the bending moments, the through thickness shear forces and the torsional moment.

The axial force and bending moments combine to give σ_{xx} . The shear stresses are a combination of the through thickness shear + torsion. To combine the shear stresses the through thickness shear stresses are rotated into components parallel and perpendicular to the surface of the section and the torsional shear stress is added to the component parallel to the surface.

$$\begin{Bmatrix} \tilde{\tau}_{xy} \\ \tilde{\tau}_{xz} \end{Bmatrix} = \mathbf{T} \begin{Bmatrix} \tau_{xy} \\ \tau_{xz} \end{Bmatrix} + \begin{Bmatrix} \tau_T \\ 0 \end{Bmatrix}$$

The von Mises stress is then calculated from

$$\sigma_{vM} = \sqrt{\sigma_{xx}^2 + 3\tilde{\tau}_{xy}^2 + 3\tilde{\tau}_{xz}^2}$$

Note: in most cases this is an over estimate of the von Mises stress.

9.9.6.7 Strain energy density

The strain energy density for a beam is a measure of how hard the beam is working. Strain energy density may be output for Beam, Bar, Tie and Strut element types.

The definition of strain energy density is:

$$SED = \int \sigma d\varepsilon$$

For a linear material this is equivalent to:

$$SED = \frac{\sigma^2}{2E}$$

which represents the strain energy density at a point in the material. For beam elements it is more convenient to integrate this over the area (whole or partial) of the section and express the result as strain energy per unit length. So considering the axial stresses only the extensional (e) and bending (b) strain energies per length for the section are:

$$SE_e = \frac{F^2}{2EA}$$

$$SE_b = \frac{M^2}{2EI}$$

Values of strain energy density may be output for the Web, the Flange or the Total value, though Web and Flange values are only output for I section and rectangular hollow sections with unmodified section properties. In the case of rectangular hollow sections “Web” refers to the sides of the section and “Flange” refers to the top and bottom of the section.

The average strain energy density is the average density along the element or member.

9.9.6.8 Cross strain energy density

The strain energy density for a beam is a measure of how hard the beam is working. The definition of strain energy density is:

$$SED = \int \sigma d\varepsilon$$

For a linear material this is equivalent to:

$$SED = \frac{\sigma^2}{2E}$$

A related quantity is the cross strain energy density. The cross terms are calculated from the forces in one analysis case (i) and the displacements in a different analysis case (j). This can be used to provide information about how to optimise a structure by indicating where stiffening will be most effective.

The definition of cross strain energy density is:

$$cSED = \int \sigma_i d\varepsilon_j$$

This quantity is useful only for linear materials where this is equivalent to:

$$cSED = \frac{\sigma_i \sigma_j}{2E}$$

For beam elements it is more convenient to integrate this over the area (whole or partial) of the section and express the result as cross strain energy per unit length. So considering the axial stresses only the extensional (e) and bending (b) cross strain energies per length for the section are:

$$SE_e = \frac{F_i F_j}{2EA}$$

$$SE_b = \frac{M_i M_j}{2EI}$$

Values of cross strain energy density may be calculated for the Web, the Flange or the Total value, though Web and Flange values are only output for I section and rectangular hollow sections with unmodified section properties. In the case of rectangular hollow sections “Web” refers to the sides of the section and “Flange” refers to the top and bottom of the section.

Note that unlike strain energy density which must always be a positive value the cross strain energy density may be negative.

These results differ from most GSA results in that they are not specific to a single analysis case so they are stored as an Element User Module. Use the “Tools | Cross Strain Energy Density Results” command to generate the user modules.

9.9.6.9 Torce lines

Torce lines are identical to thrust lines in plane frames, but in generalising thrust lines for space frames it is necessary to combine the effects of torsion with the effects of axial force (hence the name).

Eccentricities of the torce, t_y and t_z , are given by

$$t_y = -\frac{M_z - M_x \cdot F_z / F_x}{F_x}$$

$$t_z = \frac{M_y - M_x \cdot F_y / F_x}{F_x}$$

For more details refer to the note on [Torce Lines](#) in the [Technical Notes chapter](#).

9.9.7 2D element results

[2D element displacements](#)

[2D element stress results](#)

[2D element stress sign convention](#)

[Limitations of the 2D element stress calculation](#)

[Checking the 2D element stress results](#)

[Derived 2D element stress results](#)

[2D element force results](#)

[2D element force sign convention](#)

[Derived 2D element force results](#)

[2D wall element results](#)

9.9.7.1 2D element displacements

Displacements reported are element displacements; values may differ from nodal displacements when the element has end releases or is offset.

Element rotations are not reported.

9.9.7.2 2D element stress results

The method of calculating the stresses and forces in 2D elements depends to a certain extent on the solution method. Gss calculates stresses and derives the force and moment results from these. GsRelax calculates forces and moments directly.

The stresses are based on the strains, which in turn are based on the displacement gradients in the element. The stresses are most accurate when calculated at the Gaussian integration points and then extrapolated to the nodes.

2D element results (forces and stresses) are most conveniently viewed as [contours](#) or [diagrams](#). Diagram may be more useful in principal stresses with the direction information are required.

More:

[2D element stress sign convention.](#)

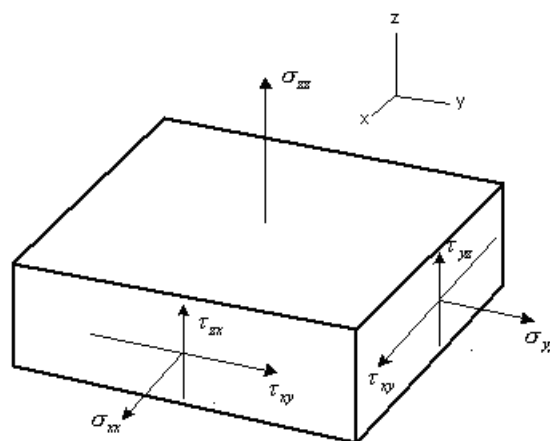
[Derived 2D element stress results](#)

[2D element force sign convention](#)

9.9.7.3 2D element stress sign convention

- Membrane stresses are positive when tensile (i.e. positive direct strain).
- Positive shear stresses result from positive shear strain.

This is illustrated as follows.



2D Element Stress Sign Convention

More:[2D element stress results](#)[Derived 2D element stress results](#)**9.9.7.4 Limitations of the 2D element stress calculation**

The stress calculations show more clearly than the displacement calculation the approximate nature of the finite element method. In particular it should be noted that the stresses are calculated by interpolation involving an extrapolation from the Gauss points to the nodes. This means that for element with high stress gradients the interpolated values will be less accurate than for elements with low stress gradients. It is important to check to see what variation of stress exists over the element and if it is large it may be necessary to refine the mesh locally to overcome this problem.

The stress is calculated on an element by element basis (unlike the displacement) so it is possible, and in fact very likely, that the stresses at a node in one element will not match up with the stress at the same node in an adjacent element. In some cases, where there is a discontinuity in the material properties or the thickness of the elements, it is correct that there is a local stress discontinuity, however in other cases this stress difference will be a measure of the error in the solution.

When contour plots of the stresses are required, or when a stress value is required for further calculation it is better to have a single value. The values from all the elements at a node can be averaged to give a single value at the node. However it should be remembered that the stresses which look smooth and continuous on the plots are in fact calculated as a series of discontinuous stress patches.

9.9.7.5 Checking the 2D element stress results

It is useful to be able to carry out some quick checks on the stresses to get an idea of the accuracy of the analysis. There is no simple answer as to how good an analysis is, but a number of qualitative rules can be established.

- check that the **stress gradients** are low within the elements, particularly in regions of high stress
- check that the **stress discontinuities** between elements are small — look at the stress errors

9.9.7.6 Derived 2D element stress results

The stress results are tensors, so in addition to looking at components of the stress tensor

$$\sigma_{xx}, \sigma_{yy}, \sigma_{zz}, \sigma_{xy}, \sigma_{yz}, \sigma_{zx}$$

it is possible to derive a number of other stress measures. Those available in GSA are:

- **maximum principal**— $\sigma_{max}, \sigma_{min}$

- **max shear**— $\sigma_{max\ shear} = \frac{1}{2}(\sigma_{max} - \sigma_{min})$

$$\sigma_{vm} = \frac{1}{\sqrt{2}} \left[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2) \right]^{1/2}$$

- **von Mises**—

- **average**— $\sigma_{av} = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$

These stress measures are useful in different circumstances.

Principal stress

- flow of stress in metals, concrete
- directions of stress relative to reinforcement

von Mises stress

- measure of distortional stress
- used in yield functions to see how close the material is to yield ($\sigma_{yield} - \sigma_{vm} = 0$)

Average stress

- checking if material is in overall compression/tension

More:

[2D element stress sign convention.](#)

[2D element force sign convention](#)

9.9.7.7 2D element force results

When dealing with elements with no thickness (such as fabrics) or composite materials (such as reinforced concrete) it is more useful to work with stress resultants or forces, than with stresses. For concrete the stress values are based on the properties of an equivalent isotropic material. The checks used for stress results noted above can also be applied to force results. For fabric elements the force resultants are calculated directly but for elements with thickness they are calculated from the in-plane and bending [stresses](#) and the element thickness.

The moment results are a tensor

$$M_x, M_y, M_{xy}$$

and the force results are a tensor (in-plane forces) and a vector (through-thickness shears)

$$N_x, N_y, N_{xy}$$

$$Q_x, Q_y$$

where:

$$\begin{aligned} M_x &= \frac{\sigma_{xx}^b t^2}{6} & M_y &= \frac{\sigma_{yy}^b t^2}{6} & M_{xy} &= \frac{\sigma_{xy}^b t^2}{6} \\ N_x &= \sigma_{xx}^p t & N_y &= \sigma_{yy}^p t & N_{xy} &= \sigma_{xy}^p t \\ Q_x &= \sigma_{xz}^p t & Q_y &= \sigma_{yz}^p t \end{aligned}$$

where the superscripts p and b refer to in-plane and bending stress terms.

The results

$$|M_x| + |M_{xy}|$$

and

$$|M_y| + |M_{yx}|$$

carry the sign of M_x and M_y respectively.

Note that the moment terms follow the Timoshenko convention in which the moment M_x is based on the stress in the x direction. With the Timoshenko convention if a slab is in compression on the top face in both the x and y directions the moments are both negative.

More:

[2D element force sign convention](#)

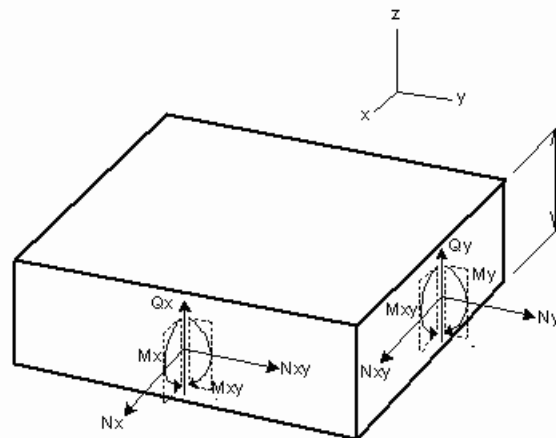
[Derived 2D element force results](#)

[2D element stress sign convention.](#)

9.9.7.8 2D element force sign convention

- Positive in-plane forces result from positive in-plane stresses.
- Positive bending moments result from positive top surface stresses relative to the bottom surface stress. (This corresponds to the Timoshenko (1964) sign convention.)

This is illustrated as follows.



2D Element Force Sign Convention

Note that moments are based on the bending stress (i.e. they are not defined as being about any axis).

More:

[2D element force results](#)

[Derived 2D element force results](#)

9.9.7.9 Derived 2D element force results

In-plane principal forces and principal moments can be calculated in a similar manner to principal stresses:

- **principal forces** $\Rightarrow$ no in-plane shear force
- **principal moments** $\Rightarrow$ no in-plane twisting moment

More:

[2D element stress sign convention.](#)

[2D element force sign convention](#)

9.9.7.10 2D wall element results

Wall element edge forces and moments are available as equivalent beam results. Wall element equivalent beam results may be tabulated in Output Views, and may be contoured and represented as diagrams in Graphic Views. They are reported with beam element results, for example as Beam

and Spring Forces and Moments.

The [1D Element Results](#) dialog allows the wall element equivalent beam or 'wall stick' to be specified as spanning either in primary or secondary direction. The primary direction is wall element x axis direction and the secondary direction is wall element y axis direction. See [2D element axes](#) for the definition of 2D element axes.

If a wall element shape is irregular or the element axis is not align with any edges, the primary direction is taken from the edge with its center located at the most negative side of the element X axis to the opposite edge. Similarly the secondary direction is taken from the edge with its center located at the most negative side of the element Y axis to the opposite edge. The edge forces and moments are relative to the center of the edges and the direction of the forces and moments are relative to the element axis (Note, the direction of the edge forces and moments are not relative to the normal direction of the edges).

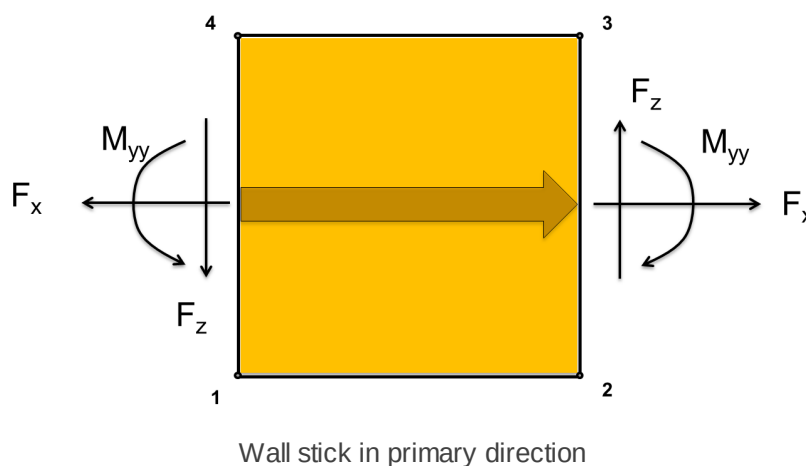
The output axis of the equivalent beam in primary direction is defined as:

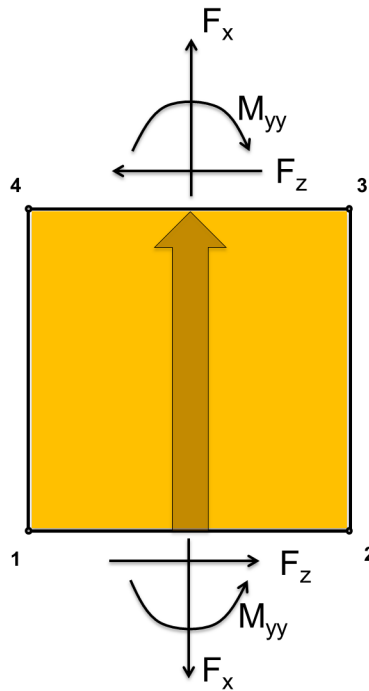
- x' – the same as wall element x axis direction
- y' – in the direction opposite wall element z axis direction
- z' – orthogonal to x' and y' (or the same as wall element local y direction)

The output axis of the equivalent beam in secondary direction is defined as:

- x' – the same as wall element y axis direction
- y' – in the direction opposite wall element z axis direction
- z' – orthogonal to x' and y' (or in the direction opposite wall element x axis direction)

The sign convention for the wall element equivalent beam forces and moments is the same as beam elements running along primary or secondary direction. The diagram below illustrate the forces and moments of 'wall stick' (equivalent beam) in primary and secondary directions. The diagram below is drawn assuming element X axis is from left to right, element Y axis is from bottom to top and element Z axis goes out of the plane, the element axis is not drawn in the diagram below.





Wall stick in secondary direction

9.9.8 Stress averaging in 2D elements

The formulation of 2D elements used in GSA means that there will be stress discontinuities between elements. Large discontinuities indicate that the mesh is not as refined as it should be. However, in the cases where the discontinuities are relatively small it is still convenient for viewing purposes to average the stresses at the nodes to give smooth contours between elements.

Stress averaging rules

In deciding which elements to include in the averaging a number of checks are made. The averaging is carried out on an element by element basis. For any 2D element the number of attached elements is established and the averaging is carried out over these elements provided the following conditions are met:

- The element must be a **2D element**.
- The element must be of the **same material** as the reference element.
- The element must be the **same thickness** as the reference element.
- The element must be the **same order** as the reference element (both linear or both parabolic).
- The element must lie in the **same plane** as the reference element (i.e. no averaging over folds).

The stress results for all the elements meeting the above criteria are then averaged to get the mean stress value.

Notes on stress averaging

The stress averaging can be used to get the “best” approximation to the stress at the node. However, stress averaging can also mask poor stress results.

The stress averaging ignores the complexities of the stress field that results from having other elements attached to the reference element, for example beam framing elements, or 2D stiffening elements.

Stress error

It is useful to be able to quantify the error in the different stress values at nodes. This can be done by contouring the stress error based on the von Mises stress in the elements meeting at a node. The stress error is the standard deviation of the element stresses:

$$e = \sqrt{\frac{\sum_n (\sigma^2) - n(\sigma_{av}^2)}{n}}$$

where n is the number of elements meeting at the node and the summation is over the n elements. The error reported at the centre of the element is the average of the nodal errors.

Part X

10 Programming and Command Line Interface

There are two methods by which GSA can be driven as a non-interactive application. These are by:

- [COM Automation](#)
- [Command File](#)

COM Automation allows commands to be issued from a separate process via a set of VBA or C++ instructions. The separate process could be a separate program or, indeed, a VBA script in a spreadsheet.

In the Command File method a series of instructions are listed in a text file. This is then submitted to GSA as a command line instruction. For example:

```
GSA commands.gwc
```

10.1 COM Automation

COM Automation allows other programs to access to GSA operations by creating an instance of GSA class "ComAuto" and calling the exported functions of this class. A log file will be created to record the execution of each of the functions. The available COM export functions are listed under [COM Export Functions](#).

Notes:

Function names are case sensitive.

Datatypes for function parameters and return values: Some programming languages like VBA are weakly typed so they can infer the type of the variable based on its usage whereas programming environments like Matlab require you to use a parameter of the correct type with the functions.

The COM interface is 'versioned', i.e. each minor and major release of GSA has COM classes specific to that release. (N.b. COM was unversioned in GSA 8.3.1 and earlier.) Instantiating these classes will specifically invoke the version of GSA they correspond to. Whilst this gives the flexibility to bind to a particular release of GSA, it is also possible to "always" bind to the latest release. The choice of whether a programmer wants to bind to a version specific COM class or version independent COM depends on how s/he invokes GSA COM (see the section '[Early and Late Binding](#)').

For early binding code:

```
Dim gsaObj As Gsa_8_4.ComAuto
Set gsaObj = New Gsa_8_4.ComAuto
```

Early binders will always need to refer to a version specific COM class. Note that, in VBA, it is necessary to add a reference to Gsa.tlb, as described in the 'Early and Late Binding' section.

For late binding code

```
Dim gsaObj As Object
Set gsaObj = CreateObject("Gsa_8_4.ComAuto")
```

... will invoke a version specific class, whereas,

```
Dim gsaObj As Object
Set gsaObj = CreateObject("Gsa.ComAuto")
```

- invokes to the current version of GSA. (To make an older version "current", simply run the GsaRegister.exe in the program files folder of the particular version with Administrator privileges.)

More:

[COM Export Functions](#)

[COM VBA Example](#)

[COM C++ Example](#)
[Early and Late Binding](#)

10.1.1 COM Export Functions

The GSA COM export functions are listed below under the following headings:

- [Core functions](#)
- [View functions](#)
- [GwaCommand function](#)
- [Data functions](#)
- [Output functions](#)
- [Case and Task functions](#)
- [List functions](#)
- [Tool functions](#)
- [Enums](#)

The use of many of these functions is demonstrated in sample Excel (XLS) files that are installed in the Samples folder in the GSA program folder. In particular, macros in the sample file "GsaComGwaSample.xls" use most of these GSA COM functions.

See also

[GSA Text \(ASCII\) File](#)

10.1.1.1 Case and Task Functions

Case and Task functions

The following functions are for extracting information on GSA load, analysis and combination cases.

long HighestCase (string caseop)

Returns the highest numbered case.

caseop – the type of case; valid entries are:

"L" – load case

"A" – analysis case

"C" – combination case

short CaseExist (string caseop, long ref)

Returns 1 if the case exists.

caseop – the type of case; valid entries are:

"L" – load case

"A" – analysis case

"C" – combination case

ref – case number

string CaseName (string caseop, long ref)

Returns the name of the case as a string.

caseop – the type of case; valid entries are:

"L" – load case
 "A" – analysis case
 "C" – combination case

ref – case number

long CaseNumPerm (string caseop, long ref)

Returns the number of permutations in the case or 0 if the case is not an enveloping case.

caseop – the type of case; valid entries are:

"A" – analysis case
 "C" – combination case

ref – case number

string CasePermDesc (string caseop, long ref, long perm)

Returns the case description of the case as a string.

caseop – the type of case; valid entries are:

"A" – analysis case
 "C" – combination case

ref – case number

perm – permutation number or 0

string CasePermString (string caseop, long ref, long perm)

Returns the case reference as a string, e.g. "C4p3".

caseop – the type of case; valid entries are:

"L" – load case
 "A" – analysis case
 "C" – combination case

ref – case number

perm – permutation number or 0

float CasePermAnalFactor (string caseop, long ref, long perm, long analref)

Returns the factor of the specified analysis case that contributes to the specified case reference as a string, e.g. "C4p3".

caseop – the type of case; valid entries are:

"A" – analysis case
 "C" – combination case

ref – case number

perm – permutation number or 0

analref – the analysis case number; the returned factor of this analysis case contributes to the case specified by **caseop**, **ref** and **perm**

short CaseResultsExist (string caseop, long ref, long perm)

Returns 1 if results exist for the case.

caseop – the type of case; valid entries are:

"A" – analysis case

"C" – combination case

ref – case number

perm – permutation number or 0

long CaseTask (long ref)

Returns the reference number of the analysis task that is parent to the analysis case.

ref – analysis case number

short TaskStatus (long ref)

Returns the status of the analysis task. Returns a status, as follows:

0 – task exists and has been analysed

1 – no GSA file is open

2 – task does not exist

3 – task exists but has not been analysed

ref – analysis task number

10.1.1.2 Core Functions

Core functions

short NewFile ()

Open a new model. Returns a status, as follows:

0 – OK

1 – failed to open

short Open (string filename)

Open a GWB, GWA or CSV file. Returns a status, as follows:

0 – OK

1 – failed to open

filename – the name of the file to be opened, including path and extension.

short Save ()

Save the data to the default file (i.e. overwriting the file that was opened or last saved). Returns a status, as follows:

0 – OK

1 – no GSA file is open

2 – no default path is available; use SaveAs

3 – failed to save

short SaveAs (string filename)

Save the data to GWB, GWA or CSV file. Returns a status, as follows:

0 – OK

1 – no GSA file is open

2 – invalid file extension

3 – failed to save

filename – the name of the file to be saved, including path and extension.

short Close ()

Close the current file. Returns a status, as follows:

- 0 – OK
- 1 – no GSA file is open

short Analyse (long task)

If no analysis tasks exist and task is zero: do linear static analysis of each specified load case.

If analysis tasks exist: analyse all analysis tasks that have not been analysed.

Returns a status, as follows:

- 0 – OK, analysis attempted
(use CaseResultsExist or TaskStatus to establish whether the analysis succeeded in producing results)
- 1 – no GSA file is open
- 2 – failed to attempt analysis

task – analysis task number (if task is zero, all tasks are considered) (default: zero)

short Delete (string option)

Deletes results. Returns a status, as follows:

- 0 – OK
- 1 – no GSA file is open
- 2 – invalid option
- 3 – that data is not present; – no action taken

option – valid settings are:

RESULTS – delete all results but not analysis cases.

RESULTS_AND_CASES – delete all results and analysis cases.

10.1.1.3 Data Functions**Data functions****short NodeCoord(long ref, double x, double y, double z)**

Retrieves the position of a node in global coordinates. Returns a status, as follows:

- 0 – OK
- 1 – node not found

ref – node reference

x – x coordinate of the node (double*)

y – y coordinate of the node (double*)

z – z coordinate of the node (double*)

long Gen_NodeAt (double x, double y, double z, double tol)

Returns the node number of the newly generated node or the existing node for the given coordinate(x, y, z)

x – x coordinate of the node
y – y coordinate of the node
z – z coordinate of the node
tol – tolerance to use existing node

string Gen_SectionMatchDesc (string sectdesc, long flags)

Returns valid GSA section description from the given tentative section description (sectdesc)

sectdesc – tentative section description

flags – compound flag; valid settings are:

Enum Gen_SectionMatchDesc_Flags

SEC_INCL_SS = &H1 ' include superseded
 sections in search

SEC_ATTEMPT_STD = &H2 ' attempt to assemble a
 standard section from tentative section description (sectdesc)

End Enum

long MembNumElem (long ref)

Returns the number of elements associated with the given member (ref).

ref – member number

long MembElemNum (long ref, long index)

Returns the element number for the given member (ref) and element index.

ref – member number

index – element index for the member (zero based)

long ElemMembNum (long ref)

Returns the member number of associated with the given element (ref).

ref – element number

short Nodes(SAFEARRAY(long)* nodeRefs, SAFEARRAY(GsaNode)* nodes)

Fetches an array of GsaNode objects given an array of node references.

nodeRefs – SAFEARRAY of valid node references

nodes – SAFEARRAY of [GsaNode](#) objects associated with the node references

Returns a status as follows

0 – OK

1 – No file open or invalid input

For example, the following VB.NET snippet retrieves all nodes in the GSA model

```
Dim nodeRefs() As Integer
s = gsaObj.EntitiesInList("all", GsaEntity.NODE, nodeRefs)
Debug.Assert(Not s.Equals(0) And Not (nodeRefs Is Nothing))
Dim nodes() As GsaNode
s = gsaObj.Nodes(nodeRefs, nodes)
```

short SetNodes(SAFEARRAY(GsaNode) nodes, VARIANT_BOOL Overwrite)

Sets an array of [GsaNodes](#) into the GSA model data.

nodes – SAFEARRAY of GsaNode objects.

Overwrite – Boolean flag to indicate if existing nodes, if any, are to be overwritten

short Elements(SAFEARRAY(long)* elemRefs, SAFEARRAY(GsaElement)* elems)

Fetches an array of [GsaElement](#) objects given an array of element references.

elemRefs – SAFEARRAY of valid element references

elems – SAFEARRAY of GsaElement objects associated with the element references

Returns a status as follows

0 – OK

1 – No file open or invalid input

For an example, see documentation for the function `Nodes ( . . . )`.

short SetElements(SAFEARRAY(GsaElement) elems, VARIANT_BOOL Overwrite)

Sets an array of [GsaElement](#) objects into the GSA model data.

nodes – SAFEARRAY of GsaElement objects.

Overwrite – Boolean flag to indicate if existing elements, if any, are to be overwritten

short Sections(SAFEARRAY(long)* sectRefs, SAFEARRAY(GsaSection)* sections)

Fetches an array of [GsaSection](#) objects given an array of section references.

sectRefs – SAFEARRAY of valid section references

sections – SAFEARRAY of GsaSection objects associated with the section references

Returns a status as follows

0 – OK

1 – No file open or invalid input

For an example, see documentation for the function `Nodes ( . . . )`.

short SetSections(SAFEARRAY(GsaSection) sections, VARIANT_BOOL Overwrite)

Sets an array of [GsaSection](#) objects into the GSA model data.

nodes – SAFEARRAY of GsaSection objects.

Overwrite – Boolean flag to indicate if existing elements, if any, are to be overwritten

short NodeConnectedEnt(GsaEntity entityType, long nodeRef, SAFEARRAY(long) entRefs)

Fetches the indices of elements or members connected to a node.

entityType – enum of type [GsaEntity](#). Has to be ELEMENT or MEMBER.

nodeRef – reference of the node to query for

entRefs – array holding the references to connected entities. (Output)

Returns a status as follows

- 0 – OK
- 1 – No file open or invalid input
- 2 – Node does not exist

short Gen_RegionMeshCheck (long ref, long* error, long* warning, string* message)

Check the validity of the given region (ref). Returns a status, as follows:

- 0 – no errors or warnings
- 1 – some errors or warnings

ref – region reference number (if ref is zero, all regions are checked)

error – the number of errors generated in running this function (long*) (Output)

warning – the number of warnings generated in running this function (long*) (Output)

message – the message explaining the status of running this function (string*) (Output)

short Gen_RegionMeshGen (long ref, long* error, long* warning, string* message)

Generate meshes for the given region (ref). Returns a status, as follows:

- 0 – at least one region meshed successfully
- 1 – no region meshed

ref – region reference number (if ref is zero, meshes are generated for all regions)

error – the number of errors in running this function (long*) (Output)

warning – the number of warnings in running this function (long*) (Output)

message – the message explaining the status of running this function (string*) (Output)

short Gen_RegionMeshDel (long ref)

Delete the mesh for the given region (ref). Returns a status, as follows:

- 0 – mesh for at least one region deleted successfully
- 1 – no region mesh deleted

ref – region reference number (if ref is zero, all regions are deleted)

short Tool_UpdateElemSections ()

Returns true if successful

Call this function to update element section properties according to corresponding member's section properties.

short Tool_ResetMemberSections ()

Returns true if successful

Call this function to reset member section properties according to associated elements' section properties.

short Renumber_Init (GsaEntity option)

Initialises the list of entities that are to be renumbered for a specific entity option. Call this before calling Renumber_SetTo for the option.

option – enum [GsaEntity](#)

Returns a status, as follows:

- 0 – OK
- 1 – no GSA file is open

short Renumber_SetTo(GsaEntity option, long oldRef, long newRef)

Specifies an entity that is to be renumbered and its new number. Renumber_SetTo can be called several times before calling Renumber_Go, to build up a list of entities to be renumbered in one operation. Call Renumber_Init for the entity option before calling this function.

option – enum [GsaEntity](#)

oldRef – the reference to the existing entity

newRef – the new number of the entity

Returns a status, as follows:

- 0 – OK
- 1 – no GSA file is open
- 2 – Renumber_Init not called for this option

short Renumber_Go (GsaEntity option)

Does the renumber operation. Call Renumber_SetTo to specify what is to be renumbered, before calling this.

option – enum [GsaEntity](#)

Returns a status, as follows:

- 0 – OK
- 1 – no GSA file is open
- 2 – Renumber_Init not called for this option

sid functions

Sid for each module record can be set and read though the GwaCommand using their [format](#) in GWA files. But this requires a lot of string parsing and hence the following functions are provided to set and get sids directly for each valid module record. For more information on sids, refer to [String IDs](#)

short WriteSidTagValue(string sKey, long iRecord, string sTag, string sValue)

Writes a Tag-Value pair to the sid of the module record specified.

Returns a status, as follows:

- 0 – OK, successfully written
- 1 – otherwise

sKey – the keyword of the module

iRecord – the index of the record

sTag – the tag to be written to the sid

sValue – the value associated with the above tag

This function writes a Tag-Value pair if none exists inside the sid of the record. If there's already a tag present, the associated value is overwritten with the new value. Eg.(VB .NET)

```
Dim iSuccess as Integer = gsaObj.WriteSidTagValue("SEC_BEAM", 12, "RVT", "Structural I
```

string GetSidTagValue(string sKey, long iRecord, string sTag)

Returns the value for a tag stored inside the sid of the module record specified

sKey – the keyword of the module

iRecord – the index of the record

sTag – the tag for which value is to be retrieved from the sid

An example of usage(VB .NET)

```
Dim sValue as String = gsaObj.GetSidTagValue("MEMBER", 3, "RVT")
```

Notes:

- The `WriteSidTagValue` function sets the sid in the standard format specified in the documentation. Concordantly, the `GetSidTagValue` retrieves a value from sid only if it is in the same format.
- Keyword for model sid is "SID".
- An authoring application using the sid feature must write a model sid. Otherwise sids written to module records will not persist.
- It's the authoring application's responsibility to ensure that the record exists for the module on which it is being called – otherwise GSA might throw an exception

10.1.1.4 GwaCommand Function

GwaCommand function

GWA commands are commands that interact directly with the data in a GSA session. This allows the user to modify any of the data in a GSA file. GSA data is stored in modules that are one of the following types:

specification – single record module (e.g. General Specification)

ordered table modules – multi-record but can include gaps in the records (e.g. Nodes)

collection table modules – multi-record without gaps in the records (e.g. Beam Loads)

Generally collection table modules are those that are not cross referenced with other modules, so, for example, switching a couple of load records does not affect the data integrity while switching a couple of node records does.

The syntax of the command is based on GWA syntax and the units follow the GWA unit syntax; – refer to the “[GSA Text \(ASCII\) File](#)” chapter for details.

Note that GwaCommand always operates in High Precision, ignoring the 'GWA Numeric Precision' preference setting.

variant GwaCommand (string command)

Issue a GWA command to GSA.

command – a command in GWA format with fields separated by a tab or the list separator (typically a comma or semi-colon). The commands that can be used are:

key_word, {data} – this is used to insert a record of data (e.g. writing a complete set of data to a GSA model).

SET, key_word, {data} – this is similar to the previous command and inserts a record of data (for specification and ordered table modules). For a node this will overwrite the node (if it is already defined) as the node number is part of the data – but for a load it will simply append to the module as the load records do not have a record number).

SET_AT, record, key_word, {data} – set a particular data record (for collection table modules) this allows, for example, load records to be

modified and overwritten as the record number is specified.

ADD, key_word, {data} – add a data record at the end of the module (for collection table modules).

GET, key_word, {data} – return a record of data.

GET_ALL, key_word – return all records of data.

HIGHEST, key_word – return the number of the highest record number.

BLANK, key_word, low_record [, high_record] – blank a record or a range of records (without reordering subsequent records).

DELETE, key_word, low_record [, high_record] – delete a record or a range of records (reordering subsequent records to close the gap).

EXIST, key_word, record – return true if the record exists.

LOCKED, key_word, record – return true if the record is locked.

– where items in [] are optional and items in { } represent a list of values.

Examples of **command**:

(NB. these are only illustrations, for the exact syntax of the GWA string, refer to [GSA Text \(ASCII\) File](#))

To create a node 25 at (10,4.5,8)

```
SET, NODE, 25, , NO_RGB, 10, 4.5, 8
```

or

```
UNIT_DATA, LENGTH , cm, 100
```

```
NODE, 25, , NO_RGB, 1000, 450, 800
```

or

```
UNIT_DATA, LENGTH , cm, 100
```

```
NODE.1, 25, 1000, 450, 800
```

To set beam load 1 to be a UDL of –1000 in z on elements 1 to 10 for case10 in global axes

```
SET, 1, LOAD_BEAM, 1 to 10, 10, GLOBAL, NO, Z, -1000
```

or to append this beam load

```
ADD, , LOAD_BEAM, 1 to 10, 10, GLOBAL, NO, Z, -1000
```

To check if element 11 exists

```
EXIST, EL, 11
```

To create a beam element 11 with property 1, group 101 and nodes 12 and 22 and orientation node 3

```
SET, EL, 11, , NO_RGB, BEAM, 1, 101, 12, 22, 3
```

To return the data for material 3

```
GET, MAT, 3
```

To get the displacements for node 25 in case 4

```
GET, DISP, 25, 4
```

To check the highest node

```
HIGHEST, NODE
```

To see if an axis 4 is locked

```
LOCKED, AXIS, 4
```

To blank element 40

```

        BLANK, EL, 40
    or to delete the element
        DELETE, EL, 40

```

Notes on using GwaCommand:

- The GwaCommand fully exposes the GSA data structure, so care should be taken to ensure that the data is not corrupted.
- Regarding GWA keyword versioning, both the SET and GET commands recognise the keyword version. If the keyword version is not specified then the SET and GET commands assume the current version. If the keyword version is specified in a GET command then the keyword in the returned string includes the keyword version.
- Before GSA 8.4: when creating **elements** the entity type was specified in the keyword; so to SET a beam element the keyword was EL_BEAM but to GET an element, or to otherwise inquire about an element, the keyword was simply EL. Now the keyword for setting, getting or inquiring about an element is EL; however, EL.1 (EL version 1) can be used to GET the element data in EL_BEAM (etc.) format, and EL_BEAM (etc.) can still be used to SET elements.
- GSA uses the current locale settings for parsing GwaCommand strings. To override this behaviour see SetLocale function.
- To check the highest **user module** or to delete a user module refer to the respective user module title record. E.g.

```

        HIGHEST, USER_MOD_NODE_TITLE
        DELETE, USER_MOD_ELEM_TITLE, 3

```

String IDs (sIDs) and GwaCommand:

- The model sID is accessed by using the SID key_word.
- Object sIDs are appended to the object key_word.

```

        SET, SID, {tag1:value1}{tag2:value2}, 11, 1, 101, 12, 22, 3

```

Refer to [Program Fundamentals: — String IDs](#) for information on sID formatting.

10.1.1.5 List Functions

List functions

long IsItemIncluded (string option, long ref, string list)

Returns 1 if the item reference is included in the list

option – valid settings are:

ITEM – a general list.

NODE – list is a list of nodes.

ELEM – list is a list of elements.

MEMBER – list is a list of members.

CASE_ANAL – list is a list of cases and ref is an analysis case reference.

CASE_COMB – list is a list of cases and ref is a combination case reference.

GRID_PT – list is a list of grid points.

ref – item to be checked for inclusion

list – the list description (e.g. 1 3 5 or 1 to 10 not 7) - note this is different from the list name.

short EntitiesInList (string sList, enum GsaEntity* listType, SAFEARRAY (long)* arrayEntities)

Retrieves entities from a list description or a saved list.

sList – list description or saved list reference (passed as a string)

listType – input or output parameter depending on whether sList is a list description or a saved list.

if sList is a simple description (e.g. "1 to 10"), listType is an input parameter specifying the type of list

if sList is a saved list reference (e.g. "1"), listType is an output parameter returning the type of the saved list

arrayEntities – the array of all valid entities in the list. (Output)

Returns a status as follows:

0 – OK

1 – no GSA file open

2 – saved list does not exist

3 – no items in the list

10.1.1.6 Output Functions

Output functions

The following functions are for extracting processed data, for example, derived enveloped stresses.

short Output_Init (long flags, string axis, string case, long dataref, long num1dpos)

Initialises the output functions. **Call this before calling any other "Output_" functions. Returns a status, as follows:**

0 – OK

1 – no GSA file is open

3 – invalid axis

4 – invalid case

5 – invalid dataref

flags – compound flag; valid settings are:

Enum Output_Init_Flags

OP_INIT_2D_BOTTOM = &H1 ' output
2D stresses at bottom layer

OP_INIT_2D_MIDDLE = &H2 ' output
2D stresses at middle layer

OP_INIT_2D_TOP = &H4 ' output
2D stresses at top layer

```

OP_INIT_2D_BENDING = &H8          ' output
2D stresses at bending layer

OP_INIT_2D_AVGE = &H10             ' average
2D element stresses at nodes

OP_INIT_1D_AUTO_PTS = &H20         '
calculate 1D results at interesting points

OP_INIT_INFINITY = &H40            ' return
infinity and NaN values as such, else as zero

OP_INIT_1D_WALL_RES_SECONDARY = &H80 '
output secondary stick of wall equivalent beam results, else
primary

```

End Enum

E.g. OP_INIT_2D_TOP Or OP_INIT_2D_AVGE ' 2D stresses at top layer, averaged at nodes

axis – output axis; enter the name of a standard axis or the number of a user defined axis; examples of valid entries:

```

"default" – the default for the data being extracted
"global"
"local"

```

case – the output case, ignored if not relevant; CasePermString may be used to collate this string; examples of valid entries:

```

"L1"
"A3"
"C3"
"C4max" – assumes C4 is an envelope
"C4min" – <ditto>
"C4abs" – <ditto>
"C4signabs" – <ditto>
"C4p3" – <ditto>

```

dataref – an integer data reference; refer to file "Output_DataRef.txt" in the Docs folder for available options; for example:

14003001 refers to beam element axial stress

num1dpos – the number of equidistant internal positions along 1D elements to be considered for 1D element results, in addition to the automatic interesting positions if specified in flags

short Output_SetStage (long stage)

Returns a status, as follows:

```

0 – OK
1 – no GSA file is open
2 – Output_Init not called

```

stage – the analysis stage to be considered; only relevant for input data (e.g. properties) since results are for the stage that was analysed; enter 0 for "whole model"

string Output_DataTitle (long flags)

Returns the title of the data specified in the last call to Output_Init, as a string.

flags – compound flag; valid settings are:

1 – full title (otherwise the abbreviated title is returned)

short Output_IsDataRef (long flags)

Returns 1 if the flags condition applies to the data specified in the last call to Output_Init.

flags – compound flag; valid settings are:

Enum Output_IsDataRef_Flags

OP_IS_AND = &H1

otherwise OR

OP_IS_PER_REC = &H2

OP_IS_PER_NODE = &H4

OP_IS_PER_ELEM = &H8

OP_IS_PER_MEMB = &H10

OP_IS_PER_1D_DISP = &H20

OP_IS_PER_1D_FRC = &H40

OP_IS_PER_TOPO = &H80

OP_IS_AT_CENTRE = &H100

End Enum

E.g. OP_IS_PER_1D_DISP Or OP_IS_PER_1D_FRC ' data is reported at 1D element internal displacement OR internal force positions.

E.g. OP_IS_AND Or OP_IS_PER_ELEM Or OP_IS_PER_TOPO ' data is reported per element AND per node per element.

string Output_UnitString ()

Returns the units of the data specified in the last call to Output_Init, as a string.

float Output_UnitFactor ()

Returns the factor to convert the data specified in the last call to Output_Init from SI to the current model units.

short Output_DataExist (long ref)

Returns 1 if the data specified in the last call to Output_Init exists for the specified item.

ref – the record or node or element or member number to be considered

long Output_NumElemPos (long ref)

Returns the number of positions on the element or member for which the data specified in the last call to Output_Init are available. For 1D elements this will be the end positions plus the internal positions, based on the arguments supplied in the last call to Output_Init. For 2D elements this will be the number of nodal positions on the element plus, for some data options, the centre value. ***This must be called before Output_1DElemPos and Output_Extract for 1D element results.***

ref – the element or member number to be considered

float Output_1DElemPos (long pos)

Returns the position along a 1D element as a proportion of the element length for specified position number. Call Output_NumElemPos before calling this and after calling Output_Init.

pos – the position number to be considered; zero based (i.e. “0” is the first position and “Output_NumElemPos – 1” is the last)

variant Output_Extract (long ref, long pos)

Returns the requested data for the specified axis and case in the current model units. Output_NumElemPos should be called before calling this for 1D element data, and after calling Output_Init.

ref – the record or node or element or member number to be considered

pos – the position number to be considered; zero based (i.e. “0” is the first position and “Output_NumElemPos – 1” is the last)

Note:

- NaN and Infinity are returned as zero unless the OP_INIT_INFINITY flag is set in the Output_Init call.
- When NaN or Infinity is returned, an HRESULT of S_FALSE or E_ATL_VALUE_TOO_LARGE is raised, respectively. (This might result in an error being thrown in VBA.) Additionally a warning is also written to the log file created.
- While extracting nodal forces on elements (REF_FORCE_NODAL), ref is the node reference and pos is the element reference. The element reference can be determined from the [NodeConnectedEnt](#) function.

long Output_Extract_CurPerm ()

Returns the envelope permutation that gave the data returned by the last Output_Extract call. This is only relevant if the current case, set in Output_Init, is a “Cnmax”, “Cnmin”, “Cnabs” or “Cnsignabs” case.

The following functions return ALL result values for a node or an element from a single function call.

short Output_Init_Arr (long iFlags, string sAxis, string sCase, enum ResHeader header, long num1dpos)

Initializes the Output Array API for a specified case, axis, header and flags.

flags – compound flag; valid settings are:

Enum Output_Init_Flags

OP_INIT_2D_BOTTOM = &H1	' output
2D stresses at bottom layer	
OP_INIT_2D_MIDDLE = &H2	' output
2D stresses at middle layer	
OP_INIT_2D_TOP = &H4	' output
2D stresses at top layer	
OP_INIT_2D_BENDING = &H8	' output
2D stresses at bending layer	
OP_INIT_2D_AVGE = &H10	' average
2D element stresses at nodes	
OP_INIT_1D_AUTO_PTS = &H20	'
calculate 1D results at interesting points	
OP_INIT_INFINITY = &H40	' return
infinity and NaN values as such, else as zero	
OP_INIT_1D_WALL_RES_SECONDARY = &H80	'
output secondary stick of wall equivalent beam results, else primary	

End Enum

E.g. OP_INIT_2D_TOP Or OP_INIT_2D_AVGE ' 2D stresses at top layer, averaged at nodes

axis – output axis; enter the name of a standard axis or the number of a user defined axis; examples of valid entries:

"default" – the default for the data being extracted

"global"

"local"

case – the output case, ignored if not relevant; CasePermString may be used to collate this string; examples of valid entries:

"L1"

"A3"

"C3"

"C4max" – assumes C4 is an envelope

"C4min" – <ditto>

"C4abs" – <ditto>

"C4signabs" – <ditto>

"C4p3" – <ditto>

header -- enum of the type, defined by [ResHeader](#)

num1dpos – the number of equidistant internal positions along 1D elements to be considered for 1D element results, in addition to the automatic interesting positions if specified in flags

Returns the following status

0 – OK

1 – no GSA file is open

3 – invalid axis

4 – invalid case

5 – invalid dataref

**short Output_Extract_Arr(long iRef, SAFEARRAY(struct GsaResults)
arrayResults, long numComponents)**

Returns the output data for a node, element or member, as an array.

iRef – the entity reference to return results for

arrayResults – array of struct [GsaResults](#) (Output)

numComponents – number of result components for the header in question (Output)

The function Output_Extract_Arr returns an array of results for the element or node in question. For an element, the arrayResults array consists of an array of GsaResults objects. The number of objects returned is equal to one of the following:

- number of intermediate points on 1D elements
- topology points on 2D elements
- 1 if the entity in question is a node

Each GsaResults object has 2 members

- a dynamically allocated array called `dynaResults`. This contains double values corresponding to results for each component for the given header (as specified in the `Output_Init_Arr` function).
- an integer member `NumComponents` that contains the number of components in `dynaResults`.

For e.g. the call

```
gsaObj.Output_Set_Init(Output_Init_Flags.OP_INIT_1D_AUTO_PTS, "default",
"A1", ResHeader.REF_DISP, 2)
```

initializes the results API with `REF_DISP`, the nodal displacements. From the file `Output_DataRef.txt`, we note that `REF_DISP` has the following components:

```
REF_DISP_DX, REF_DISP_DY, REF_DISP_DZ, REF_DISP_TRANS, REF_DISP_RXX,
REF_DISP_RYY, REF_DISP_RZZ, REF_DISP_ROT, REF_DISP_DXY
```

Then, the call

```
gsaObj.Output_Extract_Arr(iNode, arrRes, numComp)
```

populates `arrRes` with a single struct that contains `dynaResults` of length 9 and value of `NumComponents` is set to 9.

The order of components in `dynaResults` is the same as their order in the `DataRef` enums.

short `Output_Extract_CutAssembly(long assembRef, VARIANT_BOOL Avg2DStress, BSTR sCase, BSTR sAxis, SAFEARRAY(struct GsaResults)* arrayResults)`

Fetches 'Cut Section Forces' results for a given assembly. See documentation for the [Cut Section Forces](#) for more explanation of the parameters.

assembRef – the assembly to extract forces for.

Avg2DStress – use averaged 2D stresses

start – first offset (set to 0 if no offsets are to be defined)

end – last offset (set to 0 if no offsets are to be defined)

sCase – case description

sAxis – axis definition

arrayResults – SAFEARRAY of `GsaResults` object where

GsaResults.dynaResults stores the x, y, z xx, yy, zz results in that order,

GsaResults.Pos stores the position of the cut.

For example:

```
Dim results() As GsaResults
gsa.Output_Extract_CutAssembly(1, False, 0, 0, "A1", "1", results)
```

10.1.1.7 Tool Functions

Tool Functions

short `SetLocale (Locale locale)`

Set the locale for Decimal and List Separators for use by the `GwaCommand` function.

option – enum `Locale`. Valid options are:

`LOC_SYSTEM` – Use the system default setting.

LOC_EN_GB – Use English.(' and '.)

Returns a status, as follows:

0 – OK

1 – invalid Locale

The Locale setting is culture and language dependent and varies from one machine to the other. GwaCommand strings use decimal and list separators for input and output. Call this function to set GwaCommand to operator in a locale independent way.

long NumArg(string line)

Returns the number of arguments in a line

line – a string of comma separated arguments

For example NODE, 25, 10, 4.5, 8 would return 5

string Arg(long index, string line)

Returns as a string the argument at the index position in the line

index – index in line (zero based)

line – the line to be decoded

For example the third argument from NODE, 25, 10, 4.5, 8 would return 4.5

string VersionString()

Returns the version of the current instance of GSA.

10.1.1.8 View Functions

View functions

UpdateViews ()

Refreshes all GSA views currently displayed.

short PrintView (string option)

Print saved or preferred Graphic Views or Output Views. Returns a status, as follows:

0 – OK

1 – no GSA file is open

2 – invalid argument

option – valid settings are:

ALL_PGV – print all preferred Graphic Views.

ALL_SGV – print all saved Graphic Views.

ALL_POV – print all preferred Output Views.

ALL_SOV – print all saved Output Views.

ALL_LST – print all View Lists.

TAGGED_PGV – print tagged preferred Graphic Views.

TAGGED_SGV – print tagged saved Graphic Views.

TAGGED_POV – print tagged preferred Output Views.

TAGGED_SOV – print tagged saved Output Views.

TAGGED_LST – print tagged View Lists.

<name> – print the first view or view list found with the specified name.

short SaveViewToFile (string option, string filetype)

Save saved or preferred Graphic Views or Output Views to file. Returns a status, as follows:

- 0 – OK
- 1 – no GSA file is open
- 2 – invalid argument

option – valid settings are:

- ALL_PGV – save all preferred Graphic Views.
- ALL_SGV – save all saved Graphic Views.
- ALL_POV – save all preferred Output Views.
- ALL_SOV – save all saved Output Views.
- ALL_LST – save all View Lists.
- TAGGED_PGV – save tagged preferred Graphic Views.
- TAGGED_SGV – save tagged saved Graphic Views.
- TAGGED_POV – save tagged preferred Output Views.
- TAGGED_SOV – save tagged saved Output Views.
- TAGGED_LST – save tagged View Lists.
- <name> – save the first view or view list found with the specified name.

filetype – valid settings are:

- WMF – save Graphic Views to WMF file.
- PNG – save Graphic Views to PNG file.
- JPG – save Graphic Views to Jpeg file.
- TXT – save Output View to tab delimited TXT file.
- CSV – save Output View to comma delimited CSV file.
- HTM – save Output View to HTML file.

integer HighestView (string option)

Return the highest numbered saved or preferred Graphic View or Output View.

option – valid settings are:

- PGV – highest numbered preferred Graphic View.
- SGV – highest numbered saved Graphic View.
- POV – highest numbered preferred Output View.
- SOV – highest numbered saved Output View.
- LST – highest numbered saved View List.

short ViewExist (string option, long ref)

Returns 1 if the saved or preferred Graphic View or Output View exists.

option – valid settings are:

- PGV – preferred Graphic View.
- SGV – saved Graphic View.
- POV – preferred Output View.
- SOV – saved Output View.

LST – saved View List.

ref – view number (1 based)

string ViewName (string option, long ref)

Returns the name of the saved or preferred Graphic View or Output View.

option – valid settings are:

PGV – preferred Graphic View.

SGV – saved Graphic View.

POV – preferred Output View.

SOV – saved Output View.

LST – saved View List.

ref – view number (1 based)

short CopyView (string sViewType, long idxView, long* idxNewView)

Creates a copy of an existing view and returns the index of the new view.

sViewType – valid settings are:

SGV – saved Graphic View

SOV – saved Output View

idxView – view reference

Returns a status, as follows:

0 – Success

1 – file not open

2 – view does not exist

short RenameView (string sViewType, long idxView, string sNewName)

Renames an existing view.

sViewType – valid settings are:

SGV – saved Graphic view

SOV – saved Output View

idxView – view reference

sNewName – new name

Returns a status, as follows:

0 – Success

1 – file not open

2 – view does not exist

short DeleteView (string sViewType, long idxView)

Deletes a view.

sViewType – valid options are:

SGV – saved Graphic view

SOV – saved Output view

idxView – view reference

Returns a status, as follows:

- 0 – Success
- 1 – file not open
- 2 – view does not exist

short GetViewCaseList (string sViewType, long idxView, string* sList)

Retrieves the case list associated with a view.

sViewType – valid options are:

- SGV – saved Graphic view
- SOV – saved Output view

idxView – view reference

sList – case list (Output)

Returns a status, as follows:

- 0 – Success
- 1 – file not open
- 2 – view does not exist

short SetViewCaseList (string sViewType, long idxView, string sList)

Sets the case list in a saved view.

sViewType – valid options are:

- SGV – saved Graphic view
- SOV – saved Output view

idxView – view reference

sList – case list

Returns a status, as follows:

- 0 – Success
- 1 – file not open
- 2 – view does not exist

short GetViewDisplayList (string sViewType, long idxView, GsaEntity* listType, string sList)

Retrieves the entity display list associated with a view.

sViewType – valid options are:

- SGV – saved Graphic view
- SOV – saved Output view

idxView – view reference

listType – the entity type of the list, specified as an enum. This parameter can either be an input or output parameter depending on the type of view being queried. If **sViewType** is "SGV" (graphic view), then **listType** is an output parameter returning the currently active list type. If **sViewType** is "SOV" (output view), then **listType** is an input parameter specifying the entity list to be returned.

sList – entity list (Output)

Returns a status, as follows:

- 0 – Success
- 1 – file not open

2 – view does not exist

short SetViewDisplayList (string sViewType, long idxView, GsaEntity listType, string* sList)

Sets the entity list associated with a view.

sViewType – valid options are:

SGV – saved Graphic view

SOV – saved Output view

idxView – view reference

listType – the entity type of the list, specified as an enum.

sList – entity list

Returns a status, as follows:

0 – Success

1 – file not open

2 – view does not exist

short RescaleViewData (long idxView)

Rescales a graphic view.

idxView – view reference (graphic view only)

Returns a status, as follows:

0 – Success

1 – file not open

2 – view does not exist

10.1.1.9 Enums and Structs

Enum GsaEntity

NODE
ELEMENT
MEMBER
LINE
AREA
REGION

Enum ResHeader

REF_DISP
REF_VEL
REF_ACC
REF_REAC
REF_FORCE_CONSTR
REF_FORCE_NODAL
REF_MASS_NODAL
REF_SOIL_NODAL
REF_DISP_EL0D
REF_FORCE_EL0D
REF_DISP_EL1D
REF_FORCE_EL1D
REF_STRESS_EL1D
REF_STRESS_EL1D_DRV

```
REF_STRAIN_EL1D
REF_SED_EL1D
REF_SED_AVG_EL1D
REF_STL_UTIL
REF_DISP_EL2D
REF_FORCE_EL2D_DRV
REF_MOMENT_EL2D_PRJ
REF_FORCE_EL2D_PRJ
REF_STRESS_EL2D_DRV
REF_STRESS_EL2D_AX
REF_STRESS_EL2D_PRJ
REF_RC_SLAB_REINF
```

Struct GsaResults

```
int NumComponents
SAFEARRAY(double) dynaResults
```

Struct GsaNode

```
int Ref
BSTR Name
int Color
SAFEARRAY(double) Coor
int Restraint
SAFEARRAY(double) Stiffness
```

Struct GsaElement

```
int Ref
BSTR Name
int Color
int eType
int Property
int Group
int NumTopo
SAFEARRAY(int) Topo
double Beta
int OrientNode
```

Struct GsaSection

```
int Ref
BSTR Name
int Color
BSTR SectDesc
int Material
```

10.1.2 COM VBA Example

The following is an example VBA script to run GSA remotely.

```
Sub RunGsa2()
    Dim iName0, iName1, iName2 As String
    Dim GsaAuto As Object
    Set GsaAuto = CreateObject("Gsa.ComAuto")
    iName0 = "C:\Temp\gsa_com0.gwa"
    iName1 = "C:\Temp\gsa_com1.gwa"
    iName2 = "C:\Temp\gsa_com2.gwb"
    GsaAuto.Open (iName0)
    GsaAuto.Analyse
    GsaAuto.Save
    GsaAuto.PrintView ("TAGGED_SGV")
    GsaAuto.Delete ("RESULTS")
    GsaAuto.SaveAs (iName1)
    GsaAuto.Delete ("RESULTS_AND_CASES")
    GsaAuto.SaveAs (iName2)
    GsaAuto.Close
    Set GsaAuto = Nothing
End Sub
```

10.1.3 COM C++ Example

The following is an example C++ code to run GSA remotely.

```
#include <string>
#import "C:\Program Files\Oasys\GSA 8.4\GSA.tlb" no_namespace

void CGsaComClientApp::invokeGsa(CString filename, CString
analysed_filename)
{
    if(FAILED(CoInitializeEx(0, COINIT_APARTMENTTHREADED)))
        return;

    IComAutoPtr pObj(__uuidof(ComAuto));
    short ret_code = 0;

    _bstr_t bsFileName = (LPCTSTR)filename;
    ret_code = pObj->Open(bsFileName);
    if(ret_code ==1)
        return;

    _bstr_t bsContent(_T("RESULTS"));
    ret_code = pObj->Delete(bsContent);
    ASSERT(ret_code != 1); // file not open!

    // try getting node data out
    _variant_t result;
    result = pObj->GwaCommand("HIGHEST, NODE");
```

```

ASSERT(result.vt == VT_INT || result.vt == VT_I4 );
int nNode = result.intVal;
TRACE("Number of nodes: %d\n", nNode);

for(int iNode = 1; iNode <=nNode; iNode++)
{
    CString command;
    command.Format("GET, NODE, %i", iNode);
    result = pObj->GwaCommand(command.AllocSysString());
    ASSERT(result.vt == VT_BSTR);
    CString nodeString = (CString)result.bstrVal;
    TRACE("%s\n", nodeString);
}

_variant_t vCase(0L);
ret_code = pObj->Analyse(vCase);
ASSERT(ret_code ==0);

_bstr_t bsAnalysedFileName = (LPCTSTR)analysed_filename;
ret_code = pObj->SaveAs(bsAnalysedFileName);
ASSERT(ret_code ==0);

pObj->Close();
}

```

10.1.4 Early and Late Binding

In simplest terms, early and late binding refer to when COM programmers choose to let the compiler* know that the object created is of type Gsa.ComAuto.

In early binding, the object is declared to be of type Gsa.ComAuto and then invoked**.

```

Dim gsaObj As Gsa_8_4.ComAuto
Set gsaObj = New Gsa_8_4.ComAuto

```

In late binding, the object is simply declared to be of the type 'Object' and it's only when the code is run does the compiler know that it belongs to the type Gsa.ComAuto.

```

Dim gsaObj As Object
Set gsaObj = CreateObject("Gsa.ComAuto")

```

When a particular approach is to be adopted over the other depends on particular needs(see reference 2) but broadly late binding allows for the code to be set up and prototyped quickly and is recommended if the programmer doesn't want to be bothered with the version of GSA he is using. Early binding on the other hand lets the code run faster and gives a better handle on the version of class being invoked.

References:

1. <http://visualbasic.about.com/od/usingvbnet/a/earlybind.htm> [about.com]
2. <http://support.microsoft.com/kb/245115> [microsoft.com]

*Compiler/interpreter. In case of Excel, it's VBA.

** In VBA, this must be preceded by adding a reference to Gsa.tlb, from 'Tools | References'. Gsa.tlb can be found in the GSA program files folder. Other IDE's have equivalent methods.

10.2 Command File

The command file allows access to certain (basic) GSA operations by specifying these in a text file called a command file, identified by a GWC extension. The command file can then be run from the Windows “Run” command by typing

```
gsa <filename>
```

On machines that have access to several versions of GSA it is necessary to be explicit in specifying the version of GSA that is to be run. For example:

```
"c:\program files\oasys\gsa 8.1\gsa" c:\data\ex.gwc
```

The double quotes are necessary when the path includes spaces.

Alternatively the command file can be run when in GSA by selecting a GWC file in the “File | Open” command.

A log file will be created to record the execution of each of the commands in the GWC file.

The rules for a command file are as follows:

- The command file consists of a series of records, each record consisting of a command followed by any required arguments in parenthesis and separated by commas.
- There may only be one command per line and only one line per command.
- The commands are not case sensitive.
- Text following a comment marker, //, is ignored.
- Commands can optionally finish with a semi-colon character.
- White spaces are ignored except within quoted strings.
- Blank lines (no space or tab characters) are acceptable.
- Where command arguments are strings these should be enclosed in double quotes (e.g. "test.gwb")

More:

[Command File Commands](#)

[Command File Example](#)

10.2.1 Command File Commands

The GWC file commands are listed below under the following headings:

- [Core commands](#)
- [View commands](#)

Core commands

```
{
```

Special command to mark the start of a command sequence.

```
}
```

Special command to mark the end of a command sequence.

Open (filename)

Open a GWB, GWA or CSV file. Returns a status, as follows:

0 – OK

1 – failed to open

filename – the name of the file to be opened, including path and extension.

Save ()

Save the data to the default file (i.e. overwriting the file that was opened or last saved). Returns a status, as follows:

- 0 – OK
- 1 – no GSA file is open
- 2 – no default path is available; use SaveAs
- 3 – failed to save

SaveAs (filename)

Save the data to GWB, GWA or CSV file. Returns a status, as follows:

- 0 – OK
- 1 – no GSA file is open
- 2 – invalid file extension
- 3 – failed to save

filename – the name of the file to be saved, including path and extension.

Close ()

Close the current file. Returns a status, as follows:

- 0 – OK
- 1 – no GSA file is open

Exit ()

Close the file and exit GSA.

Analyse (task)

If no analysis tasks exist and task is zero: do linear static analysis of each specified load case.

If analysis tasks exist: analyse all analysis tasks that have not been analysed.

Returns a status, as follows:

- 0 – OK, analysis attempted
(use CaseResultsExist or TaskStatus to establish whether the analysis succeeded in producing results)
- 1 – no GSA file is open
- 2 – failed to attempt analysis

task – analysis task number (if task is zero, all tasks are considered) (default: zero)

Delete (option)

Deletes results. Returns a status, as follows:

- 0 – OK
- 1 – no GSA file is open
- 2 – invalid option
- 3 – that data is not present; – no action taken

option – valid settings are:

RESULTS – delete all results but not analysis cases.

RESULTS_AND_CASES – delete all results and analysis cases.

View commands

PrintView (option)

Print saved or preferred Graphic Views or Output Views. Returns a status, as follows:

0 – OK

1 – no GSA file is open

2 – invalid argument

option – valid settings are:

ALL_PGV – print all preferred Graphic Views.

ALL_SGV – print all saved Graphic Views.

ALL_POV – print all preferred Output Views.

ALL_SOV – print all saved Output Views.

ALL_LST – print all View Lists.

TAGGED_PGV – print tagged preferred Graphic Views.

TAGGED_SGV – print tagged saved Graphic Views.

TAGGED_POV – print tagged preferred Output Views.

TAGGED_SOV – print tagged saved Output Views.

TAGGED_LST – print tagged View Lists.

<name> – print the first view or view list found with the specified name.

The **option** needs to be double quoted, e.g. PrintView("ALL_PGV");

SaveViewToFile (option, filetype)

Save saved or preferred Graphic Views or Output Views to file. Returns a status, as follows:

0 – OK

1 – no GSA file is open

2 – invalid argument

option – valid settings are:

ALL_PGV – save all preferred Graphic Views.

ALL_SGV – save all saved Graphic Views.

ALL_POV – save all preferred Output Views.

ALL_SOV – save all saved Output Views.

ALL_LST – save all View Lists.

TAGGED_PGV – save tagged preferred Graphic Views.

TAGGED_SGV – save tagged saved Graphic Views.

TAGGED_POV – save tagged preferred Output Views.

TAGGED_SOV – save tagged saved Output Views.

TAGGED_LST – save tagged View Lists.

<name> – save the first view or view list found with the specified name.

filetype – valid settings are:

WMF – save Graphic Views to WMF file.

PNG – save Graphic Views to PNG file.

JPG – save Graphic Views to Jpeg file.

TXT – save Output View to tab delimited TXT file.

CSV – save Output View to comma delimited CSV file.

HTM – save Output View to HTML file.

The **option** and **filetype** need to be double quoted, e.g.
SaveViewToFile("ALL_SOVS", "TXT");

10.2.2 Command File Example

The following is an example of a command file.

```
// GSA command (GWC) file to:
//   open a model
//   delete any results
//   re-analyse
//   save the model
//   close the model
//   exit GSA

{
    open("test_01.gwb");
    // make sure there are no results present
    delete(RESULTS_AND_CASES);
    analyse();
    save();                      // overwrites the
                                // original file

    close();
    exit();
}
```

Part XI

11 Interaction with Other Software

GSA is only one of the programs that an engineer will use so it is important to be able to exchange information between GSA and other software tools such as spreadsheets, CAD programs, etc. for both model creation and manipulation, and for result post-processing and design.

More:

[Export options](#)

[Import options](#)

[Graphic View Save Options](#)

[Links With 3D Modelling Packages](#)

11.1 Export options

The export options are available from the “File | Export” menu command.

More:

[Input data](#)

[GSA 6.x](#)

[LS-DYNA](#)

[SAP2000](#)

[NASTRAN](#)

[ANSYS ASAS](#)

[OpenSees](#)

[AutoCAD](#)

[CIMsteel](#)

[Pdisp](#)

[ADC](#)

[Steel Member\(s\)](#)

[Footfall Response](#)

[Text output](#)

11.1.1 Input data

The input data can be exported as a GWA file — the GSA [ASCII file format](#). This file can contain all the input data for the model in a tab separated file. Each record starts with a keyword which defines the content of the record. The option is given to export data in either SI or model units.

Three different file extensions can be used:

- **GWA** - this is used as a means of transferring data from GSA in one version or on one machine to another. These files are tab separated and country independent.
- **TXT** - this is used to transfer data from GSA to other programs such as spreadsheets. These files are tab separated and the decimal point is determined from the Windows regional settings. So for example in the UK the decimal point will be a "." but in France this would be a ",".
- **CSV** - this is similar to the TXT option and is normally used to transfer data from GSA to other programs such as spreadsheets. These files use the list separator and the decimal point as determined from the Windows regional settings. So for example in the UK the decimal point and

list separator would be a "." and "," respectively but in France these would be a ";" and "," respectively.

11.1.2 GSA 6.x

To go to non-Windows version of GSA there is the option to export GSA 6.x file as GSI files.

11.1.3 LS-DYNA

There are two export options for LS-DYNA - the keyword file and the plot file.

Keyword file

This exports model data to an LS-DYNA keyword (.key) file. When exporting the user has control over the following:

- Export beam shape information
- Load case selection
- Seismic beams

Exporting beam shape information - this allows sections to be output as geometry (shape and dimensions). If this is not checked only the section properties (A , I_{yy} , I_{zz} , etc.) are output.

Load case selection - loads from selected cases are output. not all load types can be handled by LS-DYNA so only the node loads, beam loads and face loads output. LS-DYNA does not allow for the different load patterns that can be accommodated in GSA, so in creating the LS-DYNA data file the total load on the element is calculated and this is applied as a UDL in LS-DYNA.

Seismic beams - for structural applications it is often useful to export beams as seismic beams. This option allows this to happen. if this is selected a list of properties (not elements) is selected which are to be output as seismic beams. As the behaviour of seismic beams is asymmetric (plastic hinges forming only at end 2) there is an option to split the GSA beam elements in two, with a new node formed at the centre of the existing element. This becomes the first topology item in the two new elements. Seismic beams assume a yield stress of 275MPa. the GSA model is left unchanged by this operation.

This feature does not create a complete LS-DYNA data and some GSA features are excluded so the file is not thoroughly tested and should be used with caution. Any problems should be reported to Oasys.

The export includes:

GSA Feature	LS-DYNA Keyword
Title	*TITLE
Axes	*DEFINE_COORDINATE_SYSTEM
Nodes	*NODE, *BOUNDARY_SPC_NODE
Elements (beams, 2D elements, springs, links, mass)	*ELEMENT_BEAM, *ELEMENT_SHELL, *ELEMENT_DISCRETE, *ELEMENT_BEAM, *MAT_RIGID, *ELEMENT_MASS, *ELEMENT_INERTIA
Properties (beams, 2D elements, springs)	*PART, *SECTION_BEAM_TITLE, *INTEGRATION_BEAM, *SECTION_SHELL, *SECTION_DISCRETE, *MAT_SPRING_NONLINEAR_ELASTIC
Materials	*MAT_ELASTIC, *MAT_SEISMIC_BEAM,

GSA Feature	LS-DYNA Keyword
	*MAT_ORTHOTROPIC_ELASTIC, *MAT_FABRIC
Constraints (joints, constraint equations, rigid constraints, tied interfaces)	*CONSTRAINED_LINEAR_LOCAL, *CONSTRAINED_NODAL_RIGID_BODY, *CONTACT_TIED_SHELL_EDGE_TO_SURFACE_ID
Loads (node, beam, face, gravity)	*LOAD_NODE_POINT, *LOAD_BEAM_ELEMENT, *LOAD_SHELL_ELEMENT, *LOAD_BODY_X, *LOAD_BODY_Y, *LOAD_BODY_Z
Load curves	*DEFINE_CURVE
Lists	*SET_NODE_LIST, *SET_BEAM, *SET_SHELL_LIST

Typically the material and section properties will require updating to include the non-linear properties required by LS-DYNA. In general beam sections, shell properties, etc. will map to parts in the LS-DYNA model details of the mapping. The details of the mapping between the GSA and LS-DYNA models are recorded in a separate file with the same name as the keyword file but with a .txt extension appended.

DYNA Part Index

For most elements the part corresponds to the element property but in the case of link elements it is the element number and for rigid constraints it is the constraint number.

DYNA Part	Part type	Reference	
1	Beam	1	(property)
2	Shell	1	(property)
3	Spring	1	(property)
4	Link	13	(element)
5	Link	14	(element)
6	Rigid	1	(constraint)

Plot file

This exports model results to an LS-DYNA plot (.ptf) file for display in D3PLOT. The plot file contains the basic geometry and results for nodes and beam & shell elements. All states are written to a single plot file.

When using this feature it can be useful to also export a keyword file (with no seismic beams) as this can be read by PRIMER to generate a ztf file which can be used to pass extra model information to D3PLOT.

Details of LS-DYNA, D3PLOT and PRIMER at www.oasys-software.com/dyna/

11.1.4 SAP2000

The model data can be exported to a SAP2000 S2K file.

The user can select:

- To export sections as defined, section shapes or section properties.

The following restrictions apply:

- The S2K file is written in SAP2000 v6 format.
- Only geometry, properties and loading are exported.

- Spacer elements are not exported.
- Standard materials are appended to user materials.
- Additional mass on 2D elements is ignored.
- Pinned conditions on rigid constraints are ignored.

This option is not thoroughly tested and should be used with caution. Any problems should be reported to Oasys.

11.1.5 NASTRAN

The model data can be exported to a NASTRAN bulk data file.

Export

The user can select:

- Small or large field format.
- Static, modal dynamic or buckling solution options and number of modes for eigenvalue solutions.
- Which analysis stage should be exported.
- To export section properties or section shapes.

Restrictions

The following restrictions apply:

- Spacer elements not exported.
- Spring elements in non-global directions are approximate. The spring is assumed to act in the closest global direction.
- In spring elements non-axial/torsional effects are ignored.
- No distinction is made between applied displacements and settlements.
- The axis is ignored for applied displacements.
- Beam loads in user axes are transferred in global axes.
- Face load axes are ignored and pressure assumed normal to surface.
- Edge loads on 2D elements are ignored.
- Pre-stress and in-plane loads are converted to equivalent thermal loads.

CBAR & CBEAM elements

The principal difference between CBARs and CBEAMs is that the shear centre, neutral axis and centre of mass are all coincident for a CBAR element, but can be defined separately for a CBEAM.

Also, for SOL106 – $P\delta$ analysis – only CBEAM elements may be used.

CBAR elements will generally take PBAR or PBARL property cards, CBEAM elements will generally take PBEAM or PBEAML property cards. The 'L' suffix indicates that the property is defined by its dimensions, rather than defining the section properties explicitly. This is useful when element stress output is required, as NASTRAN automatically chooses the main stress recovery points of the section, or if you want to optimize on a particular dimension. See the NASTRAN manual for further information.

When converting GSA models into NASTRAN models the option is given writing either of the four property formats mentioned above. Using the PBEAML card is the only way to ensure that the shear centre, centre of mass and neutral axis are correctly defined for sections that do not have two axes of symmetry, such as angles, channels or tees. Sections that are defined explicitly in GSA are written as PBEAM or PBAR cards, regardless of whether section shape information is requested.

The disadvantage of using PBEAML cards is that some pre-processors may not be able to read and rewrite them.

11.1.6 ANSYS ASAS

The model data can be exported as an ANSYS ASAS data set.

The **System Data Area** specifies the amount of memory (in integer words) to be used by the ASAS run. Typical values are between 30000 and 1000000. (Default: 1000000)

A user-defined **Preliminary Data File** may be specified, or **default preliminary data** may be referenced.

The following restrictions apply:

- Spacer and Cable elements are not supported.
- Only rotational and translational springs are supported.
- 2D element releases are ignored.
- Only Explicit, I, RHS, CHS, Channel, L and T, standard and catalogue sections are supported.

Sections should be specified as rolled or welded. (Default: welded)

11.1.7 OpenSees

The model data can be exported to an OpenSees .tcl file.

Export

The user can select:

- Export bars as Truss or Corotational truss.
- Export steel beams as Elastic, Nonlinear or Displacement.
- Export concrete beams as Elastic, Nonlinear or Displacement.
- Export beam sections as Elastic or Fiber.

Restrictions

The following restrictions apply:

- Spacer elements are not exported.
- Standard materials are appended to user materials.
- Additional mass on 2D elements is ignored.
- Pinned conditions on rigid constraints are ignored.

11.1.8 AutoCAD

GSA can export DWG/DXF files that can be read by CAD packages such as AutoCAD and Microstation.

The [CAD Export Options dialog](#) allows the user to specify the content of the exported CAD file. A target version of AutoCAD may be selected; depending on the version selected there will be some variation in the way in which the GSA details are exported. GSA data is stored in SI units, but these may not be convenient units for the CAD program, so the user may select appropriate length units. The user can also choose to export from either the GSA Analysis Layer or the GSA Design Layer. Beam elements or members may be exported either as lines or as lines and 3D sections.

The model can be exported as either a wire-frame model or as wire-frame with 3D sections. Each

property can be assigned to a different layer and the 3D sections can be written to separate layers from the wire-frame model. The restraints on a model are exported to a Restraint layer. The 3D sections are written as blocks consisting of 3D faces. Alternatively, elements can be exported based on their group numbers, wherein entities created from those elements will have colours based on the element group.

If a section property in GSA is named "Main Beam" and has the description "CAT%UC%UC152x152x23%19860501" then the elements with this property will be represented in wire-frame style in a layer named "Main_Beam{CAT%UC%UC152x152x23%19860501}", the elements will be represented as 3D in a layer named "3D_Main_Beam" and a block will be created with the section information called BM_MAIN_BEAM.

This option exports the whole GSA model and should not be confused with the Graphic View "Graphics | Save Image | Save DXF" command that saves the current graphic image, including diagrams, deformations etc. to DXF.

Legacy Option

The option described above is the current and preferred method. The *superseded* [DXF Export Options dialog](#) remains available.

11.1.9 CIMsteel

GSA data and analysis results can be exported in CIMSteel (CIS/1) STEP file format.

The following elements and the relevant data (e.g. its property, load and results etc) will not be exported since they are not supported by CIS Standard

- 2D elements
- Ground spring elements
- Spring elements
- Mass elements
- Link elements
- Cable elements
- Spacer elements

The following load types will not be exported since they are not supported by the CIS Standard

- Gravity loads
- Thermal loads
- Distortional loads
- Support settlements
- Applied nodal displacements
- Beam poly-loads
- Beam pre-stressed loads

The following features will not be exported since they are not supported by the CIS Standard

- Rigid constraints
- Joints
- Constraint equations
- User defined axes

If a model has elements, features and/or loads etc, that cannot be exported the analysis results will not be exported, since the results are meaningless to the exported data. As the analysis results in GSA can relate to several load cases they are exported as Combined Results. Results for combination cases are not exported.

Bar, Tie and Strut elements in GSA are all exported as Beam elements with releases at both ends.

11.1.10 Pdisp

This exports the [Pdisp](#) data (compatible with Pdisp 17.7.2) used in a GsRaft model in the Windows Pdisp format. If GsRaft results exist when exporting Pdisp data the equivalent soil loads will be calculated and they will be exported along with the other Pdisp data.

To use these features go to the “Tools | Preferences” (Ctrl+F7) menu command, select “Advanced Features” and ensure that the “Raft analysis” option is enabled.

11.1.11 ADC

This exports member data for design in [ADC](#)/AdBeam.

The GSA model must contain concrete beam or column members and analysis results for this option to work.

Having set up concrete members, RC member design properties and RC Member bar layouts the “[ADC AdBeam Export](#)” dialog allows the user to set various parameters and select a list of beams to export. The export creates an AdBeam sub-frame model for each beam.

The following restrictions apply:

- Each member must be oriented to point in the same direction with the x axes pointing along the beam.
- The GSA model can include any type of loading but only beam loads are exported to AdBeam (excluding distributed moments).
- Only dead and imposed loads are exported.
- Gravity loading is transferred into an equivalent weight density for AdBeam.
- The effect of loading from secondary beams is ignored.
- The moment and shear results are exported only for the cases which can be analysed in GSA. The alternate span loads must be analysed in AdBeam.
- Minor axis bending and shear, axial loads and torsion in the beams are ignored.
- Offsets are ignored.
- The beam member properties in the design layer may differ from the corresponding element properties in the analysis layer.

11.1.12 AdSec

This exports section data for design in [AdSec](#)/AdBeam.

The GSA model must contain either concrete beam or column members, or assemblies for this option to work. If results are present forces and moment can be extracted from these and set as loads in AdSec. If members are chosen, one file is created for each member. For beam members three sections are exported (left, middle and right) and for columns a single section is exported. If an assembly is chosen a section is created for each position specified.

For an assembly GSA determines the elements cut at the specified position(s). The polygons corresponding to these elements are then assembled into large polygons and then into sections.

elements of the same material are merged into a single section while element of different materials form separate component sections. Where a section has several components GSA will assemble this into a compound section.

The following restrictions apply:

- The export may fail if materials are not specified as concrete or steel.
- The combination cases must contain only a single permutation.
- The design code in GSA may at time be unsupported in AdSec leading to a failure to read the file.

11.1.13 Steel Member(s)

Steel member input data can be exported to a csv file using this option. The input data comprises all the information supplied to the member code checking modules in text form. The resulting file can be opened by the Steel Design application, and the members checked.

11.1.14 Footfall Response

This exports data for use in the Footfall Response spreadsheet. The user is prompted for the nodes to export. The data is exported as a csv file containing sets of frequency, modal mass and nodal z displacements for each mode in a form that can be interpreted by the spreadsheet.

This will only work where there is at least one set of modal results.

11.1.15 Text output

Text output is available from Table Views and Output views. There are three file formats supported for text output

- **TXT**—tab delimited file for import to spreadsheets
- **CSV**—comma delimited file for import to spreadsheets
- **HTML**—web pages for display in browsers

These output the data displayed in the current view, in the selected file format.

For output views the export options is also available from the "[Output Wizard](#)". This allows the data or results to be exported without generating the output view first. (The option to copy output directly to the clipboard is also offered in the "Output Wizard".)

The layout of the information exported from an output view can be changed to "spreadsheet format" from the "[Further Options](#)" dialog in the Output Wizard. In this form the results for all positions on an element are output on one line per case, rather than the normal style of one line for each element position.

11.2 Import options

The import options are available from the "File | Import" menu command.

More:

[AutoCAD](#)

[CIMsteel](#)

[Pdisp](#)

11.2.1 Text files

Three different file extensions can be used:

- **GWA** - this is used as a means of transferring data from GSA in one version or on one machine to another. These files are tab separated and country independent.
- **TXT** - this is used to transfer data to GSA from other programs such as spreadsheets. These files are tab separated and the decimal point is determined from the Windows regional settings. So for example in the UK the decimal point will be a "." but in France this would be a ",".
- **CSV** - this is similar to the TXT option and is normally used to transfer data to GSA from other programs such as spreadsheets. These files use the list separator and the decimal point as determined from the Windows regional settings. So for example in the UK the decimal point and list separator would be a "." and "," respectively but in France these would be a "," and ";" respectively.

11.2.2 AutoCAD

GSA allows you to open/import CAD drawings stored in DWG or DXF formats.

The way in which AutoCAD features are interpreted will depend on the user selections in the [CAD Import wizard](#). The following points should be noted:

- **Layers** — Layers can be mapped to element properties in the property tables.
- **Sections** — Sections can be set up from layers. The section description must be given in the layer name enclosed in curly braces: { }.
- **Entity colours** — Entity colours can be used to map to element group numbers.
- **Lines** — These can be used to create 1D (two-noded) elements such as beams bars, ties, struts, etc. but can also be used to create two 0D (one-noded) elements (one at each end).
- **Points** — These can either be used to create 0D (one-noded) elements or to specify pinned restraints.
- **Arc / Circle / Ellipse** — These can be interpreted in the same way as lines, but are subdivided either by specifying an angle or a number of segments.
- **Polygon Mesh** — These can be interpreted as 2D, 1D or 0D elements. Two options are allowed for the polygon mesh; this allows, for example, quad elements to be stiffened by beam elements on the grid lines.
- **3D Face / Poly Face Mesh** — These are interpreted as 2D (3-noded triangle or four-noded quad) elements.

Note: when importing from a Polygon Mesh or 3D Face / Poly Face Mesh and 'Quad4' is selected: where two of the four vertices are coincident, a Tri3 element is created; where more than two vertices are coincident an error is reported.

- **Other options** — Other post-import options such as selecting units for import, tolerance for collapsing coincident nodes, tolerance for rounding node coordinates etc can also be specified.

CAD files can be imported into your model in 2 ways.

1. Open a DWG/DXF file — Give the 'File | Open' command and, in the Open dialog, set 'Files of type' to 'CAD Files (*.dwg, *.dxf)'.
2. Import an CAD file into an existing model — Give the 'File | Import | AutoCAD (DWG/DXF files)' menu command and, in the Import CAD dialog set 'Files of type' to DWG or DXF.

Selecting a DWG or DXF file will then open the CAD Import wizard. Follow the wizard instructions to import the drawing into a new GSA model.

Legacy Option

The option described above is the current and preferred method. The **superseded** [DXF Import Options dialog](#) remains available.

11.2.3 CIMSteel

GSA data and analysis results can be imported in CIMSteel (CIS/1) STEP file format.

A CIMSteel STEP file can either be read directly into GSA from the “File | Open” menu command or imported into an existing model using the “File | Import | CIMSteel (STEP file)” menu command. If there is any duplication when importing the file the user is asked to select the master data. This data will be kept unchanged when the data is merged.

Standard materials (UK, US and EC) cannot be imported since they are not supported by GSA. When these materials are encountered they are converted into the GSA standard “Steel” material.

When CIS standard sections are not supported by GSA, the section type will be defined as explicit and the explicit section properties will be imported, if they are available in the CIS STEP file, otherwise the section will be ignored.

The analysis results from Rigid-plastic, Elasto-plastic, Semi-rigid and Rigid partial strength analyses will be ignored since these analyses are not supported by GSA.

Standard Australian Sections cannot be imported. If the explicit section properties are available in the CIS STEP file, they will be imported, otherwise the section is ignored.

11.2.4 Pdisp

Pdisp data (VDD) files from Windows [Pdisp](#) 17.7.2 or earlier version can be imported into GSA. The analysis results in Pdisp data will be ignored in the importation since they are meaningless when imported into GSA.

To use these features go to the “Tools | Preferences” (Ctrl+F7) menu command, select “Advanced Features” and ensure that the “Raft analysis” option is enabled.

11.3 Graphic View Save Options

When working in Graphic Views it is convenient to be able to save an image of the displayed graphic for transfer to some other graphics or CAD package. The graphic image can be saved directly from Graphic Views in the following file formats:

- **WMF**—(Windows Metafiles) The native format supported by Microsoft
- **PNG**—(Portable Network Graphic) Best for images that contain mainly simple lines, text or white space.
- **JPEG**—(Joint Photographic Experts Group) Best for images that contain a lot of colour filling.
- **DXF**—For output to CAD packages. Note that this is an alternative to the [Export DXF](#). It does not carry any intelligence of the structural model but does include any adornments currently displayed in the Graphic View.
- **AVI**—(Audio Video Interleave) For recording moving graphic images.

See also:

[Working with the Program — Working with Graphic Views — Output of the graphic image — Saving the graphic image to file](#)

[Working with the Program — Working with Graphic Views — Output of the graphic image — Recording a moving graphic image](#)

11.4 Links With 3D Modelling Packages

Refer to the documentation for the specific feature for access details.

More:

[Revit](#)

11.4.1 Revit

The link between Revit and GSA is provided by the Revit plug-in, GsRevit, that is installed with GSA. After successful [GsRevit registration](#), several GSA / Revit Structure link options are offered on the *Revit Structure* 'Tools | External Tools' menu. (The options are named from the Revit perspective.)

The following options are provided:

- [Export to Oasys GSA](#) – create a new GSA model from a Revit model or synchronize an existing GSA model with changes from a Revit model.
- [Update from Oasys GSA](#) – synchronize an existing Revit model with changes from a GSA model.
- [Import from Oasys GSA](#) – create a new Revit model from a GSA model.
- [Import from Oasys GSA - An Example](#) - Import sample GSA model in to Revit.

Following the [Recommended Practice](#) will improve coordination between the GSA and Revit models.

11.4.1.1 Export to Oasys GSA

You are prompted to enter the name of the GWB file that is to be written. If the GWB file exists then you must choose whether to update the existing file or to overwrite with a new file.

GSA Layer

You must select which GSA layer to export to.

- Design Layer – GSA members are created. These will then have to be converted to elements within GSA.
- Analysis Layer – GSA elements are created. These will then have to be checked for connectivity within GSA.

See '[Recommended Practice](#)' for the pros and cons of each option.

Entities

The option is given to export the whole model or, if a selection has been made in Revit, just the selection.

Map GSA section

After the initial export there are options to map to GSA sections using the 'GSA section name' shared parameter or the 'GSA section description' shared parameter instead of the default coordination that uses the Revit symbol ID that is stored in GSA.

Properties and Materials

The option is given to export all sections (and associated materials), 2D Element properties or just those that are referenced by entities that are being exported. If 'Export all sections, 2D element properties and materials' is selected then all Structural Columns, Structural Framing types, wall and floor types are exported.

See '[Recommended Practice](#)' for information on why you might want to export all properties.

Open GSA session

If checked then the model is opened in GSA upon completion of the export.

Clear Log

The log is saved automatically on completion of the operation. The folder and file name of the log file is the same as the specified GWB file but with the extension RTF.

If 'Clear Log' is checked the log file associated with the GWB file being exported to is deleted at the start of the operation.

Notes on the Export Operation – writing a new GWB file

- Revit levels are written as GSA grid planes.
 - The GSA grid plane name is assigned the name of the level.
- Revit grids are written as GSA grid lines.
 - The GSA grid line name is assigned the name of the grid.
- Each Revit column, beam and brace is written as GSA a member or element, depending on whether 'design layer' or 'analysis layer' is selected (see above).
 - If existing nodes are not found at the required locations then new nodes are created.
- Each Revit Structural Floor and wall is written as a GSA area. The lines comprising a wall or a floor get translated as GSA lines.
 - A Revit Floor is deemed structural if the "Structural" option in its properties is checked.
 - A Revit Wall is considered structural if it has the "Enable Analytical Model" option checked in its properties and provided its structural usage is not "non-bearing".
 - Walls/Floors with openings may not translate to GSA Areas accurately and may require manual editing. Nonetheless, lines that form the opening will be exported as GSA Lines.
- The option to export Revit analytical offset of GSA member/element offset.
- Revit element orientation is translated into GSA member/element orientation.
- Revit materials associated with written columns, beams and braces are written as GSA materials.
 - The GSA material name is assigned the name of the material.
 - For steel materials:
 1. the Revit 'Minimum Yield Stress' is translated to the GSA 'Yield Stress' and
 2. the Revit 'Minimum Tensile Strength' is translated to the GSA 'Ultimate Stress'.
 - For concrete materials the Revit 'Concrete Compression' is translated to both the GSA 'Yield Stress' and 'Ultimate Stress'.
- Revit family types associated with written Structural Columns and Structure Framing are written as GSA sections.
 - GsRevit attempts to find a valid GSA section basing the section description on:
 1. the description entered in the Revit instance GSA section description shared parameter .
OR
 2. the description entered in the Revit symbol 'Type Comments' field. OR
 3. a matching entry in the 'Mappings.db3' file. OR
 4. the Revit symbol name.
 - The search for a matching GSA section is as follows:
 1. Is the section description an exact GSA section description?
 2. Does the section description approximate to a catalog section name, whether with the type at the beginning or end of the name?

- If no matching section name is found then an attempt is made to create a standard section based on the Revit symbol properties, as follows:
 1. If a depth, width and thickness are defined then a standard rectangular hollow section is created.
 2. If a depth and width are defined then a standard rectangular section is created.
- The GSA section name is assigned the Revit section description, whether the symbol 'Type Comment' or symbol name.
- Revit Floor and Wall types are written as 2D Element properties, with the same thickness and name as the Revit types. The materials associated with these types will not be exported, and 2D element properties will carry a dummy material reference of 99.

Notes on the Export Operation – updating an existing GWB file

For any Revit object that is handled by GsRevit(except floors and walls), the related object is searched for in the GSA model. The search is for the first occurrence of a reference to the Revit object in the GSA model, basing the search on the Revit object ID reference stored in the GSA objects.

- If a related GSA object is found then only the data controlled by Revit for that object is updated. Extra data is not affected.
- If a related GSA object is not found then a new object is created.
- Other data in the GSA model is not affected by the update.
- This does not apply to Floors and Walls and their types and may result in duplicate entities in GSA.

Known Issues

- Restraints are not handled.
- GSA objects that relate to deleted Revit objects are not deleted.
- Floors and walls are not imported/Updated into the Revit.

11.4.1.2 Update from Oasys GSA

You are prompted to enter the name of the GWB file from which the Revit model is to be updated.

GSA Layer

This updates from the selected GSA layer. It's best to keep this in line with the layer the model was originally exported to.

Notes on the Update Operation

When updating a Revit Model from GSA:

- Update of member/element is identified by unique ID. Additionally members/elements can be updated using the GSA member/element ID stored in the Revit shared parameter if available. If this option is checked then the instance in Revit will be first identified by unique ID otherwise by the GSA member/element ID shared parameter.
- Grid Planes / Levels: Any changes to the elevations of GSA grid planes will reflect as changes in the elevation of the related Revit levels.
- Grid Lines / Revit Grids: Changes to grid line start point will be transferred.
- GSA Beam elements/Members related to Structural Columns:
 - Columns can be moved horizontally, but not vertically.
 - Column lengths can be altered.

- Revit offsets are changed as necessary to accommodate changes in the elevation of reference levels.
- GSA Beam elements/Members related to Structural Framing: Beams/Braces
 - Beams can be moved horizontally.
 - Beam lengths can be changed.
 - Revit offsets are changed as necessary to accommodate changes in the elevation of reference levels.
- The option to update Revit analytical offset.
- Changes in beam orientations is translated
- Changes in beam releases can be translated
- Element property references: Element property references may be changed but the references must be to properties that are either
 - Related to Revit family types (i.e. a section that has been exported from Revit to GSA.) An element referring to a "Framing Section" must refer only to other Framing Sections. See '[Recommended Practice](#)' OR
 - If referencing new section property, the section in question must be imported into Revit as new Family.
- Section properties:
 - Newly created section properties can be imported into Revit. A new section can only be referenced by columns or by Framing. If it is referenced by both, it will be ignored for the import.
 - Changes to existing section properties are not reflected in the corresponding Revit Families.
- Materials: Properties of materials can be edited. However, creating new materials is not allowed since this is not supported by Revit. See '[Recommended Practice](#)'.

Known Issues

- Changes made to GSA areas and lines will not be translated, yet.
- There may be unexpected results while trying to update Revit beam systems, if their coordinates have been changed inside GSA, or while updating Revit levels whose elevations have been changed via GSA grid planes that have Revit beam systems defined on them.
- When a beam is 'hosted' on a column or where two or more beams are 'joined', changing their coordinates inside GSA may result in unexpected behavior such as the Revit elements becoming disjointed.

11.4.1.3 Import from Oasys GSA

You are prompted to enter the name of the GWB file that is to be imported. Until a valid GWB file is found, other controls on the form will stay disabled.

Import from (GSA Layer)

You must select which GSA layer to import from.

- Design Layer – GSA members are read and imported.
- Analysis Layer – GSA elements are read and imported.

Importing Members/Elements

- Import all entities – Imports all the entities (elements/members) on the selected layer.

- Import from GSA list – You can import only those elements/members that are a part of pre-saved lists in the GSA model.
- Exclude dummy elements – dummy elements are not imported, even if referenced in the list.

Grid lines

Grid lines are imported unless they have been imported previously. (Use Update from Oasys GSA to update the position of grid lines that have been imported previously.)

Datum

The 'Align imported model such that the GSA global origin is located at the Revit Project Base Point (else at the Revit Survey Point)' option determines the datum of the Revit model. (The Revit Project Base Point option is not available in Revit Structure 2012.)

Levels

- Use GSA grid planes – GsRevit will create levels at elevations corresponding to GSA's grid planes.
- Use existing levels – All elements created will be based on the existing levels in the model.

Offsets

The 'Import GSA member/element offsets to Revit analytical offsets (else ignore GSA offsets)' option may be used to import GSA offsets.

Orientation

GSA member/element orientations are imported into Revit.

Import GSA Members/Elements to Revit

- Vertical GSA members/elements can be imported as Revit columns or braces.
- Inclined GSA members/element can be imported as Revit beams, braces or slanted columns.
- Horizontal members can be imported as Revit beams or braces.
- Determine Revit element type from GSA member type. This is applicable when importing GSA from the design layer and it overrides the setting above, except where a horizontal GSA member is specified in GSA as a column member type; in this case the member will be imported as a Revit beam or brace according to the setting above.

Mappings database

This shows the column and framing mapping tables where you can specify the Revit family type of a particular Revit family to map the corresponding GSA section using either the GSA section description or name. Changes made to tables will update the mapping.db3 at the indicated at the top of the form.

GSA model section not mapped to Revit

This shows the list of GSA sections not mapped to Revit and the suggested available Revit family plus the section usage as either column or framing. The *Add suggested match to the mappings table* button will then copy the suggested matches to the mappings tables above.

Mapping option

The family symbol will be used as per the selection- when there is conflict between a family symbol that is already loaded in Revit and a family symbol identified in mapping database.

Map GSA Sections to Revit Family Types

This table displays the actual Revit Family types that will be used for the corresponding GSA sections. Please note that a given GSA section can either be set to a Revit Family for Structural columns or for Structural framing but not both. Sections referred to by both GSA columns and beams/braces will be flagged INVALID.

GsRevit will try to find the best mapping between a GSA Section and Revit Family Type based on the mappings database. If it fails to find a match, the section will be flagged <unmapped>. Members/elements referring to unmapped sections will not be imported.

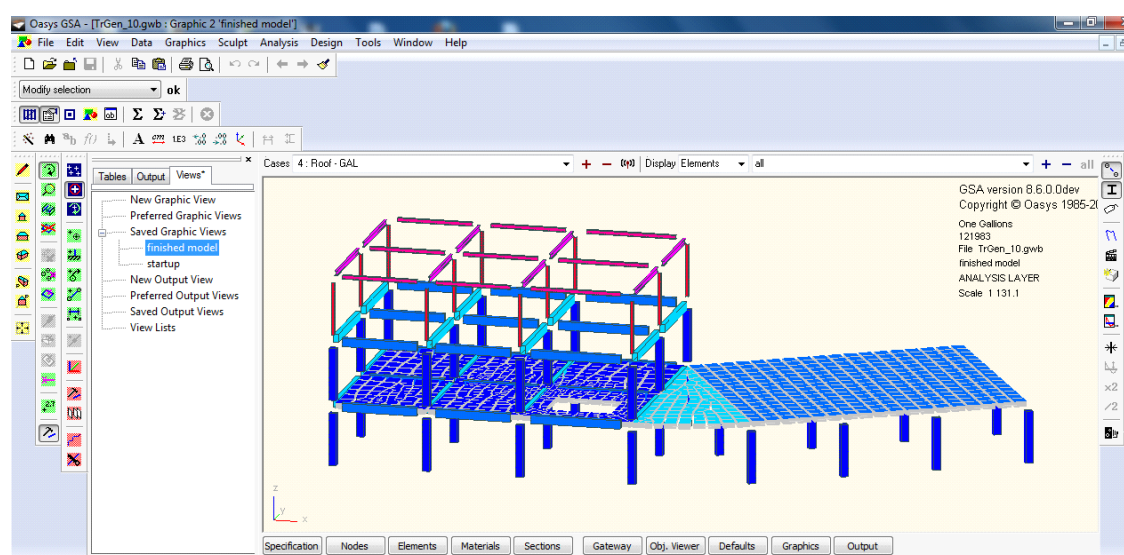
The section mapping utility looks for entries in the Mappings database. For sections that don't have an entry in the Mappings database, load the appropriate Family Type into the model before starting the import process and map them manually using the drop downs.

11.4.1.4 Import from Oasys GSA-An Example

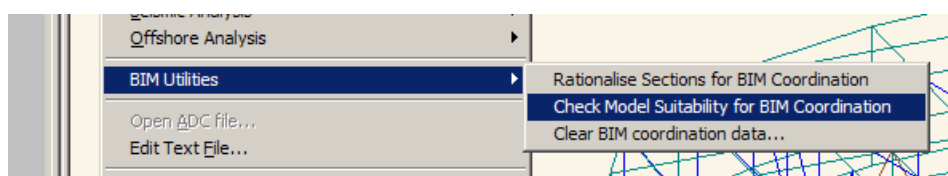
Importing a GSA model into Revit

This tutorial contains a step by step guide on importing a GSA model into Revit using the GsRevit 'Import from Oasys GSA' tool. The procedure followed for importing this model is specific to the example and the approach can differ depending on the model in hand.

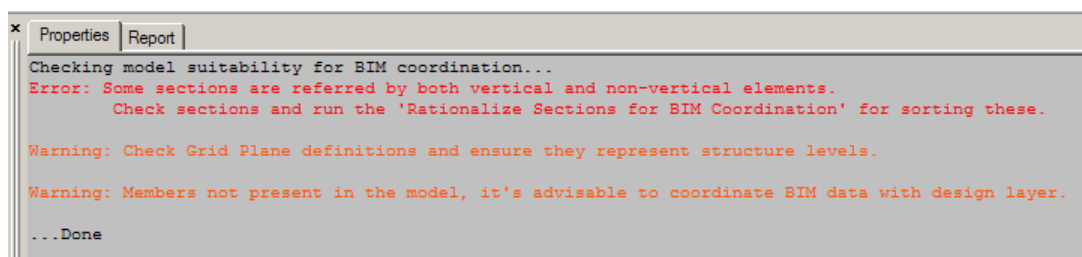
The model used in this tutorial, "TrGen_10.gwb" can be found under the Samples folder in the GSA installation folder, typically <C:\Program Files\Oasys\GSA 8.7\Samples\Training_General>.



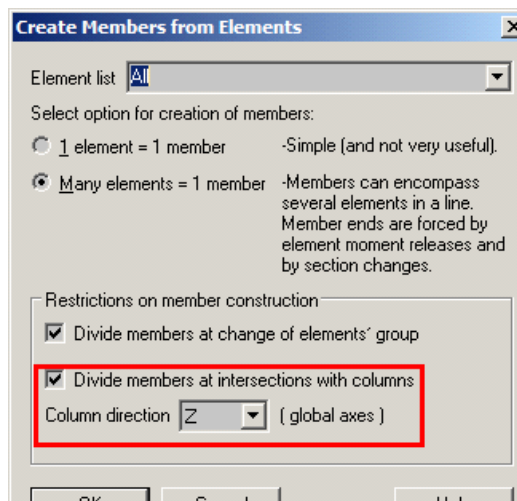
1. Open the file in GSA 8.7, or later.
2. From Tools | BIM utilities, run 'Check Model Suitability for BIM Coordination Command'



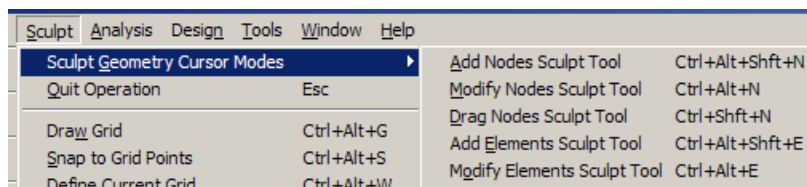
3. GSA produces the following output in the Property Viewer:



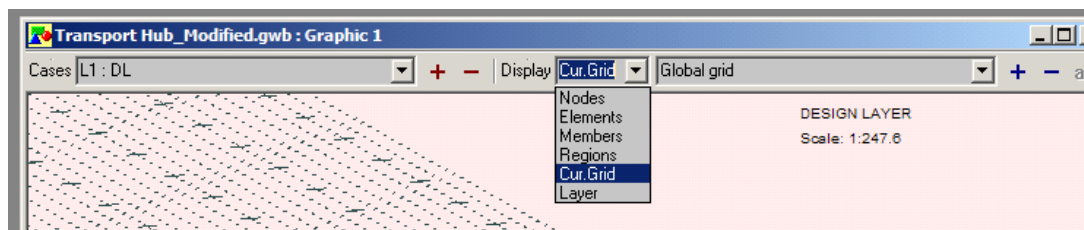
4. **Warning: Members are not present in this model**: It's advisable to coordinate with the GSA Design layer. Use 'Tools | Manipulate Model | Create Members From Elements' to automatically generate members.



Note: If the Automatic tool does not split members at the correct locations, try changing the elements list and Column direction options in tandem. Alternatively you could also use the sculpt options to create members manually wherever needed.



5. **Error: Some sections are referred by both vertical and non-vertical elements**. GSA Sections that are referred to by vertical and non-vertical members cannot be imported. However, vertical and slanted column can refer to same GSA section. Use 'Tools | BIM Utilities | Rationalize sections for BIM Coordination' to correct these.
6. **Warning: Check Grid Plane definitions and ensure they represent structure levels**. Grid planes elevations in GSA model should represent levels elevations. To graphically iterate through the grid planes in the model, first turn on the grid display from 'Sculpt | Draw Grid' or the  toolbar option, then switch to 'Cur Grid' in the Display drop down in the graphics view.



You can now loop through all grid planes in the model by using the  buttons. More on grid planes is in the manual under 'Program Fundamentals | Grid Planes'.

7. Creating new grid planes: Change to 'Select Node' cursor mode and select 3 nodes lying on the proposed plane. 'Sculpt | Create Grid Plane' will create a grid plane if one doesn't exist. See 'Sculpting | Creating grid planes graphically' in the manual.
8. Section Mapping: The section property table contains cross sections of members used in the model. These need to be mapped to Revit Families. This can be done in two ways, one by editing 'Mappings.db3' file using SQLite or by editing mapping table (Section 10) in mapping section wizard.

Transport Hub_Modified.gwb : Beam Sections				
Property	A Name	B Material	C Description	D Axes Stiffn Calcu
Defaults	Section #	Steel	Explicit	yz axes
1				
2	Section 2	Steel	CAT UB UB838x292x176	yz axes
3	Section 3	Steel	CAT UC UC356x406x634	yz axes
4	Section 4	Steel	CAT PFC PFC430x100x64	yz axes
5	Section 5	Steel	CAT ISA ISA20x20x4	yz axes
6	Section 6	Steel	CAT UC UC356x406x551	yz axes
7	Section 7 (vertical)	Steel	CAT UC UC356x406x634	yz axes
8	Section 8	Steel	STD R 100. 10.	yz axes
9	Section 7 (non-verti	Steel	CAT UC UC356x406x634	yz axes

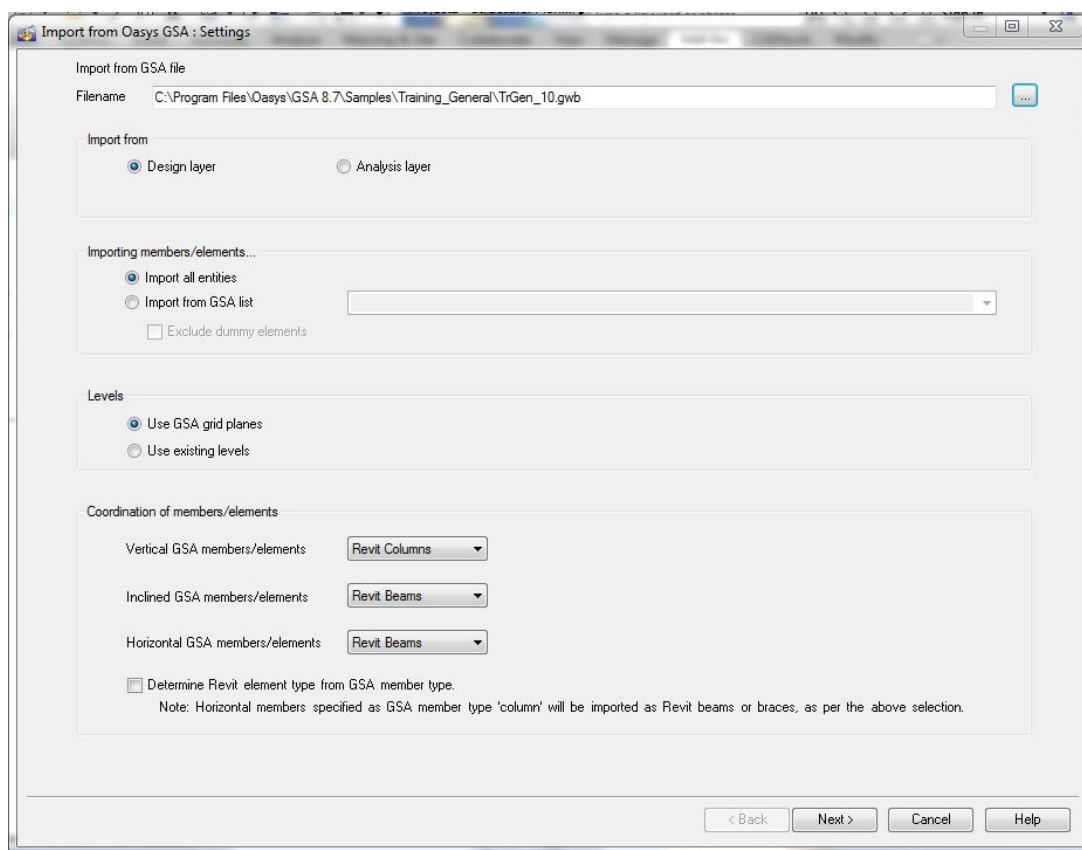
Open 'Mappings.db3' from <C:\Program Files\Oasys\GSA 8.7\mappings.db3> in SQLite personal (<http://www.sqliteexpert.com/>) and ensure all entries in the section property table are present in Framing or Columns table as the case may be:

Database: mappings Table: GsRvt_Columns File: C:\Program Files\Oasys\GSA 8.6\mappings.db3				
RecNo	GSA_Section_Description	Revit_Family_Type	Revit_Family	
Click here to define a filter				
1	CAT PFC PFC100x50x10	100x50x10PFC	PFC-Parallel Flange Channel-Column	
2	STD C 300.	300mm	Concrete-Round-Column	
3	STD R 450. 300.	300 x 450mm	M_Concrete-Rectangular-Column	
4	CAT UB UB203x133x30	203x133x30UB	UB-Universal Beam-Column	
5	CAT UB UB838x292x176	838x292x176UB	UB-Universal Beam-Column	
6	CAT UB UB762x267x197	762x267x197UB	UB-Universal Beam-Column	
7	CAT PFC PFC430x100x64	430x100x64PFC	PFC-Parallel Flange Channel-Column	
8	CAT UC UC356x406x634	356x406x634UC	UC-Universal Column-Column	
9	CAT UC UC356x406x551	356x406x551UC	UC-Universal Column-Column	

Caution: If you modify the mappings.db3 file, make sure to save it in a different location – since reinstalling GSA would overwrite this file. In this case, the path to your version of mappings.db3 can be set in GSA by ‘Tools | Preferences | Miscellaneous | File Locations | Mappings Database file’.

Save and close the mappings.db3 file.

9. In Revit, run ‘External Tools | Import From Oasys GSA’ and select the following options



10. On clicking next, “Import from GSA: Mappings” screen will appear where user can see list of mapped/unmapped GSA section. User can do mapping here and on “Update mapping database” changes will be saved in mapping database at path displayed at top.

Import from Oasys GSA : Mappings

Mapping database file location: C:\Program Files\Oasys\GSA 8.6\mappings.db3

Mapping database

Column mapping table

GSA_Section_Description	Revit_Family_Type	Revit_Family
CAT PFC PFC100x50x10	100x50x10PFC	PFC-Parallel Flange Channel-Column
STD C 300.	300mm	Concrete-Round-Column
STD R 450. 300.	300 x 450mm	M_Concrete-Rectangular-Column
CAT UB UB203x133x30	203x133x30UB	UB-Universal Beam-Column
CAT IUB IUB838x292x176	838x292x176IUB	IUB-Universal Beam-Column

Framing mapping table

GSA_Section_Description	Revit_Family_Type	Revit_Family
STD R 800. 400	400 x 800mm	Concrete-Rectangular Beam
STD A 500. 450. 300. 200.	450LB500	Precast-L Shaped Beam
STD A 600. 450. 300. 200.	450LB600	Precast-L Shaped Beam
CAT ISA ISA20x20x4	SA20X20X4	L-Angles
STD C 350.	350mm	M_Round-Column

Update mapping database

GSA model section not mapped to Revit

GSA model sections missing from mappings database	Best guess (Family Type - Family)	Usage
STD%R(mm)%:600%:300	-	Framing
*		

Add missing section to mapping table Add best guess to mapping table

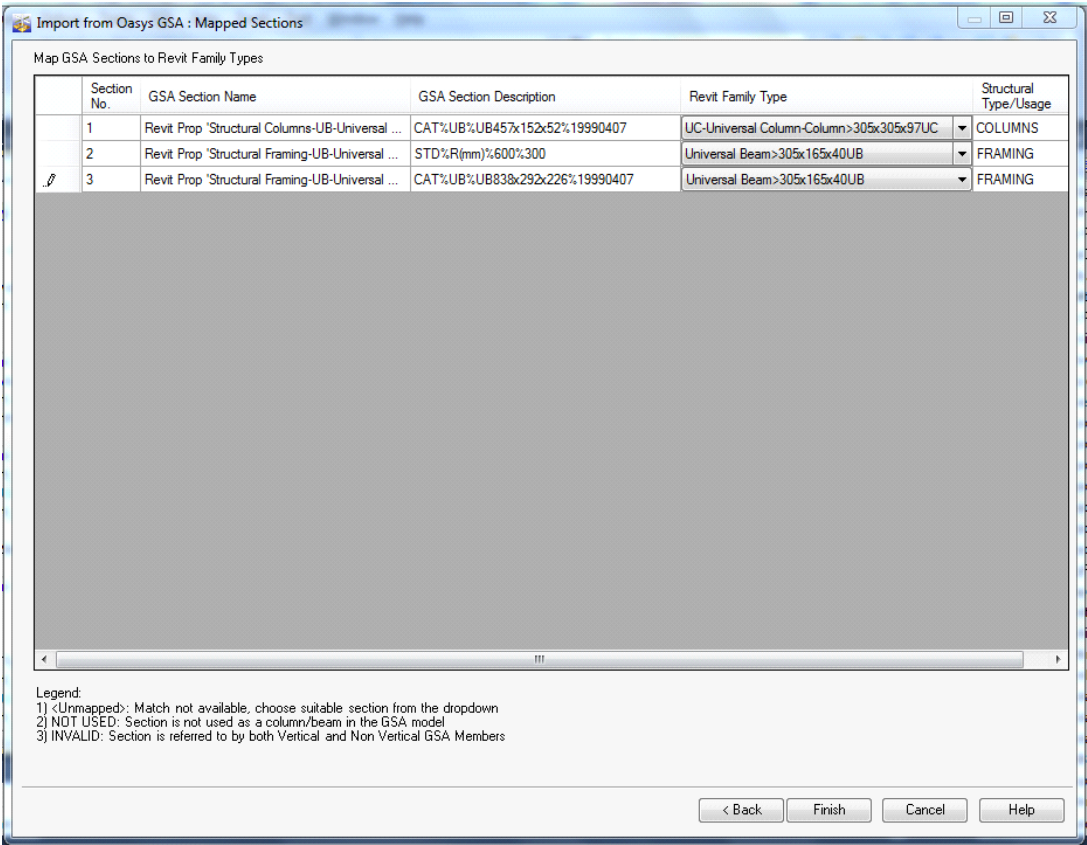
Mapping option

Where there is a conflict between a family symbol that is already loaded in Revit and a family symbol identified in the mappings database:

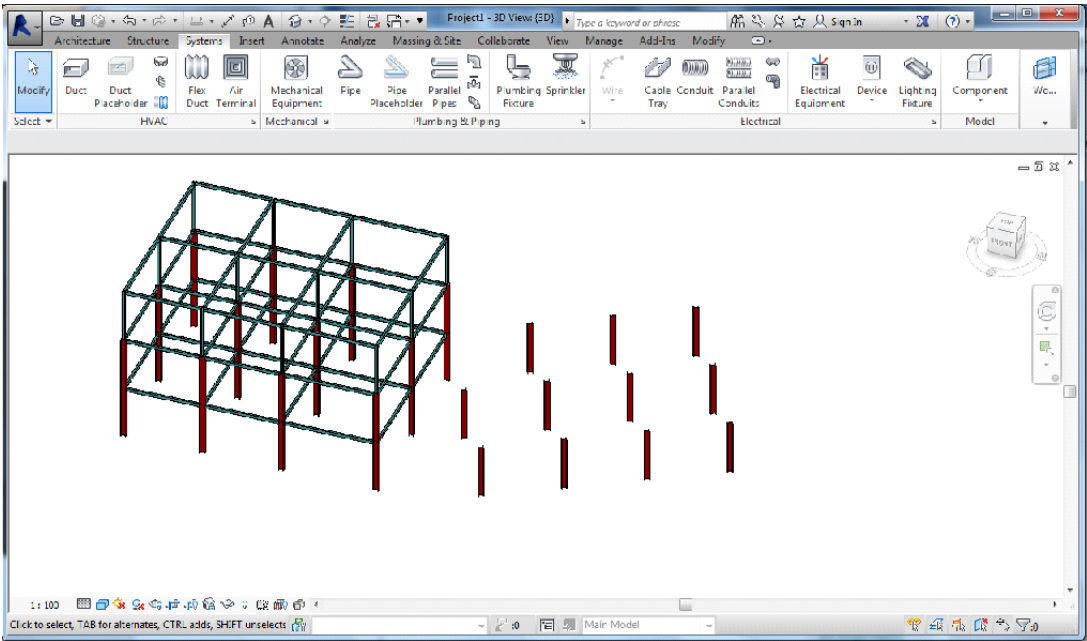
☒ Use the family symbol that is already loaded in Revit. ☐ Override the loaded family symbol with the symbol identified in the mappings database.

< Back Next > Cancel Help

11. When mapping sections, make sure no sections are <Unmapped>. Sections mapped INVALID will not be imported.



Select 'Unjoin Elements' if Revit throws up warnings.



--- oOo ---

11.4.1.5 Recommended Practice

Update From GSA and Import From GSA

The suitability of a GSA model for coordination with Revit can be examined using the tool 'Check Model suitability for BIM coordination' from the tools menu.

Mapping GSA Sections

Mapping section properties to Revit Family types is the most crucial step in the import. GsRevit looks for entries in the Mappings file to find suitable matches for GSA sections. For sections not listed in the file, load the desired (equivalent) family type before starting the import. During the import process, the appropriate family type can be manually selected from the drop-downs.

Section Property References

For a workable mappings you must ensure that each section is not referred to by elements/members that are vertical, inclined and horizontal. However, it mostly depends on how you want to import the GSA elements/members to Revit: if for example vertical and inclined members in GSA are both imported as columns in Revit then they can be referred to by same GSA section.

GSA has a tool that facilitates splitting section properties into vertical, inclined and horizontal sections. The command 'Rationalize sections for BIM coordination' in the Tools menu examines all sections in the section property table. If any section property is referred to by both vertical, inclined and horizontal entities, the tool creates a suitable new section property by copying original one and then edits the appropriate elements and members to suit.

Each Revit column, beam and brace is reflected in GSA as a single member or beam element, depending on whether Design Layer or Analysis Layer is selected. Note the following:

Member - Element coordination

It is recommended that the design layer (GSA Members) be used for coordinating with BIM models.

The commands 'Tools | Create Elements from Members' and 'Tools | Create Members from Elements' can be used to initially create Members from Elements and vice-versa, but subsequent coordination between the design and analysis layer will need to be done manually. To assist in checking consistency of section property references, the tools 'Update Element sections' or 'Reset Member sections' in the Design menu might be helpful.

Creating levels

All the new entities created by GsRevit will map to the nearest level available. Since Revit Columns span between 2 valid levels, it is important for the levels to properly reflect elevations of the column ends. GsRevit can try and create levels from GSA's grid planes, but the user can also create them manually before invoking the import.

Preparing for the import/update

1. Ensure if model is fit to be coordinated with Revit by running the 'Check Model Suitability for BIM coordination' command. Make modifications to the model if necessary.
 - 1a. Create Members from Elements.
 - 1b. Create Grid Planes. If your model doesn't contain existing grid planes, they can be created by running '[Tools | Create Storeys](#)' and '[Tools | Create Grid Planes From Storeys](#)' in succession. It's necessary to ensure Grid Planes represent column ends and level elevations.
 - 1c. Rationalize Sections, if needed, so that each section is referred only by either vertical or non-vertical entities.
2. Open a new Revit model from one of the templates for Metric or for Imperial/English units.
3. Load the appropriate family types which are used in the GSA model. If a particular section

is being used both as a column and as a beam in GSA ensure that you load them separately in the Revit model under Structural Column Family types and Structural Framing Family types.

The Revit model is now ready for receiving the GSA model file.

Note: After the import process, Revit may display warnings against errors in the models. Choose 'Unjoin Elements' in such a situation.

11.4.1.6 Registering GsRevit

Revit needs to be told that the Revit / GSA link plug-in (GsRevit) is available, where it resides on disk and what options are provided, before Revit will offer the GsRevit options. If Revit is already installed at the time GSA is installed then the GSA installation will automatically register GsRevit with Revit/Revit Structure. Note that you must run Revit/Revit structure once before installing GSA as this will create the required folders for the Addins file. Successful registration is indicated by the presence of the GSA options in the Revit Structure 'Add-Ins | External Tools' menu (and that they work!).

If Revit is installed after GSA then you can register GsRevit interactively by running GsaRegister.exe, after running the Revit Application once to create necessary folders. You will find GsaRegister.exe in the GSA program folder which may be found in the GSA program folder. The user running GsaRegister must have admin rights on the machine.

11.4.1.7 Section Mappings File

The Mappings file, Mappings.db3, is a SQLite database that contains mappings between GSA Sections and Revit Column and Framing Family Types. This file is installed at a default location in the GSA installation Program Files folder, typically, <C:\Program Files\Oasys\GSA x.y> where x and y are version numbers. The location of this file can be changed in [GSA preferences](#).

Note: The Mappings.db3 file will be overwritten whenever a new version of GSA is installed. Before making changes, copy the mappings file to a different folder and change GSA Preferences to reflect the new location.

The tables GsRvt_Framing and GsRvt_Columns contain section mappings for Revit Framing Family Types and Column Family Types respectively. A brief description of the fields -

GSA_Section_Description: Use the section description from GSA Section Property table. See [Naming convention for sections](#).

Revit_Family_Type: The name of the family type. This name must either be contained in the text file accompanying the family rfa file (as in the case of Steel Catalogue families) or they must be 'defined' in the rfa file.

Revit_Family: The name of the family file on the disk, without the extension.

GsRevit searches for family files in locations listed in Revit 'Options | File Locations | Places'.

Part XII

12 Gss Theory

[Active Degrees of Freedom](#)

[Analysis Options](#)

[Applied Displacements](#)

[Axes](#)

[Condition number](#)

[Constraints](#)

[Direction cosines](#)

[Dynamic results](#)

[Eigensolver](#)

[Elements](#)

[Error Norms](#)

[Forces in 2D Elements](#)

[Ill Conditioning](#)

[Loads on 2D elements](#)

[Loads on beams](#)

[Mass distribution](#)

[Material models](#)

[Matrix Solver](#)

[Nodal Stresses, Forces and Moments](#)

[Non-linear Analysis](#)

[Reactions](#)

[Rigid constraints](#)

[Ritz analysis](#)

[Shape functions](#)

[Stress and strain](#)

12.1 Active Degrees of Freedom

Before the stiffness matrix is assembled it is necessary to decide which degrees of freedom need to be included in the solution.

The nodes can be categorised as follows:

- **inactive**—the node does not exist.
- **non-structural**—the node is not part of the structure (e.g. orientation node).
- **active**—the node is part of the structure.

Likewise the degrees of freedom can be categorised as:

- **non-existent**—this degree of freedom does not exist because the node is undefined.
- **inactive**—this degree of freedom exists but is not used (considered like a restrained node).
- **restrained**—the degree of freedom exists and is part of the structure but it is restrained and so it is not active in the stiffness matrix.

- **slave**—the degree of freedom is constrained (through being in a rigid constraint, or by a repeat freedom) to move relative to a master degree of freedom and so it is not active in the stiffness matrix.
- **active**—this degree of freedom is active in the stiffness matrix.

In setting up a list of degrees of freedom the following operations are carried out:

1. The nodes are sorted with respect to an ordering vector (front order) to ensure efficient use of memory in the solution procedure.
2. All the nodes are assumed to be inactive.
3. Look at elements attached to nodes to see which degrees of freedom are required.
4. Remove the degrees of freedom that are restrained by single point constraints or global constraints.
5. Remove the degrees of freedom that are slaves.
6. Remove degrees of freedom that have no local stiffness.
7. Number the degrees of freedom.

These aspects are considered in more detail below.

More:

[Front Order](#)

[Active Degrees of Freedom for Elements](#)

[Restraints](#)

[Slave Degrees of Freedom](#)

[Degrees of Freedom with no Local Stiffness](#)

12.1.1 Degree of Freedom Ordering

The way in which the degrees of freedom are ordered is important in keeping the size of the stiffness matrix to a minimum. The methods offered in GSA are

- **Geometric**—the degrees of freedom are ordered along a vector. By default this vector is along the longest dimension of the model but can be adjusted by the user.
- **Reverse Cuthill-McKee**—use the bandwidth minimizer of the same name,. This looks at the connectivity of the structure by using graphs and reorders the degrees of freedom so as to minimize the bandwidth.
- **Cuthill-McKee**—similar to the Reverse Cuthill-McKee method (but generally less optimal).

The normal Gss solver is a skyline solver and the storage depends on the individual column profiles. With this type of solver minimum bandwidth may not coincide with minimum matrix size.

12.1.2 Active Degrees of Freedom for Elements

The degrees of freedom are made active based on the elements attached at the nodes. The degrees of freedom will depend on the element type: These are summarised in the table below:

Element	Active degrees of freedom
bar	3 translational
beam	3 translational + 3 rotational
translational spring	3 translational

rotational spring	3 rotational
lumped mass	3 translational
lumped mass with inertia	3 translational + 3 rotational
2D plane stress, plane strain, axisymmetric	Bilinear formulation: 2 translational Allman-Cook formulation: 2 translational + 1 rotational
2D membrane, shear panel	2 translational if in global plane 3 translational otherwise
2D bending	1 translational + 2 rotational if in global plane 3 translational + 3 rotational otherwise
2D shell	3 translational + 2 rotational if in global plane 3 translational + 3 rotational otherwise
2D wall element	3 translational + 3 rotational
3D brick, wedge, tetra	3 translational

12.1.3 Restraints

Restraints can be applied to a node. These can either be for individual nodes or globally. These simply remove the relevant degrees of freedom from the active list.

12.1.4 Slave Degrees of Freedom

Degrees of freedom which are referenced as slave degrees of freedom in rigid elements or in joints/ constraint equations are removed from the list of active degrees of freedom and a constraint equation is set up relating the displacement of the slave relative to the master degree of freedom. For a joint these equations are of the form

$$u_{slave} = u_{master}$$

For a rigid constraint or link slave the degrees of freedom are more complicated (see [Rigid Constraints](#) and [Link Elements](#))

12.1.5 Degrees of Freedom with no Local Stiffness

It is possible to construct a model and find that there is no stiffness associated with particular degrees of freedom, either for translation or rotation. For example a model made up of shell elements in a general plane 6 degrees of freedom will be assigned per node, but there is only stiffness in 5 of these. There are a number of approaches to avoid this problem.

Geometry based automatic constraints

At each node in the structure the attached elements are identified. A pseudo stiffness matrix associated with rotations is set up with a value of one on the diagonal if the element is stiff in that direction or zero if there is no stiffness. All off-diagonal terms are set to zero. The pseudo stiffness is transformed into the nodal axis system (so the off-diagonal terms are, in general, no longer zero) and added to a nodal pseudo stiffness matrix.

Once this has been done for all the attached elements an eigenvalue analysis of the resulting pseudo stiffness is carried out to reveal the principal pseudo stiffnesses and their directions. If any of the principal pseudo stiffnesses are less than the pre-set “flatness tolerance” then those degrees

of freedom are removed from the solution and an appropriate rotation to apply to the stiffness matrix at the node is stored.

Stiffness based automatic constraints

This is similar to the geometry based automatic constraints but instead of a value of one or zero assigned to degrees of freedom the actual stiffness matrix is used. The resulting stiffness matrix is the same as would result from restraining the whole model except from the rotations at the node of interest.

Again an eigenvalue analysis is carried out to reveal the principal stiffnesses and their directions. If any of the principal stiffnesses are less than the pre-set “stiffness tolerance” then those degrees of freedom are removed from the solution and an appropriate rotation to apply to the stiffness matrix at the node is stored.

Artificial stiffness in shells

An alternative and cruder approach is to make sure that there is some stiffness in all directions by applying an artificial stiffness in the directions that are not stiff. This is done by constructing the element stiffness matrix for shell elements and then replacing the zeros on the leading diagonal with a value of 1/1000th of the minimum non-zero stiffness on the diagonal.

Since this approach introduces an artificial stiffness term that has not physical basis it should be used with care.

12.2 Analysis Options

[Static & Static P-delta](#)

[Modal & modal P-delta dynamics](#)

[Ritz Analysis](#)

[Buckling](#)

[Model Stability Analysis](#)

12.2.1 Static & Static P-delta

Static

The static analysis is concerned with the solution of the linear system of equations for the displacements, u , given the applied loads. The applied loads give the load or force vector, f . The elements contribute stiffness, K , so the system of equations is

$$f = Ku$$

Static P-delta

The static P-delta analysis is similar to the static except that a first pass is done to calculate the forces in the elements. From these forces the differential stiffness can be calculated. The stiffness of the structure can therefore be modified to take account of the loading and the displacements are then the solution of

$$f = (K + K_g)u$$

The options allow for

- a single load case to be used as the P-delta load case
- each load case to be its own P-delta load case

In the first case the first pass of the analysis solves

$$\mathbf{f}_{p\delta} = \mathbf{K}\mathbf{u}_{p\delta} \rightarrow \mathbf{K}_g$$

then for all the load vectors

$$\mathbf{F} = (\mathbf{K} + \mathbf{K}_g)\mathbf{U}$$

where $\mathbf{F}$ is a matrix of the load vectors for all the load cases.

In the second case there is a one for correspondence between P-delta load case and analysis case so

$$\mathbf{f}_i = \mathbf{K}\mathbf{u}_i \rightarrow \mathbf{K}_{gi}$$

then

$$\mathbf{f}_i = (\mathbf{K} + \mathbf{K}_{gi})\mathbf{u}_i$$

12.2.2 Modal & Modal P-delta Dynamics

Modal

The modal dynamic analysis is concerned with the calculation of the natural frequencies and the mode shapes of the structure. As in the static analysis a stiffness matrix can be constructed, but in a modal dynamic analysis a mass matrix is also constructed. The free vibration of the model is then given by

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{K}\mathbf{u} = 0$$

The natural frequencies are then given when

$$|\mathbf{K} - \lambda\mathbf{M}| = 0$$

The eigenvalue problem is then

$$\mathbf{K}\boldsymbol{\varphi} = \mathbf{M}\boldsymbol{\varphi}\Lambda$$

where $\{\lambda, \boldsymbol{\varphi}\}$ are the eigenpairs – eigenvalues (the diagonal terms are the square of the free vibration frequencies) and the eigenvectors (the columns are the mode shapes) respectively.

Modal P-delta

The modal P-delta is similar to the normal modal analysis but takes into account that loading on the structure will affect its natural frequencies and mode shapes. In the same way as a static P-delta analysis a first pass is carried out from which the differential stiffness can be calculated. This is used to modify the stiffness matrix so the eigenproblem is modified to

$$(\mathbf{K} + \mathbf{K}_g)\boldsymbol{\varphi} = \mathbf{M}\boldsymbol{\varphi}\Lambda$$

12.2.3 Ritz Analysis

Often the use of modal analysis requires a large number of modes to be calculated in order to capture the dynamic characteristics of the structure. This is particularly the case when the horizontal and vertical stiffnesses of the structure are significantly different (while the mass is the same). One way to circumvent this problem is to use Ritz (or Rayleigh-Ritz) analysis which yield approximate eigenvalues. While these are approximate they have some useful characteristics

The eigenvalues (natural frequencies) are upper bounds to the true eigenvalues

The mode shapes are linear combinations of the exact eigenvectors

The number of Ritz vectors required to capture the dynamic characteristics of the structure is

usually significantly less than that required for a proper eigenvalue analysis.

Ritz analysis method

A set of trial vectors based initially on gravity loads in each of the x/y/z directions. The subsequent trial vectors are created from these with the condition that they are orthogonal to the previous vectors. The assumption is that we can get approximations to the eigenvectors by taking a linear combination of the trial vectors.

So for trial vectors

$$\mathbf{X}_m = [\mathbf{x}_1 \quad \mathbf{x}_2 \quad \cdots \quad \mathbf{x}_m]$$

Let

$$\boldsymbol{\varphi} = \mathbf{X}_m \mathbf{s} = \sum_{i=1}^m x_i \mathbf{s}_i$$

and if the approximation to the eigenvalue is λ , the residual associated with the approximating pair $\{\lambda, \boldsymbol{\varphi}\}$ is given by

$$\mathbf{r} = \mathbf{K}\boldsymbol{\varphi} - \lambda \mathbf{M}\boldsymbol{\varphi}$$

The Rayleigh –Ritz method requires the residual vector be orthogonal to each of the trial vectors, so

$$\mathbf{X}_m^T \mathbf{r} = \mathbf{X}_m^T \mathbf{K}\boldsymbol{\varphi} - \lambda \mathbf{X}_m^T \mathbf{M}\boldsymbol{\varphi} = 0$$

Substituting for $\boldsymbol{\varphi}$ from above gives

$$\mathbf{X}_m^T \mathbf{K} \mathbf{X}_m \mathbf{s} - \lambda \mathbf{X}_m^T \mathbf{M} \mathbf{X}_m \mathbf{s} = 0$$

or

$$\mathbf{K}_m \mathbf{s} - \lambda \mathbf{M}_m \mathbf{s} = 0$$

with

$$\mathbf{K}_m = \mathbf{X}_m^T \mathbf{K} \mathbf{X}_m$$

$$\mathbf{M}_m = \mathbf{X}_m^T \mathbf{M} \mathbf{X}_m$$

This eigenproblem is then solved for the eigenpairs $\{\lambda, \mathbf{s}\}$ and then the approximate eigenvectors are evaluated from

$$\boldsymbol{\varphi} = \mathbf{X}_m \mathbf{s} = \sum_{i=1}^m x_i \mathbf{s}_i$$

Ritz trial vectors

The algorithm as applied in a single direction is as follows:

Create a load vector $\mathbf{f}$ corresponding to a gravity load in the direction of interest

Solve for first vector

solve for $\mathbf{X}_i^*$

normalize $\mathbf{M}$

Solve for additional vectors

$$\mathbf{K}\mathbf{X}_i^* = \mathbf{M}_{\text{for}}^{\text{solve}} \mathbf{X}_i^*$$

$$c_j = \mathbf{X}_j^T \mathbf{M}_{\text{for}} \quad j = 1, \dots, i-1$$

$$\mathbf{X}_i = \mathbf{X}_i^* - \text{orthogonalize} \mathbf{M}$$

$$\mathbf{X}_i^T \mathbf{M} \mathbf{X}_i = \text{normalize} \mathbf{M}$$

12.2.4 Buckling

The problem in this case is to determine critical buckling loads (Eulerian buckling load) of the structure. The assumption is that the differential stiffness matrix is a linear function of applied load. The aim of the buckling analysis is to calculate the factor that can be applied to load before the structure buckles. At buckling the determinant of the sum of the elastic stiffness and the critical differential (or geometric) stiffness is zero.

$$|\mathbf{K} + \mathbf{K}_{g,crit}| = 0$$

Using the assumption of differential stiffness a linear function of loads gives

$$\mathbf{K}_{g,crit} = \lambda \mathbf{K}_g$$

so the equation is

$$|\mathbf{K} + \lambda \mathbf{K}_g| = 0$$

and the eigenvalue problem is then

$$\mathbf{K} \boldsymbol{\varphi} = -\mathbf{K}_g \boldsymbol{\varphi} \Lambda$$

12.2.5 Long Term Analysis

Long term analysis is not a different type of analysis as such. Instead it is an analysis where creep is taken into account for concrete materials. A creep coefficient is specified and this is used in the analysis to give effective E and G values for the concrete materials.

$$E_{eff} = \frac{E}{(1 + \phi)}$$

$$G_{eff} = \frac{G}{(1 + \phi)}$$

12.2.6 Model Stability Analysis

When a structural model is ill-conditioned (as reported by the [condition number](#) estimate) it could be a result modeling errors in the model. These errors could be of two types:

- Some elements may not be well connected or could be badly restrained, e.g. beam elements spinning about their axis.
- Some elements very stiff compared with all other elements in the model, e.g. a beam element of

short length but a large section.

To detect such errors, model stability analysis, which is a qualitative analysis intended to reveal the causes of ill conditioning in models, can be useful. The analysis calculates the smallest and largest eigenvalues and corresponding eigenvectors of the stiffness matrix, i.e. it solves the problem

$$\mathbf{Ku} = \lambda$$

for eigenvalues λ and eigenvectors u . For each mode that is requested, element virtual energies are calculated for each element in the model. These are defined as follows.

The virtual strain energy s_e for large eigenpairs where

$$s_e = \mathbf{u}_e^T \mathbf{K}_e \mathbf{u}_e$$

and virtual kinetic energy v_e for small eigenpairs, defined as

$$v_e = \mathbf{u}_e^T \mathbf{u}_e$$

The virtual energies can be plotted onto elements as contours. Typically, for an ill conditioned model, a handful of elements will have large relative values of virtual energies.

- Where the ill conditioning is caused from badly restrained elements, such elements will have large relative virtual kinetic energies .
- If the ill conditioning is from the presence of elements with disproportionately large stiffnesses, then these elements will have large virtual strain.

The analysis also reports, in increasing order, the eigenvalues computed. For the case of badly restrained elements, there is usually a *gap* in the smallest eigenvalues. The number of smallest eigenpairs to be examined is given by the number of eigenvalues between zero and the gap.

The model stability analysis option is based on research published in [\[38\]](#).

See step by step guide for more detail on how to use this tool.

12.3 Applied Displacements

Use of Lagrange multipliers

The basic equations for a static analysis are:

$$\mathbf{Ku} = \mathbf{f}$$

The applied displacements are applied by using Lagrange multipliers. The basic concept is that the structure matrix can be modified to apply a displacement condition

$$\begin{bmatrix} \mathbf{K} & \mathbf{e} \\ \mathbf{e}^T & 0 \end{bmatrix} \begin{Bmatrix} \mathbf{u} \\ \lambda \end{Bmatrix} = \begin{Bmatrix} \mathbf{f} \\ \tilde{\mathbf{u}} \end{Bmatrix}$$

where λ is the vector of Lagrange multipliers, $\tilde{\mathbf{u}}$ are the applied displacements and $\mathbf{e}$ is a matrix whose columns are zero except for the rows representing the applied displacements, where the values will be the components of the applied displacement. E.g. if an applied displacement of (10, 10, 0) is applied this will give components of (0.707, 0.707, 0) in $\mathbf{e}$.

This can be extended for multiple applied displacements as follows

$$\begin{bmatrix} \mathbf{K} & \mathbf{e}_1 & \mathbf{e}_2 \\ \mathbf{e}_1^T & 0 & 0 \\ \mathbf{e}_2^T & 0 & 0 \end{bmatrix} \begin{Bmatrix} \mathbf{u} \\ \lambda_1 \\ \lambda_2 \end{Bmatrix} = \begin{Bmatrix} \mathbf{f} \\ \tilde{\mathbf{u}}_1 \\ \tilde{\mathbf{u}}_2 \end{Bmatrix}$$

or

$$\begin{bmatrix} \mathbf{K} & \mathbf{E} \\ \mathbf{E}^T & 0 \end{bmatrix} \begin{Bmatrix} \mathbf{u} \\ \Lambda \end{Bmatrix} = \begin{Bmatrix} \mathbf{f} \\ \tilde{\mathbf{U}} \end{Bmatrix}$$

The additional degrees of freedom to solve for in the Lagrange multipliers are the forces at the applied displacements and thus can be used to calculate the effects of the displacements on the structure. Expanding the matrix equations gives

$$\mathbf{K}\mathbf{u} + \mathbf{E}\Lambda = \mathbf{f} \quad \mathbf{E}^T\mathbf{u} = \tilde{\mathbf{U}}$$

Thus u can be expressed as

$$\mathbf{u} = \mathbf{K}^{-1}(\mathbf{f} - \mathbf{E}\Lambda)$$

so that

$$(\mathbf{E}^T\mathbf{K}^{-1}\mathbf{f} - \tilde{\mathbf{U}}) = (\mathbf{E}^T\mathbf{K}^{-1}\mathbf{E})\Lambda$$

or

$$\mathbf{f}_m = \mathbf{K}_m\Lambda$$

gives a solution for Λ and then the previous equation can be used to establish u .

12.4 Axes

Axes can be either Cartesian, cylindrical or spherical. The coordinates in these are:

- **Cartesian**—(x, y, z)
- **Cylindrical**—(r, θ, z)
- **Spherical**—(r, θ, ϕ)

An axis is defined by three vectors irrespective of axis type. These define the location and basic orientation. The x axis vector is any vector pointing in the positive x axis direction. The xy plane vector is any vector in the xy plane of the axis that is not parallel with the x axis vector. The axes are then constructed as follows:

$$\mathbf{x} = |\mathbf{x}_{axis}| \quad \mathbf{z} = |\mathbf{x}_{axis} \times \mathbf{xy}_{plane}|$$

12.5 Condition number

As discussed in '[ill conditioning](#)', while solving linear equations of the type

$$\mathbf{K}\mathbf{x} = \mathbf{f}$$

for given loads $\{\mathbf{f}\}$ and stiffness $[\mathbf{K}]$ in (say) linear static analysis, approximations are introduced in the solution because all calculations are carried out in finite precision arithmetic. This becomes important when $[\mathbf{K}]$ is ill-conditioned because there is a possibility of these approximations leading to large errors in the displacements. The extent of these errors can be quantified by the 'condition number' of the stiffness matrix.

The condition number of a matrix (with respect to inversion) measures worst-case of changes in $\{x\}$ corresponding to small changes in $[K]$ or $\{f\}$. It can be calculated using the product of norm of the matrix times the norm of its inverse.

$$\kappa(K) = \|K\| \cdot \|K^{-1}\|$$

where $\|\cdot\|$ is a subordinate matrix norm (see reference[29] for all definitions).

If K is a symmetric matrix, the condition number $\kappa(K)$ can be shown to be the ratio of its maximum and minimum eigenvalues $\lambda_{\max}$ and $\lambda_{\min}$.

$$\kappa(K) = \frac{\lambda_{\max}}{\lambda_{\min}}$$

The minimum value of $\kappa(K)$ is 1 and the maximum value is infinity. If the condition number is small, the computed solution x is reliable (i.e. a reliable approximation to the true solution of $[K]\{x\}=\{f\}$). If the condition number is large, (i.e. if the matrix is almost singular) the results cannot be trusted.

GSA computes a lower bound approximation to the 1-norm condition number of K and this is reported as part of the solver output. This can be used to evaluate the accuracy of the solution both qualitatively and quantitatively. The (qualitative) rule of thumb for accuracy in $\{x\}$ is

$$\text{number of digits of accuracy in } \{x\} = 16 - \log \kappa.$$

In general any stiffness matrix with condition number above 10^{15} can produce results with no accuracy at all. Any results produced from matrices with condition number greater than 10^{10} must be treated with caution.

Where a model is ill-conditioned, Model Stability analysis can help detect the causes of ill conditioning.

For a given condition number, we can also compute the maximum relative error in $\{x\}$. The max. relative error in x is defined as the maximum ratio of norms of error in x to x , i.e.,

$$\text{max relative error} = \frac{\|\Delta x\|}{\|x\|}.$$

Given a matrix K with condition number κ , the maximum relative error in $\{x\}$ when solving $[K]\{x\} = \{f\}$ is

$$\frac{2\varepsilon\kappa}{1 - \varepsilon\kappa},$$

where ε is the constant 'unit-roundoff' and is equal to $1.11\text{e-}16$ for double precision floating point numbers. The maximum relative error is computed and reported as part of solver output. Ideally, this should be small ($\ll 1$), since a small relative error indicates a reliable solution but as $\kappa \rightarrow 1/\varepsilon$, the relative error grows rapidly.

GSA calculates the condition number using Higham and Tisseur's block 1-Norm condition number estimation algorithm[30].

These are reported in the Analysis Details output.

12.6 Constraints

Constraints are a way of setting up mathematical relationships between different parts of the model. Node(s) are constrained to behave in a particular way relative to other node(s). This takes no account of the stiffness so is useful for avoiding potential ill-conditioning problems, but should not be used to over-constrain a model.

Constraints are based on constraint equations. These are a way to constrain a degree of freedom to

move in a prescribed manner relative to other degrees of freedom. Constraint equations define the movement of a slave degree of freedom relative to a list of master degrees of freedom. For example to constrain a node mid-way between 2 other nodes to move as if it is fixed between the nodes would require a set of constraint equations of the form

$$u_{x_s} = 0.5u_{x_{m1}} + 0.5u_{x_{m2}}$$

Note: the constraint equations provide a mathematically convenient way of linking degrees of freedom, which are not necessarily compatible with equilibrium.

More:

[Joints](#)

[Rigid constraints](#)

[Constraint equations](#)

Tied Interface

12.6.1 Joints

Joints (or repeat freedoms) give a convenient way of connecting degrees of freedom without required an element. These simply link the specified degrees of freedom. They are useful for modelling pinned connections where the x, y and z degrees of freedom are repeated.

$$u_{s,i} = u_{m,i}$$

Any load applied to a joint is transferred directly to the master so these constraints should be used with care as moment mismatches can arise due to the load being applied to the structure at a different point to the point of application of the load.

12.6.2 Rigid constraints

Rigid constraints constrain a collection of nodes to move together as a rigid body.

The master node is the node that is retained in the solution. All the other nodes (those in the slave list) are related back to the master node.

$$\varphi_s = \mathbf{T} \varphi_m$$

The degrees of freedom at the slaves that are linked will depend on the type of rigid constraint. The constraint allows the slave nodes to be fixed (where the rotations at the slave node depend on the rotation of the master) or pinned (where the rotations at the slave node are independent of the rotation of the master). The master node is always has the rotations linked to the rest of the structure. The constraint can act in all directions or be restricted to act in a plane (xy, yz or zx) where the nodes are rigidly connected for motion in the plane but are independent for out of plane motion.

The slave master transformations are summarised below:

All directions

$$\begin{bmatrix} 1 & 0 & 0 & 0 & z & -y \\ 0 & 1 & 0 & -z & 0 & x \\ 0 & 0 & 1 & y & -x & 0 \\ 0 & 0 & 0 & \delta & 0 & 0 \\ 0 & 0 & 0 & 0 & \delta & 0 \\ 0 & 0 & 0 & 0 & 0 & \delta \end{bmatrix} \quad \text{where } \delta = 1/0 \text{ if } \textit{fixed/pinned}$$

Plane (xy plane as example)

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & -y \\ 0 & 1 & 0 & 0 & 0 & x \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & \delta \end{bmatrix} \quad \text{where } \delta = 1/0 \text{ if } \textit{fixed/pinned}$$

Plate (z plate as example)

$$\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & y & -x & 0 \\ 0 & 0 & 0 & \delta & 0 & 0 \\ 0 & 0 & 0 & 0 & \delta & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \quad \text{where } \delta = 1/0 \text{ if } \textit{fixed/pinned}$$

Loads applied to slave nodes of a rigid constraint will be correctly transferred to the master degrees of freedom as a force + moment so no spurious moments result.

The constraint equations for a rigid constraint assume small displacements. When large displacements are applied to a rigid constraint the constraint equations no longer apply and the links between slave and master get stretched. This effect can be noticeable in a dynamic analysis where the results are scaled to an artificially large value. When these are scaled to realistic value this error should be insignificant.

12.6.3 Constraint equations

Constraint equations represent the most fundamental way of constraining one node relative to a set of other nodes. Joints and rigid constraints can all be represented as sets of constraint equations. The displacement (or rotation) of the slave node is related to that at the masters

$$u_{s,i} = \sum_m a_m u_{m,j}$$

12.6.4 Tied Interface

A tied interface is used to join dissimilar meshes. The basic definition can use slave nodes or elements connected to a master set of elements.

GSA will establish which nodes and which elements are to be connected and will set up the

appropriate set of constraint equations. For the j^{th} direction

$$u_j = \sum_i a_{ij} u_{ij}$$

For the translations we can establish a set of these connection the x, y and z directions

$$\begin{Bmatrix} u_x \\ u_y \\ u_z \end{Bmatrix} = \sum_i \begin{bmatrix} a_{ix} & 0 & 0 \\ 0 & a_{iy} & 0 \\ 0 & 0 & a_{iz} \end{bmatrix} \begin{Bmatrix} u_{ix} \\ u_{iy} \\ u_{iz} \end{Bmatrix}$$

For a node connected to 2D elements we link the drilling degree of freedom to the displacements of the 2D element nodes so the relationship between slave and masters for rotations in a system local to the 2D element is

$$\begin{Bmatrix} \theta_x \\ \theta_y \\ \theta_z \end{Bmatrix} = \sum_i \left(\begin{bmatrix} a_{ix} & 0 & 0 \\ 0 & a_{iy} & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} \theta_{ix} \\ \theta_{iy} \\ \theta_{iz} \end{Bmatrix} + \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ b_{ix} & b_{iy} & 0 \end{bmatrix} \begin{Bmatrix} u_{ix} \\ u_{iy} \\ u_{iz} \end{Bmatrix} \right)$$

or

$$\begin{Bmatrix} \theta_x \\ \theta_y \\ \theta_z \end{Bmatrix} = \sum_i \left(\mathbf{A}_i \begin{Bmatrix} \theta_{ix} \\ \theta_{iy} \\ \theta_{iz} \end{Bmatrix} + \mathbf{B}_i \begin{Bmatrix} u_{ix} \\ u_{iy} \\ u_{iz} \end{Bmatrix} \right)$$

This needs to be resolved into global and/or nodal directions using the 2D element direction cosines and the nodal direction cosines giving

$$\begin{Bmatrix} \theta_{Nx} \\ \theta_{Ny} \\ \theta_{Nz} \end{Bmatrix} = \sum_i \left(\begin{bmatrix} \mathbf{N}_{DC}^T \mathbf{E}_{DC} \mathbf{A}_i \mathbf{E}_{DC}^T \mathbf{N}_{i,DC} \end{bmatrix} \begin{Bmatrix} \theta_{Nix} \\ \theta_{Niy} \\ \theta_{Niz} \end{Bmatrix} + \begin{bmatrix} \mathbf{N}_{DC}^T \mathbf{E}_{DC} \mathbf{B}_i \mathbf{E}_{DC}^T \mathbf{N}_{i,DC} \end{bmatrix} \begin{Bmatrix} u_{Nix} \\ u_{Niy} \\ u_{Niz} \end{Bmatrix} \right)$$

or in more compact form

$$\boldsymbol{\theta}_G = \sum_i \left(\mathbf{N}^T \mathbf{E}_i \mathbf{A}_i \mathbf{E}_i^T \mathbf{N}_i \boldsymbol{\theta}_{Ni} + \mathbf{N}^T \mathbf{E}_i \mathbf{B}_i \mathbf{E}_i^T \mathbf{N}_i \mathbf{u}_{Gi} \right)$$

12.7 Creep Analysis

GSA offers a simplified approach to creep analysis. This is available by using analysis stages. In a stage definition it is possible to specify the creep coefficient for concrete materials.

This option allows the concrete properties to be modified for a long term analysis as follows:

$$E \rightarrow \frac{E}{(1 + \phi)}, \quad G \rightarrow \frac{G}{(1 + \phi)}$$

Only concrete materials are considered for creep.

12.8 Direction cosines

The direction cosines describe the relationship between different axis systems, typically how a local axis relates to a global axis. The direction cosine array is assembled from vectors in the local axis directions as follows:

$$\mathbf{T}_{DC} = [\mathbf{x} \mid \mathbf{y} \mid \mathbf{z}]$$

The transformations for vectors and tensors are then:

$$\begin{aligned} \mathbf{v}_l &= \mathbf{T}_{DC}^T \mathbf{v}_g & \mathbf{t}_l &= \mathbf{T}_{DC}^T \mathbf{t}_g \mathbf{T}_{DC} \\ \mathbf{v}_g &= \mathbf{T}_{DC} \mathbf{v}_l & \mathbf{t}_g &= \mathbf{T}_{DC} \mathbf{t}_l \mathbf{T}_{DC}^T \end{aligned}$$

12.9 Dynamic and Mass Results

[Dynamic Results](#)

[Axis transformations](#)

[Mass Details](#)

12.9.1 Dynamic Results

In dynamic problems the most basic results are modes. These are described by a frequency and shape. The mode shape is stored as the displacement results. (Not just displacements but any of the results can be thought of as mode shapes). The modal displacements are of arbitrary magnitude, so some choice has to be made about the scaling of the values. The two main schemes are to normalize so that the modal mass (see below) is unity or so that the maximum displacement is unity. In GSS the latter option is used by default but the user can select to normalize the modal mass instead.

From the modal results some other information can be derived which characterises the mode. These are described below.

Modal mass

The modal mass for mode i is defined as

$$\hat{m}_i = \varphi_i^T \mathbf{M} \varphi_i$$

The more the mass of the structure that is mobilised by this mode the greater the modal mass.

Modal stiffness

The modal stiffness for mode i is defined as

$$\hat{k}_i = \varphi_i^T \mathbf{K} \varphi_i$$

The modal mass and modal stiffness are related through the frequency

$$2\pi f_i = \omega_i = \sqrt{\hat{k}_i / \hat{m}_i}$$

Modal damping

The modal damping is calculated from

$$\xi_i = k \frac{\sum_e \xi_e \varphi_e^T \mathbf{K}_e \varphi_e}{\varphi_e^T \mathbf{K} \varphi_e} + m \frac{\sum_e \xi_e \varphi_e^T \mathbf{M}_e \varphi_e}{\varphi_e^T \mathbf{M} \varphi_e}$$

where the factor k is in the range 0:1 and

$$m = 1 - k$$

Participation factors

The participation factors are defined for each mode i in direction j as

$$\Gamma_{i,j} = \frac{\varphi_i^T \mathbf{M} \varphi_{\text{rigid},j}}{\varphi_i^T \mathbf{M} \varphi_i}$$

where

$$\varphi_{\text{rigid},j}$$

is a vector corresponding to unit (dimensionless) rigid body motion in the j direction. Thus the translational participation factors have dimensions of 1/length and the rotational participation factors are dimensionless.

The modal acceleration is then defined in terms of the participation factor and the spectral acceleration as

$$\ddot{q}_{i,j} = \Gamma_{i,j} a_{\text{spectral}_i}$$

and the actual acceleration of a node is then

$$\ddot{\mathbf{x}} = \ddot{\mathbf{q}} \boldsymbol{\phi}$$

Effective mass

The effective mass is related to the participation factors and the modal mass. It is defined for direction i as

$$m_{\text{eff},i,j} = \frac{(\varphi_i^T \mathbf{M} \varphi_{\text{rigid},j})^2}{\varphi_i^T \mathbf{M} \varphi_i}$$

The translational effective masses have dimensions of mass while the rotational effective masses have dimensions of inertia.

12.9.2 Axis transformations

There are two transformations of interest. The transformation of a point and the transformation of a vector, either from global to a user defined or local axis or from the user defined to global axis.

For axis defined local axis directions $\mathbf{x}$, $\mathbf{y}$ and $\mathbf{z}$, the transformation to rotate into the local axis is

$$\mathbf{T} = \begin{bmatrix} x_x & x_y & x_z \\ y_x & y_y & y_z \\ z_x & z_y & z_z \end{bmatrix}$$

with the origin, $\mathbf{o}$, of the local axis the transformation of a point, $\mathbf{p}$, is

$$\mathbf{p}' = \mathbf{T}(\mathbf{p} - \mathbf{o})$$

If the local axis is cylindrical then the transformed point $\mathbf{p}'$ is converted to cylindrical coordinates where

$$r = \sqrt{p_x^2 + p_y^2} \quad \theta = \cos^{-1}\left(\frac{p_x}{r}\right) \quad z = p_z$$

or for a spherical axis the transformed point $\mathbf{p}'$ is converted to spherical coordinates where

$$r = \sqrt{p_x^2 + p_y^2 + p_z^2} \quad \theta = \cos^{-1}\left(\frac{p_z}{r}\right) \quad \phi = \tan^{-1}\left(\frac{p_y}{p_x}\right)$$

The transformation of a vector, $\mathbf{v}$, uses the same transformation matrix $\mathbf{T}$ plus a further rotation, subscript $\mathbf{r}$, relating to the location of the vector, $\mathbf{p}$, in the local axis system. For a Cartesian axis the rotation is

$$\mathbf{T}_r = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

for a cylindrical axis it is

$$\mathbf{T}_r = \begin{bmatrix} \cos \theta & -\sin \theta & 0 \\ \sin \theta & \cos \theta & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

and for a spherical axis it is

$$\mathbf{T}_r = \begin{bmatrix} \sin \theta \cos \phi & \sin \theta \sin \phi & \cos \theta \\ \cos \theta \cos \phi & \cos \theta \sin \phi & -\sin \theta \\ -\sin \phi & \cos \phi & 0 \end{bmatrix}$$

This gives the transformed vector as

$$\mathbf{v}' = \mathbf{T}_r \cdot \mathbf{T} \cdot \mathbf{v}$$

12.9.3 Mass Details

The model can be thought of as a collection of nodes with mass (a scalar) and inertia (a tensor). The total mass of the system is then a summation over all the nodes:

$$m = \sum_i m_i$$

and the centre of mass is:

$$\mathbf{c} = \frac{\sum_i m_i \mathbf{x}_i}{\sum_i m}$$

The inertia has two components: the inertia of the nodes and the contribution from the offset of the mass of individual nodes. Summing the terms over all the nodes gives the inertia tensor:

$$\hat{\mathbf{I}}_m = \begin{bmatrix} \sum_i \{I_{xx,i} + m_i(y_i^2 + z_i^2)\} & \sum_i \{I_{xy,i} - m_i x_i y_i\} & \sum_i \{I_{xz,i} - m_i x_i z_i\} \\ \sum_i \{I_{yx,i} - m_i x_i y_i\} & \sum_i \{I_{yy,i} + m_i(x_i^2 + z_i^2)\} & \sum_i \{I_{yz,i} - m_i y_i z_i\} \\ \sum_i \{I_{zx,i} - m_i x_i z_i\} & \sum_i \{I_{zy,i} - m_i y_i z_i\} & \sum_i \{I_{zz,i} + m_i(x_i^2 + y_i^2)\} \end{bmatrix}$$

Note - in most cases the inertia of the nodes can be ignored, in which case this reduces to

$$\hat{\mathbf{I}}_m = \begin{bmatrix} \sum_i m_i(y_i^2 + z_i^2) & -\sum_i m_i x_i y_i & -\sum_i m_i x_i z_i \\ -\sum_i m_i x_i y_i & \sum_i m_i(x_i^2 + z_i^2) & -\sum_i m_i y_i z_i \\ -\sum_i m_i x_i z_i & -\sum_i m_i y_i z_i & \sum_i m_i(x_i^2 + y_i^2) \end{bmatrix}$$

The minimum inertia is about the centre of mass

$$\mathbf{c} = (x_c, y_c, z_c)$$

so the inertia defined above can be transformed to the centre of mass to give

$$\mathbf{I}_m = \hat{\mathbf{I}}_m - \begin{bmatrix} m(y_c^2 + z_c^2) & -mx_c y_c & -mx_c z_c \\ -mx_c y_c & m(x_c^2 + z_c^2) & -my_c z_c \\ -mx_c z_c & -my_c z_c & m(x_c^2 + y_c^2) \end{bmatrix}$$

12.10 Eigensolver

The standard eigenproblem can be described as

$$[K]\{\phi\} = \lambda\{\phi\}$$

where $[K]$ is the stiffness matrix of order n . The eigenpair is denoted as

$$(\lambda_i, \phi_i)$$

where the eigenvalues are ordered according to the magnitudes. The solution for p eigenpairs can be written

$$[K][\Phi] = [\Phi][\Lambda]$$

where $[\Phi]$ is an $n \times p$ matrix with the columns equal to the p eigenvectors and $[\Lambda]$ is a $p \times p$ diagonal matrix with the corresponding eigenvalues.

In most engineering applications it is the generalized eigenproblem that is of interest. For dynamics problem this can be written as

$$[K]\{\phi\} = \lambda[M]\{\phi\}$$

where $[M]$ is the mass matrix, or in buckling problems as

$$[K]\{\phi\} = \lambda[K_g]\{\phi\}$$

In both these cases we are interested in the eigenvalues (square of the natural frequencies and load factors respectively) and the eigenvectors (mode shapes).

The solution of eigenproblems is a large topic covered in many texts. The method used in GSS is the subspace iteration method. This method was developed by Bathe and consists of the following

1. For $k = 1, 2, \dots$, iterate

$$[K][\bar{X}_{k+1}] = [M][X_k]$$

2. Find the projections of the operators $[K]$ and $[M]$

$$[K_{k+1}] = [\bar{X}_{k+1}^T][K][\bar{X}_{k+1}] \quad [M_{k+1}] = [\bar{X}_{k+1}^T][M][\bar{X}_{k+1}]$$

3. Solve for the eigensystem of the projected system

$$[K_{k+1}][Q_{k+1}] = [M_{k+1}][Q_{k+1}][\Lambda_{k+1}]$$

4. Find an improved approximation to the eigenvalues

$$[X_{k+1}] = [\bar{X}_{k+1}][Q_{k+1}]$$

5. Then, provided that the vectors in $[X_1]$ are not orthogonal to one of the required eigenvectors

$$[\Lambda_{k+1}] \rightarrow [\Lambda] \text{ and } [\bar{X}_{k+1}] \rightarrow [\Phi] \text{ as } k \rightarrow \infty$$

The solution of the projected eigenproblem is carried out using the Jacobi method.

The approach above calculates the eigenvalues (frequencies or load factors) closest to zero. At times it is useful to be able to calculate eigenvalues in a different part of the spectrum. This can be done by introducing a shift into the calculation. The generalized eigenproblem can be rewritten

$$[K] - \mu[M] = (\lambda - \mu)[M]$$

where μ is a shift. This then gives eigenvalues in the vicinity of μ .

12.11 Elements

The element types are:

- [Bar, Tie and Strut](#) (2 nodes)
- [Beam](#) (2 nodes)
- [Spring](#) (2 nodes)
- [Grounded Spring](#) (1 node)
- [Lumped mass](#) (1 node)
- [Linear quad](#) (4 nodes)
- [Parabolic quad](#) (8 nodes)
- [Linear triangle](#) (3 nodes)
- [Parabolic triangle](#) (6 nodes)
- Linear tetra (4 nodes)
- Parabolic tetra (10 nodes)
- Linear wedge (6 nodes)
- Parabolic wedge (15 nodes)

- Linear brick (8 nodes)
- Parabolic brick (20 nodes)

Bar and Beam elements refer to the same beam property sets. Quad4, Quad8, Tri3 and Tri6 elements refer to the same 2D property sets. All other element types have unique property sets.

More:

[Topology](#)

[Local axes](#)

[Bar elements](#)

[Beam elements](#)

[Spring elements](#)

[Mass elements](#)

[2D isoparametric elements](#)

[Link elements](#)

[Releases](#)

[Offsets](#)

[Transformations](#)

[Mass Options](#)

12.11.1 Topology

The element's topology with reference to a plan view on the element is:

- **for 2D elements**—anticlockwise around the corner nodes, followed by anticlockwise around the mid-side nodes.
- **for 3D elements**—anticlockwise around the base corner nodes, followed by anticlockwise around the top corner nodes, followed by anticlockwise around the base mid-side nodes, anticlockwise around the side mid-side nodes and finally anticlockwise around the top mid-side nodes.

12.11.2 Local axes

The local axes are based on pseudo nodes (ie the flexible part of the element). For a 2 node element (beam, bar or spring) a vector $\mathbf{u}$ is defined from the first node to the second node. The local y axis is defined by either a third node and/or an orientation angle. If there is no orientation angle, a vector $\mathbf{v}$ is defined from the first node to the third node. For a 2D element (linear or parabolic) a vector $\mathbf{u}$ is defined from the first corner node to the second corner node, and $\mathbf{v}$ is defined from the first corner node to the last corner node. The local axes can then be calculated using:

$$x = \frac{\mathbf{u}}{\|\mathbf{u}\|} \quad z = \frac{\mathbf{x} \times \mathbf{v}}{\|\mathbf{x} \times \mathbf{v}\|} \quad y = z \times x$$

If a beam has an orientation angle defined and no third node, the local x axis is defined by the vector $\mathbf{u}$ and the local z axis is defined by a rotation of the z axis by the orientation angle from the global z axis, or in the case of a vertical element a rotation of the y axis by the orientation angle from the global y axis. If a third node is defined the local axes as defined above are rotated by the orientation angle about the local x axis.

12.11.3 Bar elements

Bar elements are considered to transfer axial loads only. The stiffness matrix for a bar element is therefore

$$\mathbf{K}_e = \begin{bmatrix} \frac{AE}{l} & 0 & 0 & -\frac{AE}{l} & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ \frac{AE}{l} & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

where A , E and l are the area, the Young's modulus and element length respectively and the degrees of freedom are the axial degrees of freedom at the nodes.

The mass matrix for the bar is

$$\mathbf{M}_e = \begin{bmatrix} \frac{\rho Al}{3} & 0 & 0 & \frac{\rho Al}{6} & 0 & 0 \\ 0 & \frac{\rho Al}{3} & 0 & 0 & \frac{\rho Al}{6} & 0 \\ 0 & 0 & \frac{\rho Al}{3} & 0 & 0 & \frac{\rho Al}{6} \\ 0 & 0 & 0 & \frac{\rho Al}{3} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{\rho Al}{3} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{\rho Al}{3} \end{bmatrix}$$

where ρ , A and l are the density, the area and the element length respectively.

The geometric stiffness is

$$\mathbf{K}_{g,e} = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & \frac{F_x}{l} & 0 & 0 & -\frac{F_x}{l} & 0 \\ 0 & 0 & \frac{F_x}{l} & 0 & 0 & -\frac{F_x}{l} \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{F_x}{l} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{F_x}{l} & 0 \end{bmatrix}$$

where l is the element length and F is the axial force in the element.

12.11.4 Beam elements

Beam elements have six degrees of freedom at nodes, and can either include or exclude shear deformations. The shear effects are ignored if the shear factor is set to zero.

The stiffness matrix for a beam, ignoring shear effects, is

$$\mathbf{K}_e = \begin{bmatrix} \frac{AE}{l} & 0 & 0 & 0 & 0 & 0 & -\frac{AE}{l} & 0 & 0 & 0 & 0 & 0 \\ 0 & \frac{12EI_{xx}}{l^3} & 0 & 0 & 0 & \frac{6EI_{xx}}{l^2} & 0 & -\frac{12EI_{xx}}{l^3} & 0 & 0 & 0 & \frac{6EI_{xx}}{l^2} \\ 0 & 0 & \frac{12EI_{yy}}{l^3} & 0 & -\frac{6EI_{yy}}{l^2} & 0 & 0 & 0 & -\frac{12EI_{yy}}{l^3} & 0 & -\frac{6EI_{yy}}{l^2} & 0 \\ 0 & 0 & 0 & \frac{GJ}{l} & 0 & 0 & 0 & 0 & 0 & \frac{GJ}{l} & 0 & 0 \\ 0 & 0 & -\frac{6EI_{yy}}{l^2} & 0 & \frac{4EI_{yy}}{l} & 0 & 0 & 0 & \frac{6EI_{yy}}{l^2} & 0 & \frac{2EI_{yy}}{l} & 0 \\ 0 & \frac{6EI_{xx}}{l^2} & 0 & 0 & 0 & \frac{4EI_{xx}}{l} & 0 & -\frac{6EI_{xx}}{l^2} & 0 & 0 & 0 & \frac{2EI_{xx}}{l} \\ -\frac{AE}{l} & 0 & 0 & 0 & 0 & 0 & \frac{AE}{l} & 0 & 0 & 0 & 0 & 0 \\ 0 & -\frac{12EI_{xx}}{l^3} & 0 & 0 & 0 & 0 & 0 & \frac{12EI_{xx}}{l^3} & 0 & 0 & 0 & -\frac{6EI_{xx}}{l^2} \\ 0 & 0 & \frac{12EI_{yy}}{l^3} & 0 & -\frac{6EI_{yy}}{l^2} & 0 & 0 & 0 & \frac{12EI_{yy}}{l^3} & 0 & \frac{6EI_{yy}}{l^2} & 0 \\ 0 & 0 & 0 & \frac{GJ}{l} & 0 & 0 & 0 & 0 & 0 & \frac{GJ}{l} & 0 & 0 \\ 0 & \frac{2EI_{yy}}{l} & 0 & 0 & \frac{4EI_{yy}}{l} & 0 & 0 & 0 & 0 & 0 & \frac{4EI_{yy}}{l} & 0 \\ 0 & \frac{2EI_{xx}}{l} & 0 & 0 & 0 & \frac{4EI_{xx}}{l} & 0 & 0 & 0 & 0 & 0 & \frac{4EI_{xx}}{l} \end{bmatrix}$$

where A , I_{yy} , I_{zz} , J , E , G and l are the area, the length, the second moments of area, the torsion constant, the Young's modulus, the shear modulus and the length respectively. If shear effects are to be included the bending terms are modified by a factor including the shear area factors

$$\alpha = 12 \frac{EI_{ii}}{l^2 GA_s} \quad A_s = Ak_j$$

as follows

$$\begin{aligned} \frac{2EI}{l} &\rightarrow \frac{2-\alpha}{1+\alpha} \frac{EI}{l} & \frac{6EI}{l^2} &\rightarrow \frac{6}{1+\alpha} \frac{EI}{l^2} \\ \frac{4EI}{l} &\rightarrow \frac{4+\alpha}{1+\alpha} \frac{EI}{l} & \frac{12EI}{l^3} &\rightarrow \frac{12}{1+\alpha} \frac{EI}{l^3} \end{aligned}$$

The above is true for symmetric sections, but when the section is non-symmetric there are deflections orthogonal to the loading. If these effects are to be taken into account the local ("yy" and "zz") directions must be replaced with the principal directions. In this case the stiffness matrix is calculated using the principal 2nd moments of area and then a rotation of the stiffness is applied to transform to the element local directions.

The consistent mass matrix for a beam, derived from the element shape functions is

$$\mathbf{M}_e = \frac{\rho A l}{420} \begin{bmatrix} 140 & 0 & 0 & 0 & 0 & 0 & 70 & 0 & 0 & 0 & 0 & 0 \\ & 156 & 0 & 0 & 0 & 22l & 0 & 54 & 0 & 0 & 0 & -13l \\ & & 156 & 0 & -22l & 0 & 0 & 0 & 54 & 0 & 13l & 0 \\ & & & 140 \frac{J}{A} & 0 & 0 & 0 & 0 & 0 & 70 \frac{J}{A} & 0 & 0 \\ & & & & 4l^2 & 0 & 0 & 0 & -13l & 0 & -3l^2 & 0 \\ & & & & & 4l^2 & 0 & 13l & 0 & 0 & 0 & -3l^2 \\ & & & & & & 140 & 0 & 0 & 0 & 0 & 0 \\ & & & & & & & 156 & 0 & 0 & 0 & -22l \\ & & & & & & & & 156 & 0 & 22l & 0 \\ & & & & & & & & & 140 \frac{J}{A} & 0 & 0 \\ & & & & & & & & & & 4l^2 & 0 \\ & & & & & & & & & & & 4l^2 \end{bmatrix}$$

where

$$J = I_{yy} + I_{zz}$$

and ρ , A and l are density, area and length respectively.

In many cases it is simpler to use a lumped mass approach where all the mass is assumed to be lumped at the nodes. The lumped mass matrix is

$$\mathbf{M}_e = \frac{\rho A l}{2} \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & & & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & & & & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & & & & & 1 & 0 & 0 & 0 & 0 & 0 \\ & & & & & & & 1 & 0 & 0 & 0 & 0 \\ & & & & & & & & 1 & 0 & 0 & 0 \\ & & & & & & & & & 0 & 0 & 0 \\ & & & & & & & & & & 0 & 0 \\ & & & & & & & & & & & 0 \end{bmatrix}$$

When the consistent mass matrix is used the errors in the axial terms are reduced if the average of the consistent and lumped values are used for these terms. This results in the 140 and 70 values being replaced by 175 and 35. These modified values are used in GSA.

The geometric stiffness matrix for a beam is based on the loads on the beam. These are

$$\mathbf{K}_{g,e} = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & \frac{6F_x}{5l} & 0 & \frac{M_{y1}}{l} & 0 & \frac{F_x}{10} & 0 & -\frac{6F_x}{5l} & 0 & \frac{M_{y2}}{l} & 0 & \frac{F_x}{10} \\ & & \frac{6F_x}{5l} & \frac{M_{z1}}{l} & -\frac{F_x}{10} & 0 & 0 & 0 & -\frac{6F_x}{5l} & \frac{M_{z2}}{l} & -\frac{F_x}{10} & 0 \\ & & & \frac{F_x I_{xx}}{Al} & -\frac{V_y l}{6} & -\frac{V_z l}{6} & 0 & -\frac{M_{y1}}{l} & -\frac{M_{z1}}{l} & -\frac{F_x I_{xx}}{Al} & \frac{V_y l}{6} & \frac{V_z l}{6} \\ & & & & \frac{2F_x l}{15} & 0 & 0 & 0 & \frac{F_x}{10} & \frac{V_y l}{6} & -\frac{F_x l}{30} & 0 \\ & & & & & \frac{2F_x l}{15} & 0 & -\frac{F_x}{10} & 0 & \frac{V_z l}{6} & 0 & -\frac{F_x l}{30} \\ & & & & & & 0 & 0 & 0 & 0 & 0 & 0 \\ & & & & & & & \frac{6F_x}{5l} & 0 & -\frac{M_{y2}}{l} & 0 & -\frac{F_x}{10} \\ & & & & & & & & \frac{6F_x}{5l} & -\frac{M_{z2}}{l} & -\frac{F_x}{10} & 0 \\ & & & & & & & & & \frac{F_x I_{xx}}{Al} & \frac{V_y l}{6} & \frac{V_z l}{6} \\ & & & & & & & & & & \frac{2F_x l}{15} & 0 \\ & & & & & & & & & & & \frac{2F_x l}{15} \end{bmatrix}$$

where l is the length, A is the area, and

$$I_{xx} = I_{yy} + I_{zz}$$

12.11.5 Spring elements

Spring elements have either translational or rotational stiffness, and depending on their type they will have different actions:

- **axial**—axial stiffness only
- **axial + shear**—axial stiffness + shear stiffness
- **general translational**—stiffness in all three translational directions
- **torsional**—torsional stiffness only
- **torsion + bending**—torsional stiffness + bending stiffness
- **general rotational**—stiffness in all three rotational directions

Axial / torsional springs only have stiffness in the local x/xx direction.

$$k_x = k_{axial} \quad k_y = 0 \quad k_z = 0$$

Axial + shear / torsion + bending springs have an axial/torsional stiffness and also a shear/bending stiffness which applies in both the local y and z directions.

$$k_x = k_{axial} \quad k_y = k_{shear} \quad k_z = k_{shear}$$

The general spring has three stiffnesses defined in local x, y and z directions.

$$k_x = k_{axial} \quad k_y = k_{y\ shear} \quad k_z = k_{z\ shear}$$

The local directions for a spring element are defined as follows. The axis is used to define local directions. If the axis is "local" the local x direction is from node 1 to node 2 of the flexible part of the element. The local y and z are then defined in an analogous manner to a beam element. If the spring has zero length the local directions default to global.

$$\mathbf{K}_e = \begin{bmatrix} k_x & 0 & 0 & -k_x & 0 & 0 \\ & k_y & 0 & 0 & -k_y & 0 \\ & & k_z & 0 & 0 & -k_z \\ & & & k_x & 0 & 0 \\ & & & & k_y & 0 \\ & & & & & k_z \end{bmatrix}$$

12.11.6 Mass elements

Mass elements can have only mass and inertia. In the normal case the mass matrix is

$$\mathbf{M}_e = \begin{bmatrix} m & & \\ & m & \\ & & m \end{bmatrix}$$

but if mass modifiers are included the mass matrix becomes

$$\mathbf{M}_e = \begin{bmatrix} m_x & & \\ & m_y & \\ & & m_z \end{bmatrix}$$

Inertia is a tensor quantity with the following terms

$$\mathbf{I}_e = \begin{bmatrix} I_{xx} & I_{xy} & I_{xz} \\ & I_{yy} & I_{yz} \\ & & I_{zz} \end{bmatrix}$$

where the terms in the inertia tensor are defined as

$$\begin{aligned} I_{xx} &= \int_V \rho [(y - y_a)^2 + (z - z_a)^2] dV \\ I_{yy} &= \int_V \rho [(z - z_a)^2 + (x - x_a)^2] dV \\ I_{zz} &= \int_V \rho [(x - x_a)^2 + (y - y_a)^2] dV \\ I_{xy} &= I_{yx} = \int_V \rho (x - x_a)(y - y_a) dV \\ I_{yz} &= I_{zy} = \int_V \rho (y - y_a)(z - z_a) dV \\ I_{zx} &= I_{xz} = \int_V \rho (z - z_a)(x - x_a) dV \end{aligned}$$

where (x_a, y_a, z_a) are the coordinates of the centre of mass. If non-zero values are specified for the off-diagonal terms, it is important that these are consistent with the diagonal terms. If this is not done the principal inertia values can become negative. The inertia matrix is never modified for directions.

12.11.7 2D elements

The formulation of 2D plate and shell elements considers both in-plane and transverse (out-of-plane) displacements. Following Mindlin-Reissner plate theory, in addition to the bending strains we consider the effect of transverse shear strain in our complete expression for the element strain

$$\begin{Bmatrix} \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial w}{\partial x} - \beta_x \\ \frac{\partial w}{\partial y} - \beta_y \end{Bmatrix}$$

where w is the out-of-plane displacement and β is introduced as an independent variable to express the section rotations (i.e. rotations of the transverse normals about the local x and y axes).

We can define separate material matrices C_b and C_s that relate stress and strain for the pure bending and shear strains respectively and so the pure bending moments and shear forces can be written

$$\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xy} \end{Bmatrix} = -z \mathbf{C}_b \begin{Bmatrix} \frac{\partial \beta_x}{\partial x} \\ \frac{\partial \beta_y}{\partial y} \\ \frac{\partial \beta_x}{\partial y} + \frac{\partial \beta_y}{\partial x} \end{Bmatrix}$$

and

$$\begin{Bmatrix} \tau_{xz} \\ \tau_{yz} \end{Bmatrix} = \mathbf{C}_s \begin{Bmatrix} \frac{\partial w}{\partial x} - \beta_x \\ \frac{\partial w}{\partial y} - \beta_y \end{Bmatrix}$$

respectively.

In this way we can then express the local element stiffness matrix as a summation of the in-plane and out-of-plane stiffness's

$$\mathbf{K} = \int_V (\mathbf{B}_\kappa^T \mathbf{C}_b \mathbf{B}_\kappa + \mathbf{B}_\gamma^T \mathbf{C}_s \mathbf{B}_\gamma) dV$$

where B_κ and B_γ represent the strain-displacement transforms for the bending and shear components respectively.

While brief, this outlines the basic approach to the Mindlin-Reissner 2D element stiffness formulation. In GSA we label this concisely as the Mindlin formulation.

12.11.7.1 2D isoparametric elements

An irregularly shaped continuum defined by a boundary and areas of loading has to be subdivided into a mesh of finite elements. The size and shape of the element is chosen so that the approximate stiffness implied by the finite element is close enough to the actual stiffness of the continuum in that region. Thus the finite element size and shape will be determined by the type of element being used, the shape of the boundary if in a region close to the boundary, the loading applied and the experience of the user.

Element stiffness

To generate a stiffness matrix for a curvilinear quadrilateral or triangular element a new approach must be used. Most finite element codes used an approach based on **isoparametric** or similar elements.

In an **isoparametric** element the element displacements are interpolated in the same way as the geometry, eg a plane stress element. In a **superparametric** or **degenerate isoparametric** element the interpolation on the geometry is of a higher order than the interpolation of the displacements, eg a plate element. In a **subparametric** element the interpolation of the geometry is of a lower order than the interpolation of the displacements, e.g. an eight noded straight sided quadrilateral element, where a different geometric interpolation function is used for the geometry from that for the displacements. The term **isoparametric** is often used as a general term to cover all these element types.

For a plane stress problem, we can establish a material matrix $\mathbf{C}$ that relates stress and strain. The displacements in a local coordinate system are

$$\mathbf{u} = \{u, v\}$$

the strains are

$$\boldsymbol{\epsilon} = \{\epsilon_{xx}, \epsilon_{yy}, \gamma_{xy}\}$$

and the stresses are

$$\boldsymbol{\sigma} = \{\sigma_{xx}, \sigma_{yy}, \tau_{xy}\}$$

For an elastic-isotropic material the material matrix is

$$\mathbf{C} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}$$

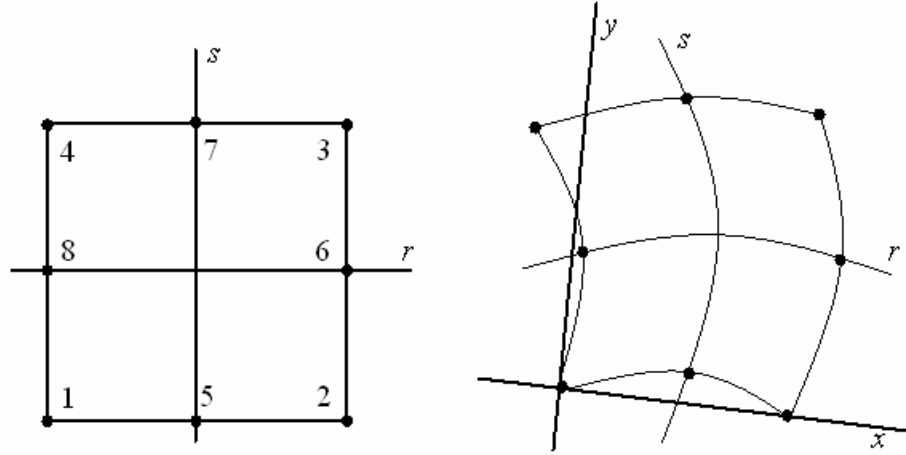
where E , ν are the Young's modulus and Poisson's ratio respectively.

Note that there is an out of plane strain (ϵ_{zz}), which we can ignore as it plays no part in the element formulation.

The strains are defined in terms of the displacements as

$$\epsilon_{xx} = \frac{\partial u}{\partial x} \quad \epsilon_{yy} = \frac{\partial v}{\partial y} \quad \gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}$$

The simplest elements to consider are the 4 noded and 8 noded quadrilateral elements, of which the 4 noded element can be considered a simplification of the 8 noded element. A typical 8 noded element is shown below. The element has an arbitrary local coordinate system based on the location of the nodes and the element property axis (x, y), and a natural (curvilinear) coordinate system (r, s) based on the topology of the nodes.



Natural and local coordinate systems

We can set up interpolation functions to interpolate the geometry as follows

$$x = \sum h_i x_i \quad y = \sum h_i y_i$$

where the h_i are the interpolation functions defined as follows

$$\begin{aligned} h_1 &= \frac{1}{4}(1+r)(1+s) - \frac{1}{2}[h_5 + h_8] & h_5 &= \frac{1}{2}(1-r^2)(1+s) \\ h_2 &= \frac{1}{4}(1-r)(1+s) - \frac{1}{2}[h_6 + h_5] & h_6 &= \frac{1}{2}(1-s^2)(1-r) \\ h_3 &= \frac{1}{4}(1-r)(1-s) - \frac{1}{2}[h_7 + h_6] & h_7 &= \frac{1}{2}(1-r^2)(1-s) \\ h_4 &= \frac{1}{4}(1+r)(1-s) - \frac{1}{2}[h_8 + h_7] & h_8 &= \frac{1}{2}(1-s^2)(1+r) \end{aligned}$$

for an 8 noded quadrilateral element and the first four terms without the terms [...] relate to 4 noded quadrilateral element. These interpolation functions are chosen so that

$$\mathbf{h}_i = \begin{Bmatrix} 1 \\ 0 \end{Bmatrix} \text{ at node } \begin{Bmatrix} i \\ j \end{Bmatrix}$$

As the elements are isoparametric we use the same interpolation function for the displacements so the displacement in the element is related to the nodal displacements by

$$u = \sum \mathbf{h}_i^T \mathbf{u}_i \quad v = \sum \mathbf{h}_i^T \mathbf{v}_i$$

To evaluate the stiffness matrix we need the strain-displacement transformation matrix. The element displacements are obtained in terms of derivatives of the element displacements with respect to the local coordinate system (x, y) . Because the elements displacements are in the natural coordinate system (r, s) we need to relate the derivatives in the local coordinate system to those in the natural coordinate system. We can write an equation for the derivative with respect to x in terms of derivatives with respect to r and s

$$\frac{\partial}{\partial x} = \frac{\partial}{\partial r} \frac{\partial r}{\partial x} + \frac{\partial}{\partial s} \frac{\partial s}{\partial x}$$

to establish these derivatives we use the chain rule to set up the following relationship

$$\begin{bmatrix} \frac{\partial}{\partial r} \\ \frac{\partial}{\partial s} \end{bmatrix} = \begin{bmatrix} \frac{\partial x}{\partial r} & \frac{\partial y}{\partial r} \\ \frac{\partial x}{\partial s} & \frac{\partial y}{\partial s} \end{bmatrix} \begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{bmatrix}$$

or in matrix notation

$$\frac{\partial}{\partial \mathbf{r}} = \mathbf{J} \frac{\partial}{\partial \mathbf{x}}$$

where $\mathbf{J}$ is the Jacobian operator relating natural coordinate derivatives to local coordinate derivatives. Given that we know x and y in terms of the interpolation functions the Jacobian operator is easily found

$$\frac{\partial}{\partial \mathbf{x}} = \mathbf{J}^{-1} \frac{\partial}{\partial \mathbf{r}}$$

This requires that the inverse of the Jacobian exist, which requires that there is a one to one correspondence between natural and local coordinates. This will be the case provided the element is not grossly distorted from a square and that it does not fold back on itself.

We can then evaluate

$$\frac{\partial u}{\partial x}, \frac{\partial u}{\partial y}, \frac{\partial v}{\partial x}, \frac{\partial v}{\partial y}$$

and thus we can construct the strain-displacement transformation matrix, $\mathbf{B}$

$$\boldsymbol{\varepsilon} = \mathbf{B}\mathbf{u}$$

where $\mathbf{u}$ is the vector of nodal displacements. The element stiffness corresponding to the local element degrees of freedom is then

$$\mathbf{K} = \int_V \mathbf{B}^T \mathbf{C} \mathbf{B} dV$$

The elements of $\mathbf{B}$ are functions of the natural coordinate system, r, s . Therefore the volume integration extends over the natural coordinate volume so the volume differential needs to be written in terms of the natural coordinates

$$dV = \det \mathbf{J} dr ds$$

The volume integral is not normally amenable to an explicit integration so normally a numerical integration technique is used. The integral can be written

$$\mathbf{K} = \int_V \mathbf{F} dr ds$$

where

$$\mathbf{F} = \mathbf{B}^T \mathbf{C} \mathbf{B} \det \mathbf{J}$$

and the integral is performed in the natural coordinate system of the element. This is convenient as the limits of the integration are then ± 1 . The stiffness can then be calculated

$$\mathbf{K} = \sum \alpha_{ij} \mathbf{F}_{ij}$$

where the matrix $\mathbf{F}$ is evaluated at the Gaussian integration points (r_i, s_i) and α_{ij} are Gaussian weights.

In a similar way the mass matrix and the load vectors are established.

$$\begin{aligned}\mathbf{M} &= \int_V \rho \mathbf{H}^T \mathbf{H} dV & \mathbf{R}_i &= \int_S \mathbf{H}_s^T \mathbf{f}_s dS \\ \mathbf{R}_b &= \int_V \mathbf{H}^T \mathbf{f}_b dV & \mathbf{R}_s &= \int_V \mathbf{B}^T \boldsymbol{\tau}_i dV\end{aligned}$$

where $\mathbf{H}$ is a matrix of interpolation functions and $\mathbf{R}_b$, $\mathbf{R}_s$, $\mathbf{R}_i$ are the body force vector, surface force vector and initial stress vector respectively.

Geometric stiffness matrix of shell element

The geometric stiffness matrix is calculated from:

$$\mathbf{K}_g = \int_A \mathbf{G}^T \mathbf{N} \mathbf{G} dA = \int_r \int_s \mathbf{G}^T \mathbf{N} \mathbf{G} \det \mathbf{J} dr ds$$

where:

$$\mathbf{N} = \begin{bmatrix} N_{xx} & N_{xy} \\ N_{yx} & N_{yy} \end{bmatrix}$$

N_{xx} and N_{yy} are the in-plane forces of the shell element in x and y directions respectively

N_{xy} and N_{yx} are the in-plane shear forces of the shell element and they are equal

$$\mathbf{G} = [\mathbf{G}_1 \quad \mathbf{G}_2 \quad \cdots \quad \mathbf{G}_n]$$

and n is the number of nodes of the elements

$$\mathbf{G}_i = \begin{bmatrix} 0 & 0 & \frac{\partial h_i}{\partial x} & 0 & 0 & 0 \\ 0 & 0 & \frac{\partial h_i}{\partial y} & 0 & 0 & 0 \end{bmatrix}$$

for $i = 1, 2, 3 \dots n$

12.11.7.2 MITC element formulation

We find that the Mindlin formulation is an effective approach for 2D parabolic elements where the 8-noded element accommodates sufficient terms in the stiffness matrix to sufficiently define the behaviour of the element numerically. However the same formulation defined over a linear element becomes noticeably more problematic where the absence of available terms in our 4-noded stiffness matrix leads to numerical difficulties in expressing the same element behaviour. Specifically we find a difficulty in attempting to represent the transverse shear strain terms. Analytically as before we can express the transverse shear strain as

$$\begin{Bmatrix} \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial w}{\partial x} - \beta_x \\ \frac{\partial w}{\partial y} - \beta_y \end{Bmatrix}$$

although numerically, the difference in the order of terms for the shear strain may lead to artificial stiffening of the element where the shear terms are numerically constrained from approaching zero.

See reference 1 in the [bibliography](#) for further information. This restriction would be particularly noticeable where the thickness of the plate is small.

A widely practiced remedy is to under-integrate the shear term and while effective, its use is at the cost of reduced accuracy and stability for the element. The problem of stability alone is often of greatest concern where the phenomenon of [hourglassing](#) can become apparent in elements where the thickness to length ratio is large.

An alternative formulation was put forward by Bathe and Dvorkin (ref. 28, [bibliography](#)) and has been found to be especially effective at resolving these difficulties. The formulation is extendable to higher order elements although we find the approach is most effective when resolving the difficulties most apparent in linear elements. The formulation is based upon the theory of Mixed Interpolated Tensorial Components (MITC). For the pure displacement-based Mindlin formulation we use the independent variables

$$w = \sum_{i=1}^n \mathbf{h}_i^T \mathbf{w}_i$$

$$\beta = \sum_{i=1}^n \mathbf{h}_i^T \boldsymbol{\theta}_i$$

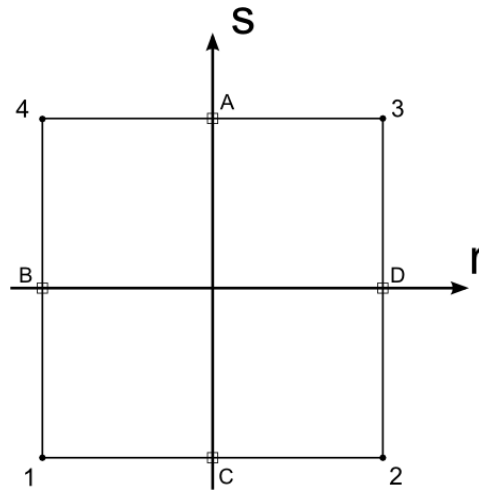
where Bathe now introduces a separate independent variable to represent the transverse shear term

$$\gamma = \sum_{i=1}^n \mathbf{h}_i^{*T} \boldsymbol{\gamma}_{P_i}$$

We use h^* to represent an additional set of interpolation functions for our new variable γ which we find by a direct evaluation of the shear strain at points P_i , that is

$$\gamma_{P_i} = \left(\frac{\delta w}{\delta x} - \beta \right)_{P_i}$$

For a linear 2D element we obtain direct values for the shear strain at four points A, B, C, D on the element and so we evaluate the displacement and section rotations at these points through direct interpolation.



We can then construct the interpolations

$$\gamma_{rz} = \frac{1}{2}(1+s)\gamma_{rz}^A + \frac{1}{2}(1-s)\gamma_{rz}^C$$

$$\gamma_{rz} = \frac{1}{2}(1+r)\gamma_{sz}^D + \frac{1}{2}(1-r)\gamma_{sz}^B$$

using direct values for the shear strains obtained at the four points. This replaces our original expression for the shear terms and we continue to construct the local stiffness matrix as normal in a similar approach as before in [2D isoparametric elements](#). It is lastly worthwhile to note that the interpolation above assumes our element is in the isoparametric coordinate system. Further transformations are necessary to interpolate the shear strains directly through an arbitrary element in local space. Indeed, when this is done the element shows considerably improved predictive capabilities for distorted elements.

12.11.7.3 Wall element formulation

Wall element is a super element, internally it is meshed by 4x4 quad8 shell elements, therefore the size of wall elements can be relatively large without losing the accuracy of analysis results. Typically a wall from one floor to the next can be modeled by a single wall element rather than a shell element mesh. The internal nodes of wall elements are condensed out and do not contribute to the number of degree of freedom in global analysis, but the middle nodes along the edges are considered in global analysis, but they are hidden for users, the middle nodes along edges do contribute to the total number of degree of freedom of the model.

As wall elements are large, the constraints at the hidden nodes along the edges will affect wall element stiffness and analysis results. The constraints of the middle nodes along the edges are considered in the following way, if the two corner nodes of an edge have the same nodal constraints, e.g. both nodes have x and zz constraints, then all the middle nodes along this edge will be constrained in x and zz. However if only one of the two nodes of the edge has constraint in a certain direction, the middle nodes along this edge will be free, e.g. if one corner node has x constraint and the other corner node has zz constraint, then all the middle nodes along this edge will be free.

The above constraint principle also applies to rigid constraints, link elements and joints, i.e. if both corner nodes of an edge are in the same rigid constraint, the same link or the same joint, all the hidden nodes along the edge will be also in the same rigid constraint, the same link or the same joint respectively. This will guarantee there is no incompatibility between the displacements along the edge and the two edge node.

12.11.8 Link elements

Link elements are different to the other element types in that they apply a constraint between a pair of nodes. The master node is the first node specified in the element topology and this is the node that is retained in the solution. The other node (the slave node) is related back to the master node.

$$\mathbf{u}_s = \mathbf{T}\mathbf{u}_m$$

The degrees of freedom at the slaves that are linked will depend on the type of link. The link allows the slave node to be fixed (where the rotations at the slave node depend on the rotation of the master) or pinned (where the rotations at the slave node are independent of the rotation of the master). The master node is always has the rotations linked to the rest of the structure. The links can act in all directions or be restricted to act in a plane (xy, yz or zx) where the nodes are rigidly connected for motion in the plane but are independent for out of plane motion.

The constraint conditions are summarised below:

All directions

$$\begin{bmatrix} 1 & 0 & 0 & 0 & z & -y \\ 0 & 1 & 0 & -z & 0 & x \\ 0 & 0 & 1 & y & -x & 0 \\ 0 & 0 & 0 & \delta & 0 & 0 \\ 0 & 0 & 0 & 0 & \delta & 0 \\ 0 & 0 & 0 & 0 & 0 & \delta \end{bmatrix} \quad \text{where } \delta = 1/0 \text{ if } \textit{fixed/pinned}$$

Plane (xy plane as example)

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & -y \\ 0 & 1 & 0 & 0 & 0 & x \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & \delta \end{bmatrix} \quad \text{where } \delta = 1/0 \text{ if } \textit{fixed/pinned}$$

Loads applied to link elements will be correctly transferred to the master degree of freedom as a force + moment so no spurious moments result.

More:

[Inertia properties of link elements](#)

12.11.8.1 Inertia properties of link elements

The inertia properties of a link element can be calculated from the masses at the nodes attached to the rigid element as follows. The mass is

$$m = \sum m_i$$

and the coordinates of the centre of mass are

$$x_c = \frac{\sum m_i x_i}{m} \quad y_c = \frac{\sum m_i y_i}{m} \quad z_c = \frac{\sum m_i z_i}{m}$$

The inertia about the global origin is then

$$\begin{aligned}\bar{I}_{xx} &= \sum m_i (y_i^2 + z_i^2) & \bar{I}_{xy} &= -\sum m_i x_i y_i \\ \bar{I}_{yy} &= \sum m_i (z_i^2 + x_i^2) & \bar{I}_{yz} &= -\sum m_i y_i z_i \\ \bar{I}_{zz} &= \sum m_i (x_i^2 + y_i^2) & \bar{I}_{zx} &= -\sum m_i z_i x_i\end{aligned}$$

and relative to the centre of mass (x_c, y_c, z_c) this is

$$\begin{aligned}\bar{I}_{xx} &= \sum m_i ((y_i - y_c)^2 + (z_i - z_c)^2) & \bar{I}_{xy} &= -\sum m_i (x_i - x_c)(y_i - y_c) \\ \bar{I}_{yy} &= \sum m_i ((z_i - z_c)^2 + (x_i - x_c)^2) & \bar{I}_{yz} &= -\sum m_i (y_i - y_c)(z_i - z_c) \\ \bar{I}_{zz} &= \sum m_i ((x_i - x_c)^2 + (y_i - y_c)^2) & \bar{I}_{zx} &= -\sum m_i (z_i - z_c)(x_i - x_c)\end{aligned}$$

The constraint equations for a link element assume small displacements. When large displacements are applied to a link element the constraint equations no longer apply and the links between slave and master get stretched. This effect can be noticeable in a dynamic analysis where the results are scaled to an artificially large value. When these are scaled to realistic value this error should be insignificant.

12.11.9 Releases

Releases are applied at the nodes (or pseudo nodes in the case of offset elements) so that the elements do not have any moment connection. This can be applied to the elements by recognising that any moment applied to node is not resisted by the element. This condition is used to partition the element stiffness matrix.

$$\begin{Bmatrix} \mathbf{f}_s \\ \mathbf{f}_e \end{Bmatrix} = \begin{bmatrix} \mathbf{k}_{ss} & \mathbf{k}_{se} \\ \mathbf{k}_{es} & \mathbf{k}_{ee} \end{bmatrix} \begin{Bmatrix} \mathbf{u}_s \\ \mathbf{u}_e \end{Bmatrix}$$

Where the subscripts s refer to the structure and e refer to the element. Once partitioned the degrees of freedom related to the structure are combined into the structure stiffness matrix while the element degrees of freedom are included in the structure stiffness matrix (but do not interact with the stiffness matrix for any other element).

In the case of stiffnesses at the releases we add the stiffness terms for the release into the partitioned stiffness matrix. So in the case of a beam with releases at end 2 and stiffness associated with the released degrees of freedom the matrix is partitioned as below.

$$\begin{bmatrix} \mathbf{k}_{s11} & \mathbf{k}_{s12} \\ \mathbf{k}_{s12} & \mathbf{k}_{s22} \end{bmatrix} \rightarrow \begin{bmatrix} \mathbf{k}_{s11} & \tilde{\mathbf{k}}_{s12} & \tilde{\mathbf{k}}_{e12} \\ \tilde{\mathbf{k}}_{s12} & \tilde{\mathbf{k}}_{s22} & \\ \tilde{\mathbf{k}}_{e12} & & \tilde{\mathbf{k}}_{e22} \end{bmatrix} + \begin{bmatrix} & & \\ & \mathbf{k}_r & -\mathbf{k}_r \\ -\mathbf{k}_r & & \mathbf{k}_r \end{bmatrix}$$

The stiffness terms at nodes where releases are applied are split, so

$$\mathbf{k}_{sij} \rightarrow \tilde{\mathbf{k}}_{sij} + \tilde{\mathbf{k}}_{eij}$$

The release stiffness terms are similar to a spring stiffness matrix:

$$\mathbf{k}_r = \begin{bmatrix} k_x & & & & & \\ & k_y & & & & \\ & & k_z & & & \\ & & & k_{xx} & & \\ & & & & k_{yy} & \\ & & & & & k_{zz} \end{bmatrix}$$

12.11.1(Offsets)

Element offsets are defined as global vectors relative to the nodes. These locate pseudo nodes that define the flexible part of the element. Element stiffnesses are calculated for the flexible part and modified to give the required nodal stiffnesses. Release conditions are applied before the offsets (i.e. they are applied to the pseudo nodes). The element local axes are defined with respect to the flexible part of the element.

The offsets can then be considered as rigid links and a constraint equation set established linking pseudo nodes to actual nodes

$$\varphi_{pseudo} = \mathbf{T} \varphi_{node}$$

where the transformation is based on the location of the pseudo node (x, y, z) relative to the actual node is

$$\mathbf{T} = \begin{bmatrix} 1 & 0 & 0 & 0 & z & -y \\ 0 & 1 & 0 & -z & 0 & x \\ 0 & 0 & 1 & y & -x & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

The displacements at the pseudo nodes can then be related to the displacements at the actual nodes.

$$\mathbf{u}_{pseudo} = \mathbf{T} \mathbf{u}_{node}$$

The forces at the nodes can be related to the forces at the pseudo nodes.

$$\mathbf{f}_{node} = \mathbf{T}^T \mathbf{f}_{pseudo}$$

The element stiffness can then be modified on a node by node basis using

$$\begin{aligned} \mathbf{f}_{i,pseudo} &= \mathbf{K}_{ij} \mathbf{u}_{j,pseudo} \\ \mathbf{f}_{i,node} &= \mathbf{T}_i^T \mathbf{K}_{ij} \mathbf{T}_j \mathbf{u}_{j,node} \end{aligned}$$

12.11.1 Transformations

The element matrices are set up in the element local axes. In the structure matrix these need to be in nodal axes. Thus two transformations are required first from element axes to global

$$\begin{bmatrix} \mathbf{T}_{DC,e} & & & \\ & \mathbf{T}_{DC,e} & & \\ & & \mathbf{T}_{DC,e} & \\ & & & \ddots \end{bmatrix} \begin{bmatrix} \mathbf{K}_{11} & \mathbf{K}_{12} & \mathbf{K}_{13} & \cdots \\ \mathbf{K}_{21} & \mathbf{K}_{22} & \mathbf{K}_{23} & \\ \mathbf{K}_{31} & \mathbf{K}_{32} & \mathbf{K}_{33} & \\ \vdots & & & \ddots \end{bmatrix} \begin{bmatrix} \mathbf{T}_{DC,e}^T & & & \\ & \mathbf{T}_{DC,e}^T & & \\ & & \mathbf{T}_{DC,e}^T & \\ & & & \ddots \end{bmatrix}$$

followed by a transformation to the nodal axes:

$$\begin{bmatrix} \mathbf{T}_{DC,n1t}^T & & & \\ & \mathbf{T}_{DC,n1r}^T & & \\ & & \mathbf{T}_{DC,n2t}^T & \\ & & & \ddots \end{bmatrix} \begin{bmatrix} \mathbf{K}_{11} & \mathbf{K}_{12} & \mathbf{K}_{13} & \cdots \\ \mathbf{K}_{21} & \mathbf{K}_{22} & \mathbf{K}_{23} & \\ \mathbf{K}_{31} & \mathbf{K}_{32} & \mathbf{K}_{33} & \\ \vdots & & & \ddots \end{bmatrix} \begin{bmatrix} \mathbf{T}_{DC,n1t} & & & \\ & \mathbf{T}_{DC,n1r} & & \\ & & \mathbf{T}_{DC,n2t} & \\ & & & \ddots \end{bmatrix}$$

where the subscripts e , n_{it} , n_{ir} relate to elements, node translations and node rotations respectively. In general the nodes act in global directions so the second transformation is not required.

12.11.1 Mass Options

The mass matrix for an element can be derived as described above. This is known as a consistent mass matrix. In many situations it is convenient to simplify the mass matrix. One way of doing this is to diagonalize the mass matrix. In this case all the terms relating to rotations are zeroed and then a row summation is carried out on the remaining entries.

$$m_{ii} = \sum_j m_{ij}$$

This has the effect of lumping all the mass at the nodes. The other simplification that is used is to ignore the mass of all elements except lumped masses.

With both these simplifications it may be possible to diagonalize the structure mass matrix, however this is not possible if the inertias of lumped mass elements are non-zero or if there are rigid elements in the structure. (Rigid elements generate off diagonal terms when the masses are replaced by inertias at the master.)

12.12 Error Norms

When using GSA it is important that the user can be confident in the accuracy of the results. There are a number of checks that the user can carry out to check accuracy, however GSA provides some measure of the accuracy of the displacement solution. This is reported in the Analysis Details output. The definition of the error norm is different for static and modal dynamic or buckling results.

Statics

In static analysis the error norm is not calculated unless the model shows signs of being ill-conditioned. The calculation is as follows

1. Calculate the residual.
2. Solve for the displacements resulting from the residual and compare these with the actual displacements.
3. Calculate the error norm.

Thus

$$\mathbf{f}_R = \mathbf{f} - \mathbf{K}\mathbf{u}$$

$$\mathbf{u}_R = \mathbf{K}^{-1}\mathbf{f}_R$$

$$e = \|\mathbf{u}_R\|/\|\mathbf{u}\|$$

where

$$\|\mathbf{u}\| = \sqrt{\sum u_i^2}$$

Dynamics and buckling

In a dynamic analysis the error norm is always calculated as follows:

$$e = \frac{\|\mathbf{K}\mathbf{u} - \lambda\mathbf{M}\mathbf{u}\|}{\|\mathbf{K}\mathbf{u}\|}$$

and in the case of buckling:

$$e = \frac{\|\mathbf{K}\mathbf{u} + \lambda\mathbf{K}_g\mathbf{u}\|}{\|\mathbf{K}\mathbf{u}\|}$$

12.13 Forces in 2D Elements

The Timoshenko¹ convention is used for forces in 2D elements. This means that a moment M_x is based on the stress in the x direction. With the Timoshenko convention if a slab is in compression on the top face in both the x and y directions the moments are both negative. Consequently starting from the assumption that tensile stress is positive, we have the following relationships for the forces:

$$\begin{aligned} N_x &= \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{xx} dz & N_y &= \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{yy} dz & N_{xy} &= \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{yx} dz \\ Q_x &= \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{xz} dz & Q_y &= \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{yz} dz \end{aligned}$$

and moments:

$$M_x = \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{xx} z dz \quad M_y = \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{yy} z dz \quad M_{xy} = \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{yx} z dz$$

Following from this a plate that has a positive in-plane stress in x/y will have a positive force resultant and a positive bending stress in x/y (i.e. positive stress at the top surface relative to the bottom surface) will have a positive moment.

When the structure is linear these simplify to:

$$\begin{aligned} N_x &= \sigma_{xx}^p t & N_y &= \sigma_{yy}^p t & N_{xy} &= \sigma_{xy}^p t \\ Q_x &= \sigma_{xz} t & Q_y &= \sigma_{yz} t \end{aligned}$$

and

$$M_x = \sigma_{xx}^b \frac{t^2}{6} \quad M_y = \sigma_{yy}^b \frac{t^2}{6} \quad M_{xy} = \sigma_{xy}^b \frac{t^2}{6}$$

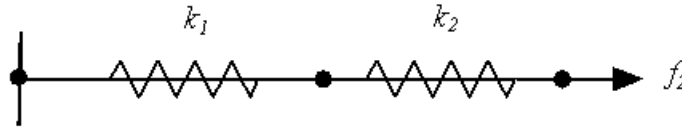
where the superscripts p and b refer to in-plane and bending stress terms.

When in-plane and bending thickness modifiers are used the in-plane forces are based on the in-plane thickness and the moments and shear forces are based on the bending thickness. Stresses are always based on the actual thickness of the element.

12.14 III Conditioning

In the vast majority of cases the solver will give a correct solution to the problem. However, some problems are by nature ill-conditioned in which case small changes in the input data can lead to more significant changes in the results.

Taking a simple example to look at ill-conditioning; consider a simple two spring system, where the springs are connected in series. The stiffness of the first spring is k_1 and that of the second is k_2 and we assume that k_2 is much greater than k_1 ($k_2 \gg k_1$).



In this case the equations describing the system is

$$\begin{Bmatrix} 0 \\ f_2 \end{Bmatrix} = \begin{bmatrix} k_1 + k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

As in a solver based on a Gaussian elimination technique, we use these equations to arrive at a relationship between u_2 and u_1 :

$$u_2 = \frac{(f_2 + k_2 u_1)}{k_2}$$

which when substituted in the other equation gives:

$$(k_1 + k_2)u_1 - k_2 \left(\frac{(f_2 + k_2 u_1)}{k_2} \right) = 0$$

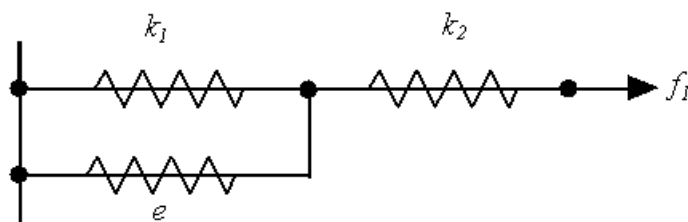
or

$$[k_1 + (k_2 - k_2)]u_1 = f_2$$

With exact arithmetic the term $(k_2 - k_2)$ would be zero, however, if k_2 is large compared with k_1 and due to limited precision, some error will be introduced in the calculation. If this error is denoted by e , then the equation we have is

$$(k_1 + e)u_1 = f_2$$

We have then a system as shown below where the error is like adding a third spring, which acts in parallel with k_1 .



The expected reaction is f_2 , but the reaction that is calculated is:

$$r = k_1 u_1 = \frac{k_1 f_2}{k_1 + e}$$

Thus the reaction is in error by a factor

$$\frac{k_1}{k_1 + e}$$

This, however, gives us a way of checking for ill-conditioning. We can sum the applied forces and compare these with the sum of the reactions. If these totals are significantly different then there may well be an ill-conditioning problem. This data is available in the “Global Results — Total Loads and Reactions”. Note the total load and moment results may well show a discrepancy if joints with non co-incident nodes are used. However, the [condition number](#) estimation gives a more rigorous way of querying the conditioning of the model.

In addition to the load/reaction check a further check on the accuracy of the solution is carried out in Gss when ill-conditioning is suspected. In this case the error is calculated as follows:

1. Calculate the residual.
2. Solve for the displacements resulting from the residual and compare these with the actual displacements.
3. Calculate the error norm.

Thus

$$\mathbf{f}_R = \mathbf{f} - \mathbf{K}\mathbf{u}$$

$$\mathbf{u}_R = \mathbf{K}^{-1}\mathbf{f}_R$$

$$e = \frac{\|\mathbf{u}_R\|}{\|\mathbf{u}\|}$$

where

$$\|\mathbf{u}\| = \sqrt{\sum u_i^2}$$

12.15 Loads on 2D elements

In a similar way the mass matrix and the load vectors are established.

$$\mathbf{R}_b = \int_V \mathbf{H}^T \mathbf{f}_b dV \quad \mathbf{R}_i = \int_S \mathbf{H}_s^T \mathbf{f}_s dS \quad \mathbf{R}_s$$

where $\mathbf{H}$ is a matrix of interpolation functions and $\mathbf{R}_b$, $\mathbf{R}_s$, $\mathbf{R}_i$ are the body force vector, surface force vector and initial stress vector respectively.

More:

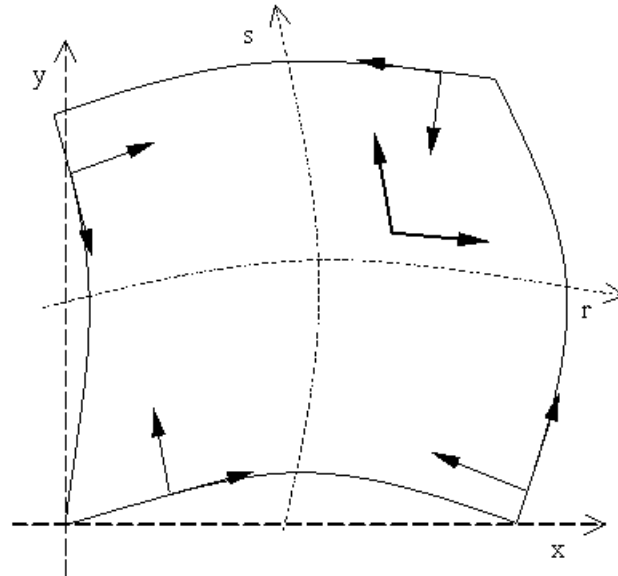
[Face and edge loads](#)

[Thermal Loads](#)

[In-plane loads](#)

12.15.1 Face and edge loads

Loads on 2D elements can be applied in local, global or user axes. The direction of the loading is straightforward for global and user directions but for local direction it is related to the face or edge geometry.



In local directions the face loads are applied with the x direction pointing in the element r direction, the y direction pointing in the element s direction and the z direction through the thickness (element t) direction. This means that the x and y directions are not necessarily orthogonal.

In local directions the edge loads are applied with the x direction pointing along the edge, the y direction pointing into the element normal to the edge and the z direction through the thickness (element t) direction.

12.15.2 Thermal Loads

Thermal loads can be either a constant temperature rise in the whole element or a temperature gradient, varying over the surface element. To evaluate the equivalent nodal forces the temperatures have to be converted to strains, using the temperature coefficient of expansion. The strains are then related to the stress through the material matrix and then the internal stresses are integrated over the element. The thermal effects are the same in all directions so there are no shear strains introduced.

For in-plane effects (constant temperature) the strain is

$$\varepsilon_i = [\varepsilon_{xx}, \varepsilon_{yy}, \varepsilon_{xy}] = [\alpha T, \alpha T, 0]$$

For bending effects (temperature gradients) the strain and stress are

$$\varepsilon_b = [\kappa_{xx}, \kappa_{yy}, \kappa_{xy}] = \left[\frac{4\alpha T}{t}, \frac{4\alpha T}{t}, 0 \right]$$

These are converted to stresses through the material matrix

$$\sigma_i = C_i \varepsilon_i \quad \sigma_b = C_b \varepsilon_b$$

12.15.3 In-plane loads

In-plane loads break down into two categories, pre-stress loads and initial strains. Pre-stress loads are defined as a force per unit width in either x, y or both x and y directions, along with an offset from the neutral axis. Initial strains are defined in either x, y or both x and y directions, but no offsets are permitted. The prestress loads can be converted directly to a set of element stresses, which are integrated over the element to get the nodal forces. The offset force gives rise to a moment on the element so

$$\sigma_i = \frac{F}{t} \quad \sigma_b = \frac{6M}{t^2} = \frac{6Fz}{t^2}$$

The initial strains can be converted to stresses in a manner analogous to that used for thermal loads.

12.16 Loads on beams

The loads on beam elements have to be transformed to a set of nodal forces and moments. These depend on the load type and the end conditions (pinned/fixed) on the beam.

More:

[Fully fixed beams](#)

[Gravity Loads](#)

[Pin ended beams](#)

[Projected loads](#)

12.16.1 Fully fixed beams

The nodal forces corresponding to beam load depends on the load type and if the beam formulation includes shear deformations. The load types break down into mechanical, thermal, pre-stress and distortion loads.

More:

[Mechanical loads](#)

[Thermal Loads](#)

[Pre-stress loads and lack of fit](#)

[Member distortion](#)

[Load direction](#)

12.16.1.1 Mechanical loads

The reference mechanical load is the point load; all the other mechanical load types can be established by integrating the results for a point load over the loaded part of the beam.

The basic approach to calculating the load on the beam for a force at position *a* is to consider the beam split at *a* into two separate beams. Flexibility matrices can be established for axial, torsional and flexural loading

$$f_{ax} = \frac{l}{AE} \quad f_{tors} = \frac{l}{GJ} \quad f_{flex_{ii}} = \frac{1}{EI_{ii}} \begin{bmatrix} \frac{l}{4} & \frac{l^2}{6} \\ \frac{l^2}{6} & \frac{l^3}{12} + \frac{EI_{ii}l}{GA_{s_{ii}}} \end{bmatrix}$$

There must be continuity of displacement and rotation between the two beams and the forces and moment must balance the applied load. This allows a set of equations to be set up for the sub-beams a and b which can be solved for the shear force and bending moment at the loaded point.

$$\mathbf{f}_a \mathbf{w}_a = \mathbf{f}_b \mathbf{w}_b \quad \mathbf{w} = \mathbf{w}_a + \mathbf{w}_b$$

Where the vector $\{w\}$ is respectively for unit force and unit moment

$$\mathbf{w}_f = \begin{Bmatrix} 1 \\ 0 \end{Bmatrix} \quad \mathbf{w}_m = \begin{Bmatrix} 0 \\ 1 \end{Bmatrix}$$

Once the force and moment at the loaded point have been established the end forces and moments (and hence the equivalent nodal forces) result from equilibrium of the two sub-beams.

The general distributed loading in the patch load, varying in linearly in intensity from position a to position b . The nodal forces and moments are then given by integrating the results for a point load

$$f_i = \int_a^b (w_1 + w_2 x) f_{pi}(x) dx$$

where $f_{pi}(x)$ is the force due to a point load at x , and

$$w_1 = \frac{w_a b - w_b a}{b - a} \quad w_2 = \frac{w_b - w_a}{b - a}$$

The tri-linear load option is simply a repeated set of patch loads.

12.16.1.2 Thermal Loads

The thermal loads allow for the introduction of load due to temperature variations in the beam elements. For an axial thermal loading the expansion of the element leads to axial forces, for thermal gradients through the thickness of the element both axial forces and end moments are induced.

$$f_x = AE \cdot \alpha T_{na}$$

$$m_{yy} = EI_{yy} \cdot \alpha \frac{dT}{dz}$$

$$m_{zz} = EI_{zz} \cdot \alpha \frac{dT}{dy}$$

where α is the temperature coefficient of expansion, T is temperature and T_{na} is the temperature rise at the neutral axis.

12.16.1.3 Pre-stress loads and lack of fit

Pre-stress and lack of fit loads are similar, but the definition differs. A pre-stress is considered as a force applied to the element at some position in the section relative to the neutral axis. A lack of fit is assumed to affect only the axial terms relating to the element. For a pre-stress of force and offsets of y and z the resulting nodal forces and moments are

$$\begin{aligned}f_x &= f_p \\m_{yy} &= f_p z \\m_{zz} &= -f_p y\end{aligned}$$

For the lack of fit and the initial strain the nodal forces are respectively

$$\begin{aligned}f_x &= \frac{AE}{l} \Delta l \\f_x &= AE \varepsilon\end{aligned}$$

12.16.1.4 Member distortion

A member distortion is a discontinuity along the length of the element. As for a mechanical load the element can be considered as two elements where in this case the displacement or rotation discontinuity is taken into account at the interface between the two parts.

12.16.1.5 Load direction

The mechanical loads are calculated assuming that the load is applied in all three directions. These then have to be factor depending on the actual direction of the load. In the case of a load in the beam local axes the factor is unity in the direction of the load and zero in the other directions. In the general case the factor are found by projecting a unit vector in the direction of the load onto the local axes of the element. Given the vector in the direction of the load (subscript l), then the factors are the components on the element vector (subscript e) are arrived at as follows

$$\begin{aligned}\mathbf{v}_g &= \mathbf{T}_{DC,a} \mathbf{v}_l \\ \mathbf{v}_e &= \mathbf{T}_{DC,e}^T \mathbf{v}_g\end{aligned}$$

or

$$\mathbf{v}_e = \mathbf{T}_{DC,e}^T \mathbf{T}_{DC,a} \mathbf{v}_l$$

12.16.2 Gravity Loads

The gravity load can be considered as a uniform load acting in all three element local directions. The factors that apply in the three directions are derived by rotating the global factors

$$\mathbf{v}_e = \mathbf{T}_{DC,e}^T$$

12.16.3 Pin ended beams

When the ends of beam are not fully fixed, some adjustment has to be made to the forces and moments that are applied at the nodes and the pinned end cannot sustain any moment. The simplest case is when both ends are pinned so

$$m_1 = m_2 = 0$$

The forces and moments are then modified to maintain equilibrium as follows

$$f_1 \rightarrow f_1 \pm \frac{(m_1 + m_2)}{l} \text{ for } y/z \quad m_1 \rightarrow 0$$

$$f_2 \rightarrow f_2 \mp \frac{(m_1 + m_2)}{l} \text{ for } y/z \quad m_2 \rightarrow 0$$

When the element is pinned at one end only the corrections depend on the material properties in the general case

pin at end 1	pin at end 2
$f_1 \rightarrow f_1 \pm \left(\frac{6}{4 + \alpha}\right) \frac{m_1}{l} \text{ for } y/z$	$f_1 \rightarrow f_1 \mp \left(\frac{6}{4 + \alpha}\right) \frac{m_2}{l} \text{ for } y/z$
$f_2 \rightarrow f_2 \mp \left(\frac{6}{4 + \alpha}\right) \frac{m_1}{l} \text{ for } y/z$	$f_2 \rightarrow f_2 \pm \left(\frac{6}{4 + \alpha}\right) \frac{m_2}{l} \text{ for } y/z$
$m_1 \rightarrow 0$	$m_1 \rightarrow m_1 - \left(\frac{2 - \alpha}{4 + \alpha}\right) m_2$
$m_2 \rightarrow m_2 - \left(\frac{2 - \alpha}{4 + \alpha}\right) m_1$	$m_2 \rightarrow 0$

where

$$\alpha = 12 \frac{EI_{ii}}{GA_s l} \quad A_s = Ak_j$$

For a simple beam this reduces to

pin at end 1	pin at end 2
$f_1 \rightarrow f_1 \pm \frac{3m_1}{2l} \text{ for } y/z$	$f_1 \rightarrow f_1 \mp \frac{3m_2}{2l} \text{ for } y/z$
$f_2 \rightarrow f_2 \mp \frac{3m_1}{2l} \text{ for } y/z$	$f_2 \rightarrow f_2 \pm \frac{3m_2}{2l} \text{ for } y/z$
$m_1 \rightarrow 0$	$m_1 \rightarrow m_1 - \frac{m_2}{2}$
$m_2 \rightarrow m_2 - \frac{m_1}{2}$	$m_2 \rightarrow 0$

12.16.4 Projected loads

When the load on a beam is distributed the total load can be based on the actual length of the element or on the projected length of the element. Thus a load normal to the x axis of the element would have a projected length equal to the beam length while a load parallel to the x axis of the beam would have a projected length of zero. In the general case of a load in the a direction defined by a unit vector w and where the beam load x axis is denoted x the factor that has to be applied to the load intensity, defined in terms of the angle between the two vectors, is

$$\text{factor} = \sin \theta = \sqrt{1 - (x \cdot w)^2}$$

12.17 Mass distribution

The calculation of the mass and inertia of the structure are as follows

$$m = \sum m_i$$

$$x_c = \frac{\sum (m_i x_i)}{m}$$

$$\mathbf{I} = \begin{Bmatrix} \sum I_{xx} + \sum m_i (y_i^2 + z_i^2) \\ \sum I_{yy} + \sum m_i (z_i^2 + x_i^2) \\ \sum I_{zz} + \sum m_i (x_i^2 + y_i^2) \\ \sum I_{xy} + \sum m_i (x_i y_i) \\ \sum I_{yz} + \sum m_i (y_i z_i) \\ \sum I_{zx} + \sum m_i (z_i x_i) \end{Bmatrix}$$

where the summations are over all the nodes and x , y and z are the coordinates of the node relative to the centre of mass.

If the mass option is set to ignore the element mass, this calculation is only carried out over the mass elements. If an additional mass due to load is set the load vector resulting from the load description is calculated and the required component is extracted scaled and converted to mass

$$m_i = s \frac{f_{ij}}{g}$$

where s is the scale factor, j is the specified component and g is the gravity value.

12.18 Material models

Isotropic material

The material properties are

E - Young's modulus

G - Shear modulus

ν - Poisson's ratio

The general elasticity matrix is

$$\mathbf{C}^{-1} = \begin{bmatrix} \frac{1}{E} & \frac{-\nu}{E} & \frac{-\nu}{E} & 0 & 0 & 0 \\ \frac{-\nu}{E} & \frac{1}{E} & \frac{-\nu}{E} & 0 & 0 & 0 \\ \frac{-\nu}{E} & \frac{-\nu}{E} & \frac{1}{E} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G} \end{bmatrix}$$

The shear modulus is related to the Young's modulus and Poisson's ratio through

$$G = \frac{E}{2(1 + \nu)}$$

Orthotropic material

The material properties are

E_x - Young's modulus in x direction

E_y - Young's modulus in y direction

E_z - Young's modulus in z direction

ν_{xy} - Poisson's ratio, y direction strain generated by unit strain in x direction

ν_{yz} - Poisson's ratio, z direction strain generated by unit strain in y direction

ν_{zx} - Poisson's ratio, x direction strain generated by unit strain in z direction

The other three Poisson's ratios (ν_{yx} , ν_{zy} , ν_{xz}) can be obtained from the following relationships.

$$\frac{E_x}{E_y} = \frac{\nu_{xy}}{\nu_{yx}} \quad \frac{E_y}{E_z} = \frac{\nu_{yz}}{\nu_{zy}} \quad \frac{E_z}{E_x} = \frac{\nu_{zx}}{\nu_{xz}}$$

G_{xy} - Shear modulus in xy plane

G_{yz} - Shear modulus in yz plane

G_{zx} - Shear modulus in zx plane

The general elasticity matrix is

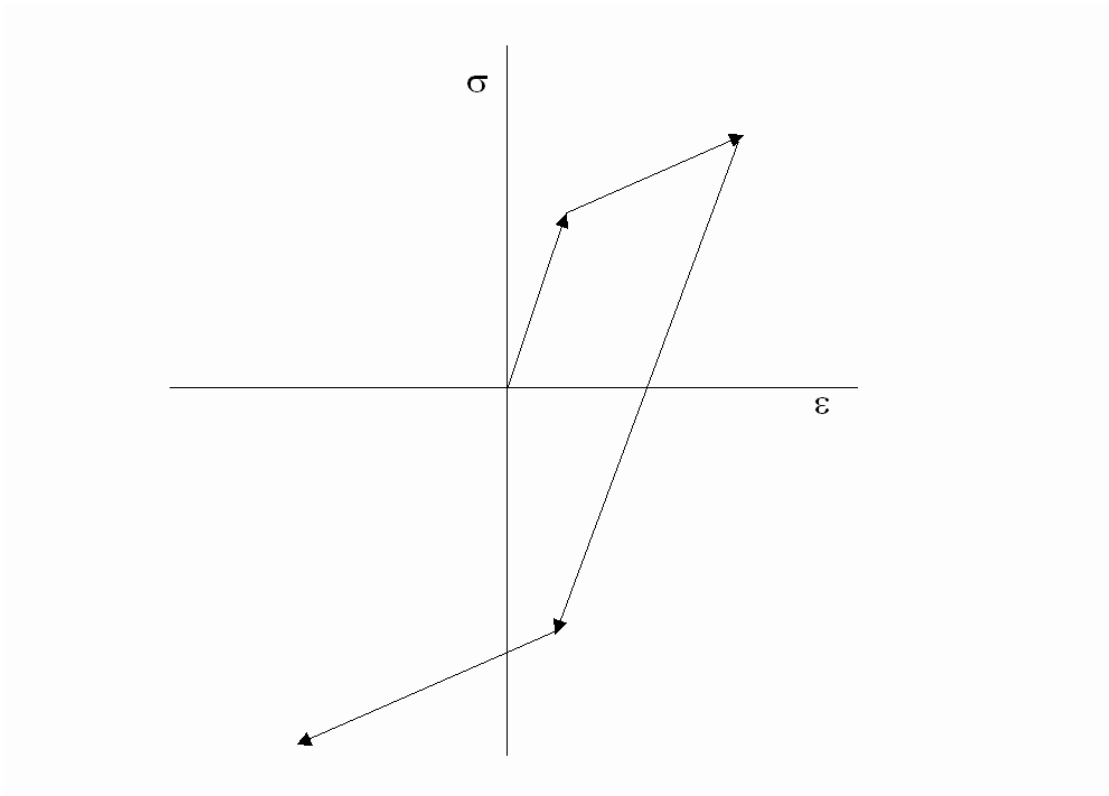
$$\mathbf{C}^{-1} = \begin{bmatrix} \frac{1}{E_x} & \frac{-\nu_{yx}}{E_y} & \frac{-\nu_{zx}}{E_z} & 0 & 0 & 0 \\ \frac{-\nu_{xy}}{E_x} & \frac{1}{E_y} & \frac{-\nu_{zy}}{E_z} & 0 & 0 & 0 \\ \frac{-\nu_{xz}}{E_x} & \frac{-\nu_{yz}}{E_y} & \frac{1}{E_z} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{xy}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{yz}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{zx}} \end{bmatrix}$$

The von Mises yield condition is

$$\sigma_y - \frac{1}{\sqrt{2}} \left[\frac{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}{6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)} \right]^{\frac{1}{2}} = 0$$

The yield and ultimate stress and the hardening parameters are not used in linear analysis. For non-linear analysis in GSA, only the yield stress is used and ultimate stress and hardening parameters are ignored.

In general, material yielding follows the line defined by the hardening modulus. Either isotropic hardening ($\beta=1$) or kinematic hardening ($\beta=0$) can be defined.



If the hardening parameter, β , is 1 (isotropic hardening as shown above) the yield stress retains its maximum value on reversal of stress. A value of 0 corresponds to kinematic hardening, where the diameter of the yield surface remains constant so on reversal of stress the material yield when the stress reversal is twice the original yield stress.

12.19 Matrix Solver

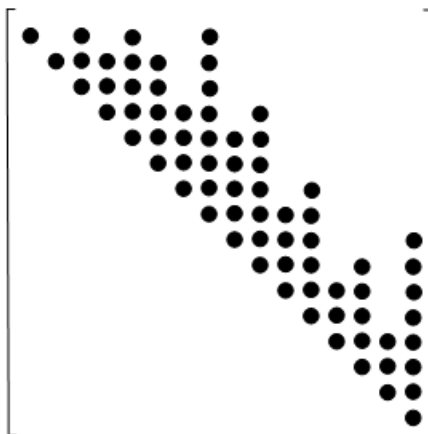
The general equation to be solved in GSS is

$$\{f\} = [K]\{u\}$$

In all cases the fact that the matrix $[K]$ is symmetric and relatively sparse is exploited in the solution.

Active Column Solver (default solver)

By default the stiffness matrix is stored in skyline form



With skyline storage only the above diagonal elements and that exist below the "skyline" are stored. Some or many of the entries within the skyline may be non-zero however in the factorization there is fill-in which leads to these entries becoming non-zero. This type of storage works well with an active column solver where the factored matrix

$$[K] = [L][D][L]^T$$

can be stored within the same skyline profile.

Once the matrix is factorized the solution of the equations is a straightforward back substitution in two passes.

Conjugate Gradient Solver

As the problem size increases the skyline storage become more inefficient as only a small proportion of the entries may be non-zero. The conjugate gradient solver exploits the sparsity of the matrix to the full by keeping an index of the non-zero terms in the matrix. This means that the factorizing which produced fill in is no longer an attractive solution procedure so a different solution is used. In this case we switch to a conjugate gradient solver with a preconditioner.

The significant difference from the active column solver is that the conjugate gradient method is iterative. In theory the solution will converge in no more iterations than the number of degrees of freedom in the solution, however rounding in the calculations means that this cannot be guaranteed. Moreover the aim is to get the solution to converge in a much smaller number of iterations so a preconditioner is used to give faster convergence. The line Jacobi preconditioner is recommended in GSA.

The conjugate gradient method and the use of preconditioners are described in many text books and the user is directed to the bibliography for further information.

Sparse Direct Solver

The sparse direct solver is another option with exploits the sparsity of the structure matrices. The solution method is similar to the active column solver in that the solution method is direct although the actual methods used are somewhat different. The sparse direct solver makes use of the approximate minimum degree ([AMD](#)) algorithm to order the degrees of freedom. This method is useful in minimizing the amount of fill when factorizing the matrices. The actual factorizing uses a sparse [LDLT](#) algorithm.

These algorithms have been developed at the University of Florida CISE (<http://www.cise.ufl.edu/research/sparse/>).

Sparse Parallel Direct Solver

The sparse parallel direct solver uses a similar storage scheme as the sparse direct solver but factorizes the matrix in parallel, utilising multiple cores in CPUs. This makes use of the 'Pardiso'

solver from Intel Math Kernel Library. Pardiso uses METIS based reordering for reducing fill-in and employs Bunch-Kauffman based pivoting for a sparse LDL^T factorization.

12.20 Nodal Stresses, Forces and Moments

In analysis of finite elements the stresses, forces or moments are calculated at Gauss points, and have to be converted from these Gauss point values to nodal values. The way in which this is achieved depends on the type of elements and the way in which the Gaussian integration is carried out. There are two resulting situations

- the number of Gauss points does not exceed the number of nodes
- the number of Gauss points is greater than the number of nodes

In the first situation it is sufficient to extrapolate the values from the Gauss points. In the second situation it is necessary to fit a function through the points to get to the nodal values.

Basic Results

The basic "stress" results in GSA are in-plane forces ($N - N_x, N_y$ and N_{xy}), through thickness forces ($Q - Q_x, Q_y$) and moments ($M - M_x, M_y$ and M_{xy}). These are values per unit width. The in-plane and bending stress can be calculated from these as

$$\begin{aligned}\sigma_{b,xx} &= \frac{M_x}{t^2/6}, & \sigma_{b,yy} &= \frac{M_y}{t^2/6}, & \sigma_{b,xy} &= \frac{M_{xy}}{t^2/6} \\ \sigma_{p,xx} &= \frac{N_x}{t}, & \sigma_{b,yy} &= \frac{N_y}{t}, & \sigma_{b,xy} &= \frac{N_{xy}}{t} \\ \sigma_{p,xz} &= \frac{Q_x}{t}, & \sigma_{b,yz} &= \frac{Q_y}{t}\end{aligned}$$

When modifiers are applied to the in-plane and bending thickness. These modified thicknesses are used to calculate the N , Q and M terms but the stresses are always based on the actual element thickness.

More:

[Direct extrapolation](#)

[Least squares extrapolation](#)

12.20.1 Direct extrapolation

In the case of direct extrapolation a function is chosen to represent the variation of stress over the element based on the number of Gauss points. In practice this is used when there are 1, 3 or 4 Gauss points. The corresponding polynomial functions are

$$f = a \quad f = a + bx + cy \quad f = a + bx + cy + dxy$$

For 1 Gauss point the values are assumed to be constant over the whole element so the Gauss point values are simply copied to the nodes.

For 3 Gauss points the values at the Gauss points are known so the following set of equations can be set up

$$f_0 = a + bx_0 + cy_0 \quad f_1 = a + bx_1 + cy_1 \quad f_2 = a + bx_2 + cy_2$$

This can be used to calculate the coefficients a , b and c

$$c = \frac{(f_0 - f_1)(x_0 - x_2) - (f_0 - f_2)(x_0 - x_1)}{(y_0 - y_1)(x_0 - x_2) - (y_0 - y_2)(x_0 - x_1)}$$

$$b = \frac{(f_0 - f_1) - c(y_0 - y_1)}{(x_0 - x_1)}$$

$$a = f_0 - bx_0 - cy_0$$

For 4 Gauss points a similar approach can be used, but in this case the locations of the Gauss points are at

$$r, s = \pm \frac{1}{\sqrt{3}}$$

so if

$$x = \frac{1}{\sqrt{3}} \quad y = \frac{1}{\sqrt{3}}$$

the equations can be written in the form

$$\begin{aligned} f_0 &= a - bx - cy + dxy & f_2 &= a + bx + cy + dxy \\ f_1 &= a + bx - cy - dxy & f_3 &= a - bx + cy - dxy \end{aligned}$$

This can then be solved for the coefficients a , b , c and d

$$\begin{aligned} a &= \frac{(f_0 + f_1 + f_2 + f_3)}{4} & c &= \frac{(-f_0 - f_1 + f_2 + f_3)}{4y} \\ b &= \frac{(-f_0 + f_1 + f_2 - f_3)}{4x} & d &= \frac{(f_0 - f_1 + f_2 - f_3)}{4xy} \end{aligned}$$

Once these are established the polynomial functions can be used to establish the values at any position on the element.

12.20.2 Least squares extrapolation

In this case a function chosen to fit through the points would imply a higher order polynomial than the one used to interpolate the geometry, so a least squares approach is used to find the polynomial to map the stresses from the Gauss points to the nodes. The interpolation functions used for 6 node and 8 node elements are respectively

$$\begin{aligned} f &= a_0 + a_1x + a_2y + a_3xy + a_4x^2 + a_5y^2 \\ f &= a_0 + a_1x + a_2y + a_3xy + a_4x^2 + a_5y^2 + a_6x^2y + a_7xy^2 \end{aligned}$$

The square of the error for any point is then

$$e^2 = (p(x, y) - f)^2$$

This is summed over all n Gauss points and then the derivatives with respect to the coefficients are set to zero (selecting the coefficients that minimise the error). This leads to the matrix equation for 8 node elements

$$\begin{bmatrix} n & \sum x & \sum y & \sum xy & \sum x^2 & \sum y^2 & \sum x^2 y & \sum xy^2 \\ & \sum x^2 & \sum xy & \sum x^2 y & \sum x^3 & \sum xy^2 & \sum x^3 y & \sum x^2 y^2 \\ & & \sum y^2 & \sum xy^2 & \sum x^2 y & \sum y^3 & \sum x^2 y^2 & \sum xy^3 \\ & & & \sum x^2 y^2 & \sum x^3 y & \sum xy^3 & \sum x^3 y^2 & \sum x^2 y^3 \\ & & & & \sum x^4 & \sum x^2 y^2 & \sum x^4 y & \sum x^3 y^2 \\ & & & & & \sum y^4 & \sum x^2 y^3 & \sum xy^4 \\ & & & & & & \sum x^4 y^2 & \sum x^3 y^3 \\ & & & & & & & \sum x^2 y^4 \end{bmatrix} \begin{Bmatrix} a_0 \\ a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \\ a_6 \\ a_7 \end{Bmatrix} = \begin{Bmatrix} \sum f \\ \sum fx \\ \sum fy \\ \sum fxy \\ \sum fx^2 \\ \sum fy^2 \\ \sum fx^2 y \\ \sum fxy^2 \end{Bmatrix}$$

The 6 node version is the same except that the a_6 and a_7 terms are ignored. This can then be solved for the coefficients.

12.21 Non-linear Analysis

Static

The static analysis is concerned with the solution of the system of equations for the displacements, u , given the applied loads. For many analyses a straightforward linear analysis is all that is required but when a model contains tie, strut or cable elements a non-linear analysis is required.

The applied loads give the load or force vector, f_{App} . The elements contribute stiffness, K . An iterative solution procedure can then be established:

1. Solve

$$\mathbf{f}_{add} = \mathbf{K}$$

1. Create a non-linear force vector to remove the effect of the non-linear effects

$$\mathbf{f}_{NI}$$

1. Solve the revised set of equations

$$\mathbf{f}_{add} + \mathbf{f}_{NL} = \mathbf{I}$$

1. Check for convergence.

$$\|u_i\| - \|u_{i-1}\| < \epsilon$$

1. If not converged go to 2.

12.22 Reactions

The reactions are the out of balance forces that remain at nodes which have a fixed displacement boundary condition. For the structure as a whole the equilibrium equation can be written

$$\mathbf{f}_r = \mathbf{K}\mathbf{u} - \mathbf{f}_a$$

The vector $\mathbf{f}_a$ can be established by calculating all the loads at the nodes. The term

$$\mathbf{K}\mathbf{u}$$

is the product of element stiffnesses and element displacements.

This equation holds for the structure as a whole, however there is no reason why this calculation cannot be carried out on an element by element basis, resulting in a set of forces at each node

$$\mathbf{f}_{ri} = \mathbf{f}_{si} - \mathbf{f}_{ai}$$

where the f_{si} are the internal forces at the node i .

12.23 Shape functions

The isoparametric elements rely on shape functions to describe the geometry and displacement fields. It is often necessary to calculate other terms that depend on the shape functions, such as directions in the element. A two dimensional element has a local coordinate system (in the plane of the element in x, y space) and a natural coordinate system (that maps the element to a square or triangle in r, s space).

Vectors in r, s, t directions

It is often required to calculate vectors in specific directions (for example for loading normal to a surface or along the edge of an element). If a vector v_r that is in the “ r ” direction is required in terms of (x, y) this is calculated as follows.

The position in (x, y) coordinates is

$$x = h_i \hat{x}_i \quad y = h_i \hat{y}_i$$

where the interpolation functions are functions of (r, s) .

Choosing a point at $(r + \delta r, s)$, the slope of the line is

$$\begin{aligned} \text{slope} &= \frac{y(r + \delta r, s) - y(r, s)}{x(r + \delta r, s) - x(r, s)} \\ &= \frac{y(r + \delta r, s) - y(r, s)}{\delta r} \cdot \frac{\delta r}{x(r + \delta r, s) - x(r, s)} \end{aligned}$$

which in the limit as $r \rightarrow 0$ gives

$$\text{slope} = \frac{\partial y}{\partial r} \frac{\partial r}{\partial x}$$

so the components of the vector and the corresponding “ s ” vector are

$$v_r = \left(\frac{\partial x}{\partial r}, \frac{\partial y}{\partial r} \right) \quad v_s = \left(\frac{\partial x}{\partial s}, \frac{\partial y}{\partial s} \right)$$

Note that v_r and v_s need not be orthogonal.

The “ t ” vector or normal to the element is then

$$v_t = v_r \times v_s$$

12.24 Stress and strain

[Stress definitions](#)

[Stress in 2D elements](#)

[Strain definitions](#)

[Stress-strain relationships](#)

12.24.1 Stress definitions

Stress can be considered as a tensor quantity whose components can be represented in matrix form as

$$\sigma = \begin{bmatrix} \sigma_{xx} & \sigma_{yx} & \sigma_{zx} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix}$$

where each term corresponds to a force per unit area. The following notation for the stress components is common

$$\begin{aligned} \sigma_x &= \sigma_{xx} & \sigma_y &= \sigma_{yy} & \sigma_z &= \sigma_{zz} \\ \tau_{xy} &= \sigma_{xy} & \tau_{yz} &= \sigma_{yz} & \tau_{zx} &= \sigma_{zx} \end{aligned}$$

The principal stresses $\sigma_1, \sigma_2, \sigma_3$ are calculated as the roots (S) of the cubic

$$S^3 - I_1 S^2 + I_2 S - I_3 = 0$$

where the terms I_1, I_2, I_3 are stress invariants defined as

$$\begin{aligned} I_1 &= \sigma_x + \sigma_y + \sigma_z \\ I_2 &= \sigma_x \sigma_y + \sigma_y \sigma_z + \sigma_z \sigma_x - \tau_{xy}^2 - \tau_{yz}^2 - \tau_{zx}^2 \\ I_3 &= \sigma_x \sigma_y \sigma_z + 2\tau_{xy} \tau_{yz} \tau_{zx} - \sigma_x \tau_{yz}^2 - \sigma_y \tau_{zx}^2 - \sigma_z \tau_{xy}^2 \end{aligned}$$

Alternatively the principal stress equation can be written

$$\begin{aligned} S^3 - (\sigma_x + \sigma_y + \sigma_z)S^2 + \\ (\sigma_x \sigma_y + \sigma_y \sigma_z + \sigma_z \sigma_x - \tau_{xy}^2 - \tau_{yz}^2 - \tau_{zx}^2)S - \\ (\sigma_x \sigma_y \sigma_z + 2\tau_{xy} \tau_{yz} \tau_{zx} - \sigma_x \tau_{yz}^2 - \sigma_y \tau_{zx}^2 - \sigma_z \tau_{xy}^2) = 0 \end{aligned}$$

The maximum shear stress is calculated from the principal stress as

$$\sigma_{max\ shear} = \frac{1}{2}(\sigma_1 - \sigma_3)$$

Two other stress measures that are used are the average or hydrostatic stress and the von Mises stress; these are defined as

$$\begin{aligned} \sigma_{av} &= \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z) \\ \sigma_{vm} &= \frac{1}{\sqrt{2}} \left[\frac{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2}{6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)} \right]^{1/2} \end{aligned}$$

12.24.2 Stress in 2D elements

The stress in 2D elements is calculated via the strains. The strains are calculated from the displacements using the strain displacement relationship **B** (see [2D elements](#)). Using the interpolation functions these can be calculated at any point in the element. Once the strains are calculated the stress can be calculated using the material elastic matrix **C** for example for an elastic isotropic material the material matrix is

$$\mathbf{C} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

Thus the strains are

$$\boldsymbol{\varepsilon} = \mathbf{B}\mathbf{u}$$

and the stresses are

$$\boldsymbol{\sigma} = \mathbf{C}\boldsymbol{\varepsilon} = \mathbf{C}\mathbf{B}\mathbf{u}$$

This can be used to evaluate the stress at any point in the element. However the stress is based on the strain which in turn is based on the displacement gradients in the element. Thus some of the strain terms in an element that has a parabolic displacement field are linear. It has been found that the best stress results are obtained by evaluating the stress at particular points (the points used for the element integration) and extrapolating the results to the nodes.

In order to have good stress results the mesh will have to be finer than the mesh required for the displacement solution and the stress results are likely to be influenced by high displacement gradients in the element.

12.24.3 Strain definitions

The normal definitions of strain used are as follows:

$$\begin{aligned} \varepsilon_x &= \frac{\partial u}{\partial x} & \varepsilon_y &= \frac{\partial v}{\partial y} & \varepsilon_z &= \frac{\partial w}{\partial z} \\ \gamma_{xy} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & \gamma_{yz} &= \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} & \gamma_{zx} &= \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \end{aligned}$$

An alternative definition which fits more neatly in tensor form is

$$\begin{aligned} \varepsilon_{xx} &= \frac{\partial u}{\partial x} & \varepsilon_{yy} &= \frac{\partial v}{\partial y} & \varepsilon_{zz} &= \frac{\partial w}{\partial z} \\ \varepsilon_{xy} &= \frac{1}{2} \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) & \varepsilon_{yz} &= \frac{1}{2} \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) & \varepsilon_{zx} &= \frac{1}{2} \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right) \end{aligned}$$

with the strain tensor defined as

$$\boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{xx} & \varepsilon_{yx} & \varepsilon_{zx} \\ \varepsilon_{xy} & \varepsilon_{yy} & \varepsilon_{yz} \\ \varepsilon_{zx} & \varepsilon_{zy} & \varepsilon_{zz} \end{bmatrix}$$

The calculation of principal strains $\varepsilon_1, \varepsilon_2, \varepsilon_3$ follows from

$$\begin{aligned} E^3 - (\varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz})E^2 + \\ (\varepsilon_{xx}\varepsilon_{yy} + \varepsilon_{yy}\varepsilon_{zz} + \varepsilon_{zz}\varepsilon_{xx} - \varepsilon_{xy}^2 - \varepsilon_{yz}^2 - \varepsilon_{zx}^2)E - \\ (\varepsilon_{xx}\varepsilon_{yy}\varepsilon_{zz} + 2\varepsilon_{xy}\varepsilon_{yz}\varepsilon_{zx} - \varepsilon_{xx}\varepsilon_{yz}^2 - \varepsilon_{yy}\varepsilon_{zx}^2 - \varepsilon_{zz}\varepsilon_{xy}^2) = 0 \end{aligned}$$

The maximum shear strain is calculated from the principal strain as

$$\varepsilon_{max\ shear} = \frac{1}{2}(\varepsilon_1 - \varepsilon_3)$$

or

$$\gamma_{max\ shear} = (\varepsilon_1 - \varepsilon_3)$$

In a similar way to the definitions of average and von Mises stress a volumetric and effective strain can be calculated as

$$\varepsilon_{av} = \frac{1}{3}(\varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz})$$

$$\varepsilon_{vm} = \frac{1}{\sqrt{2}(1+\nu)} \left[(\varepsilon_{xx} - \varepsilon_{yy})^2 + (\varepsilon_{yy} - \varepsilon_{zz})^2 + (\varepsilon_{zz} - \varepsilon_{xx})^2 + 6(\varepsilon_{xy}^2 + \varepsilon_{yz}^2 + \varepsilon_{zx}^2) \right]^{1/2}$$

12.24.4 Stress-strain relationships

The relationship between stress and strain depends on the type of problem

Problem	Displacements	Strain	Stress
Bar	u	ε_x	σ_x
Beam	w	κ_{xx}	M_{xx}
Plane stress	u, v	$\varepsilon_{xx}, \varepsilon_{yy}, \gamma_{xy}$	$\sigma_{xx}, \sigma_{yy}, \tau_{xy}$
Plane strain	u, v	$\varepsilon_{xx}, \varepsilon_{yy}, \gamma_{xy}$	$\sigma_{xx}, \sigma_{yy}, \tau_{xy}$
Axisymmetric	u, v	$\varepsilon_{xx}, \varepsilon_{yy}, \varepsilon_{zz}, \gamma_{xy}$	$\sigma_{xx}, \sigma_{yy}, \sigma_{zz}, \tau_{xy}$
General	u, v, w	$\varepsilon_{xx}, \varepsilon_{yy}, \varepsilon_{zz}, \gamma_{xy}, \gamma_{yz}, \gamma_{zx}$	$\sigma_{xx}, \sigma_{yy}, \sigma_{zz}, \tau_{xy}, \tau_{yz}, \tau_{zx}$
Plate bending	w	$\kappa_{xx}, \kappa_{yy}, \kappa_{xy}$	M_{xx}, M_{yy}, M_{xy}

Where for beam and plate bending problems the relationship is between moment and curvature

$$\kappa_{xx} = -\frac{\partial^2 w}{\partial x^2} \quad \kappa_{yy} = -\frac{\partial^2 w}{\partial y^2} \quad \kappa_{xy} = -\frac{\partial^2 w}{\partial x \partial y}$$

The stress-strain matrices for isotropic materials:

Problem Stress-strain matrix

Bar	E
Beam bending	EI

Plane stress

$$\frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}$$

Plane strain

$$\frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{1-\nu} & 0 \\ \frac{\nu}{1-\nu} & 1 & 0 \\ 0 & 0 & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix}$$

Axisymmetric

$$\frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{1-\nu} & \frac{\nu}{1-\nu} & 0 \\ \frac{\nu}{1-\nu} & 1 & \frac{\nu}{1-\nu} & 0 \\ \frac{\nu}{1-\nu} & \frac{\nu}{1-\nu} & 1 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix}$$

General

$$\frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{1-\nu} & \frac{\nu}{1-\nu} & 0 & 0 & 0 \\ \frac{\nu}{1-\nu} & 1 & \frac{\nu}{1-\nu} & 0 & 0 & 0 \\ \frac{\nu}{1-\nu} & \frac{\nu}{1-\nu} & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2(1-\nu)} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2(1-\nu)} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix}$$

Plate
bending

$$\frac{Et^3}{12(1-\nu^2)} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}$$

The stress-strain matrices for orthotropic materials:

Pro Stress-strain matrix
ble
m

Plan
e
stres
s

$$\frac{1}{1-\nu_{xy}\nu_{yx}} \begin{bmatrix} E_x & \nu_{xy}E_y & 0 \\ \nu_{yx}E_x & E_y & 0 \\ 0 & 0 & G_{xy} \end{bmatrix}$$

Plane strain

$$\begin{bmatrix} \frac{E_x(1-\nu_{yz}\nu_{zy})}{1-\nu_{xy}\nu_{yx}-\nu_{zx}(\nu_{xy}\nu_{yz}+\nu_{xz})-\nu_{zy}(\nu_{xz}\nu_{yx}+\nu_{yz})} & \frac{E_y(\nu_{xy}-\nu_{zy}\nu_{xz})}{1-\nu_{xy}\nu_{yx}-\nu_{zx}(\nu_{xy}\nu_{yz}+\nu_{xz})-\nu_{zy}(\nu_{xz}\nu_{yx}+\nu_{yz})} & 0 \\ \frac{E_x(\nu_{yx}-\nu_{zx}\nu_{yz})}{1-\nu_{xy}\nu_{yx}-\nu_{zx}(\nu_{xy}\nu_{yz}+\nu_{xz})-\nu_{zy}(\nu_{xz}\nu_{yx}+\nu_{yz})} & \frac{E_y(1-\nu_{xz}\nu_{zx})}{1-\nu_{xy}\nu_{yx}-\nu_{zx}(\nu_{xy}\nu_{yz}+\nu_{xz})-\nu_{zy}(\nu_{xz}\nu_{yx}+\nu_{yz})} & 0 \\ 0 & 0 & G_{xy} \end{bmatrix}$$

Axisymmetric

$$\begin{bmatrix} \frac{1}{E_x} & \frac{-\nu_{yx}}{E_y} & \frac{-\nu_{\theta x}}{E_\theta} & 0 \\ \frac{-\nu_{xy}}{E_x} & \frac{1}{E_y} & \frac{-\nu_{\theta y}}{E_\theta} & 0 \\ \frac{-\nu_{x\theta}}{E_x} & \frac{-\nu_{y\theta}}{E_y} & \frac{1}{E_\theta} & 0 \\ 0 & 0 & 0 & \frac{1}{G_{xy}} \end{bmatrix}^{-1}$$

General

$$\begin{bmatrix} \frac{1}{E_x} & \frac{-\nu_{yx}}{E_y} & \frac{-\nu_{zx}}{E_z} & 0 & 0 & 0 \\ \frac{-\nu_{xy}}{E_x} & \frac{1}{E_y} & \frac{-\nu_{zy}}{E_z} & 0 & 0 & 0 \\ \frac{-\nu_{xz}}{E_x} & \frac{-\nu_{yz}}{E_y} & \frac{1}{E_z} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{xy}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{yz}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{zx}} \end{bmatrix}^{-1}$$

Plate
bend
ing

$$\frac{t^3}{12} \begin{bmatrix} \frac{E_x}{1-\nu_{xy}\nu_{yx}} & \frac{\nu_{yx}E_y}{1-\nu_{xy}\nu_{yx}} & 0 \\ \frac{\nu_{xy}E_x}{1-\nu_{xy}\nu_{yx}} & \frac{E_y}{1-\nu_{xy}\nu_{yx}} & 0 \\ 0 & 0 & G_{xy} \end{bmatrix}$$

Part XIII

13 GsRelax Theory

[Dynamic Relaxation: Solution method used by GsRelax](#)

[Element Behaviour](#)

[Non-linear behaviour and buckling](#)

13.1 Dynamic Relaxation: Solution method used by GsRelax

The solution technique used in GsRelax solver is Dynamic Relaxation. Dynamic relaxation is an analysis method for non-linear statically loaded structures direct integration dynamic analysis technique. In dynamic relaxation analysis it is assumed that the loads are acting on the structure suddenly, therefore the structure is excited to vibrate around the equilibrium position and eventually come to rest on the equilibrium position. In order to simulate the vibration, mass and inertia are needed for each of the free nodes. In dynamic relaxation analysis, artificial mass and inertia are used which are constructed according to the nodal translational stiffness and rotational stiffness. If there is no damping applied to the structure, the oscillation of the structure will go forever, therefore, damping is required to allow the vibration to come to rest at equilibrium position. There are two types of damping in GsRelax analysis, one is viscous damping and another one is kinetic damping, kinetic damping is an artificial damping which will reposition the nodes according to the change of system kinetic energy.

More:

[Damping](#)

[Solution Process](#)

[Fictitious masses and inertia used in dynamic relaxation analysis](#)

[Definition of Control Parameters: residuals](#)

13.1.1 Damping

Two types of artificial damping are used — Kinetic Damping & Viscous Damping. There are two types of viscous damping, one is viscous damping and one is artificial viscous damping. Viscous damping will apply the specified (or automatically selected) percentage of the critical damping to the system. Artificial viscous damping will artificially reduce the velocity at each cycle by the specified (or automatically selected) percentage of velocity in previous cycle. Once artificial viscous damping is used, kinetic damping will be disabled automatically. By applying one or both of these artificial damping methods, the vibration will gradually come to rest at the equilibrium position and this will be the solution given by dynamic-relaxation analysis.

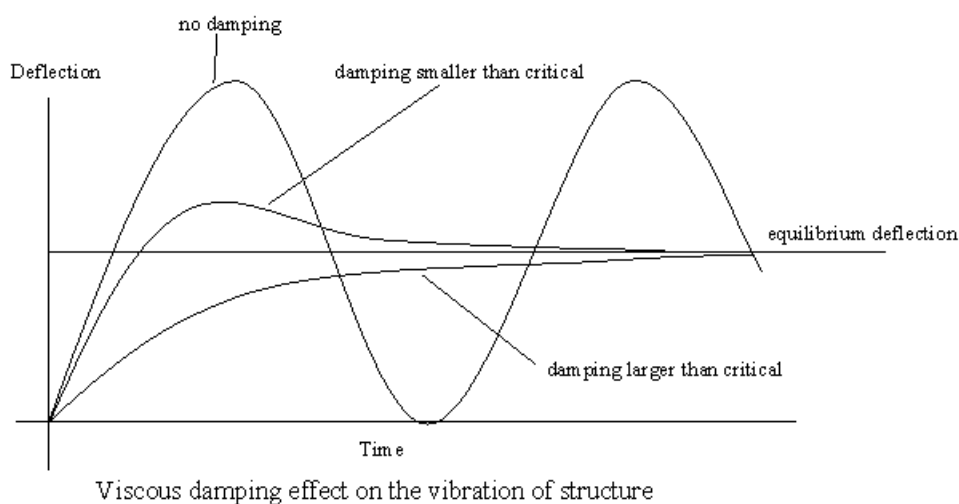
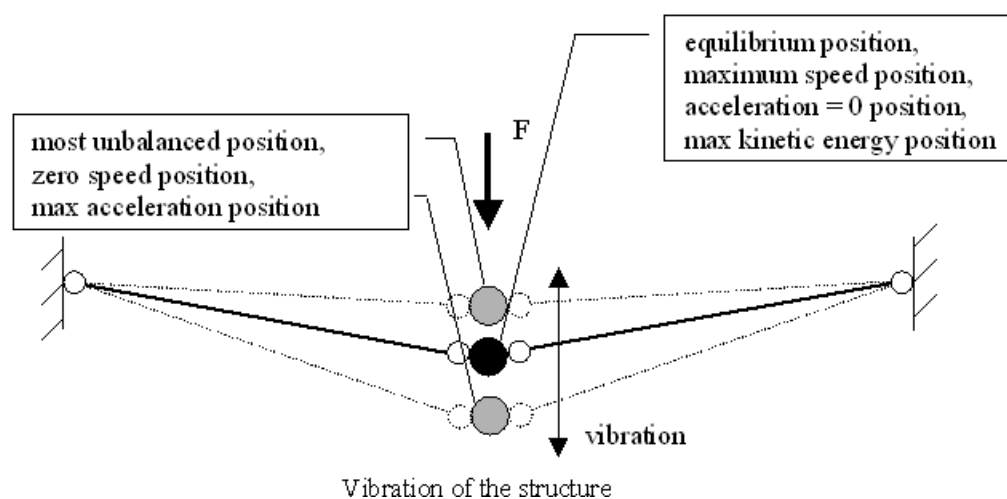
The structure below shows the effect of viscous damping on the dynamic relaxation analysis process. The oscillation of the structure eventually comes to rest at the static equilibrium position if viscous damping is applied. The problem with viscous damping is that it is not an easy task to estimate the critical damping of the structure.

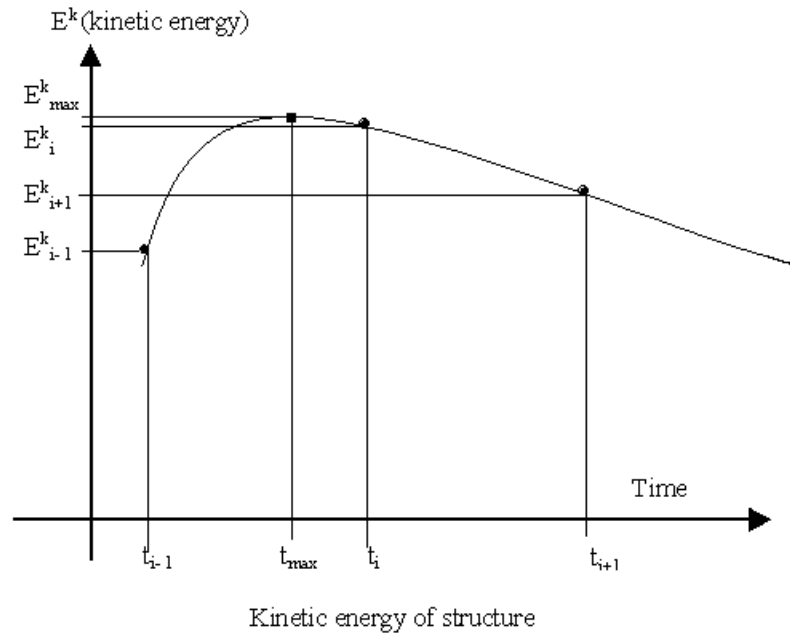
Kinetic damping is unrelated to conventional concepts of damping used in structural dynamic analysis. It is an artificial control to reduce the magnitude of the vibration in order to make it come to rest. It is based on the behaviour of structures with only one degree of freedom or the vibration of a multiple degree of freedom structure in a single mode. For these cases it is known that the structure's kinetic energy reaches a maximum at the static equilibrium position.

The structure's kinetic energy is monitored in the analysis at each time increment. The Kinetic energy increases as the nodes approach equilibrium position and starts to decrease once the nodes are away from equilibrium position. Once the kinetic energy starts to decrease, an estimate of the equilibrium position of the nodes can be interpolated from the previous nodal positions and kinetic energies.

At this point the kinetic damping process is applied. The vibration is stopped and the nodes repositioned to correspond to the maximum kinetic energy. Assuming the relationship between structural kinetic energy and time is a parabola, then the moment at which the kinetic energy peaked can be calculated. Based on the previous nodal displacements and rotations, the equilibrium positions of the nodes can be estimated. After shifting the nodes to their optimum positions, the analysis will restart again by resetting the time, speed and acceleration to be zero.

Since it is unlikely that a multiple degree of freedom structure will vibrate in a single mode, it is impossible to find the equilibrium position just by reaching the maximum kinetic energy of the structure once or twice. Nevertheless, previous experience has shown that the use of kinetic damping is very efficient in searching for the equilibrium position in dynamic relaxation analysis.





13.1.2 Solution Process

The following steps are adopted in GsRelax analysis.

1. Compute equivalent nodal forces and moments. In this process, member loads are converted into nodal force or moments. These are the forces that initiate vibration.
2. Construct dummy mass and dummy inertia for the unrestrained (active) nodes according to the translational and rotational stiffness of the members at the nodes
3. Compute the acceleration, speed and displacement for each node at each cycle.
4. Compute a new nodal position and rotation for each node at each cycle; update the nodal stiffness and member force acting on the nodes.
5. Check the force and moment residuals at each node at the current position.
6. If no residual exceeds the limit, the analysis is considered to have converged and the final position is considered as the equilibrium position of the structure.
7. If any residual is not satisfied, the analysis is continued to the next step.
8. Compute the total kinetic energy of the structure. If the kinetic energy at a cycle overshoots the maximum, it is considered that the equilibrium position has been passed. Therefore, all nodes will be re-positioned so that they are closer to the equilibrium position. Reset the speed and acceleration to be zero and let the structure start to vibrate again from the new position.
9. After analysis has been converged, the element forces, moments and stresses are calculated according to the final equilibrium position of the nodes.

13.1.3 Fictitious masses and inertia used in dynamic relaxation analysis

As mentioned in previous sections, nodal masses and inertia are needed in Dynamic Relaxation analysis. To speed up and simplify dynamic relaxation analysis, fictitious (dummy) masses and Inertia rather than real masses and inertia are used in GSA dynamic relaxation analysis. The fictitious masses and inertia are generated in the GsRelax solver and there is no need for users to define them. However, fictitious masses and inertia can be adjusted pre and during analysis by applying dummy mass and inertia factors and/or dummy mass and inertia power.

The fictitious masses and inertias calculated by the program are designed to be small enough for convergence to be reasonably fast but large enough to prevent nodes shifting too much in one cycle, which causes the solution method to become unstable. To this end, it is logical to take the fictitious masses and inertia proportional to the nodal translational stiffness and rotational stiffness respectively. From previous experience, it is found that the best estimate of the fictitious masses and inertia are two times the nodal translational stiffness and rotational stiffness respectively and they are calculated as follows:

Fictitious mass of a node = $2 \times$ Sum of translational stiffness of the elements connected to the node

Fictitious inertia of a node = $2 \times$ Sum of rotational stiffness of the elements connected to the node

13.1.4 Definition of Control Parameters: residuals

The iterative dynamic relaxation process continues until convergence criteria (unbalanced nodal force and moment) are met. If this does not happen, the iteration will continue for a maximum number of cycles, or analysis time in minutes.

It is almost impossible to achieve 100% accurate results in non-linear analysis, so an acceptable residual (tolerance) force and moment should be specified. The residual may be absolute or relative.

If relative residual is selected, the actual force residual and moment residual at each node are calculated from:

$$\Delta F = \rho_f \frac{\sum F}{n}$$

$$\Delta M = 5\rho_m \frac{\sum F}{n}$$

in which:

ΔF & ΔM = force residual and moment residual respectively

ρ_f & ρ_m = relative force residual and relative moment residual respectively

$\sum F$ = the sum of the total imposed loads including both nodal and member loads

n = the number of nodes in the structure

If there is no imposed load, e.g. a structure subjected only to support settlement, the force residual and moment residual are calculated from:

$$\Delta F = 0.01\rho_f \frac{\sum K_F}{n}$$

$$\Delta M = 0.01\rho_m \frac{\sum K_M}{n}$$

in which:

ΔF & ΔM = force residual and moment residual respectively

ρ_f & ρ_m = relative force residual and relative moment residual respectively

$\sum K_F$ & $\sum K_M$ = the sum of nodal translational stiffness and rotational stiffness of all the nodes in the structure.

n = the number of nodes in the structure

If an absolute residual is selected, the specified force residual and moment residual will be used in the analysis.

13.2 Element Behaviour

[Beams](#)

[Fabric](#)

[Spacers and Sliding Cables](#)

13.2.1 Beams

Beam section properties can be specified explicitly, or a standard shape or catalogue section can be input. The default setting given in the analysis wizard is that slenderness effects are to be considered. This means that the stiffness of individual beam elements will reduce as the axial load increases. Thus making it unnecessary to add additional nodes along the length of beam elements to simulate buckling effects. The theory used is below.

In the case of explicitly defined section properties it is assumed that I_{yy} and I_{zz} are principal axes (i.e. I_{yz} is zero) and that the section centroid and shear centre both lie on the line joining the beam end nodes. Also, for explicitly defined sections, approximations are made when considering plasticity.

In GSS $f_{y_{tension}} = f_{y_{compression}} = f_y$. The following text explicitly includes $f_{y_{tension}}$ and $f_{y_{compression}}$ for completeness.

More:

[Axial force](#)

[Shear and Torsion](#)

[Axial force — flexural stiffness interaction](#)

[Material plasticity — yielding of beams with explicit section properties](#)

[Material plasticity — yielding of beams defined as a standard shape](#)

13.2.1.1 Axial force

The axial force of a beam is first calculated as

$$F_x = \text{prestress} + AE\varepsilon$$

If this force is greater than the yield capacity in tension it is set to the yield capacity in tension; if it is less than the yield capacity in compression it is set to the yield capacity in compression. The yield capacities are

$$A \times f_{y_{compression}} \quad \text{and} \quad A \times f_{y_{tension}}$$

where:

F_x = axial force

A = area

E = modulus

ε = strain

$f_{y_{compression}}$ = compressive yield stress

$f_{y_{tension}}$ = tension yield stress

The strain is calculated as

$$\varepsilon = \frac{(\text{current distance between end nodes}) - \text{unstressed length}}{\text{unstressed length}}$$

The unstressed length is the initial distance between the end nodes (or the “initial length” as specified by the user) modified for temperature.

13.2.1.2 Shear and Torsion

The shear modulus of a beam is assumed to be

$$G = \frac{E}{2(1 + \nu)}$$

where:

G = shear modulus

E = Young's modulus

ν = Poisson's ratio

The shear strain caused by a shear force is considered to be uniform over the whole beam for planes normal to a principal axis. The shear strain between a principal axis and the local beam x axis is taken as

$$\varepsilon_{xy} = \frac{\sigma_{xy}}{G}$$

and the effective shear stress is taken as

$$\sigma_{xy} = \frac{\text{total shear force acting along a principal axis}}{(\text{section area}) \times (\text{shear factor})}$$

The K_y shear factor applies to the shear force acting along the principal axis closest to the local beam y axis.

The K_z shear factor applies to the shear force acting along the principal axis closest to the local beam z axis.

The angle by which a beam is twisted about its local x axis is simply considered to be

$$\frac{(\text{torsion moment}) \times (\text{beam length})}{(\text{torsion constant}) \times (\text{shear modulus})}$$

13.2.1.3 Axial force - flexural stiffness interaction

If slenderness effects are to be considered the bending stiffness of a beam is modified according to the axial load by using Livesey's “stability functions” (see for example M.R. Horne & W. Merchant “The Stability of Frames” Pergamon 1965).

For a continuous/continuous beam within the elastic range the bending moment at end 1 is taken as

$$M_1 = \frac{EI(S \cdot \theta_1 + SC \cdot \theta_2)}{l}$$

EQUATION A

S and SC are derived from series

$$S = \frac{\left(\frac{2}{3!} - K \times \frac{4}{5!} + K^2 \times \frac{6}{7!} - K^3 \times \frac{8}{9!} \dots\right)}{\left(\frac{2}{4!} - K \times \frac{4}{6!} + K^2 \times \frac{6}{8!} - K^3 \times \frac{8}{10!} \dots\right)}$$

$$SC = \frac{\left(\frac{1}{3!} - K \times \frac{4}{5!} + K^2 \times \frac{6}{7!} - K^3 \times \frac{8}{9!} \dots\right)}{\left(\frac{2}{4!} - K \times \frac{4}{6!} + K^2 \times \frac{6}{8!} - K^3 \times \frac{8}{10!} \dots\right)}$$

where

$$K = \frac{(\text{axial load}) \times l^2}{EI}$$

and compression is positive.

For a continuous/pinned beam within the elastic range the bending moment at end 1 is taken as

$$M_1 = \frac{EI \cdot S'' \cdot \theta_1}{l} + C \cdot M_2$$

EQUATION B

S'' and C are derived from series

$$S'' = \frac{\left(1 - K \times \frac{4}{3!} + K^2 \times \frac{6}{5!} - K^3 \times \frac{8}{7!} \dots\right)}{\left(\frac{2}{3!} - K \times \frac{4}{5!} + K^2 \times \frac{6}{7!} - K^3 \times \frac{8}{9!} \dots\right)}$$

$$C = \frac{\left(\frac{1}{3!} - K \times \frac{4}{5!} + K^2 \times \frac{6}{7!} - K^3 \times \frac{8}{9!} \dots\right)}{\left(\frac{2}{3!} - K \times \frac{4}{5!} + K^2 \times \frac{6}{7!} - K^3 \times \frac{8}{9!} \dots\right)}$$

where

$$K = \frac{(\text{axial load}) \times l^2}{EI}$$

$$M_2 = \text{moment at end 2}$$

and compression is positive.

These series are used to precalculate S, SC, S'' and C for ten values of K. During calculation cycles values of S, SC, S'' and C are interpolated for the current value of K.

13.2.1.4 Material plasticity - yielding of beams with explicit section properties

Bending Moments about the principal axes are limited to the following value

$$0.45 \times (f_{y_{TENSION}} - f_{y_{COMPRESSION}}) \times \sqrt{(A \times I_p)}$$

For equal yield stresses this is a good approximation to the plastic bending moment capacity.

The axial force is computed as above.

The calculations for the axial force and for the bending moments about the principal axes are all performed independently. Beams are assumed to behave elastically up to the limiting force or bending moment. Thus plastic behaviour is only modelled with any degree of realism for cases

where either

- Only axial forces are significant

or

- Only bending about one principal axis is significant, the tensile and compressive yield stresses are similar and the transition between first yield and full plasticity can be ignored.

If the bending moment at one end of a beam has been limited to the plastic moment capacity, the bending moment at the other end is obtained by using EQUATION B

$$bm = \frac{EI \times S'' \times \theta}{length} + C \times (plasticbm)$$

This bending moment is in turn limited to the plastic moment capacity.

13.2.1.5 Material plasticity - yielding of beams defined as a standard shape

If a yield stress is specified for a beam, the program calculates the tensile and compressive yield forces of the section, which are taken to be

$$area \times fy_{tension} \quad \text{and} \quad area \times fy_{compression}$$

The program then constructs a “look up” table for each shape before the commencement of calculation cycles.

The “look up” table contains values of

- bending moment causing first yield (i.e. the lowest bending moment at which with elastic behaviour yield stress is attained in tension or compression at one point in the section)
- plastic bending moment (i.e. the bending moment with the section on one side of the neutral axis at the tensile yield stress and on the other side of the neutral axis at compressive yield stress).

for

- nine values of axial force equally spaced between the tensile and compressive yield axial loads of the section.
- angles of applied moment at intervals of 15 degrees from 0 to 345 degrees with reference to the principal axis that is nearest to the beam local y axis.

During calculation cycles the program computes the bending moment at first yield and the plastic bending moment in a beam for the current axial force and angle of applied bending moment by linear interpolation between the values in the “look up” table (both bending moments are of course zero when the axial force equals the tensile or compressive yield force, and the axial force of a beam is limited to values between the tensile and compressive yield forces)

The program initially calculates the forces and bending moments at each end of a beam assuming elastic behaviour. If the net bending moment at the first end is greater than the moment causing first yield then the bending moment is modified according to the formula

$$new \text{ } bm = (bm \text{ at first yield}) + 0.5(old \text{ } bm - bm \text{ at first yield})$$

Equation C

If the bending moment at the first end of a beam is modified, the bending moment at the second end is obtained by using *Equation B*

$$bm \text{ at } 2_{nd} \text{ end} = \frac{EI \times S'' \theta}{length} + C \times (modified \text{ } bm \text{ at } 1_{st} \text{ end})$$

If the bending moment at the second end exceeds that at first yield, it is modified in the same way

as was the one at the first end, and the bending moment at the first end is obtained by using *Equation B*

$$bm \text{ at } 1_{st} \text{ end} = \frac{EI \times S''\theta}{length} + C \times (\text{modified } bm \text{ at } 2_{nd} \text{ end})$$

If this bending moment is greater than the moment causing first yield, the whole process is repeated until the bending moments cease to be modified.

Equation C is equivalent to halving the stiffness of a beam at first yield.

13.2.2 Fabric

[Stress Computation](#)

[Determination of shear modulus](#)

13.2.2.1 Stress Computation

The warp and weft directions are assumed to be perpendicular. The direct and shear strains are first computed for the warp and weft directions assuming uniform strains over each triangle and the stresses are calculated from the equations

$$\begin{aligned} \text{A} \quad \sigma_{xx} &= \frac{E_{xx}\varepsilon_{xx} + E_{yy}\nu_{yx}\varepsilon_{yy}}{1 - \nu_{xy}\nu_{yx}} \\ \text{B} \quad \sigma_{yy} &= \frac{E_{yy}\varepsilon_{yy} + E_{xx}\nu_{xy}\varepsilon_{xx}}{1 - \nu_{xy}\nu_{yx}} \\ \sigma_{xy} &= G\varepsilon_{xy} \end{aligned}$$

where

σ_{xx} = warp stress

ε_{xx} = warp strain

E_{xx} = warp modulus

ν_{xy} = -(ratio of weft to warp strain) for pure warp stress

σ_{yy} = weft stress

ε_{yy} = weft strain

E_{yy} = weft modulus

ν_{yx} = -(ratio of warp to weft strain) for pure weft stress

σ_{xy} = shear stress for warp/weft axes

ε_{xy} = shear strain for warp/weft axes

G = shear modulus.

The principal stresses are then computed. If a triangle is set to take no compression, compressive principal stresses are set to zero.

The forces exerted by the triangle are calculated from the principal stresses.

Equations **A** and **B** are obtained by rewriting

$$\varepsilon_{xx} = \frac{\sigma_{xx} - \nu_{yx} \sigma_{yy}}{E_{xx}}$$

and

$$\varepsilon_{yy} = \frac{\sigma_{yy} - \nu_{xy} \sigma_{xx}}{E_{yy}}$$

Poisson's ratio for pure warp stress (ν_{xy}) is defined in the material table. ν_{yx} , the Poisson's ratio for pure weft stress is calculated by GsRelax from

$$\nu_{yx} = \frac{E_{xx} \nu_{xy}}{E_{yy}}$$

13.2.2.2 Determination of shear modulus

If no shear modulus is specified it is calculated as

$$G = \frac{0.5(E_{xx} + E_{yy})}{2(1 + 0.5(\nu_{xy} + \nu_{yx}))}$$

where

G = shear modulus

E_{xx} = warp modulus

E_{yy} = weft modulus

ν_{xy} = -(ratio of weft to warp strain) for pure warp stress

ν_{yx} = -(ratio of warp to weft strain) for pure weft stress

For isotropic materials where $E_{xx} = E_{yy}$ and $\nu_{xy} = \nu_{yx}$, this is equivalent to

$$G = \frac{E}{2(1 + \nu)}$$

where

G = shear modulus

E = Young's modulus

ν = Poisson's ratio

This corresponds to elastic behaviour.

13.2.3 Spacers and Sliding Cables

[Definition of super-elements or chains](#)

[Sliding Cables](#)

[Spacer elements](#)

[Spacer types: Controlling node position](#)

[Geodesic Spacers](#)

[Free Spacers](#)

[Bar Spacer](#)

[Spacer leg length type: Controlling node spacing along spacers](#)

13.2.3.1 Definition of super-elements or chains

Spacers and Cable elements are intended to be used as multi-noded super-elements or chains in GsRelax. These super-elements can be called SPACER and Sliding-Cable elements to differentiate them from their constituent links. To define the chain, or super-element, the program looks for spacer or cable elements with the same property number, and joins common nodes. So all the elements in a continuous length of SPACER chain or Sliding-Cable chain must have a common property number, unique to that chain.

In other words each super-element should have a unique property number. All the elements in a super-element should have this property number and it that should be different from that of other Super-elements. This is necessary for GsRelax to recognise each of the individual elements as a leg of the super-element. A Super element cannot be discontinuous or bifurcated.

With Spacer elements, the order for the input of elements is important as the “ratio” feature adjusts the relative distances between nodes and makes the first element the control. I.e. if the ratio is greater than one, the first element will be the smallest. If it is less than one, the first element will be the largest. The nodes for a spacer element should be defined so that they can be jointed together to form a SPACER chain by a head-to-tail sequence.

Generally Spacer chains should not cross each other. If Spacers are required in two directions, the Spacers in the main direction should be carried through from one edge of a surface to the other and the Spacers in the subsidiary direction should join nodes in adjacent main Spacers.

13.2.3.2 Sliding Cables

Sliding cables are a hybrid form of Tie element, intended for use in GsRelax analysis of “real” (not “form-finding”) structural models.

The Sliding-Cable, composed of a number of Cable elements, is considered as a single element in GsRelax. The Sliding-Cable can freely slide around the middle nodes of the element as if the Sliding-Cable is joined to the middle node by a pulley. As a result, the tensile forces of all the Cable elements in a Sliding-Cable are the same. If a clamp is needed along the length of a Sliding Cable, two Sliding Cable elements should be defined, joining at the clamped node.

The Sliding-Cable property is defined by unit length stiffness which is equal to EA where:

E = material Young's modulus

A = cable cross section area

The tensile force in the cable is calculated from:

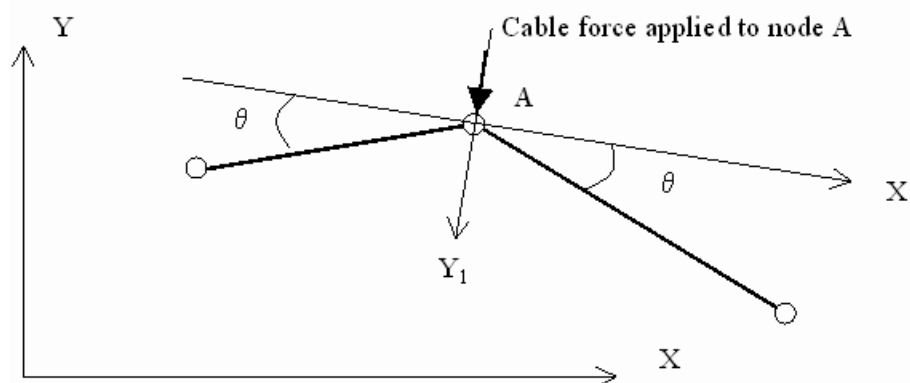
$$F = \begin{cases} \frac{L - L_0}{L_0} EA & \text{if } L > L_0 \\ 0 & \text{if } L \leq L_0 \end{cases}$$

in which:

L_0 = sum of the unstressed length of all the legs (cable elements) of the Sliding-Cable

L = sum of the deformed length of all the legs of the Sliding-Cable

The nodal normal direction in a Sliding-cable is defined by the line that is within the plane defined by the two legs and it evenly divides the angle composed by the two legs. Only the normal component of force is transferred between the nodes along a Sliding-cable and the cable.



For the portion of Sliding-cable element shown above assume the cable elements are in the XY plane.

$$\text{Resultant Cable force at node A} = T (X_1, Y_1, Z_1)$$

The Sliding-cable force is constant along its length. Therefore the components of the cable elements along X_1 at node A are equal and opposite. And the resultant force $X_1 = 0$

The component along Y_1 is applied to node A.

The component along $Z_1 = 0.0$ as the cables are in the XY plane.

$$\text{Force at node A} = T (0, Y_1, 0)$$

Free SPACERS and Sliding-Cables can be thought of as opposites in the way in which forces are transferred to their intermediate nodes. Free Spacers only apply tangential forces in the plane of the spacer, Whereas Sliding cables only apply normal forces.

13.2.3.3 Spacer elements

Spacer element is designed for soap-film form-finding only and it will be ignored in all other analyses even though it may have been defined in the model. In soap-film form-finding analysis, spacer elements are used to make up a multiple node super element called SPACER. A SPACER is made up of a chain of spacer elements that have the same property number. SPACER is used to maintain or adjust the nodal distance along the spacer element chain as desired in the form-found structure. SPACER should lie over 2D element surface or along bar or tie elements with soap-film form-finding properties (i.e. elements with zero stiffness)

All SPACERS can be considered as chains of bars with initial lengths being set to about half of their actual lengths (to ensure that the Spacers remain in tension under reasonable conditions), the stiffness (AE) of spacer elements are defined in spacer property table. To reduce the influence of the spacer elements on the form-found shape of the structure, the stiffness of spacer elements should be as small as possible as long as it can maintain the required nodal spacings. Depending on the spacer type, one or two components of the spacer forces may be suppressed, so spacer elements will only control the nodal spacing and not affect the form-found shape.

As some components of spacer forces are suppressed, the nodes attached to geodesic and free spacer elements are not be in equilibrium at the end of form-finding analysis. In this respect spacers differ from other elements.

13.2.3.4 Spacer types: Controlling node position

There are three types of spacer elements, Geodesic, Free, and Bar. The type of spacer element is defined in the spacer property table. They differ in the way that the force exerted by the spacer on the internal nodes is treated. For each internal node, a vector defines which components of the resultant spacer force on the node are taken into account. This force pulls the node along the

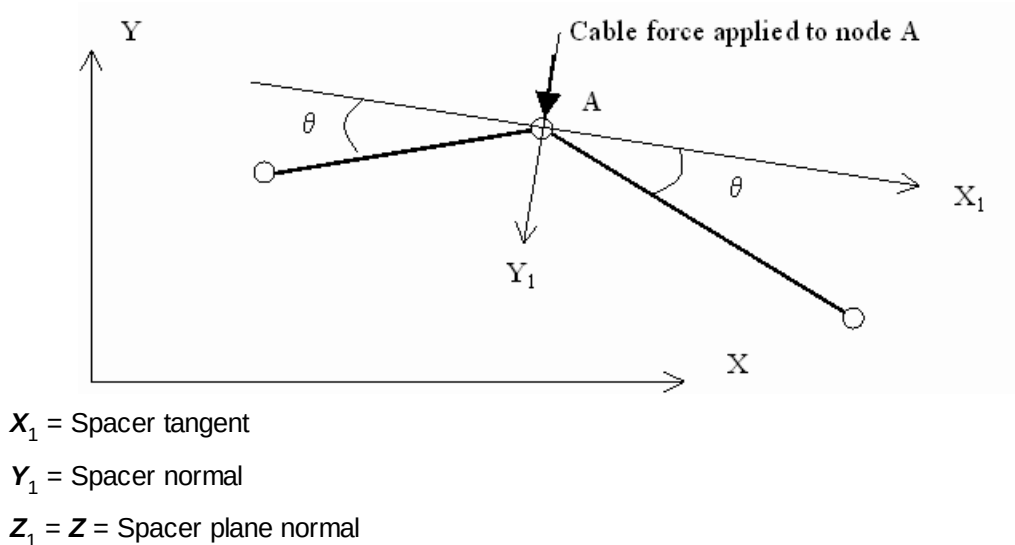
spacer vector. The remaining components are ignored.

The spacer types also differ in the way in which the spacing rules (defined as spacer leg length type) are applied within the program.

Generally Geodesic SPACERS should be used along 2D element surface. Free SPACERS should be used to control the nodal spacing along in bar or tie elements. Similar to Free spacer, Bar SPACERS can be used to control nodal spacing along bar or tie elements, the difference between Free and Bar elements is that no component of Bar spacer element forces will be suppressed.

Bar and Free SPACERS simply adjust the spacing of their nodes along the coincident bar or tie elements. Geodesic SPACERS shift their internal nodes over the surface so as to minimize the overall length between the end nodes (a geodesic is the shortest route over a surface between two points) and also maintain the spacing of their nodes.

To understand the different spacer types, it is useful to consider a short length of SPACER chain composed of two spacer elements joined at the common node.

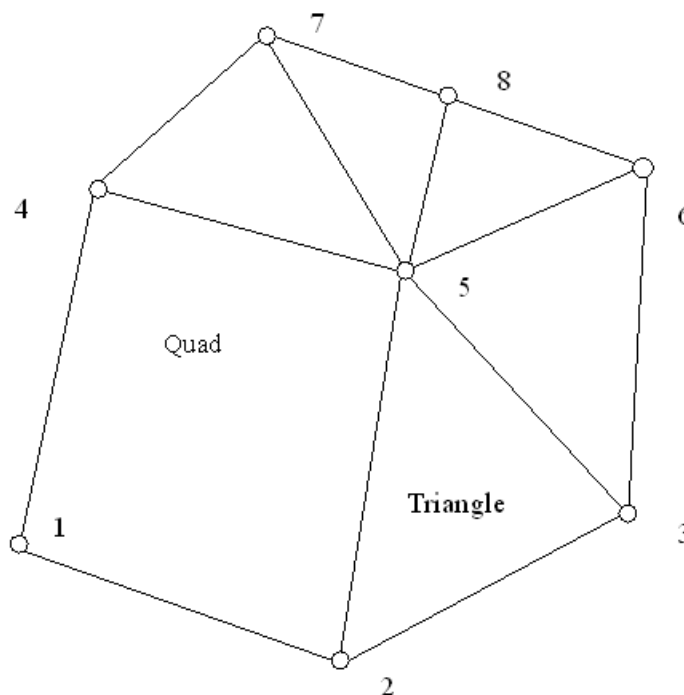


13.2.3.5 Geodesic Spacers

For Geodesic Spacers the component of each Spacer nodes reaction which is normal to the surface of the adjacent soap-film Triangles and Quads is suppressed and so the Spacers do not affect the form-found surface; they only control the position of nodes on the surface which is formed. The tension of the Geodesic Spacers ensures that the Spacers' nodes shift to form a geodesic upon the soap-film surface that is simultaneously formed by the Triangles and Quads.

The program calculates the geodesic spacer normal as follows. The program searches for Triangles and Quads that are connected to the internal nodes of each Geodesic Spacer and attempts to form a rosette of elements around each node. Failure causes a program error. The nodes of the elements in the rosette are used as "control nodes" to calculate the normal of the surface on which the spacer lies.

Initially, the normal direction is defined from the element geometry of the surrounding quads and triangles. This is then adjusted during analysis according to the displacements of the surrounding nodes



The initial normal of geodesic spacer at node 5 in the above example is equal to:

$$\mathbf{V}_{\text{normal}} = (\mathbf{C}_7 - \mathbf{C}_2) \times (\mathbf{C}_3 - \mathbf{C}_4) + (\mathbf{C}_8 - \mathbf{C}_3) \times (\mathbf{C}_6 - \mathbf{C}_7)$$

Note:

$\mathbf{V}_{\text{normal}}$ is the normal vector at node 5.

$\mathbf{C}_2$, $\mathbf{C}_3$, $\mathbf{C}_4$, $\mathbf{C}_6$, $\mathbf{C}_7$ and $\mathbf{C}_8$ are 3 dimensional vectors representing the coordinate of the nodes.

$\times$ represents vector product.

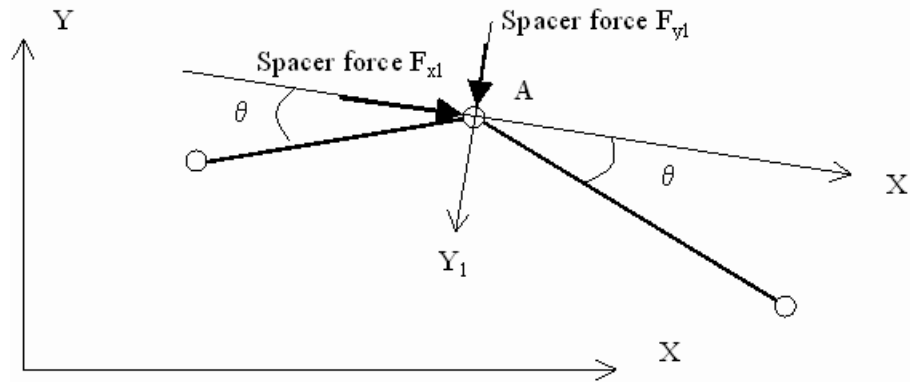
Later in the form-finding analysis, the normal will be rotated according to the displacement of the nodes. The degree of the rotation is calculated assuming all the surrounding nodes (2, 3, 4, 6, 7 and 8) are connected to the spacer node (5) by beams. The beams are pin-connected at the outer side and fixed at node 5. The EI of the beams is proportional to the sum of the angles each side of the beam. The fixed end moment for each beam at the end of node 5 is calculated. The resultant of the moment at node 5 will rotate node 5, and this rotation will be the one that rotates the spacer normal at node 5.

The normals of the end nodes of a Geodesic Spacer are found (if required) by first computing the normal as for an internal node except that

- The Rosette of Triangles and Quads may be incomplete.
- The current normal is only rotated about an x axis lying in the plane of the normal and the end spacer leg.

The normal of the next internal node is then projected onto the plane of the rotated end normal and the end spacer leg, and reflected about the plane normal to the end spacer leg.

Normals are calculated every 10 cycles.



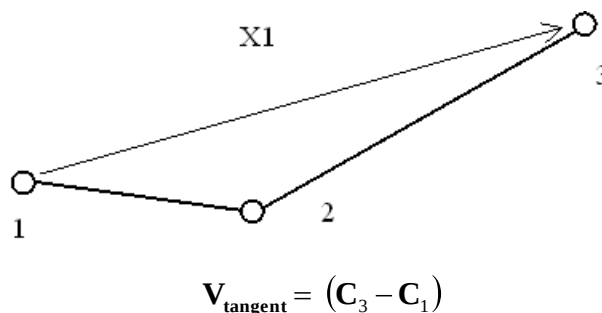
For the above portion of geodesic SPACER, assume the normal of the plane of soap film triangles and quads joined to node A is in the $Z = Z_1$ direction. The spacer force applied at node A is $(F_{x1}, F_{y1}, 0)$. So the spacer will move node A in the X_1 & Y_1 direction.

The geodesic SPACER applies the Spacer leg length rules at the start of analysis. The node positions are adjusted at the start of the analysis to meet the rules by factoring the unstressed element lengths. On convergence, the node positions may not meet the leg length rules exactly. If this proves a problem, the use of "greasy pole" restraints or Bar Spacers could be considered for a final stage form-finding analysis starting from the converged form-found shape. With both options, out-of-balance forces will exist when converting from the soap film to the "real" model, and care is needed in their interpretation.

13.2.3.6 Free Spacers

For Free Spacers only the component of a nodal reaction that acts along a SPACER's tangent is preserved. So a Free Spacer only influences the spacing of nodes, along the length of a 1D form-finding element.

The initial tangent of free spacer at node 2 in the following example is equal to:



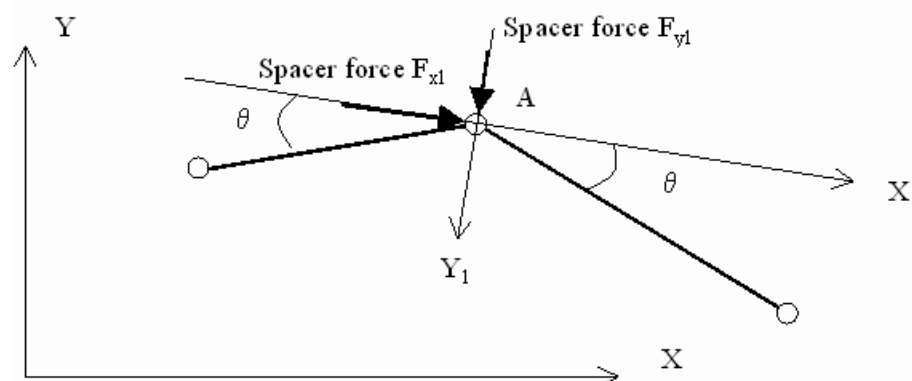
Note:

1. $\mathbf{V}_{\text{tangent}}$ is the tangent vector at node 2 ($= X_1$).
2. $\mathbf{C}_3$ and $\mathbf{C}_1$ are 3 dimensional vectors representing the coordinates of the nodes 1 and 3.

Later in the form-finding analysis, the normal will be rotated according to the displacement of the 3 nodes. The degree of the rotation is calculated assuming nodes 1 and 3 are connected to the spacer node (2) by beams. The beams are pin-connected at the nodes 1 and 3 and fixed at node 2. The EIs of the 2 beams are equal. The fixed end moment for the 2 beam at the end of node 2 is calculated. The resultant of the moment at node 2 will rotate node 2, and this rotation is the one that rotates the spacer tangent vector at node 2. The rotation will be about the normal of the plane of the adjacent spacer legs.

The tangents of the end nodes of a Free Spacer are found (if required) by reflecting the tangent of the next internal node about the plane normal to the end spacer leg. Tangents are calculated every

10 cycles.



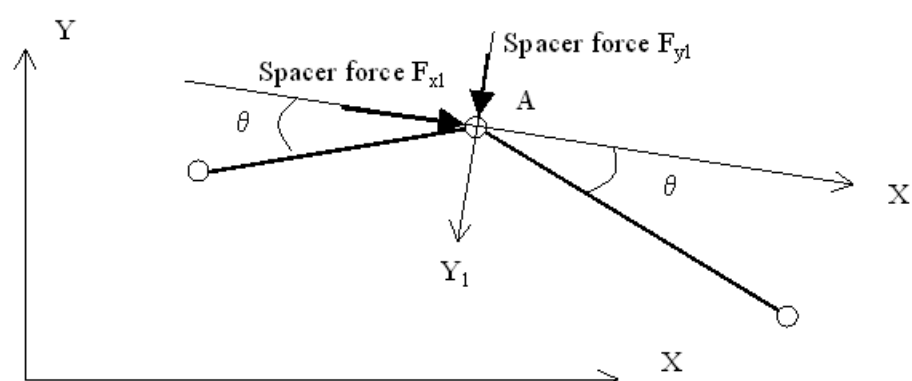
For the above portion of Free SPACER, assume the normal of the plane of the three spacer nodes is in the $Z = Z_1$ direction. The spacer force applied at node A is $(F_{x1}, 0, 0)$. So the spacer will move node A in the X_1 direction only.

The Free SPACER applies the Spacer leg length rules at the start of analysis. The node positions are adjusted at the start of the analysis to meet the rules by factoring the unstressed element lengths. On convergence, the node positions may not meet the leg length rules exactly. If this proves to be a problem, consider replacing the Free Spacer with a Bar Spacer for a final stage form finding analysis starting from the converged form-found shape.

13.2.3.7 Bar Spacer

Bar SPACERS should run parallel to 1D tie elements with soap-film properties, and the two elements should have common nodes. No component of a Bar SPACER force is suppressed so a Bar SPACER influences the spacing of nodes, along the length of the soap-film tie, and the position of the soap-film tie. The soap-film tie has zero stiffness, but the Bar Spacer has a small stiffness. Therefore forces may vary along the Bar SPACER. To maintain equilibrium post form-finding, Bar spacer forces should be added to the tie prestress in the final model as spacer elements are only considered during form-finding analysis. The effect of the Bar SPACER on the final form-found shape increases with the Bar Spacer stiffness.

The bar SPACER is, in effect, a multi-noded bar super-element, or chain, with a low stiffness that complies with spacing rules for the nodes along its length (defined by the spacer leg length type)



For the above portion of Bar SPACER, assume the normal of the plane of the three spacer nodes is in the $Z = Z_1$ direction. The spacer force applied at node A is $(F_{x1}, F_{y1}, 0)$. So the spacer will move node A in the X_1 and Y_1 direction.

The Bar spacer applies the Spacer leg length rules by repeated analysis. The node positions are adjusted at the start of the analysis to meet the rules. GsRelax is run. On convergence, the node

positions are adjusted along the length of the Bar Spacer, and the analysis repeated. This continues until the spacing rules are met.

The position of the nodes is adjusted by factoring the unstressed element lengths.

13.2.3.8 Spacer leg length type: Controlling node spacing along spacers

There are five options to control nodal spacing along a SPACER, which can be selected when defining spacer properties. These spacing options are proportional, ratio, projected ratio on XY plane, projected ratio along X axis and projected length proportional to original projected length on XY plane. If spacer type is Free and Geodesic, the initial spacer leg length will be as defined above, but the final spacer leg length at the end of form-finding analysis may not be exact the same as defined. If spacer type is Bar, the final spacer leg length will be the same as defined and the difference of the final spacer leg length and the defined spacer leg length will be smaller than the tolerance defined, this means the spacer leg lengths are checked on convergence, if the spacer leg lengths do not meet the requirements, the analysis will restart until the leg length rules are met exactly.

The five options of spacer leg length types are:

- Proportional: the final length of the SPACER legs will be proportional to their original length. To achieve this, the initial length of each spacer leg is set to half the initial distance between end nodes.
- Ratio: the SPACER leg length ratio will be equal to that specified, e.g. if Ratio 2 is defined, the final leg length of the 2nd element will be twice as long as the leg length of the first and so on. If the Ratio 1 is specified, the final length of all the spacer legs will be equal.
- Projected-ratio on XY plane: The projected length of each spacer leg onto the X-Y plane of the specified axis will form a geometric series, i.e. the projected length of the second leg over the projected length of the first leg will be equal to the ratio specified, the same applies to the following legs. If the ratio is 1.0, then the projected length of all spacer legs will be equal.
- Projected-ratio on along X axis: The projected length of each spacer leg along the X axis of the specified axis will form a geometric series, i.e. the projected length of the second leg over the projected length of the first leg will be equal to the ratio specified, the same applies to the following legs. If the ratio is 1.0, then the projected length of all spacer legs will be equal.
- Projected leg length on XY plane proportional to the original projected leg length: The projected length of each spacer leg onto the X-Y plane of the specified axis will be proportional to the original projected length. The final projected lengths are related to the original projected length.

13.3 Non-linear behaviour and buckling

[P-Delta effects, Eigenvalue buckling and GsRelax](#)

[Individual Member Buckling](#)

13.3.1 P-Delta effects, Eigenvalue buckling and GsRelax

These are three distinct analysis paths within GSA. P-Delta effects and eigenbuckling are formally applied procedures analysed with the linear elastic solver GSS.

However GsRelax, by its very nature models P-delta and buckling effects. GsRelax allows large deflection behaviour to occur and finds a state of force equilibrium based on the deflected position of the nodes and elements. Therefore secondary forces and moments due to deflections are generated automatically. Also GsRelax analysis will show the change in stiffness of the whole structure as loading increases, and reduce the stiffness of a beam as the axial load increases (even within the length of one element-see above). This can result in a reduction in the effective restraint of compression elements, or a change in load carrying characteristics from compression to bending

as elements deflect. By repeated analysis, or by using the automatic increment option, load deflection plots of key nodes can be produced to investigate buckling. Because of the solution procedure, material non-linearity and geometrical non-linearity effects will be combined together automatically. However, the study of these effects using GsRelax can be a time consuming process, requiring much interpretation. For this reason, the linear approximations often provide good enough results and should be used for most structures.

13.3.2 Individual Member Buckling

Use this option to estimate the maximum axial load capacity of one or a chain of beam elements in a model, the maximum axial load capacity can then be used to evaluate the effective length of the beam elements in question. In member buckling analysis, the model is first analysed with the imposed loads (e.g. $1.4L1 + 1.6L2$) which is the same as normal nonlinear analysis. If this analysis fails, the analysis will be abandoned and no member buckling analysis will be carried out. If it is successful, further analysis of the whole model will be carried out with the imposed loads being kept constant and a given compressive axial forces for the beam element(s) that are to be estimated for their maximum axial load capacity. The given compressive axial force are increased in steps and the whole model is analysed for every step of the given compressive axial force. Since the compressive element forces are increased, the beam element(s) will become weaker, this will also weaken the whole model, therefore an axial force versus displacement relationship/curve can be obtained. From this axial force-displacement curve, the maximum axial load capacity of the element(s) can be estimated, then the effective length of the element(s) can be evaluated.

To do member buckling analysis, the beam(s) should be given elastic material property as the effective length is obtained assuming the material is not yielding. This analysis can estimate the maximum axial load capacity for one beam element or a chain of beam element in a model. If it is a chain of beam elements, the elements must be connected one of another.

Part XIV

14 GsRelax Analysis

A number of different analyses can be performed on the same data file using the GsRelax solver. The basic analysis option is nonlinear static analysis in which all imposed loads are applied and a nonlinear analysis is carried out. This is similar to linear elastic static analysis except that material and geometrical nonlinear effects are considered. Also the post processing of results is different from linear static analysis as the principle of superposition is invalid in nonlinear analysis. Therefore results should not be combined, if results for combined load cases are required, those combination cases have to be analysed individually.

Non-linear analysis can be used to show the change in stiffness of the whole model as loading increases. This can be used to investigate near-collapse large deflection behaviour, plastic behaviour, and buckling. Automatic load increment and Individual member buckling analysis options are offered to investigate these issues.

As the GsRelax solver uses an iterative solution method non-convergence can occur. This may be due to inappropriate damping, or an inadequate number of iterations. However it will also occur if the model is a mechanism (possibly due to a modelling error), or is approaching model capacity. Non-convergence problems should be investigated. With automatic load increment analysis, GsRelax monitors the convergence rate for each analysis. If an initial run at low load converges the program stores the information about the convergence process. If convergence becomes significantly harder to achieve as the load increases it is likely that the model has reached its capacity.

Note that the model capacity is dependent on the simplifications used in the modelling process (particularly material properties, and node positions), and may not have any relation to the ultimate capacity of the real structure defined by codes of practise.

Another use of the non-linear solver is for Form-finding. This process will transform a data file with approximate geometry and prestress loading into a data file with an accurate 3 dimensional form in equilibrium with the boundary conditions, prestress and loading (if any). Selecting form finding analysis options tells the GsRelax solver to use form finding properties for the elements where these are defined, and to create new data representing the final equilibrium geometry and prestress. This new data overwrites the original data.

More:

[Static Non-linear Analysis Options](#)

[GsRelax analysis cases](#)

[GsRelax convergence and damping](#)

14.1 Static Non-linear Analysis Options

[Single increment](#)

[Automatic load increment](#)

[Individual Member Buckling](#)

[Form-finding options](#)

14.1.1 Single increment

This is the default option (i.e. 'Automatic load increment' box is not checked). In this option 100% of the loads will be applied to the structure at the start of the analysis. The analysis will then run for the number of cycles specified or until convergence is achieved. No intermediate results will be stored. If convergence has not been achieved after a significant number of cycles, it is possible that the applied loads exceed the load capacity of the model or the model has not been built appropriately, e.g. there are very soft and very stiff elements in the model. The [non-convergence](#) can be

investigated by looking at the changes in the residual forces during analysis.

14.1.2 Automatic load increment

To do automatic load increment analysis, the "Automatic load increment" box needs to be checked on the analysis wizard. In automatic increment analysis, the program is searching for the load capacity of the structure by increasing the imposed loads (but not internal loads such as thermal and pre-stress loads etc, internal loads are constant during this analysis) until one of the termination criteria is met, e.g. displacement reaches the maximum limit or convergence cannot be achieved. This option can also be used to investigate the load level at which the serviceability criteria (deflections or rotations at key nodes) are met.

In this option, the magnitude of each load increment (except the first increment) is automatically selected by the program. The first load increment is specified as a percentage of the total imposed loads. The following load increment is chosen by the program based on the likelihood of the convergence at the previous load level. The loads keep increasing monotonically and the whole model is analysed for each of the load level until one of the termination criteria is met or convergence cannot be achieved. The termination criteria can be nodal deflection/rotation or a load level expressed by load factor, the termination criteria are defined in the [Analysis wizard of GsRelax analysis](#). The automatic load increment scheme works as follows:

- 1) Do the first increment analysis (the increment level is known) and record the number (N) of cycles being used.
- 2) Determine the limit of cycles allowed for the following increment analyses, $5 \times N$ is used.
- 3) Determine the magnitude of load increment.
 - a) If the previous three or more increment analyses are all converged within the limited number of cycles, the current increment will be $1.328 \times \text{Last\_increment}$
 - b) If the previous two increment analyses are all converged within the limited number of cycles, the current increment will be $1.236 \times \text{Last\_increment}$
 - c) If only the last increment analysis is converged within the limited number of cycles, the current increment will be $1.15 \times \text{Last\_increment}$
 - d) If current analysis cannot converge within the limited number of cycles, the current increment will be reduced by a factor of 0.618, if it still cannot converge, the increment will be reduced again by the same factor.
 - e) If the increment determined above is larger than the maximum increment allowed, the maximum increment will be used.
 - f) If the increment determined above is smaller than the minimum increment specified, the analysis will be terminated.
- 4) Do GsRelax analysis under the current load level (previous loads plus the current load increment, previous nodal displacements (if analysis converges) are used as the start displacements for current analysis to speed up the process.
- 5) Check displacement and/or load factor limit, if satisfied, stop the analysis, otherwise Go to Step 3.

The final converged load level could be the model's load capacity if the analysis stops due to that: (1) very large displacement/rotation criteria is met; (2) load increment is smaller than the minimum specified. It is recommended to plot the load factor-displacement/rotation curve to evaluate the load capacity of the whole model.

Note: In automatic load increment analysis, only external loads (such as imposed nodal loads and element loads) are increased automatically, the internal loads (such as thermal, pre-stress, distortion and nodal settlements) are constant during the analysis.

14.1.3 Individual Member Buckling

In this option, the effective length of an element or a chain of compressive Beam elements in a structure can be estimated. Elements specified should be beam elements. This is useful for estimating the degree of restraint offered by the whole structure under a particular load condition. This is particularly relevant to non-linear structures where the degree of restraint offered by other parts of the model varies with load case.

The program analyses the structure with 100% imposed loads. This is called the first step analysis. If this converges, the structure is well conditioned; the number of cycles required for convergence under applied loads is stored as a measure of the stability of the structure. The program stores the equilibrium element forces and adds a small disturbance moment to the member under investigation since buckling may not occur without a small disturbance to the member.

The next step is to re-analyse the model with the imposed loads from the initial analysis but with a factored axial force in the element(s) under consideration. This is repeated until the element(s) in question buckles.

Buckling is deemed to occur when the analysis fails to converge within 5 times the number of cycles that achieved the convergence in the first step analysis. If snap through occurs in the analysis, the buckling load will not be given directly by the analysis, but the buckling load can be found out by plotting load – deflection curve using the global results for this analysis. If a load deflection curve is not available, check the results carefully.

14.1.4 Form-finding options

In these analyses, 100% of imposed loads are applied at the start of the analysis. However there may be no external loading required as the form can be generated by finding equilibrium geometry for the pre-stress state. If no imposed loading is required, input a blank load case.

The analysis proceeds as for the single increment option except the appropriate [form-finding properties](#) are used for elements where these have been defined.

More:

[Soap film form-finding](#)

[Force density form-finding](#)

[Form-finding using normal element properties](#)

14.1.4.1 Soap film form-finding

The commonest method of form-finding is with Triangles or Quads which have been given a uniform pretension and whose stiffnesses have been set to zero. Because the elements have no stiffness, the pretension is unaffected by changes of strain and so is “locked in” to the elements. They therefore behave like a soap film. If the pressure on the elements is zero they will form the “least area” surface within their boundary. A popular example of such a surface is the hyperbolic paraboloid; this is formed in a square in which two opposite corners have been raised. If the elements have a uniform pressure load, they will simulate an inflated bubble. The elements will only converge within conditions where a soap film could be formed; each point on their perimeter must be prevented from shifting inwards and they cannot be restrained internally at an isolated node since this would act like a pinpoint in a soap film. Equally in a surface that is being formed between a ring and a large base, the elements below the ring may form a narrow neck and collapse inwards if the ring is too small or too high.

If the pressure on the elements is zero, the shape that is form found is not affected by the size of the pretension and the pretension may be set to any reasonable value. If the elements have a pressure load, the curvature of the surface formed depends on the pretension. The greater the pretension, the flatter will be the form found surface. The pretension that will provide a given curvature can be estimated from the following equations.

For a spherical surface

$$\text{pretension} = \text{pressure} \times \frac{(\text{radius of curvature})}{2}$$

For a cylindrical surface

$$\text{pretension} = \text{pressure} \times (\text{radius of curvature})$$

A problem with “soap film” form-finding is that nodes that are only attached to Triangles and Quads tend to wander over the surface that is being formed. One method of controlling them is to put them on “greasy poles” i.e. restrain them so that they can only move in a line. The line should be approximately normal to the surface being formed. However Spacers offer a better means of controlling nodes. Advantages of using Spacers are that they position nodes on smooth curves with regular spacing and that the positioning of the nodes adapts automatically when the boundary of a surface is changed. Also if one uses a geodesic Spacer to form the boundary of a strip from a surface that will be flattened to make a cutting pattern, the curvature of the edges of the cutting pattern will be minimised and fabric wastage reduced.

Soap Film boundaries

In the absence of external load, the soap film surface is entirely dependent on its boundary conditions. Sometimes the boundary is a fixed line or point representing connection to a rigid structural beam or support point. More often, the boundary is a constant tension tie (defined using 1D soap film properties) which controls the edges of the surface of soap film triangles and quads. In reality this represents clamping the fabric to a cable spanning between fixed boundary points. The boundary cables should not be confused with the sliding cable element.

In the form finding process these boundary cables can be thought of as elastic bands. They are used when the precise length of the members they represent is not critical. They are modelled by a line of Ties whose Young's modulus has been set to zero using 1D soap film properties. The pre-stress in the ties is set to a constant tension using 1D soap film properties. The ties' forces are unaffected by their lengths and their nodes will shift to a position where the ties' forces are in equilibrium with the tension of the surface of which they form the edge.

As with 2D soap film form finding elements, nodes in constant tension cables may need to be prevented from wandering. They can be controlled by duplicating the Ties with a “free” spacer or by restraining them to move in a plane approximately normal to the cable.

The required pre-stress in the cable depends on the desired curvature of the cable. For a boundary cable to “soap film” Triangles and Quads

$$\text{prestress} \times \text{area} = (\text{soap film tension}) \times \frac{(\text{radius of curvature})}{2}$$

14.1.4.2 Force density form-finding

Force density form-finding is a method of form-finding cable networks which originated over 20 years ago in Germany. The tension in each link of the network is made proportional to its length. The resulting stiffness matrix is valid for any displacements and so the method enables linear structural analysis programmes to form-find. The form that is found is that giving the minimum strain energy i.e. the form which minimises the sum of the squares of the lengths of the links multiplied by the stiffnesses of the links.

1D elements—the force density form finding properties set the value of force/length for succeeding bars.

2D elements—force density form finding properties can be used to set the values of (force per width)/area for triangles or for the triangles into which succeeding quads are split. This allows fabric surfaces with different warp and weft tensions to be form-found but the shapes of the elements are generally very irregular.

14.1.4.3 Form-finding using normal element properties

This form-finding analysis is the same as normal non-linear analysis, but at the end of the analysis, there will be no analysis results being produced, instead, the model will be modified and a new load case will be generated to represent the state of the structure at the end of form-finding analysis. Since the deformed geometry and the newly generated load case are representing the state of the structure at the end of form-finding, so the actual analysis results can be obtained by analysing the deformed structure with the original loads used in form-finding analysis plus the new load, this analysis option can be used for continuation analysis and staged analysis. To achieve this, a form-finding analysis need to be carried out, then modifying the structure if necessary and then do the final stage analysis. If multiple stage analysis is required, further form-finding analysis is required after the previous form-finding analysis.

14.2 GsRelax analysis cases

The loading to be analysed by the GsRelax solver can be in the form

$$1.4L1+1.6L2$$

or

$$1.5A3$$

or

$$1.6C1$$

Note that each GsRelax run relates to a single analysis case and that results **should not** be combined. Combinations of loads and factored analysis cases should be run through the solver. If the task is form-finding with no imposed loading, a blank load case can be input. Note that in the case of incremental load options, GsRelax itself generates analysis cases for each load stage.

14.3 GsRelax convergence and damping

GsRelax functions iteratively. It simulates a process of damped vibration in small time increments (cycles). Fictitious masses and inertias are computed for each free node.

At each cycle the forces and moments which elements exert on each node are summed for the current displacements. The linear and angular accelerations of each node are computed from its fictitious mass and inertia, damping is applied to the node's current linear and angular velocities and the node's shifts and rotations are calculated for the cycle.

This process is repeated until it is terminated by the user or the out-of-balance forces and moments (residuals) at every free node are less than target values. This is called "convergence". The target values are defined in the GsRelax Control Parameters box, which is part of the Analysis Wizard. The box can also be called up during analysis.

If the damping is too high or the fictitious masses and inertias of the nodes are too large, their shifts and rotations at each cycle will be very small and many cycles will be needed to achieve a result. If on the other hand the damping is too low or the masses and inertias are too small, the simulated damped vibration becomes unstable. It is found that in order to achieve fast convergence without the solution method becoming unstable, a node's fictitious mass and inertia should be directly proportional to the stiffnesses of the elements connected to it.

The two cases of an unstable structure and of unstable simulated damped vibration can be distinguished by inspecting the results. When the structure is unstable, the element forces change little from cycle to cycle and the shifts of the nodes at each cycle may be very large but do not vary significantly from cycle to cycle. If the simulated damped vibration is unstable, the forces and nodal displacements oscillate wildly between cycles and usually increase to enormous values. The third case of stable simulated damped vibration converging to a stable solution can be recognised because the residuals and the shifts of the nodes decrease overall from cycle to cycle.

It should be noted that at least as far as GsRelax is concerned, very few structures are so unstable that they do not eventually converge to a solution. Generally secondary effects become operative with large deflections and allow the structure to reach some kind of equilibrium.

More:

[Monitoring the iterative Process](#)

[Interrupting analysis](#)

[Optimising convergence](#)

[Termination of Analysis](#)

14.3.1 Monitoring the iterative Process

During the analysis, the program tests the out-of-balance forces and moments (residuals) at each free node against target values (or Control Parameters). The frequency of this test can be adjusted. To save computing time this should not be done every cycle.

During GsRelax analysis, analysis progress information will be shown in the progress dialog box. This allows users to assess the progress of the run. In particular they can check that the simulated damped vibration has not become unstable and that the residuals are decreasing. There are three options for progress information: full progress information, summary progress information and no progress information. From the progress dialog box it is possible to reset the residual target, dummy mass factor etc or even to stop the analysis by clicking the relevant button

Summary of the analysis progress information includes the current cycles, maximum cycles, current residuals and residual targets.

Full analysis progress information includes the above plus, the maximum displacement and rotations.

Also a graphics view will be periodically updated to show the deflected shape as the analysis progresses. The frequency of updating the progress dialog box and updating the graphic view can be set prior to the analysis. The analysis speed is slightly affected by the frequency of updating the progress information, but the speed is significantly affected by the frequency of updating the graphic view especially for large structures. It is recommended not to update the graphic view too frequently for medium and large size structures.

14.3.2 Interrupting analysis

GsRelax analysis can be interrupted at any time during the analysis by clicking the stop button on the progress dialog box. When the stop button is clicked, there are three options. To continue, abandon or terminate the analysis. Terminating the analysis will mean that the current deformed shape and internal forces of the structure will be saved as result, even though the specified residual targets have not been met. The terminate option should be used cautiously and make sure that the non-converged results are not misused since they will be inaccurate. Alternatively adjust the target residual values to accept a less accurate solution.

14.3.3 Optimising convergence

By default automatic adjustment of dummy mass factors, dummy inertia factors, and damping occurs. However if convergence problems occur, the user can override these values and disable the automatic adjustment. This can be done before analysis in the Convergence control box accessed by selecting "Advanced" in the Analysis wizard. It can also be done from the progress dialog box.

More:

[Mass and inertia factors](#)

[Viscous damping](#)

[Kinetic Damping](#)

[Modelling for Optimum Convergence](#)

14.3.3.1 Mass and inertia factors

The user can modify the fictitious/dummy nodal masses and inertias used by the program linearly or exponentially. Specifying dummy mass and Inertia factors to modify dummy mass and inertia linearly, specifying dummy mass and inertia powers to modify them exponentially or specifying both factors and powers to modify dummy masses and inertias in combination.

In general, smaller dummy mass and inertia factors should be used for rigid model and larger dummy mass and inertia factors for flexible model. If dummy mass and inertia power is kept to 1.0, a dummy mass factor smaller than 1.0 could help to speed up the convergence, but it may also cause the analysis to diverge. Dummy mass factor smaller than 0.5 should rarely be used as it will be very likely to cause the analysis to diverge.

Dummy mass and inertia power should normally be 1.0, but if a model has very stiff elements and very soft elements, i.e. the difference between element stiffness is very high, a dummy mass power smaller than 1.0 may be used to help the convergence, in this case, a larger than 1.0 dummy mass factor should be used to prevent analysis to diverge, e.g. if a dummy mass power of 0.5 is used, the dummy mass factor needed could be as high as $1.0E7$ to prevent the analysis to diverge.

If an analysis is not converging or difficult to converge, trial adjusting these parameters may help, the better combination of the dummy mass factors and dummy mass powers can only be found from watching the analysis progress and adjusting these parameters.

Dummy mass factors and dummy mass power can also be adjusted during the analysis by clicking the Control button on the progress dialog box according to the residual information shown. If the maximum residual reduces too slowly it may be necessary to reduce the dummy mass factor. Otherwise if the maximum residual does not tend to reduce but jumps around, this may suggest that a larger dummy mass factor or damping is needed.

14.3.3.2 Viscous damping

There are two types of viscous damping, one is viscous damping and one is artificial viscous damping. Viscous damping will apply the specified (or automatically selected) percentage of the critical damping to the system and artificial viscous damping will artificially reduce the velocity at each cycle by the specified (or automatically selected) percentage. Once artificial viscous damping is used, kinetic damping will be disabled automatically. It is possible to modify the viscous damping applied each cycle to the linear and angular velocities of nodes. The damping is specified as a percentage (a percentage of 2 causes velocities to be factored by 0.98 each cycle if it is artificial viscous damping or applies 2% of the critical damping to the system if it is viscous damping) and is applied separately to linear and angular velocities. The default percentages of damping is automatically selected by the program, if this does not help the analysis to converge, a fixed percentage of damping can be specified or altered during the analysis. Recommended percentage of damping are 0.1 to 20 percent for linear velocities and 1 to 40 percent for angular velocities.

The more closely a structure approximates to a tuning fork, piano string, drum membrane or anything else that has a single vibration mode, the greater the damping required for fast convergence. If such structures are insufficiently damped they will oscillate instead of coming to rest in their position of equilibrium. However most real structures, at least as simulated by the program, do not vibrate in a simple mode and require little damping.

It is possible to speed up the convergence of a typical complex structure by specifying a small damping percentage. In particular the damping should be small if a structure has to deflect significantly to achieve equilibrium, since the "terminal" velocity of the nodes as they shift towards their equilibrium position depends directly on the damping. The speed of convergence can be improved by factors of 10 or more, by reducing the damping in this case. However, the user must be prepared for the solution process to "blow up" if this is done.

With viscous damping, the theoretical ideal for a structure that vibrates in a simple mode is to have low damping initially and to increase the damping as the structure approaches its equilibrium position.

The damping for angular velocities will only have an effect if the structure contains beams. The damping for angular velocities should generally be greater than the damping for linear velocities. Since typically displacements are of a greater order than rotations, target values are met more quickly for moment residuals than for force residuals and convergence is not significantly delayed by increasing the damping for angular velocities.

Generally high damping (5 to 20%) should be used for soap film form-finding. If too low damping is used, the solution process is likely to “blow up”, and even with high damping convergence is usually quite rapid. Low damping (0.1 to 2%) should be used for force density form-finding.

14.3.3.3 Kinetic Damping

The program monitors the kinetic energy of the structure. When the maximum kinetic energy is passed, the analysis is automatically adjusted using the principles of kinetic damping outlined in the [GsRelax Theory section](#). The structure is brought to rest and the process restarted. If artificial viscous damping is used, kinetic damping will be disabled.

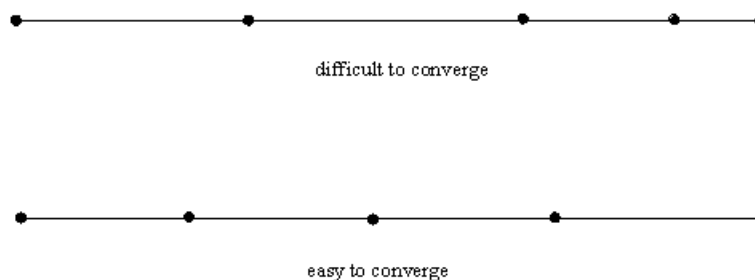
14.3.3.4 Modelling for Optimum Convergence

The speed of convergence depends primarily on the form of the structure and how it is modelled. GsRelax constructs the dummy mass and inertia of a node according to the stiffness of the elements to which it is connected. If the stiffness of some elements is much higher than that of others, the dummy mass at the relevant node will be very large. This can slow down the convergence, even causing divergence. Links and rigid constraints do not affect the dummy masses and inertias of nodes and should be used in preference to exceptionally stiff bars or beams.

If a GsRelax analysis “blows up” because the simulated damped vibration becomes unstable, the user should re-run the analysis with higher damping. If there are still problems the user should attempt to locate the spot where the forces first start to increase by watching the analysis progress dialog box. If the user is satisfied that the data file is correct and the analysis still blows up even with high damping (20%), the user should seek advice. The most likely causes of problems are extreme imposed displacements or imposed rotations, or a large difference between the specified “initial length” of a bar or a beam and the actual distance between its end nodes.

The smaller the residual target (Control Parameter) the more difficult it will be for GsRelax analysis to converge. If a larger residual is acceptable, do not try to set too small one in the analysis.

Even layouts of element meshes make the analysis converge more easily. In the example below, convergence will be quicker for the model with the most even mesh layout.



14.3.4 Termination of Analysis

The user can set single target values for force and moment residuals. Alternatively the user can specify a target Residual Percentage. These are set as GsRelax Control Parameters.

It is important not to set too rigorous target values and to monitor results. The target values may be

less than the errors that are generated by the computer. In such cases the structure does not shift significantly as cycles repeat and the residuals do not decrease. The run should be terminated since the best possible accuracy has been achieved.

If the analysis does not converge it will be stopped after the number of cycles, or a length of time specified as **termination controls**. Non convergence should be investigated as it is possible to improve convergence by adjusting damping, or modifying a badly conditioned model. It is also possible that an acceptable solution can be achieved with higher out of balance forces and that the convergence criteria should be relaxed by adjusting the **residual controls**.

In addition, limits can be set to control the automatic load increment option. With this option GsRelax is searching for the “capacity” of the model by successively increasing the load factor. The capacity can be defined as the load at which analysis ceases to converge, or, the load at which excessive deflections or rotations occur. This can also be used to investigate the load factor at which certain serviceability criteria are met.

For most structures there is a load factor beyond which the capacity becomes unimportant, as the loading will never be achieved, even considering normal safety factors. With the automatic load increment option it is important to set the max load factor at which the program should cease to search for the model capacity. Use this to avoid unnecessary analysis.

Part XV

15 Seismic Analysis

There are many different approaches to seismic analysis. The two main approaches implemented in GSA are [Equivalent Static Procedures](#) and [Response Spectrum Analysis](#). In general the response spectrum method is recommended in the various seismic codes as this is based on a dynamic analysis of the structure. However at times the simpler equivalent static method which ignores the actual dynamic response of the structure may be useful.

GSA has support for a variety of seismic codes including:

- ASCE 7 (USA)
- Eurocode 8 (Europe)
- FEMA 356 (USA)
- GB50011 (China)
- IBC (USA)
- IS1893 (India)
- Ordinanza PCM 3274 (Italy)
- UBC (USA)

15.1 Equivalent Static Procedures

In many cases simplified analysis methods using equivalent static loading are used in seismic analysis. The static loading to be considered will normally be defined with reference to a design code.

Equivalent Static

The equivalent static load is based on the base shear, which in turn depends on the period of the building. This can either be specified directly or based on the code approximation. The code period is usually defined as a function of the height of the building

$$T = C(h)^n$$

The coefficients C and n are defined in the code.

GSA uses the details defined for the response spectrum along with the weight of the building and the period to calculate the base shear, V . The base shear is then distributed over the storeys, s , in the building based on the storey weight, w and height above ground, h .

$$F_s = \frac{(V - F_t)w_s h_s}{\sum_{i=1}^n w_i h_i}$$

where F_t is a concentrated force at the top defined for some of the seismic codes, but set to zero for most codes. F_s represents the total force applied at the storey. This is then distributed across the nodes on that storey in proportion to the nodal masses.

Accidental Torsion

The equivalent static load procedures require accidental torsion loads to be considered. These are to represent the effect of a displacement of the centre of mass of the storeys. In GSA the accidental torsion loads are calculated so that the moment applied to the storey is proportional to the equivalent static storey forces F_s .

$$M_s = k d F_s$$

This depends on the building dimension d at the storey level which is determined by GSA. The factor k is specified in the seismic code (typically 5%).

GSA generates a set of nodal forces that create a moment for the storey that satisfies the equation above without generating any lateral force, so that this can be used in conjunction with the equivalent static load.

15.2 Response Spectrum Analysis

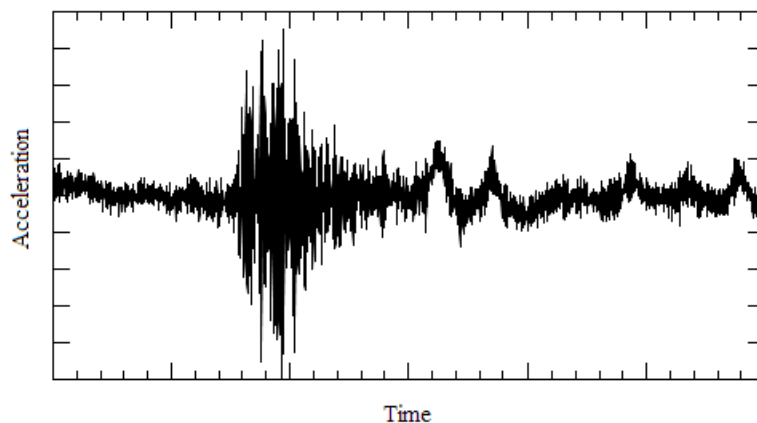
Response spectrum

A modal response spectrum analysis is a statistical approach that looks at the elastic response of the structure taking account of the response in all the calculated modes. While this gives a good estimate of maximum values of displacement and force/moment it effectively only produces an envelope of these values, so it can be difficult to examine all the results of interest or combine these results with those from other analyses.

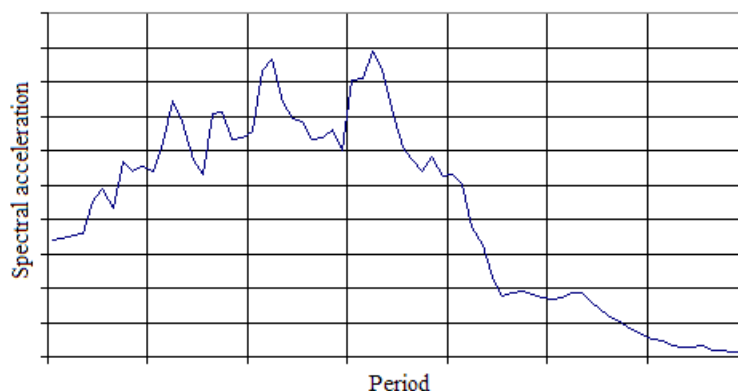
The modal analysis should contain sufficient modes to capture the dynamic characteristics of the structure. In general the sum of the effective masses (in the relevant directions) should be around 90% of the total mass of the structure. For tall structures it is typically easier to capture the effective mass in the horizontal direction than the vertical this being a reflection of the relative greater flexibility in the horizontal direction.

A response spectrum is a curve of the maximum acceleration response of a series of single degree of freedom system oscillators of different natural frequencies with a given level of damping to a given excitation.

So starting from the seismic record (acceleration time history) as shown below and a given level of damping



The response spectrum is then the maximum acceleration response of a single degree of freedom system to the excitation.



In practice the response spectrum is usually code defined. In this case the curve is an envelope of the spectra that would be calculated from a series of possible earthquakes. The actual values are determined from a selection of parameters which include such factors as the seismicity of the region, the ground conditions, the importance of the building, etc.

The response of the structure in any direction for a given mode can then be determined with reference to the frequency, participation factor and the spectral value as

$$x_i = \Gamma_i \frac{a_{spectral}(f_i)}{(2\pi f_i)^2} \phi_i$$

In this equation:

$$\Gamma_i \frac{a_{spectral}(f_i)}{(2\pi f_i)^2}$$

is known as the modal multiplier.

Modal combinations

Once the responses in each mode are calculated they must be combined over the modes. A number of different combinations are provided

- **Absolute sum** – simple combination, but overestimates the response

$$q = \sum_i |q_i|$$

- **Square Root Sum of Squares (SRSS)** – better than the absolute sum but can underestimate the response where there are closely spaced modes

$$q = \sqrt{\sum_i q_i^2}$$

- **Complete quadratic combination (CQC)** – the most commonly used method which allows for interaction between closely spaced modes

$$q = \sqrt{\sum_i \sum_j q_i \rho_{ij} q_j}$$

where

$$\rho_{ij} = \frac{8\sqrt{\xi_i \xi_j} (\xi_i + r_{ij} \xi_j) r_{ij}^{3/2}}{(1 - r_{ij}^2)^2 + 4\xi_i \xi_j r_{ij} (1 + r_{ij}^2) + 4(\xi_i^2 + \xi_j^2) r_{ij}^2}$$

$$r_{ij} = \frac{\omega_i}{\omega_j}$$

ξ_i, ξ_j = damping in mode i, j

If the damping is the same for all modes

$$\rho_{ij} = \rho_{ji} = \frac{8\xi^2 (1 + r_{ij}) r_{ij}^{3/2}}{(1 - r_{ij}^2)^2 + 4\xi^2 r_{ij} (1 + r_{ij}^2)} \quad \rho_{ii} = 1$$

- **Modified Rosenbluth** – an alternative combination, similar to CQC

$$q = \sqrt{\sum_i \sum_j q_i \rho_{ij} q_j}$$

where

$$\rho_{ij} = \frac{2\sqrt{\xi_i \xi_j}}{\xi_i + \xi_j} \left[1 + \left(\frac{f_i - f_j}{\xi_i f_i + \xi_j f_j} \right)^2 \right]^{-1}$$

The same combination can be used for any modal response, such as displacement, force or base shear.

Seismic total forces and moments

In addition to the code defined procedure GSA carries out some overall checks of total forces and moments (overturning and twisting moments).

We define an origin **o** for our calculation which is at the centre of mass on plan (x,y) and at the base level (z) of the structure.

Then, where the subscript i refers to nodes, j to modes and k the excitation direction

$$\mathbf{F}_{jk} = a_{jk} \sum_i m_i \cdot \mathbf{u}_{ij}$$

$$\mathbf{M}_{jk} = a_{jk} \sum_i (m_i \cdot \mathbf{u}_{ij} \times (\mathbf{c}_i - \mathbf{o}) + \mathbf{I}_i \cdot \boldsymbol{\theta}_{ij})$$

Where m is the nodal mass, $\mathbf{I}$ the nodal inertia, $\mathbf{u}$ and $\boldsymbol{\theta}$ the modal displacement and rotation and $\mathbf{c}$ is node coordinates and

$$a_{jk} = \Gamma_{jk} \cdot a_{spec,j}$$

Where Γ_{jk} is the participation factor for mode j in the k direction and $a_{spec,j}$ is the spectral acceleration for the frequency corresponding to mode j.

And

$$\mathbf{F}_k = \sum_{CQC} \mathbf{F}_{jk}$$

$$\mathbf{M}_k = \sum_{CQC} \mathbf{M}_{jk}$$

Then if k is the x direction the base shear from this calculation is $\mathbf{F}_{k,x}$, the overturning moment is

$M_{k,yy}$ and the torsional (twisting) moment is $M_{k,zz}$.

When the origin of the calculation is changed to the global origin then these calculations give total dynamic forces moments.

Other results

The (dynamic) base shear for a given mode can be calculated from the response spectrum and the effective mass

$$V_i = m_{eff,i} a_{spectral}(f_i)$$

The storey drift is defined as

$$i = \frac{u_{drift}}{h}$$

where h is the storey height.

Two useful results are the ratio of effective mass in the modes considered to the total mass and the base shear coefficients.

$$\text{mass ratio} = \frac{\sum_i m_{eff-i}}{M_{tot}}$$

$$\text{base shear coefficient} = \frac{V_d}{M_{tot}g} = \frac{V_d}{W_{tot}}$$

The storey inertia forces are calculated for excitation in the j^{th} direction as follows for the i^{th} storey:

$$f_x = m_i s_{code} \sum_{CQC} (\Gamma_j a_{spec} u_x)$$

$$f_y = m_i s_{code} \sum_{CQC} (\Gamma_j a_{spec} u_y)$$

$$M_z = I_{zz,i} s_{code} \sum_{CQC} (\Gamma_j a_{spec} \theta_z)$$

where s_{code} is the code scaling factor.

Limitations

The response spectrum method is essentially a statistical method which aims to give maximum responses, so it does not provide a consistent set of results but rather an envelope of the maximum results. This can lead to difficulties in interpreting and using the results of a response spectrum analysis. So

- Element forces results are from combining the modal forces, so these are not the same as calculating the results from the response displacements at the nodes.
- Storey drifts are not the same as the difference in response between floors.
- Displacements, etc. in transformed axes will be conservative (as maximum x is combined with maximum y).

Part XVI

16 Dynamic Response Analysis

Dynamic response analysis is the term used for analysis that depends on a previously analysed modal analysis. The seismic [response spectrum analysis](#) is a special case of these.

The other dynamic response options are detailed below:

[Harmonic analysis](#)

[Periodic load analysis](#)

[Linear time history analysis](#)

[Footfall analysis](#)

16.1 Harmonic Analysis

GSA harmonic analysis is to calculate the elastic structure responses to harmonic loads at steady state. Modal superposition method is used in GSA harmonic analysis.

The dynamic equation of motion is:

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{p} \sin(\omega t)$$

where

$\mathbf{M}$	mass matrix
$\mathbf{u}, \dot{\mathbf{u}}, \ddot{\mathbf{u}}$	nodal displacement, velocity & acceleration vectors
$\mathbf{C}$	damping matrix
$\mathbf{K}$	stiffness matrix
$\mathbf{p}$	load vector
ω	harmonic load frequency

From the mode shape results of a modal dynamic analysis, the nodal displacements, velocities and accelerations can be expressed as

$$\mathbf{u} = \Phi \mathbf{q}$$

$$\dot{\mathbf{u}} = \Phi \dot{\mathbf{q}}$$

$$\ddot{\mathbf{u}} = \Phi \ddot{\mathbf{q}}$$

where:

Φ	matrix of m mode shapes
$\mathbf{q}, \dot{\mathbf{q}}, \ddot{\mathbf{q}}$	displacement, velocity & acceleration vectors in modal (generalized) coordinates, the length of the vectors is m

Substituting for $\mathbf{q}$ gives

$$\mathbf{M}\Phi\ddot{\mathbf{q}} + \mathbf{C}\Phi\dot{\mathbf{q}} + \mathbf{K}\Phi\mathbf{q} = \mathbf{p} \sin(\omega t)$$

Pre-multiplying each term in this equation by the transpose of the mode shape matrix gives

$$\Phi^T \mathbf{M} \Phi \ddot{\mathbf{q}} + \Phi^T \mathbf{C} \Phi \dot{\mathbf{q}} + \Phi^T \mathbf{K} \Phi \mathbf{q} = \Phi^T \mathbf{p} \sin(\omega t)$$

According to the orthogonality relationship of the mode shapes to the mass matrix and the stiffness matrix and also assuming the mode shapes are also orthogonal to the damping matrix (e.g. Rayleigh damping), Eq. (4) becomes m uncoupled dynamic equations of motion as shown below, m is the

number of modes to be considered.

$$\begin{aligned}\varphi_1^T \mathbf{M} \varphi_1 \ddot{\mathbf{q}}_1 + \varphi_1^T \mathbf{C} \varphi_1 \dot{\mathbf{q}}_1 + \varphi_1^T \mathbf{K} \varphi_1 \mathbf{q}_1 &= \varphi_1^T \mathbf{p} \sin(\omega t) \\ \varphi_2^T \mathbf{M} \varphi_2 \ddot{\mathbf{q}}_2 + \varphi_2^T \mathbf{C} \varphi_2 \dot{\mathbf{q}}_2 + \varphi_2^T \mathbf{K} \varphi_2 \mathbf{q}_2 &= \varphi_2^T \mathbf{p} \sin(\omega t) \\ &\dots \\ \varphi_m^T \mathbf{M} \varphi_m \ddot{\mathbf{q}}_m + \varphi_m^T \mathbf{C} \varphi_m \dot{\mathbf{q}}_m + \varphi_m^T \mathbf{K} \varphi_m \mathbf{q}_m &= \varphi_m^T \mathbf{p} \sin(\omega t)\end{aligned}$$

Setting

$$\begin{aligned}\mathbf{M}_i &= \varphi_i^T \mathbf{M} \varphi_i \\ \mathbf{C}_i &= \varphi_i^T \mathbf{C} \varphi_i \\ \mathbf{K}_i &= \varphi_i^T \mathbf{K} \varphi_i \\ \mathbf{p}_i &= \varphi_i^T \mathbf{p}\end{aligned}$$

Then the m uncoupled equations can be expressed in a general form as follows

$$\mathbf{M}_i \ddot{\mathbf{q}}_i + \mathbf{C}_i \dot{\mathbf{q}}_i + \mathbf{K}_i \mathbf{q}_i = \mathbf{p}_i \sin(\omega t)$$

where all the terms are scalars. Solving this equation is equivalent to solving a single degree of freedom problem as shown below:

For the single degree of freedom problem subjected to harmonic load, the dynamic magnification factors (μ) of displacement for mode i in complex number notation is:

$$\mu = \left[\left(1 - \left(\frac{\omega}{\theta_i} \right)^2 \right) + i \left(2\xi \frac{\omega}{\theta_i} \right) \right]^{-1} = \mu_r - i \mu_i$$

where:

$$\begin{aligned}i &= \sqrt{-1} \\ \mu_r &= \frac{A}{A^2 + B^2} \\ \mu_i &= \frac{B}{A^2 + B^2} \\ A &= 1 - \left(\frac{\omega}{\theta_i} \right)^2 \\ B &= 2\xi \frac{\omega}{\theta_i}\end{aligned}$$

and θ_i is the natural frequency of mode i

The maximum displacement, velocity & acceleration of mode i in the modal coordinates are:

$$q_i = \mu \frac{P_i}{K_i} = \mu \frac{P_i}{M_i \theta_i^2}$$

$$\dot{q}_i = q_i \omega = \mu \frac{P_i \omega}{K_i} = \mu \frac{P_i \omega}{M_i \theta_i^2}$$

$$\ddot{q}_i = \dot{q}_i \omega = q_i \omega^2 = \mu \frac{P_i \omega^2}{K_i} = \mu \frac{P_i \omega^2}{M_i \theta_i^2}$$

Substituting gives the maximum actual nodal displacements, velocities & accelerations at the steady state of the forced vibration as:

$$\mathbf{u} = \sum_{i=1}^m \phi_i q_i$$

$$\dot{\mathbf{u}} = \sum_{i=1}^m \phi_i \dot{q}_i$$

$$\ddot{\mathbf{u}} = \sum_{i=1}^m \phi_i \ddot{q}_i$$

After obtaining the maximum nodal displacements, the element forces and moments etc can be calculated as in static analysis.

16.2 Periodic Load Analysis

GSA periodic load analysis is to calculate the maximum elastic structure responses to generic periodic loads at steady state. Modal superposition method is used in GSA periodic load analysis.

The dynamic equation of motion subjected to periodic loads is:

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{p}f(t)$$

where

M mass matrix

u, $\dot{u}$, $\ddot{u}$ nodal displacement, velocity & acceleration vectors

C damping matrix

K stiffness matrix

p load vector

$f(t)$ harmonic load frequency

According to Fourier Series, the periodic function of time can be expressed as a number of sine functions:

$$f(t) = \sum_{h=1}^H r_h \sin\left(\frac{h 2\pi}{T} t\right)$$

where

h Fourier term ($h=1,2,\dots, n$)

r_h	Fourier coefficient (or dynamic load factor), the values of r_h are defined by the users on dynamic load factor table. r_h can be constant or varies according to the frequencies of the periodic loads
T	the period of the periodic load, $T = 1/f$ and f is the periodic load frequency
H	the number of Fourier (harmonic) terms to be considered, a maximum of four can be considered in GSA periodic load analysis

Substituting in the first equation we can rewrite as a number of dynamic equations of motion subjected to harmonic loads:

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{p}r_h \sin\left(\frac{h2\pi}{T}t\right)$$

The maximum responses of this can be solved using [harmonic analysis](#) for each of the harmonic loads ($h = 1, 2, 3$, or 4), then the maximum responses from the periodic loads can be calculated using square root sum of the squares (SRSS) method as follows.

$$R_{\max} = \sqrt{\sum_{h=1}^H R_{\max,h}^2}$$

where

$R_{\max}$	maximum responses under the periodic loads, e.g. the maximum displacement, velocity or acceleration
$R_{h,\max}$	the maximum responses under harmonic load h , e.g. the maximum displacement, velocity or acceleration of harmonic load h

16.3 Linear Time History Analysis

GSA linear time history analysis is to calculate the transient linear structure responses to dynamic loads or base acceleration. Modal superposition method is used in GSA linear time history analysis.

The dynamic equation of motion of structure subjected to dynamic loads is:

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{p}f(t)$$

where

$\mathbf{M}$	mass matrix
$\mathbf{u}, \dot{\mathbf{u}}, \ddot{\mathbf{u}}$	nodal displacement, velocity & acceleration vectors
$\mathbf{C}$	damping matrix
$\mathbf{K}$	stiffness matrix
$\mathbf{p}$	load vector
$f(t)$	time variation of load function

If the excitation is base acceleration,

$$\mathbf{p} = \mathbf{M}\mathbf{v}$$

$\mathbf{v}$	influence vector represents the displacements of the masses resulting from static application of a unit base displacement defined by the base excitation direction

$$f(t) = \ddot{u}_g(t)$$

$u_g(t)$ the base acceleration

To use the results (mode shapes) from modal dynamic analysis, the nodal displacements, velocities and accelerations can be expressed in modal coordinates as:

$$\mathbf{u} = \Phi \mathbf{q}$$

$$\dot{\mathbf{u}} = \Phi \dot{\mathbf{q}}$$

$$\ddot{\mathbf{u}} = \Phi \ddot{\mathbf{q}}$$

where:

Φ matrix of m mode shapes

$\mathbf{q}, \dot{\mathbf{q}}, \ddot{\mathbf{q}}$ displacement, velocity & acceleration vectors in modal (generalized) coordinates, the length of the vectors is m

$$\mathbf{M}_i \ddot{\mathbf{q}}_i + \mathbf{C}_i \dot{\mathbf{q}}_i + \mathbf{K}_i \mathbf{q}_i = \mathbf{p}_i f(t)$$

Setting

$$\mathbf{M}_i = \phi_i^T \mathbf{M} \phi_i$$

$$\mathbf{C}_i = \phi_i^T \mathbf{C} \phi_i$$

$$\mathbf{K}_i = \phi_i^T \mathbf{K} \phi_i$$

$$\mathbf{p}_i = \phi_i^T \mathbf{p}$$

This gives a single degree of freedom problem and it can be solved using any of the direct numerical analysis methods such as Newmark method and central difference method (Newmark method is used in GSA). There are m such equations that are corresponding to each of the modes from the modal dynamic analysis. Superimposing the responses from each of the one degree of freedom problem given by Eq(3), the total responses of the structure can be obtained from:

$$\mathbf{u} = \sum_{i=1}^m \phi_i q_i(t)$$

$$\dot{\mathbf{u}} = \sum_{i=1}^m \phi_i \dot{q}_i(t)$$

$$\ddot{\mathbf{u}} = \sum_{i=1}^m \phi_i \ddot{q}_i(t)$$

16.4 Footfall Analysis

GSA footfall analysis (or in full footfall induced vibration analysis) is to calculate the elastic vertical nodal responses (acceleration, velocity and response factor etc) of structures to human footfall loads (excitations). The human footfall loads $F(t)$ are taken as periodical loads. According to Fourier Series, the period footfall loads can be expressed as:

$$F(t) = G \left\{ 1.0 + \sum_{h=1}^H r_h \sin\left(\frac{h\pi}{\tau} t\right) \right\}$$

where

$F(t)$	periodical footfall loads
G	the body weight of the individual, input parameter in GSA footfall analysis
h	number of Fourier terms
r_h	Fourier coefficient (or dynamic load factor), the actual values of dynamic load factors can be found from reference 24, 25 and 26 in the bibliography. Alternatively, the dynamic load factors used in footfall analysis can be viewed from footfall analysis chart view after footfall analysis. The dynamic load factors can also be defined by the users in the DLF (Dynamic Load Factor) table
T	the period of the footfall (inverse of walking frequency)
H	the number of Fourier (harmonic) terms to be considered, 4 is used for walking on floor using CCIP-016 method and SCI method etc and 2 is used for walking on stairs for Arup method

After subtracting the static weight of the individual (since it does not vary with time and does not induce any dynamic response), the dynamic part of the footfall loads are the sum of a number of harmonic loads as shown below:

$$F(t) = G \sum_{h=1}^H r_h \sin\left(\frac{h\pi}{T}\right)$$

There are two distinctive responses from the footfall excitation, the resonant (steady state) and transient. If the minimum natural frequency of a structure is higher than 4 times the highest walking frequency (see reference 24), the resonant response is normally not excited since the natural frequencies of the structure are so far from the walking (excitation) frequency, therefore the transient response is normally in control, otherwise, the resonant response is probably in control. Both resonant (steady state) and transient analyses are considered in GSA footfall analysis, so the maximum responses will always be captured.

Resonant response analysis

As footfall loads are composed of a number of harmonic loads (components), harmonic analysis is used to get the responses for each of the harmonic components of footfall loads and then to combined them to get the total responses. From one of the harmonic components (e.g. component h) of footfall loads in equation above and the given walking frequency ($1/T$), the following dynamic equation of motion can be obtained:

$$\mathbf{M}\ddot{\mathbf{u}}_h + \mathbf{C}\dot{\mathbf{u}}_h + \mathbf{K}\mathbf{u}_h = \delta_k G r_h \sin(\omega t)$$

where

$\mathbf{M}$	mass matrix
$\mathbf{u}, \dot{\mathbf{u}}, \ddot{\mathbf{u}}$	nodal displacement, velocity & acceleration vectors under harmonic load h of the footfall loads
$\mathbf{C}$	damping matrix
$\mathbf{K}$	stiffness matrix
δ_k	unit vector used to define the location of the harmonic load. All the components in this vector are zero except the k th term that corresponds to the vertical direction of the node subjected footfall load

Since the number of footfalls is limited and the full resonant response from the equation above may not always be achieved, a reduction factor $\rho_{h,m}$ for the dynamic magnification factors μ is needed to account for this non-full resonant response. According to reference 24, the reduction factor can be calculated from:

$$\rho_{h,m} = 1 - e^{-2\pi}$$

where

ζ_m	damping ratio of mode m
$N = 0.55h$	
h	harmonic load number
W	number of footfalls

Applying this reduction factor to the dynamic magnification factors (μ) in [Harmonic Analysis](#), this equation can be solved using the method described in [Harmonic Analysis Theory](#) section. Repeating this analysis, the responses from the other harmonic loads of the footfall can also be obtained. The interested results from this analysis are the total vertical acceleration and the response factor from all harmonic loads of the footfall. The total vertical acceleration is taken as the square root of the sum of squares of the accelerations from each of the harmonic analyses. The response factor for each of the harmonic loads is the ratio of the nodal acceleration to the base curve of the Root Mean Square acceleration given in reference 25 as shown below. This total response factor is then taken as the square root of the sum of squares of the response factors from each of the harmonic loads. According to this, the total acceleration and response factor can be calculated from:

$$a_i = \sqrt{\sum_{h=1}^H \ddot{u}_{i,h}^2}$$

$$f_i = \sqrt{\sum_{h=1}^H f_{i,h}^2} = \sqrt{\sum_{h=1}^H \left(\frac{\ddot{u}_{i,h}}{a_i} \right)^2}$$

where

a_i	the maximum acceleration at node i
$\ddot{u}_{i,h}$	the maximum acceleration at node i by the excitation of harmonic load h
H	the number of harmonic components of the footfall loads considered in the analysis
f_i	the response factor at node i
$f_{i,h}$	the response factor at node i by the excitation of harmonic load h
w_i	frequency weighting factor and it is a function of frequency - for standard weighting factors see Table 3 of BS6841 (Bibliography 31)

Transient response analysis

The transient response of structures to footfall forces is characterised by an initial peak velocity followed by a decaying vibration at the natural frequency of the structure until the next footfall. As the natural frequencies of the structure considered in this analysis is much higher the highest walking frequency, there is no tendency for the response to build up over time as it does in resonant response analysis. The maximum response will be at the beginning of each footfall. Each footfall is considered as an impulse to the structure, according to references 35 & 29, the design impulse can be calculated from

When walking on floor (Concrete Center/Arup method)

$$I_{des,m} = 54 \frac{f}{f_n}$$

When walking on floor (SCI P354 method)

$$I_{des,m} = 60 \frac{f^{1.43}}{f_n^{1.30}}$$

When walking on stairs (Concrete Center/Arup method)

$$I_{des,m} = \frac{1\text{E}}{f}$$

When walking on stairs (SCI P354 method)

$$I_{des,m} = 0$$

where:

- $I_{des,n}$ design footfall impulse for mode m (Ns)
- f the walking frequency (Hz)
- f_m the natural frequency of mode m of the structure (Hz)
- Q body weight of the walker (N)

According to this impulse, the peak velocity in each mode is given by:

$$\hat{v}_m = u_{e,m} u_{r,m} \frac{I_{d_i}}{N}$$

and the peak acceleration in each mode is given by:

$$\hat{\alpha}_m = 2\pi f_m \hat{v}_m = 2\pi f_m u_{e,m} u_{r,i}$$

where

- $\hat{v}_m$ peak velocity of mode m by the footfall impulse
- $\hat{\alpha}_m$ peak acceleration of mode m by the footfall impulse
- $u_{e,m}, u_{r,i}$ the vertical displacements at the excitation and response nodes respectively in mode m
- M_m modal mass of mode m

The variation of the velocity with time of each mode is given:

$$v_m(t) = \hat{v}_m e^{-2\pi\zeta f_m t} \sin(2\pi f_m t)$$

and the variation of the acceleration with time of each mode is given:

$$\alpha_m(t) = \hat{\alpha}_m e^{-2\pi\zeta f_m t} \sin(2\pi f_m t)$$

where

- ζ is the damping ratio of mode m

The final velocity and acceleration at the response node are the sum of the velocities and accelerations of all the modes, M , that are considered

$$v(t) = \sum_m^M v_m$$

$$\alpha(t) = \sum_m^M \alpha_m$$

This gives the peak velocity and peak acceleration. The root mean square velocity and root mean square acceleration can be calculated from the period of the footfall T

$$v_{RMS} = \sqrt{\frac{1}{T} \int_0^T v^2(t) dt}$$

$$\alpha_m(t) = \sqrt{\frac{1}{T} \int_0^T \alpha_m^2(t) dt}$$

The response factor at time t (t is from 0 to T and T is the period of the footfall loads) can be calculated from:

$$f_R(t) = \frac{1}{n} \sum_{m=1}^M \alpha_m(t)$$

where:

w_m the frequency weighting factor corresponding to the frequency of mode m

The final transient response factor, based on the root mean square principle, is given by

$$f_R = \sqrt{\frac{1}{T} \int_0^T f_R^2(t) dt}$$

Part XVII

17 GsRaft Analysis

GsRaft controls the solution of soil-structure interaction analysis, by iterating between a solution of the structural analysis using Gss and the soil analysis using [Pdisp](#) until convergence of nodal displacements is achieved.

Iteration

For the soil interaction nodes on raft, the analysis iterates through the following steps until convergence is reached:

1. For each soil interaction node, a spring support will be generated with default support stiffness if it does not exist. If the support spring exists, it will be used and its stiffness will be modified during the analysis. After analysis, its stiffness will be restored.
2. Run Gss linear static analysis to obtain the displacements u_{raft} and spring-support forces f_s for each of the interaction nodes
3. Calculate soil pressure p_i under each of the soil interaction nodes using

$$p_i = \frac{f_i}{A}$$

if

$$p_i < p_{min}$$

then

$$p_i = p_{min}$$

if

$$p_i > p_{max}$$

then

$$p_i = p_{max}$$

where:

A is the interaction area associated with the interaction node.

p_{min} is the minimum pressure limit of the soil

p_{max} is the maximum pressure limit of the soil

4. Run Pdisp analysis to obtain the settlements of soil, u_{soil} under the applied pressure loads
5. Check the differences between raft displacements and soil settlements, if they are smaller than the residual limit, save the results and stop the analysis, otherwise go to step 6
6. re-calculate the support spring stiffnesses according to the support spring forces and the soil settlements using the following equation and go to step 2.

$$k = \frac{f_i}{u_i}$$

Note that there is a damping coefficient, d , that can be used to modify the stiffness update, i.e. reserve some percentage of the previous stiffness. The value of damping coefficient is between 0% and 100%. If damping coefficient is specified, the new stiffness will be calculated from:

$$k = d \cdot k_{prev} + (1 - d)$$

For the soil interaction nodes on the piles, the analysis iterates through the following steps until convergence is reached:

1. For each soil interaction node, a spring support in X, Y & Z direction will be generated with support stiffness calculated from soil settlement under unit point load.
2. Run Gss linear static analysis to obtain the displacements u_{raft} and spring-support forces f_r for each of the interaction nodes
3. Calculate the soil reaction forces to the interaction nodes from:

$$\begin{aligned} f_x &= A_x \cdot c \\ f_y &= A_y \cdot c \\ f_z &= A_s \cdot c \end{aligned}$$

where:

f_x , f_y and f_z - soil reaction forces to the interaction node in X, Y and Z directions

A_x , A_y & A_s - the interaction areas for the interaction node in X, Y and vertical shear direction and they are:

$$A_x = B_v H_z; \quad A_y = B_x H_z; \quad A_s =$$

P_{xy} - the perimeter of the pile section, the value depends on the section shape, see below for the calculation of different section shapes.

σ_x , σ_y & τ_z - the stress in X, Y direction and the shear stress in vertical shear direction and they can be calculated from:

$$\sigma_x = C_x \sigma_{\max.x}; \quad \sigma_y = C_y \sigma_{\max.y}; \quad \tau_z =$$

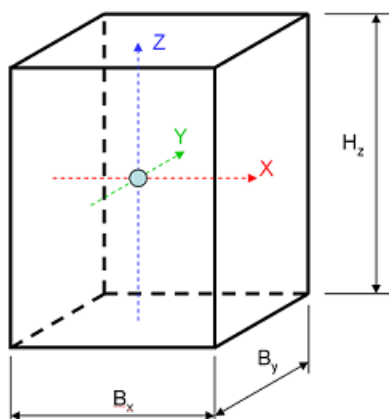
C_x , C_y & C_z - pile soil interaction coefficients and can be obtained from the Pile Soil Interaction Coefficient (PSIC) curves defined by the users as shown below.

When obtaining C_x , C_y & C_z from user defined PSIC curve, the meanings of other parameters are:

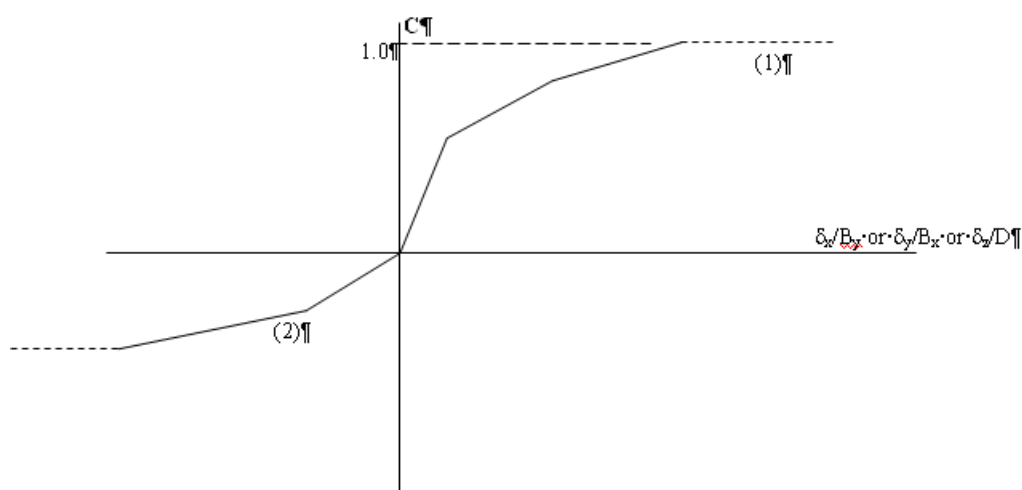
δ_x , δ_y & δ_z - the differences of the pile displacements and the soil settlements at the corresponding points.

D - equivalent diameter of the pile, see below for the calculation of pile sections other than circular.

4. Calculate the pressure loads on soil according to the soil reaction forces (f_x , f_y and f_z) and the interaction areas (A_x , A_y & A_s)
5. Do soil settlement analysis using the embedded Pdisp program
6. Calculate the compensation forces to counter balance the support spring forces due to the use of constant support spring stiffness.
7. Check convergence, if satisfied, stop, otherwise, goto Step 2.



Pile interaction node and relevant dimensions



Pile Soil Interaction Coefficient Curve

Convergence

The solution is converged if the difference of raft/pile displacements and soil settlements are smaller than the predefined acceptable residual. The residual can be defined in two ways.

- **Absolute residual** - the residual is defined directly such 0.1 mm, 1.0 mm or 5.0 mm..
- **Relative residual** - a percentage is defined that is used to calculate the actual residual based on the largest soil settlement. The actual residual is equal to the percentage defined times the largest soil settlement.

Part XVIII

18 GSBridge Theory

While Bridge Analysis uses the GSS solver, the associated pre- and post- processors merit a separate description.

[Automatic UK/HK Path Generation](#)

[Automatic EC1 Path Generation](#)

[EC1 - UK Carriageway Path load optimisation](#)

[Influence Analysis](#)

[Lane by Lane Path optimisation](#)

[UK/HK Carriageway optimisation](#)

18.1 Automatic UK/HK Path Generation

Carriageway paths are first split into Lane Paths between 2.5m and 3.65m wide in accordance with BD37. Next Vehicle Paths along the division between lanes are defined. Influence analysis is carried out on these key paths. Additional Vehicle Paths are then set up at 1.75m from the edge of each lane and at 4.25m (1.75m + 2.5m) from the edge of each lane. Influence lines on these paths are determined by linear interpolation. These will then be used during carriageway optimisation.

18.2 Automatic EC1 Path Generation

Carriageway paths are first split into Lane Paths 3m wide starting at first the extreme left then from the extreme right of the carriageway in accordance with EC1. Next Vehicle Paths along the division between lanes are defined. Influence analysis is carried out on these key paths. Additional Vehicle Paths are then set up at 4m from the edge of each lane. Influence lines on these paths are determined by linear interpolation. These will then be used during carriageway optimisation.

18.3 EC1 - UK Carriageway Path load optimisation

Carriageway Path load optimisation is performed automatically after the influence analysis. Basic Bridge Loads are derived as for lane by lane optimisation and then combined in accordance with the rules in the UK NA to EN1991-2 to give a single maximum (most positive) and minimum (most negative) Static Bridge Load case for each influence effect. The lane factors for each possible transverse position of the SV/SOV vehicle and the effect of each case are available in the EC1 Optimisation Summary output table.

18.4 Influence Analysis

For bridge loading, where large concentrated loads move across a structure, it is often useful to plot influence lines to determine the optimum position of a load to maximise an effect.

Each point on an influence line is the magnitude of a specified effect (eg moment, displacement, reaction etc) at a specified point on the structure (node or position on element) of a unit load, plotted at the point of application of the load. An influence line is the line joining these points along the line of travel of the load.

The influence analysis option in GSA produces influence lines along the centre of every path specified in the Paths module (or on key [automatically generated paths](#)), for each effect specified in the Node Influence Effect and Element Influence Effect modules.

The influence lines are produced by analysing the effect of a knife edge load (KEL) at approximately 100 points along each path, and storing the specified effects only.

All the influence lines produced by this analysis are stored in a single analysis case, and can be viewed in graphical or tabular form. As an aid to optimisation, details of each lobe of an influence line, where a lobe is generally the section of the influence line between points of zero influence, can also be tabulated. For Lane Paths only, lobes are split where the effective lobe length (the base length of a triangle with the same maximum value as the lobe) would be less than 90% of the actual lobe length.

No other results are available for this analysis case.

More:

[Lane by Lane Path optimisation](#)

[UK/HK Carriageway optimisation](#)

18.4.1 Lane-by-lane Path load optimisation

The path load optimisation tool uses the results of an influence analysis to generate a series of Static Bridge Loads, based on the loading information specified in the Path Loading module.

For each influence effect and each entry in the Path Loading module, the optimiser produces two groups of static bridge loading, to give the maximum (most positive) and minimum (most negative) effect.

Path loading can be a standard (code dependant) load type, or a user defined load type. For details of the application of standard type loading the user should refer to the relevant code, details of the versions of codes used are given in the description of Path Loading. User defined path loading can be made up of vehicles, KELs, and VUDLs.

The following rules are used for optimisation, these are based on the UK loading code (BD37) but are generally applicable:

- Unless noted otherwise in the definition of [Standard Bridge Loading](#) only one KEL or vehicle will be placed in each path, across the width of the path.
- VUDLs will always be applied over complete lobes of an influence line (except when placed outside a vehicle exclusion length, see below) and over the full width of the path.
- For Vehicle + VUDL loading, the VUDL will be placed starting at the specified exclusion length either side of the midpoint of the Vehicle (if effect at this point is adverse, or at the start of the next adverse lobe if not). Loaded length for calculating intensity of VUDL is the total length of adverse lobes, including each adverse lobe in the exclusion length.
- Intensity of VUDL will be calculated for the sum of the lengths of the adverse lobes. The worst case from 1, 2 or more lobes loaded will be determined.

18.4.2 UK/HK Carriageway Path load optimisation

Carriageway Path load optimisation is performed automatically after the influence analysis. Basic Bridge Loads are derived as for [lane by lane optimisation](#) and then combined in accordance with the rules in BD37 to give a single maximum (most positive) and minimum (most negative) Static Bridge Load case for each influence effect. The lane factors for each possible transverse position of the HB vehicle and the effect of each case are available in the UK/HK Optimisation Summary output table.

Part XIX

19 Steel Member Design

[Steel Member Restraints](#)

[Steel Restraint Properties](#)

[Steel Member Section Axes](#)

[Member Results](#)

[Steel Checks to BS5950-1:2000](#)

19.1 Steel Member Restraints

Member Restraints allow a member to have its restraint defined at one location along its length. They allow specification of restraint in two different ways: by use of a relatively user-friendly dialog box, or by an abbreviated “mnemonic” text syntax. One or more member restraints can be applied to a member by assembling them into a [Restraint Property](#).

Some of the attributes of a restraint being set in combination with others imply other restraints are also set, even though the user has not specifically stated that the additional restraint axes are restrained. For example, if both the top and bottom flanges of a member are laterally restrained, then it is clear that St. Venant torsion is also achieved at this location as well. Similarly, if the top flange is restrained and full St. Venant torsion restraint is also manually restrained, then the bottom flange is assumed also to have been restrained. With regard to warping restraint, a more conservative approach is taken, and minor axis bending restraint is not assumed to imply warping restraint of the flanges, nor is warping restraint assumed to imply any overall bending restraint of the section. This is because (using an example) although an RHS section may have its “major-axis” flanges welded to a stiff end plate (thus achieving flange warping restraint), the webs could be free. This would allow significant minor axis bending flexibility in comparison to a section that had been welded all round.

A table summarising the available restraint axes is shown below:

Restraint Axis	Possible values	Description	Comments
Compression Flange Warping	·None ·Partial ·Full	Rotational fixity of flange about its stiff axis	Advanced feature! Use with caution. These settings are applied to either the top or bottom flanges depending on the applied moment at the relevant position. (This could be different depending on the case selected).
Tension Flange Warping	·None ·Partial ·Full		
Top Flange Warping	·None ·Partial ·Full		
Bottom Flange Warping	·None ·Partial ·Full		

Torsion	·None ·Friction/ Bearing on support only ·Partial ·Full	St. Venant torsion restraint	Full restraint is assumed if both flanges are laterally restrained.
Major Axis Rotation	·None ·Partial ·Full	Major (stiff) axis restraint	
Minor Axis Rotation	·None ·Partial ·Full	Minor (weak) axis restraint	
Compression Flange Lateral	·None ·Full	Lateral restraint: restraint against movement transverse to the member, and in the plane of the flange	Advanced feature! Use with caution. These settings are applied to either the top or bottom flanges depending on the applied moment at the relevant position. (This could be different depending on the case selected).
Tension Flange Lateral	·None ·Full		
Top Flange Lateral	·None ·Full		Full restraint is assumed if the bottom flange is laterally restrained and the full torsion restraint is also applied.
Bottom Flange Lateral	·None ·Full		Full restraint is assumed if the top flange is laterally restrained and the full torsion restraint is also applied.
Major Axis Shear	·None ·Full	Restraint of shear centre against movement in the direction associated with major axis bending.	
Minor Axis Shear	·None ·Full	Restraint of shear centre against movement in the direction associated with minor axis bending.	Full restraint is assumed if both flanges are laterally restrained.

See also:

[Step By Step Guide: Member Restraints](#)

[Steel Member Design: Restraint Properties](#)

19.2 Steel Restraint Properties

Steel Restraint properties are a list of member restraints. Each is assigned to a node or span position in a member either by number or by a key word such as “all”, “end1” or “end2”. Thus a restraint property with a row referring to node 2 would apply the relevant restraint to the second node along the member (not to node number 2). Similarly, a row referring to span “all” would apply the restraint to all restraints not specifically referred to elsewhere in the member restraint.

Specific numerical and “end1” or “end2” restraint specifications take precedence over “all” specifications.

See also:

[Step by Step Guide : Steel Restraint Properties](#)

[Steel Member Design: Member Restraints](#)

19.3 Steel Member Section Axes

Steel Member Section axes are differently labelled from GSA local element axes in order to preserve the terminology used in codes. For doubly symmetric sections, the stiffer bending axis (major) is the X-X axis and the less stiff axis (minor) is the Y-Y axis. Only limited reference is made to the ZZ (torsion) axis (along the member).

19.4 Member Results

Results returned from checks of members come in a variety of forms.

Verbose text

During the member checks, much of the calculation is written to this part of the result. This is the place to look if you do not understand any of the other results a member is getting.

Brief text

Contains a summary of the exceptional (bad and interesting) things that have happened during the check.

Summary text

Textual summary of the utilisation ratios and flags that have been set.

More:

[Steel Member Design: Utilisation ratios](#)

[Steel Member Design: Flags](#)

19.4.1 Utilisation ratios

Measures of the utilisation of the member's capacity. Remember that their values will not be directly comparable from design code to design code, and that the calculation of some of them is highly non-linear. If any of them is greater than 1, then the member has failed.

All of the utilisations are the worst of their kind encountered along the member's length.

Envelopes of utilisation ratios are the maximum and minimum of the utilisation ratios calculated for each permutation in the envelope.

The ratios available are:

Ratio

Comment

Overall	worst of all the other utilisation ratios
Local Combined	local force/capacity interaction
Local Axial	
Local Moment Major	local major axis moment
Local Moment Minor	local minor axis moment
Local Shear Major	
Local Shear Minor	
Local Torsion	
Buckling Combined	buckling interaction
Buckling LT	lateral torsional buckling
Buckling Axial Major	
Buckling Axial Minor	
Buckling Axial Torsion	
Buckling Axial FT	

19.4.2 Flags

These can give a quick indication of what has happened in the member's check.

The flags that can be set are:

Flag Name	Comments
NOTIFY	
WARNING	causes NOTIFY to be set
FAIL	causes WARNING to be set
NOT CHECKED	causes WARNING to be set
FAIL LOCAL	cause FAIL to be set
FAIL BUCKLE	
FAIL BUCKLE LAT_TORS	
FAIL BUCKLE AXIAL MAJOR	
FAIL BUCKLE AXIAL MINOR	
FAIL LOCAL AXIAL	
FAIL LOCAL BENDING MAJOR	
FAIL LOCAL BENDING MINOR	
FAIL LOCAL SHEAR MAJOR	
FAIL LOCAL SHEAR MINOR	
FAIL LOCAL TORSION	
INFO LOCAL SLENDER	cause NOTIFY to be set
INFO LOCAL SLENDER WEB	
INFO LOCAL SLENDER FLANGE	
INFO MEMBER SLENDER	causes WARNING to be set

By necessity, the structure of member results is not design code specific.

19.5 Steel Checks to BS5950-1:2000

[Input Data Requirements](#)

[Local Checks](#)

[Buckling Checks](#)

[Input Data](#)

[Limitations](#)

19.5.1 Input Data Requirements

All cases provided to the checker are assumed to be ultimate limit state (i.e. the member forces are fully factored). Any non ULS cases – e.g. unfactored wind load – will be treated as if they are ULS, and so will result in non-conservative utilizations.

Member Restraints will be used to find if a member is part of non-sway frame. This is used only when calculating the Equivalent Uniform Moment Factor. If the Member Restraints are not defined and effective lengths are overridden then it is assumed that the member is part of non-sway frame.

19.5.2 Local Checks

At each end of each subspan (LH and RH), the member's section is checked for the moments and forces applied to it.

Plastic, compact, semi-compact and slender sections are supported.

Allowance for slenderness is made by the use of effective section properties, and the full design stress, which is considerably less conservative than the adoption of a lowered design stress in conjunction with the gross section properties (as used by the Blue Book [19]).

Depending on the section's properties, the method of checking may vary. For example, if the section is doubly-symmetric and compact or plastic, biaxial bending may be checked using the M_{rx} and M_{ry} method which reduces the moment capacities with increasing axial load.

Shear

Shear is checked only if the section's web(s) are not slender. Warning is given if the section is slender in shear.

Bending moments

Moments are conservatively enhanced if slenderness leads to the effective centroid of the section moving, and axial force is present.

M_{cx} and M_{cy} are always calculated (reduced by coincident shears in either direction), and compared with the applied moments. The interaction of transverse shear is not covered by BS5950-1:2000 (which is therefore potentially non-conservative). This deficiency is rectified by the use of a Von Mises type capacity reduction. Shear force parallel to webs in bending is considered in accordance with BS5950-1:2000.

M_{rx} and M_{ry} (moment capacities with coincident axial force) are calculated for plastic and compact doubly symmetric sections, and compared with the applied moments. They are not reduced by coincident shears. For this reason, their use is inhibited by "high" shears: ($F_v > 0.6P_v$) in either direction.

Axial force

Sections in tension use the user controllable Net Area Ratio to determine net area.

Slenderness is calculated differently for sections in overall tension, than for sections in

compression.

Torsion

At present, any significant torsional moment (greater than 5% of the torsional capacity) causes a warning, because torsion is not explicitly catered for by the code. The torsional capacity is calculated as follows: (1) CHS - thin wall theory is used; (2) RHS - thin wall theory is used but warping restraint is ignored; (3) I section - based on formulas in Table 20.4 in the book of "Roark's formulas for stress and strain" 5th edition.

Combined local effects

The section is checked against all the applied forces (except torsion) using the appropriate equations in BS5950-1:2000 4.8.2 and 4.8.3. The choice of equations depends on the section shape and the section's local buckling class.

19.5.3 Buckling Checks

The member is checked along its length for major axis axial buckling, minor axis axial buckling and LT buckling in turn. Once complete, relevant interactions of the buckling modes are considered.

Individual buckling mode checks

Each buckling mode check searches for appropriate buckling spans using the design restraints that have been applied to the member. Each buckling span is checked individually. Cantilever spans at either end of members are recognized and catered for.

The effective buckling length calculated can be overridden to either a multiple of the member length or to an absolute length. If this is done, the whole member length is used for determination of buckling parameters, such as equivalent uniform moment factors and applied axial forces.

Buckling interaction

During checks for the simple buckling modes (axial/major axis, axial minor axis, and LT) note is made of various parameters needed later when the buckling interaction check is carried out. These parameters are stored for each sub-span of the member. (See BS5950-1:2000 4.8.3.3)

The parameters are:

- F_{cx} maximum axial compression in the major axis buckling span
- F_{cy} maximum axial compression in the minor axis buckling span
- M_{b1} major axis bending moment capacity in the LT span
- M_{b2} opposite sense moment capacity in the LT span (for reverse curvature spans)
- M_{LT1} maximum applied major axis bending moment in the LT span
- M_{LT2} opposite sense major axis bending moment in the LT span (for reverse curvature spans)
- M_x maximum major axis moment in the major axis buckling span
- M_y maximum minor axis moment in the minor axis buckling span
- P_{cx} major axis axial buckling capacity
- P_{cy} minor axis axial buckling capacity

• Z_x	section modulus about major axis
• Z_y	section modulus about minor axis
• m_{LT}	equivalent uniform moment factor for major axis moment in relevant LT span
• m_x	equivalent uniform moment factor for major axis moment in the major axis axial span
• m_y	equivalent uniform moment factor for minor axis moment in the minor axis axial span
• m_{yx}	equivalent uniform moment factor for minor axis moment in the major axis axial span
• p_y	design stress for the section

The checker steps through the sub-spans looking at this interaction data. It only carries out an interaction check once for each permutation of buckling spans, to reduce the volume of output.

Interaction is carried out according to clauses BS5950-1:2000 4.8.3.3.1, 4.8.3.3.2 and 4.8.3.3.3 depending on the section shape. The variant (c) that allows biaxial bending moments is always used, even when there is only bending about a single axis.

19.5.4 Input Data

This section explains particular (non-obvious) significances of input data to the BS5950-1:2000 member checker. See [Step By Step Guide: Steel Design](#) and the [Steel Member Design](#) chapter for general steel design input requirements.

Max Plastic/Elastic Ratio

In order to limit the effects of plasticity at the serviceability limit state (SLS), the design code limits the maximum benefit that can be obtained at the ultimate limit state from allowing the section to become plastic in bending. To comply with BS5950-1:2000 4.2.5.1, the limit should be set to 1.2 for simply supported members and cantilevers, and 1.5 for members with moment continuity between spans. If the ratio is set to 0, the checker will assume 1.2 (which is the more conservative choice available).

Single Angle Struts

BS5950-1:2000 4.7.10.2 and Table 25 make reference to the A-A, B-B and V-V axes in relation to axial buckling. The V-V axis is the minor principal axis. The AA and BB axes are the X-X and Y-Y axes, but which is which depends on the design of the connections of the member to the rest of the structure. The checker only knows about the applied member restraints.

The rotational restraint applied by a two bolt group in clearance holes in one leg (clause 4.7.10.2 (a)) should be represented by restraining the section:

- fully about the axes of the bolts, and
- partially about an axis perpendicular to the bolts and the member.

The rotational restraints applied by a two bolt group, one of which uses a kidney shaped hole should be represented by restraining the section:

- partially about an axis perpendicular to the bolts and the member.

The rotational restraint applied by a single bolt group should be represented by leaving the member free about both the X-X and Y-Y axes.

The local capacity check needs to know which legs are connected at the location of the check (clause 4.6.3.1). This information is partly provided by the rotational restraints if they are applied as above. In cases of ambiguity, the connection restraint settings are used. Restraint settings "A"

and “B” imply that legs A and B respectively are connected. Where both legs are connected, both “A” and “B” should be set. Even if “A” and/or “B” are not set, the checker can often make assumptions using the rotational restraints alone. Examples of connections requiring use of “A” and “B” include all those consisting of a single bolt through either leg.

19.5.5 Limitations

- Torsion not fully considered.
- Unsymmetrical sections not yet supported.
- Mixed steel grades in single section not yet supported.
- Weld material disregarded in assessment of section properties.
- Web high-shear buckling treated conservatively in respect of web capacity.
- Whole structure stability issues are not considered, since the checker only knows about one member at a time. Users must construct their models to account for these effects if appropriate.

19.6 Steel Checks to EC3

[Input Data Requirements](#)

[Local Checks](#)

[Buckling Checks](#)

[Input Data](#)

[Limitations](#)

[National Annexes](#)

19.6.1 Input Data Requirement

All cases provided to the checker are assumed to be factored loads. See [BS5950-1:2000](#) for more details.

19.6.2 Local Checks

Sections are checked at the ends of every sub-span of the member for the moments and forces acting on it.

Checks are carried out for Class 1, Class 2, Class 3 and Class 4 sections. Local buckling effects for Class 4 sections are included by an effective width in accordance with EN 1993-1-5.

For Class 4 CHS sections a reduced yield stress is used to account for local buckling.

For Channel and Tee sections elastic section verification is used.

Shear

Shear is checked according to EN 1993-1-1:2005 clause 6.2.6 if the section's web(s) are not slender. A warning is given if the section is slender in shear.

Bending moment

Moment capacity is calculated in accordance with EN 1993-1-1:2005 clause 6.2.5 and compared with the applied moments. Allowance is made for shear and torsion in moment capacity according to EN 1993-1-1:2005 6.2.8.

Moment capacities are reduced for Class 1 and Class 2 sections to account for axial force as per

EN 1993-1-1:2005 6.2.9.1.

Axial force

Compression and tension capacities are calculated in accordance with EN 1993-1-1:2005 6.2.4 and 6.2.3 respectively and compared with the applied axial force. For sections in tension the user defined 'Net Area Ratio' and 'Effective net area factor (Beta)' are used to determine tension capacity.

Torsion

At present, any significant torsional moment (greater than 5% of the torsional capacity) produces a warning. See [BS5950-1:2000](#) for more details.

Combined local effects

The section is checked against all the applied forces using the equations in EN 1993-1-1:2005 6.2.9 and 6.2.10. The choice of equations depends on the section shape and the section's local buckling class.

19.6.3 Buckling Checks

The member is checked along its length for major axis axial buckling, minor axis axial buckling, torsional buckling, flexural torsional buckling and lateral torsional buckling. Interaction checks are carried out according to EN 1993-1-1:2005 6.3.3; see [National Annexes](#) for interaction factors. See [BS5950-1:2000](#) for more details on buckling checks.

19.6.4 Input Data

Effective net area factor (Beta)

For angles connected through one leg, the tensile strength of a section is affected by shear lag. To comply with EN 1993-1-8:2005 3.10.3 the Effective net area factor (Beta) is used to compute the design tensile strength. This factor is calculated for a single bolt. The effect of shear lag in tension is ignored for members with other section shapes.

Single Angle Struts

When considering [Member Restraints](#) for angle sections EC3 interprets restraint settings "A" and "B" as relating to the longer and shorter leg respectively. Where both legs are connected, both "A" and "B" should be set.

19.6.5 National Annexes

The following nationally defined parameters are used.

EN 1993-1-1:2005 Eurocode 3 (generic):

The following parameters are used for design checks to EN 1993-1-1:2005 Eurocode 3.

Clause 6.1 (1)

$$\gamma_{M0} = 1.00, \gamma_{M1} = 1.00 \text{ and } \gamma_{M2} = 1.25$$

Clause 6.3.2.2.

Table 6.3 is used for imperfection factors.

Clause 6.3.2.3 (1)

$\lambda_{LT,0} = 0.4$, $\beta = 0.75$ and table 6.5 of EN 1993-1-1:2005 are used for the lateral torsional buckling curve. For the sections not available in the table, buckling curve d is used.

Clause 6.3.2.3 (2)

$f = 1 - 0.5(1 - k_c)[1 - 2.0(\lambda_{LT} - 0.8)^2]$ and $f \leq 1.0$; Table 6.6 are used for the correction factor k_c .

Interaction factors:

Annex B is used for interaction factors. For other than doubly symmetric sections, the λ value used for calculating interaction factors in any direction (say λ_x or λ_y) is the largest value of the following three

- 1) λ_x or λ_y depending on the direction.
- 2) λ_T for torsion.
- 3) λ_{TF} for flexural torsion.

Table B.3 is used for equivalent uniform moment factors; steel checker assumes that the spans are loaded with uniform loading.

Effective Length:

Effective buckling lengths for flexural buckling are calculated in accordance with Table 22 of BS5950-1:2000.

EN 1993-1-1:2005 Eurocode 3 (UK):

The following parameters are used for design checks to the UK National Annex to EN 1993-1-1:2005 Eurocode 3.

Clause 6.1 (1)

$$\gamma_{M0} = 1.00, \gamma_{M1} = 1.00 \text{ and } \gamma_{M2} = 1.10$$

Clause 6.3.2.2.

Table 6.3 is used for imperfection factors.

Clause 6.3.2.3 (1)

For rolled sections and hot-finished and cold-formed hollow sections:

$$\lambda_{LT,0} = 0.4 \text{ and } \beta = 0.75$$

Welded sections:

$$\lambda_{LT,0} = 0.2 \text{ and } \beta = 1.00$$

Table in clause 2.17 of UK NA to EN 1993-1-1:2005 is used for the lateral torsional buckling curve. For the sections not available in the table, buckling curve d is used.

Clause 6.3.2.3 (2)

Clause NA 2.18 is used to calculation modification factor, f .

Interaction factors:

As for Eurocode 3 (generic) .

Effective Length:

Effective buckling lengths for flexural buckling are calculated in accordance with Table 22 of BS5950-1:2000.

EN 1993-1-1:2005 Eurocode 3 (NL):

The following parameters are used for design checks to the Netherlands National Annex to EN 1993-1-1:2005 Eurocode 3.

Clause 6.1 (1)

$$\gamma_{M0} = 1.00, \gamma_{M1} = 1.00 \text{ and } \gamma_{M2} = 1.25$$

Clause 6.3.2.2.

Table 6.3 is used for imperfection factors.

Clause 6.3.2.3 (1)

$\lambda_{LT,0} = 0.4$, $\beta = 0.75$ and table 6.5 of EN 1993-1-1:2005 are used for the lateral torsional buckling curve. For the sections not available in the table, buckling curve d is used.

Clause 6.3.2.3 (2)

$f = 1 - 0.5(1 - k_c)[1 - 2.0(\lambda_{LT} - 0.8)^2]$ and $f \leq 1.0$; Table 6.6 are used for the correction factor k_c .

Interaction factors:

As for Eurocode 3 (generic) .

Effective Length:

Effective buckling lengths for flexural buckling are calculated in accordance with clause 12.1.1.3 of NEN 6770.

Other checks:

In addition to the EN 1993-1-1:2005 checks, the following checks are performed.

1) Combined bending, axial and shear force is checked in accordance with the clause 11.3.1.1 to 11.3.1.3 of NEN 6770

2) Biaxial bending interaction is checked in accordance with the clause 11.3.2.1 to 11.3.2.3 of NEN 6770.

EN 1993-1-1:2005 Eurocode 3 (FR):

The following parameters are used for design checks to the NF EN 1993-1-1:2005 Eurocode 3.

Clause 6.1 (1)

$$\gamma_{M0} = 1.00, \gamma_{M1} = 1.00 \text{ and } \gamma_{M2} = 1.25$$

Clause 6.3.2.2.

Table 6.3 is used for imperfection factors.

Clause 6.3.2.3 (1)

For rolled sections (I or H sections):

$$\lambda_{LT,0} = 0.2 + 0.1(b/h), \alpha_{LT} = 0.4 - 0.2(b/h)\lambda_{LT}^2 \geq 0 \text{ and } \beta = 1.0$$

Welded sections (doubly symmetric I sections):

$$\lambda_{LT,0} = 0.3(b/h), \alpha_{LT} = 0.5 - 0.25(b/h)\lambda_{LT}^2 \geq 0 \text{ and } \beta = 1.0$$

Clause 6.3.2.3 (2)

The coefficient f is calculated as for Eurocode 3 (generic).

Note: The coefficient f is only applicable to the members without intermediate lateral restraints and no warping at the ends.

Interaction factors:

Annex A is used for interaction factors.

For other than doubly symmetric sections:

a) The χ value used in interaction equations in any direction (say χ_x or χ_y)

is the smallest value of the following three

- 1) χ_x or χ_y depending on the direction.
 - 2) χ_T for torsion.
 - 3) χ_{TF} for flexural torsion.
- b) Elastic verification is used in the interaction
- c) The equivalent uniform moment factor is limited to $C_{mi,0} \geq 1 - (N_{Ed}/N_{cr,i})$

Effective Length:

Effective buckling lengths for flexural buckling are calculated in accordance with Table 22 of BS5950-1:2000.

Elastic Critical Moment (Mcr):

For doubly symmetric I sections:

Mcr is calculated in accordance with the Annex MCR of NF EN 1993-1-1:2005. The factors C1 and C2 are taken from Table1 and Table2 of the Annex MCR.

Mcr is calculated in accordance with clause 6.2.1.4 of "The behaviour and design of steel structures to EC3" by Trahair et al. for the loading types not specified in the Annex MCR.

For other sections:

Elastic critical moment for lateral-torsion buckling is calculated in accordance with Annex I of EN 1999-1-1:2007

EN 1993-1-1:2010 Eurocode 3 (SS) - Singapore N.A to Eurocode 3:

The Singapore N.A. to Eurocode 3 i.e. "EN 1993-1-1:2010 Eurocode 3 (SS)" is identical to UK National Annex to EN 1993-1-1:2005 Eurocode 3.

19.6.6 Limitations

- Section classification is based on axial force and major axis bending moment. Minor axis bending moment is disregarded.
- Shear lag effects are not considered in bending.
- Elastic critical moment for lateral-torsion buckling is calculated in accordance with Annex I of EN 1999-1-1:2007. When the loads on the member are not found in the appropriate table of Annex I of EN 1999-1-1:2007, the worst values of C1, C2 and C3 are taken from the appropriate table assuming the ends are pinned.
- Transverse load checks not yet supported.
- Slender Angle sections not yet supported.
- Refer [BS5950-1:2000](#) for other limitations.

19.7 Steel Checks to AS4100:1998

[Input Data Requirements](#)

[Effective Section Properties](#)

[Local Checks](#)

[Buckling Checks](#)

19.7.1 Input Data Requirement

All cases provided to the checker are assumed to be factored loads including second order effects. See [BS5950-1:2000](#) for more details.

19.7.2 Effective Section Properties

Members subjected to bending are classified as Compact, Non-compact and Slender sections in accordance with AS4100:1998 clause 5.2 and Table 5.2. The effective section modulus is computed using the appropriate clause from AS4100:1998 5.2.3, 5.2.4 and 5.2.5.

Members subjected to axial compression are classified as Compact, Non-compact and Slender sections in accordance with AS4100:1998 clause 6.2.3

19.7.3 Local Checks

Sections are checked at the ends of every sub-span of the member for the moments and forces acting on it.

Shear

Shear is checked according to AS4100:1998 clause 5.11. While calculating the shear strength steel checker assumes that the section web is unstiffened.

The shear area is calculated based on clear web depth between flanges for the welded sections, and based on full web depth for rolled section. For RHS and Box sections the shear area is based on two clear web depths.

Shear capacities are reduced to account for moment in the direction of shear force according to AS4100:1998 5.12.3

Bending moment

Moment capacity at a section is calculated in accordance AS4100:1998 clause 5.2.1. See [Effective Section Properties](#) for details on calculation of effective section modulus.

Axial Force

Compression capacity at a section is calculated in accordance with AS4100:1998 clause 6.2.1. See [Effective Section Properties](#) for details on calculation of effective area of section.

Tension capacity at a section is calculated in accordance with AS4100:1998 clause 7.2 and 7.3. Sections in tension use the user defined 'Net Area Ratio'.

Torsion

At present, any significant torsional moment (greater than 5% of the torsional capacity) produces a warning.

Combined local effects

The section is checked against all the applied forces using the equations in AS4100:1998 clause 8.3.4. The choice of equations used depends on the section shape and the section's local buckling class.

Moments used in the local interaction check include the moments due to eccentricity in axial force. Eccentricity may be due to end connection and to shift in neutral axis in case of Unsymmetrical slender sections.

19.7.4 Buckling Checks

Axial Buckling

The member is checked along its length for major axis axial buckling and minor axis axial buckling according to AS4100:1998 clause 6.3.3.

Unsymmetrical sections are also checked for torsional and flexural torsional buckling according to AS4100:1998 clause 6.3.3. Lambda value will be computed in accordance with clause 6.3.4.

At present the effective length of the member is calculated from the factors available in AS4100:1998 Figure 4.6.3.2.

Lateral Torsional Buckling

Members with full lateral restraint are checked for Lateral torsional buckling in according to AS4100:1998 clause 5.3. Lateral restraints are verified in accordance with AS4100:1998 clause 5.3.2.

Lateral torsional buckling is checked in accordance with AS4100:1998 clause 5.6.1 and 5.6.2(i) for the member without continuous restraints. Effective lengths for lateral torsional buckling are calculated according to AS4100:1998 clause 5.6.3.

Buckling Interaction

Interaction checks are carried out according to AS4100:1998 clause 8.4.5.1 and 8.4.5.2 for compression and tension members respectively. Reduced moments used in compression member checks are calculated using the simplified equations in AS4100:1998 clause 8.4.2.2 and 8.4.4.1.

19.8 Steel Checks to HKSUOS:05

[Input Data Requirements](#)

[Effective Section Properties](#)

[Local Checks](#)

[Buckling Checks](#)

19.8.1 Input Data Requirements

All cases provided to the checker are assumed to be factored loads including second order p - small delta effects. See [BS5950-1:2000](#) for more details.

19.8.2 Effective Section Properties

Members are classified as Class 1, Class 2, Class 3 and Class 4 sections in accordance with HKSUOS:05 Table 7.1 and 7.2. The channel webs are classified according to BS5950-1:2000 Table 11 since HKSUOS:05 does not specify the same. Effective section properties are calculated according to the appropriate clause from HKSUOS:05 clause 11.3.4. For flanges with non-uniform compression the effective width of the plate is calculated according to EN 1993-1-5:2006 clause 4.4.

Effective Plastic Modulus

Class 3 semi-compact sections subjected to bending are designed using effective plastic modulus according HKSUOS:05 clause 7.5.

19.8.3 Local Checks

Sections are checked at the ends of every sub-span of the member for the moments and forces acting on it.

Shear

Shear is checked according to HKSUOS:05 clause 8.2.1. Shear is checked only if the section's web(s) are not slender. A warning is given if the section is slender in shear.

Bending moment

Moment capacity at a section is calculated in accordance with HKSUOS:05 clause 8.2.2. See [Effective Section Properties](#) for details on calculation of effective section modulus.

Moments are conservatively enhanced if slenderness or end connection leads to the effective centroid of the section moving, and axial force is present.

Moment capacities are reduced to take account of high shear according to HKSUOS:05 clause 8.2.2.2 and compared with the applied moments.

Axial Force

Compression capacity at a section is calculated in accordance with HKSUOS:05 clause 8.9.1. See [Effective Section Properties](#) for details on calculation of effective area of section.

Tension capacity at a section is calculated in accordance with HKSUOS:05 clause 8.6. Sections in tension use the user defined 'Net Area Ratio'.

Torsion

At present, any significant torsional moment (greater than 5% of the torsional capacity) produces a warning.

Combined local effects

The section is checked for interaction of axial, shear and moments using the equations in HKSUOS:05 clause 8.9.1 and 8.2.1.

Moments used in the local interaction check include the moments due to eccentricity in axial force. Eccentricity may be due to end connection and to shift in neutral axis.

19.8.4 Buckling Checks

Axial Buckling

The member is checked along its length for major axis axial buckling and minor axis axial buckling according to HKSUOS:05 clause 8.7. Unsymmetrical sections are checked for torsional and flexural torsional buckling according to the HKSUOS:05 clause 8.10. At present the effective length of the member is calculated from the factors available in HKSUOS:05 Table 8.6 based on the restraints on the member; this value can be overridden from the steel design properties.

Torsional and Flexural-Torsion Buckling

Unsymmetrical sections are checked for torsional and flexural torsional buckling according to the HKSUOS:05 clause 8.10

Lateral Torsional Buckling

Lateral torsional buckling is checked in accordance with HKSUOS:05 clause 8.3, Appendix 8.1 and Appendix 8.2.

Appendix 8.2 provide guidance to calculate λ_{Lt} for uniform I, H and channel with equal flanges. It also suggested to use recognized buckling analysis for other section shapes. Currently BS5950-1:2000 formulations are using to calculate λ_{Lt} for RHS and T sections.

Buckling Interaction

Interaction checks are carried out according to HKSUOS:05 clause 8.9.2. Since it is assumed that the analysis results are inclusive of second order p - small delta analysis, the amplification factor is ignored for non-sway frame members.

19.9 Steel Checks to SANS 10162-1:2011

[Input Data Requirements](#)

[Effective Section Properties](#)

[Local Checks](#)

[Buckling Checks](#)

[Limitations](#)

19.9.1 Input Data Requirements

All cases provided to the checker are assumed to be factored loads including second order elastic analysis effect. See [BS5950-1:2000](#) for other details.

19.9.2 Effective Section Properties

Members are classified as Class 1, Class 2, Class 3 and Class 4 sections in accordance with SANS 10162-1:2011 Table 3 and 4.

Effective section area is calculated in accordance with SANS 10162-1:2011 clause 13.3.3.

Effective section modulus for the section with class 3 or better web and class 4 flange is calculated in accordance with SANS 10162-1:2011 clause 13.5. If the web is class 4 and flange is class 3 or better then the revised factored moment of resistance is calculated in accordance with SANS 10162-1:2011 clause 14.3.4. Checker doesn't support the section if both the flange(s) and web are class 4.

19.9.3 Local Checks

Sections are checked at the ends of every sub-span of the member for the moments and forces acting on it.

Members with Angle sections are treated as strut and tie only member and the shear and moment capacities are assumed as zero.

Shear

For the webs of section with two flanges shear capacity is calculated using elastic analysis in accordance with SANS 10162-1:2011 clause 13.4.1.1. This clause applies for I, Channel sections with shear acting along the web and also for RHS sections.

For all other cases maximum shear stress is calculated in the section and compared with $0.66 \cdot \phi \cdot f_y$. The effect of shear buckling is not considered and this is non-conservative.

Bending moment

Moment capacity at a section is calculated in accordance with SANS 10162-1:2011 clause 13.5 and clause 14.3.4. See [Effective Section Properties](#) for details on calculation of effective section modulus. Moment capacity of a section is assumed as zero whose flanges and web are class 4

Moments are conservatively enhanced if slenderness or end connection leads to the effective centroid of the section moving, and axial force is present.

Axial Force

Compression capacity at a section is calculated in accordance with SANS 10162-1:2011 clause 13.3.1 and 13.3.4 and assuming member length as zero. See [Effective Section Properties](#) for details on calculation of effective area of section.

Tension capacity at a section is calculated in accordance with SANS 10162-1:2011 clause 13.2. Sections in tension use the user defined 'Net Area Ratio'.

Torsion

At present, any significant torsional moment (greater than 5% of the torsional capacity) produces a warning.

Combined local effects

The section is checked for interaction of axial and moments in accordance with SANS 10162-1:2011 clause 13.8.2 (b) and 13.8.3 (b).

Moments used in the local interaction check include the moments due to eccentricity in axial force. Eccentricity may be due to end connection and to shift in neutral axis.

19.9.4 Buckling Checks

Axial Buckling

The member is checked along its length for major axis axial buckling and minor axis axial buckling according to SANS 10162-1:2011 clause 13.3.1

Singly symmetric and unsymmetrical sections are also checked for torsional and flexural torsional buckling according to SANS 10162-1:2011 clause 13.3.2. Doubly symmetric sections are checked for torsional buckling in accordance with SANS 10162-1:2011 clause 13.3.2 (a).

SANS 10162-1:2011 Annex E (Figure E.1) is referred to estimate effective length factor using the restraint condition.

Bending Laterally unsupported members

Lateral torsional buckling is checked in accordance with SANS 10162-1:2011 clause 13.6.

Buckling Interaction

Interaction checks are carried out according to SANS 10162-1:2011 clause 13.8.2 and 13.8.3

19.9.5 Limitations

- Clause 15.3.4 suggest that "The factored resistances of the first compression web member and its connection shall be determined with their respective resistance factors, ϕ , multiplied by 0.85". To account for this user should explicitly enter the reduced yield strength in the steel design property.
- Web buckling, yielding and crippling checks are not implemented.
- Section with class 4 flange and class 4 webs are not supported.
- Design of cold formed sections are not supported.
- Refer [BS5950-1:2000](#) for other limitations.

19.10 Steel Checks to AISC360:05

All cases provided to the checker are assumed to be factored loads including second order small p-delta effects. See [BS5950-1:2000](#) for more details.

Section Classification

Section is classified as Compact, Non-compact and Slender in accordance with AISC360-05 Clause B4 and Table B4.1.

Tension

Section is checked at the ends of every sub-span of the member for the moments and forces acting on it.

Tension capacity at a section is calculated in accordance with AISC360-05 clause D2. Sections in tension uses the user defined 'Net Area Ratio' to compute Net Area and 'Effective net area factor (Beta)' to compute Effective Net Area.

Compression

Compression force is checked in accordance with AISC360-05 Chapter E. Checker uses appropriate equations depending upon the section shape and local buckling class. Effective length factor for the members under compression is taken in accordance with the AISC360-05 Table C-C2.2.

Bending

Section is checked for bending in accordance with AISC360-05 Chapter F using appropriate equations.

Shear

Shear is checked according to AISC360-05 Chapter G using appropriate equations. While calculating the shear strength steel checker assumes that the section web is unstiffened.

Torsion

At present, any significant torsional moment (greater than 5% of the torsional capacity) causes a warning.

Interaction checks

The section is checked against all the applied forces in accordance with AISC360-05 Clause H1 and H2. Any torsion on the section is ignored from the interaction checks.

Moments used in the interaction check includes the moments due to eccentricity in axial force. Eccentricity may be due to end connection and due to shift in neutral axis in case of Unsymmetrical slender sections.

Limitations

Limitations are the similar to for BS5950-1:2000. See BS5950-1:2000 Limitations for more details

19.11 Steel Checks to AISC LRFD

The input data requirements are the same as for BS5950-1:2000.

The effect sway of frame on the effective length of members has not been considered in member buckling check.

19.12 Steel Checks to IS800

1. IS800:1984 (Working stress method)

Input Data and requirements:

All cases provided to the checker are assumed to be working loads.

IS800:1984 does not use Max Plastic/Elastic Ratio specified in the design property.

For Member Restraints, IS800:1984 interprets connection A and B in the following manner.

- For angle sections A and B stand for longer, shorter leg respectively. Where both legs are connected, both A and B should be set.
- For I and RHS sections, A stands for the web and B for flange.

Local Checks:

Following Local checks or Section checks are carried out at each end of each subspan.

Shear strength check:

Shear is checked only for unstiffened webs. A warning is generated if the web is slender as per IS800:1984 6.7.3.1. Shear is checked to Clause 6.4.1 and 6.4.2.

Torsion checks:

At present, any significant torsional moment (greater than 5% of the torsional capacity) causes a warning, because torsion is not explicitly catered for by the code. See [BS5950-1:2000: Local checks](#) for details.

Bearing capacity checks

At present, bearing capacity checks are ignored.

Axial and bending strength checks:

Axial tensile, compressive and bending strength checks are carried as per section 4, 5 and 6 of IS800:1984 respectively without considering the effect of buckling (only local).

Combined local effects:

The section is checked against all the applied forces (except bearing) using the appropriate equations in IS800:1984 7.1.1.b, 7.1.2 and 7.1.4.

Buckling Checks

Following Buckling checks or member checks are carried out at for every member. See [BS5950-1:2000: Buckling Checks](#) for details.

Major and Minor axis axial buckling strength check:

Axial compression strength checks will be carried along major axis and minor axis as per the section 5 of IS800:1984.

Lateral Torsion buckling strength check:

The member is checked against lateral torsion buckling strength as per IS800:1984 6.2.3 and 6.2.4.

Buckling interaction strength check

These checks are carried out for the appropriate equations stipulated in section 7 of IS800:1984.

2. IS800:2007 (Working stress method)

The input data requirements and checks that are the similar to for BS5950-1:2000. See [BS5950-1:2000](#) for more details

Effective Net Area Factor (Beta)

The rupture strength of a section connected through one or more elements of section is affected by shear lag. To comply with IS800:2007 6.3.3 and 6.3.4, the Effective net area factor (Beta) is used to compute the design tensile strength of such sections to account for tearing of net section.

Part XX

20 RCSlab Theory

[Introduction](#)

[Data requirements](#)

[Other symbols used in this theory](#)

[The RCSlab sign convention](#)

[RCSlab analysis procedure](#)

[Appendix 1](#)

[Appendix 2](#)

[Appendix 3](#)

20.1 Introduction

RCSlab is a design postprocessor for reinforced concrete two-dimensional sections of uniform thickness subject to any combination of in-plane axial or shear force and out-of-plane bending moment and torsion. RCSlab is unable to allow for out-of-plane shear and through thickness forces.

The input to the postprocessor comprises applied forces and moments, reinforcement positions, section depth and material properties.

RCSlab arrives at a solution by dividing the section into several layers each with a certain state of in-plane stresses. These layered stresses are in equilibrium with the applied loads. The reinforcement requirements for each layer are then calculated and apportioned to the user specified reinforcement positions.

The results consist of areas of reinforcement for each of the four reinforcement positions specified or a message to the effect that RCSlab is unable to find a solution for the current data.

20.2 Data requirements

Each run of RCSlab obtains the following data in any consistent set of units from the GSA analysis or RCSlab design data as appropriate:

N_{xx}	Ultimate applied axial force per unit width in the x-direction
N_{yy}	Ultimate applied axial force per unit width in the y-direction
M_{xx}	Ultimate applied bending moment per unit width about the x-axis
M_{yy}	Ultimate applied bending moment per unit width about the y-axis
N_{xy}	Ultimate applied in-plane shear force per unit width
M_{xy}	Ultimate applied torsion moment per unit width
e	Minimum eccentricity ($e > 0$)
h	Section thickness ($h > 0$)
z_{t1}	Position of top reinforcement centroid in direction 1 ($0 < z_{t1} < h/2$)
z_{t2}	Position of top reinforcement centroid in direction 2 ($0 < z_{t2} < h/2$)
z_{b1}	Position of bottom reinforcement centroid in direction 1 ($-h/2 < z_{b1} < 0$)
z_{b2}	Position of bottom reinforcement centroid in direction 2 ($-h/2 < z_{b2} < 0$)

θ_1	Angle of reinforcement in direction 1, anticlockwise with respect to x-axis
θ_2	Angle of reinforcement in direction 2, anticlockwise with respect to x-axis
$A_{st1,min}$	Minimum top reinforcement to be provided in direction 1 ($0 < A_{st1,min}$)
$A_{st2,min}$	Minimum top reinforcement to be provided in direction 2 ($0 < A_{st2,min}$)
$A_{sb1,min}$	Minimum bottom reinforcement to be provided in direction 1 ($0 < A_{sb1,min}$)
$A_{sb2,min}$	Minimum bottom reinforcement to be provided in direction 2 ($0 < A_{sb2,min}$)
f_{cd1}	Uncracked compressive design strength of concrete ($f_{cd1} > 0$)
f_{cd2}	Cracked compressive design strength of concrete ($f_{cd2} > 0$)
f_t	Tensile design strength of concrete ($f_t > 0$)
ε_{ctrans}	Compressive plateau concrete strain ($\varepsilon_{ctrans} \geq 0$)
ε_{cax}	Maximum axial compressive concrete strain ($\varepsilon_{cax} \geq \varepsilon_{ctrans}$)
ε_{cu}	Maximum flexural compressive concrete strain ($\varepsilon_{cu} \geq \varepsilon_{cax}$)
b	Proportion of depth to neutral axis over which rectangular stress block acts ($b \leq 1$)
$(x/d)_{max}$	Maximum value of x/d , the ratio of neutral axis to effective depth, for flexure
	$\left(\frac{x}{d}\right)_{min} < \left(\frac{x}{d}\right)_{max} \leq 0.5 \left/ \left[b \left(0.5 + \frac{\min\{z_{t1}, z_{t2}, -z_{b1}, -z_{b2}\}}{h} \right) \right] \right.$
$(x/d)_{min}$	Minimum value of x/d , the ratio of neutral axis to effective depth, for flexure
	$(0 \leq (x/d)_{min} < (x/d)_{max})$
E_s	Elastic modulus of reinforcement
f_{yd}	Design strength of reinforcement in tension ($f_{yd} > 0$)
f_{ydc}	Design strength of reinforcement in compression ($f_{ydc} > 0$)

Within RCSlab the reinforcement positions are measured with respect to the mid-height of the section, the positions being measured positively upwards. The reinforcement angles are specified with respect to the x-axis and measured positively in an anticlockwise direction looking from above. It should be noted that the concrete is assumed to have zero tensile strength in the analysis; the tensile strength, f_t , is only used to calculate the compressive strength when tensile strains are present.

The results of each run consist of the required area of reinforcement, negative if tensile, in each direction in the top and bottom faces or an error flag indicating that a solution could not be found.

20.3 Other symbols used in this theory

f	Stress
α	Proportion of total thickness comprising each outer layer
ε	Strain
ϕ	Angle of principal compressive stress and strain,

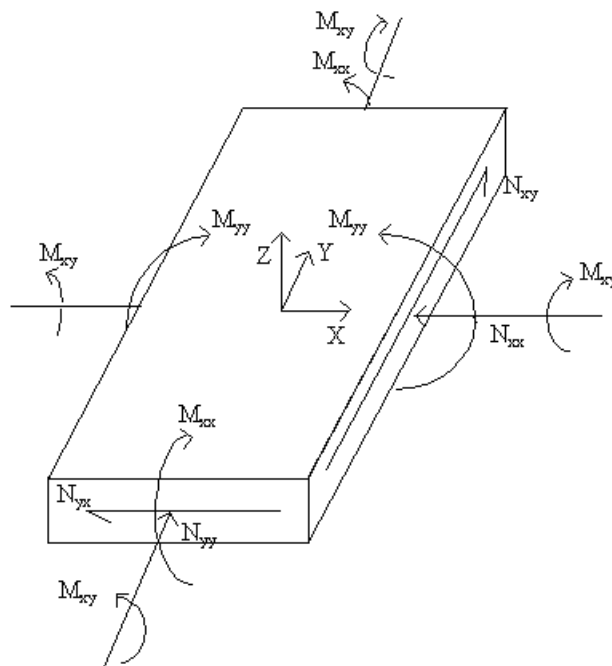
ψ	measured anticlockwise with respect to X axis Angle of reinforcement, measure anticlockwise with respect to principal compression within relevant outer layer = $\theta - \phi$

Subscripts

A	Relating to pass A
B	Relating to pass B
C	Relating to pass C or compression-only reinforcement
T	Relating to tension-only reinforcement
a	Applied
b	Bottom layer or face
c	Concrete
e	Effective
m	Middle layer
n	General direction = 1 or 2
s	Steel
t	Top layer or face
1	Direction 1
2	Direction 2
>	Principal direction with greater compression or lesser tension
<	Principal direction with greater tension or lesser compression

20.4 The RCSlab sign convention

Within RCSlab the sign convention is as shown below; note that although it is different from that used by GSA, the interface converts as necessary:



Compression and sagging moment are positive.

20.5 RCSlab analysis procedure

[Summary](#)

[Inclusion of moments resulting from minimum eccentricity](#)

[Division into layers](#)

[Applied stresses to each layer](#)

[Calculation of stress to be taken by reinforcement in a layer](#)

[Calculation of forces to be taken by reinforcement](#)

[Calculation of reinforcement strains](#)

[Determination of stress in reinforcement](#)

[Calculation of area of reinforcement](#)

[Non-iterative technique](#)

20.5.1 Summary

The section is divided into three layers, the outer ones of which have an equal thickness, selected iteratively to give the minimum area of reinforcement. In-plane forces are resisted either by the outer layers alone or, if there is sufficient in-plane compression, by all three layers. The outer layers also resist the moments resulting from out-of-plane forces. Applied forces and moments are thus resolved into three layers of in-plane stresses. The next step is to calculate the forces that the four sets of reinforcement must take to achieve acceptable concrete stresses in each layer. Finally,

reinforcement strains, and hence stresses and areas, compatible with the concrete stresses are calculated.

20.5.2 Inclusion of moments resulting from minimum eccentricity

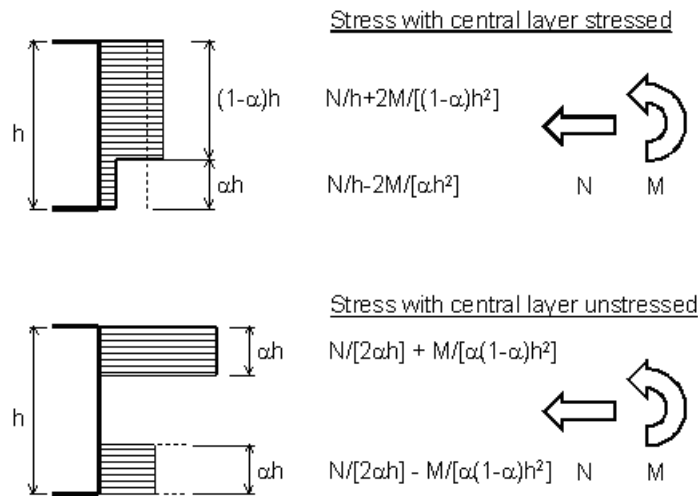
The applied moments are adjusted to take into account the minimum eccentricity of applied axial forces. If the absolute value of the applied moment exceeds the eccentricity moment for M_{xx} , M_{yy} and M_{xy} , then the applied moments are not modified. Otherwise two sets of applied moments are calculated, corresponding to eccentricities either way. For the first set, each moment is calculated as the sum of the applied moment and the product of the eccentricity and the appropriate axial force, but not greater than the eccentricity moment applied either way. For the second set, the calculation is repeated, but with a negative eccentricity. For example, if the applied and eccentricity moments were 100kNm and 150kNm respectively, two sets of design moments would be considered, one based on 150kNm and the other on -50kNm.

20.5.3 Division into layers

RCSlab divides the section thickness into three layers, each of constant stress. The outer layers both have a thickness of αh , where α is the proportion of the total section thickness h . The inner layer is therefore $(1 - 2\alpha)h$.

20.5.4 Applied stresses to each layer

Calculations are performed for the central layer both stressed and unstressed; however, results from stresses applied to the central layer are only valid if any tension reinforcement is required in both faces in a particular direction.



(In the above diagrams N represents N_{xx} , N_{yy} or N_{xy} and M represents M_{xx} , M_{yy} or M_{xy} respectively.)

The initial calculations are carried out with $\alpha = \alpha_{lim}$, where $\alpha_{lim} = 0.5$ where all three layers are

stressed and $\alpha_{lim} = b \cdot \left(\frac{x}{d} \right)_{max} \cdot \left(0.5 + \frac{\min\{z_{t1}, z_{t2}, -z_{b1}, -z_{b2}\}}{h} \right)$ where the central layer is unstressed, and the calculations are repeated for $\alpha = 0.8\alpha_0 + 0.2\alpha_{lim}$, $0.6\alpha_0 + 0.4\alpha_{lim}$, $0.4\alpha_0 + 0.6\alpha_{lim}$ and $0.2\alpha_0 + 0.8\alpha_{lim}$, where $\alpha_0 = 0$ where all three layers are stressed and

$\alpha_{lim} = b \cdot \left(\frac{x}{d} \right)_{max} \cdot \left(0.5 + \frac{\min\{z_{t1}, z_{t2}, -z_{b1}, -z_{b2}\}}{h} \right)$ where the central layer is unstressed. The value of α_{min} ,

the value of α that gives rise to the smallest total area of reinforcement, is determined. The process is repeated for successively closer values of α until the α_{min} is within 0.0032 of the true minimum value. For example, if the true value of α_{min} were 0.1234, and assuming the area of reinforcement increases approximately linearly with the difference between α and α_{min} , the program would calculate areas of reinforcement for the following values of α .

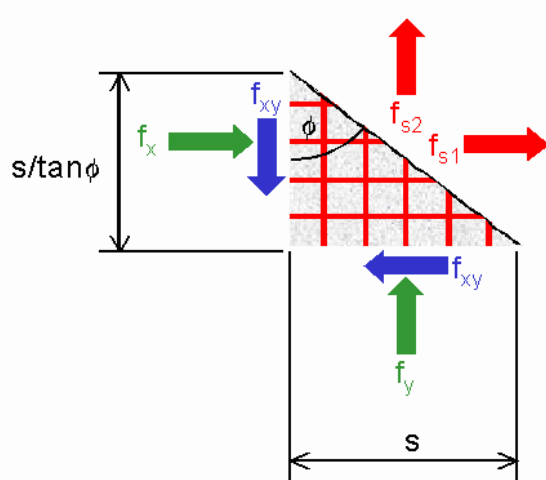
Values of α for which reinforcement areas calculated				Current value of α_{min}
0.5				0.5
0.1	0.2	0.3	0.4	0.1
0.04	0.08	0.12	0.16	0.12
0.096	0.112	0.128	0.144	0.12
0.1104	0.1168	0.1232	0.1296	0.1232

20.5.5 Calculation of stress to be taken by reinforcement in a layer

The principal applied stresses in the layer are calculated. The design strength of the concrete, f_c , depends on the extent of cracking. If neither principal stress is tensile, the uncracked concrete strength, f_u , is used. Otherwise the design strength depends on the ratio of the more tensile principal

stress, f_{min} , to the tensile strength, f_t ; the strength is calculated as $f_u \times \left[1 - \left(\frac{f_{min}}{f_t}\right)^2\right]$, but not less than the cracked concrete strength, f_{ck} . If f_{min} is not tensile and the more compressive principal stress is not greater than f_c then the reinforcement is not required to take any stress from this layer. Otherwise, the reinforcement must take sufficient stress to achieve equilibrium without overstressing the concrete or requiring it to take tension. The reinforcement stresses are chosen so that the total stress to be provided is minimised; the formulae are given in [Appendix 1](#).

The following diagrams illustrate the situation where direction 1 is aligned with x and direction 2 is aligned with y.



f_x , f_y and f_{xy} are applied in-plane stresses

To determine f_{s1} and f_{s2} , stresses to reinforcement:

- Resolve horizontally

$$s/\tan\phi \cdot (f_x + f_{s1}) = s \cdot f_{xy}$$

$$\therefore f_{s1} = f_{xy} \cdot \tan\phi - f_x$$

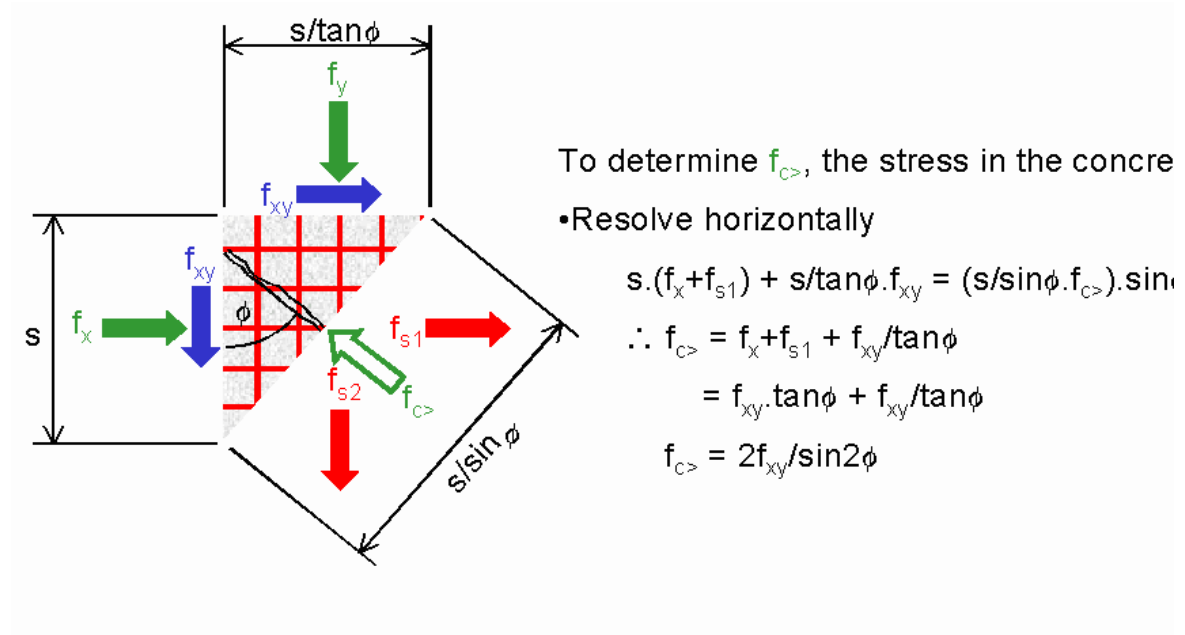
- Resolve vertically

$$s \cdot (f_{xy} + f_{s2}) = s/\tan\phi \cdot f_{xy}$$

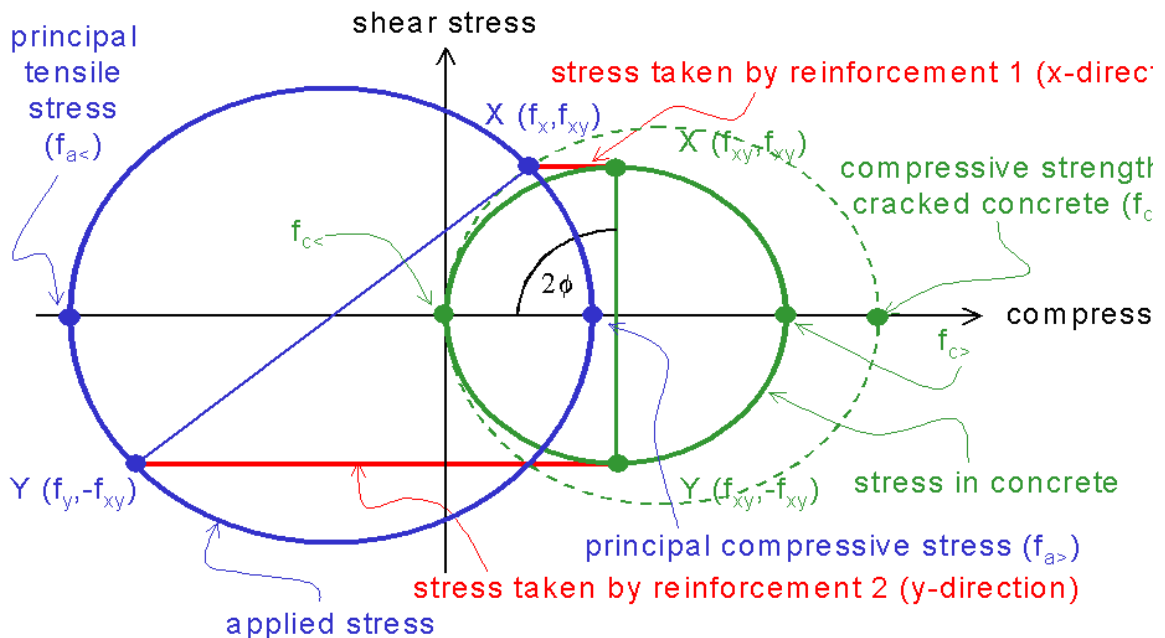
$$\therefore f_{s2} = f_{xy} / \tan\phi - f_y$$

The angle at which the cracks form under the design loads, ϕ , is not equal generally to the angle of

the initial cracks, which depends on the direction of principal tensile stress. As reinforcement yields, the angle of cracking rotates, and its value for design can be freely selected. RCSlab selects the angle that will result in the lowest total area of reinforcement. This often corresponds to the angle that gives rise to the minimum stress in the concrete.



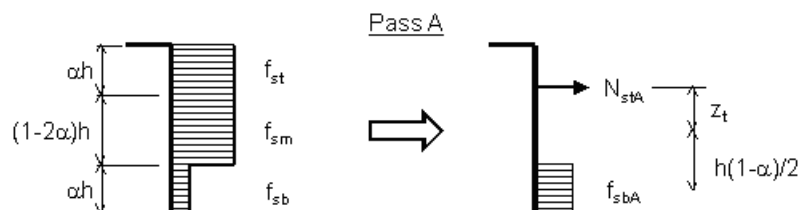
Mohr's circles provide a useful tool for determining the stress to be taken by the reinforcement and maximum stress in the concrete. The circle representing the stress in the concrete must always fall within the circle representing its compressive strength, the principal values of which are the origin and the concrete design strength.



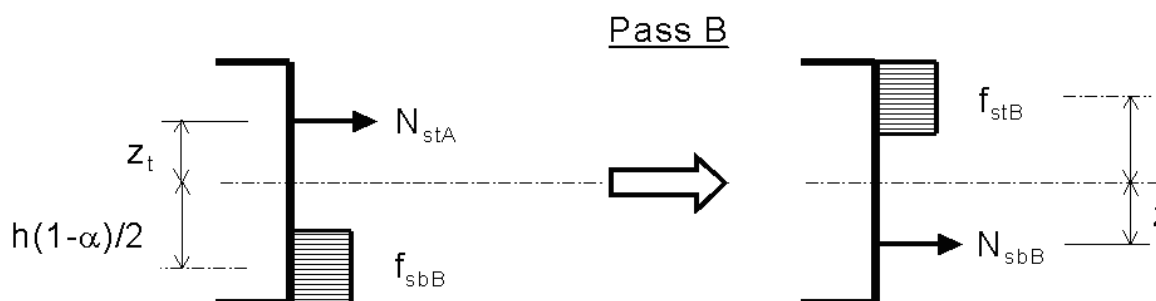
Note that for $\phi=45^\circ$, the stress taken by reinforcement 1 (x-direction) is equal to f_x and that taken by the reinforcement 2 (y-direction) is equal to f_y .

20.5.6 Calculation of forces to be taken by reinforcement

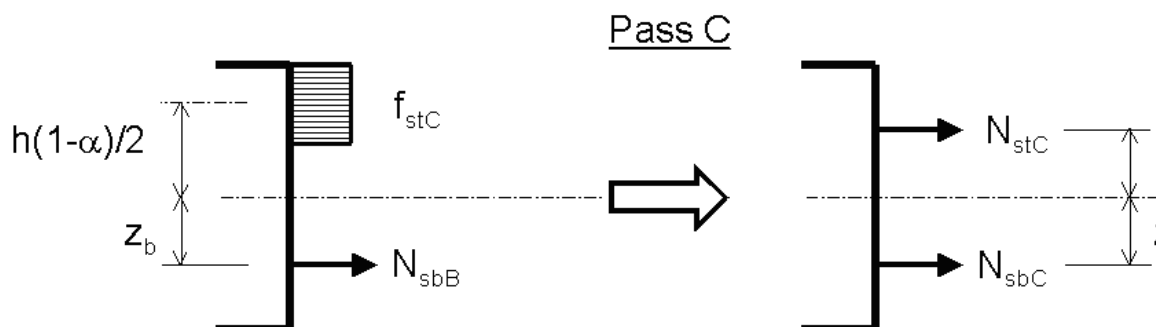
The forces in the four sets of reinforcement must be in equilibrium with the stresses that the reinforcement must take in each layer, taking into account their thicknesses and centroids. If this calculation is undertaken in one stage, anomalies could occur such as tension reinforcement being required where the surrounding concrete is in biaxial compression. To avoid this, the calculation is made in three stages.



In pass A, the force required in the top steel is calculated assuming that no bottom steel is required. This is shown diagrammatically in the figure above; there would of course be an equivalent diagram for the second steel direction. The bottom layer is reanalysed for f_{sbA} , the combination of the applied stress and the increases in stress ($f_{sbA} - f_{sb}$).



In pass B, the force required in the bottom steel is calculated assuming that no top steel is required. The top layer is reanalysed for f_{stB} , the combination of the applied stress and the increases in stress ($f_{stB} - f_{st}$).



In the final pass, C, any reinforcement, top and bottom, required to take the modified stress in the top layer is calculated. By considering the top layer last, there will be places where no top steel is required. Although pass C could result in bottom steel in locations where it would be unnecessary with further rationalisation, the reinforcement areas would be small and usually less than nominal bottom steel.

20.5.7 Calculation of reinforcement strains

Reinforcement strains for each of the four sets of reinforcement are calculated to be compatible with stress blocks.

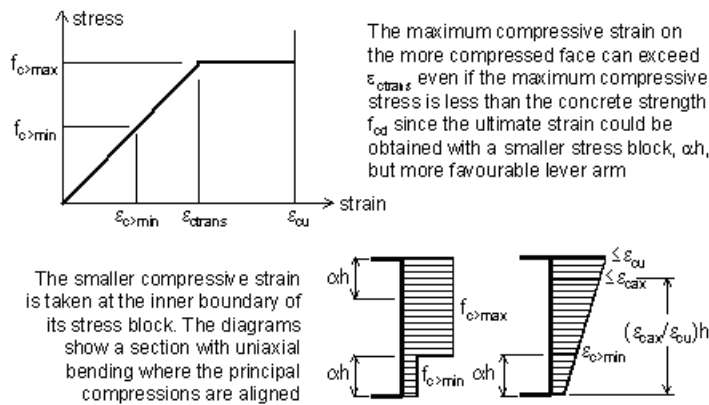
First the strains corresponding to the concrete block stresses are determined. The maximum strain is taken as the limiting flexural strain, ϵ_{cu} , in the outer layer being considered if tension reinforcement is required in the opposite face, but not in the corresponding direction on the face under consideration.

If the maximum concrete stress is the same in the two outer layers then the maximum strain is taken as the axial limiting strain, ϵ_{cax} , in both layers.

If the maximum concrete stress is greater than the maximum concrete stress in the opposite outer layer the maximum concrete strain is calculated so that it is below the limiting strain, ϵ_{cu} , and the

axial limiting strain, ϵ_{cax} , is not exceeded at a position $(1 - \epsilon_{cax}/\epsilon_{cu})h$ from the face.

If neither of the above conditions applies, the maximum strain is equal to the compressive plateau strain of concrete, ϵ_{trans} , factored by the ratio of maximum concrete stress to the maximum concrete stress in the opposite outer layer.



There are six different procedures for calculating the reinforcement strains, the choice of which depends on the relations between the reinforcement forces, summarised in the following table. Both faces must be checked for each procedure in turn since some procedures relate to the strains that have been calculated for the opposite face by an earlier procedure.

Face under consideration		Opposite face		Procedure
Direction under consideration	Other direction	Direction under consideration	Other direction	
C	T	C/N	T	A
T	C	T	C/N	
T	C/N/T	T or concrete unstressed	C/N/T	B
C	T	C/N/T	C/N/T	C
T	C/N/T	C/N/T	C/N/T	
C	C/N	C/N	T	D
C	C/N	T	C/N/T	
C	C/N	C/N	C/N	E

C = Compression reinforcement requirement

N = No reinforcement requirement

T = Tension reinforcement requirement

More:

[Procedure A](#)

[Procedure B](#)

[Procedure C](#)

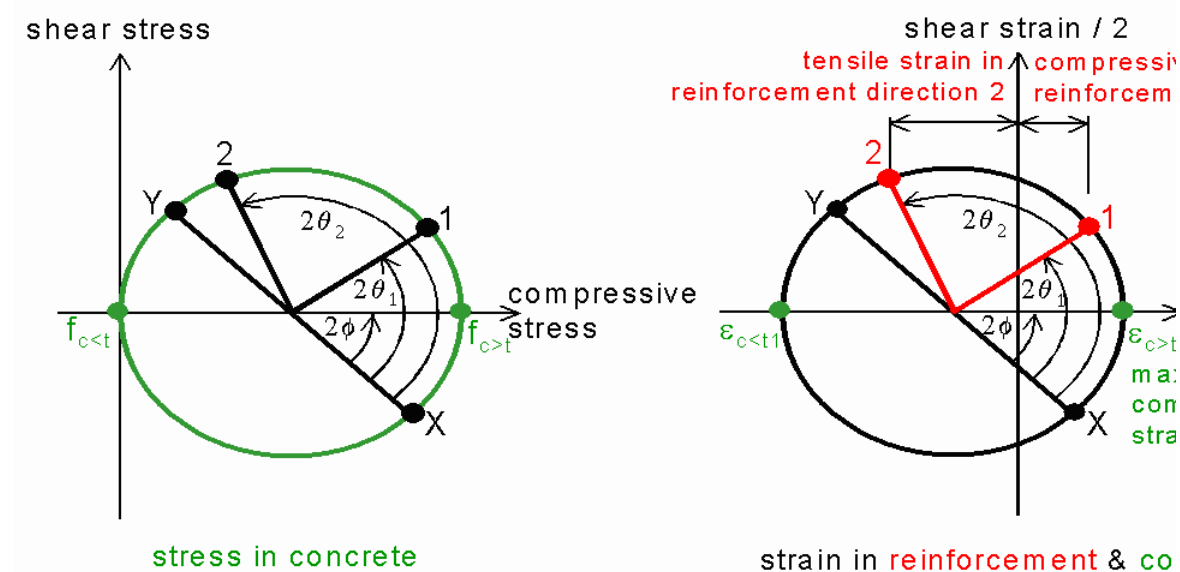
[Procedure D](#)

[Procedure E](#)

20.5.7.1 Procedure A

The calculation is made at the level of the compression reinforcement. Although the tension reinforcement will be at a slightly different level, it is assumed that the strain in it is approximately equal to the strain in its direction at this level; since the reinforcement in the opposite face in the same direction will always be in tension, this should be a reasonable assumption. The principal tensile strain at this level is calculated so that the total area of reinforcement in this layer is at a minimum. If the tensile strain is not large enough, the tensile reinforcement will be working at a low stress, or even be in compression. If the tensile strain is too large, the compressive reinforcement will be working at a low stress, or even be in tension.

Relations between stresses and strains at the level of the top reinforcement direction 1, with procedure A



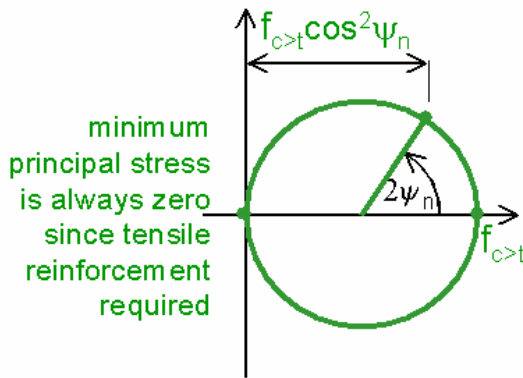
Note that principal stresses and strains are aligned

The notation assumes that this is the top layer with compression reinforcement in

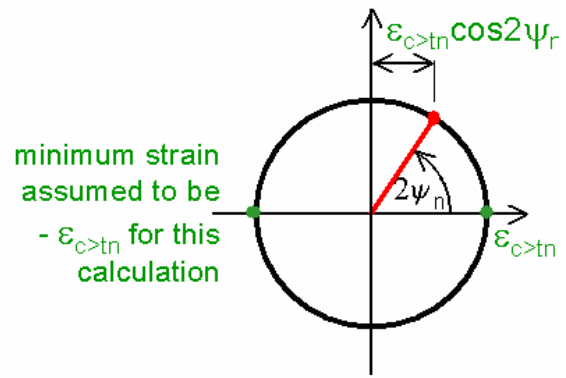
To estimate the principal compressive strain at the level of the top reinforcement, the difference in the direction of principal compression in the top and bottom layers has to be considered.

The final stage of calculating reinforcement stresses makes allowance for the area of concrete displaced by compression bars by subtracting the concrete stress at the level of the reinforcement from the steel stress. To compensate for this when determining the optimum tensile strain with procedure A, a force, estimated to equal the force in displaced concrete is added to compressions. This is achieved by factoring the compression by:

$$1 + \frac{f_{c>t} \cos^2 \psi_n}{\min(f_s, E_s \varepsilon_{c>tn} \cos 2\psi_n)} \text{ where } \psi_n = (\theta_1 - \phi) \text{ or } (\theta_2 - \phi) \text{ as appropriate}$$



stress in concrete



strain in reinforcement & concrete

The final stage of calculating reinforcement stresses makes allowance for the area of concrete displaced by compression bars by subtracting the concrete stress at the level of the reinforcement from the steel stress. To compensate for this when determining the optimum tensile strain with procedure A, a force, estimated to equal the force in displaced concrete is added to compressions. This is achieved by factoring the compression by:

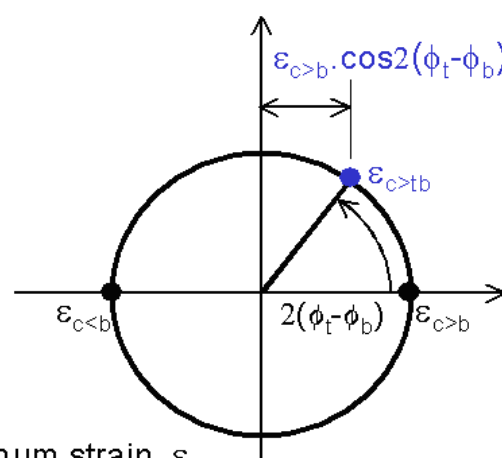
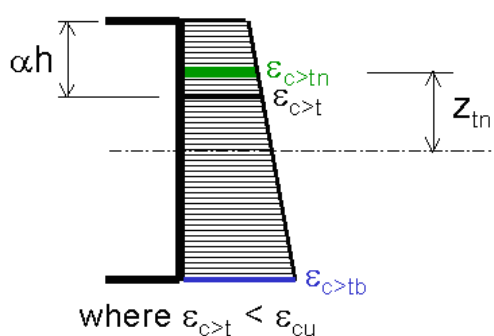
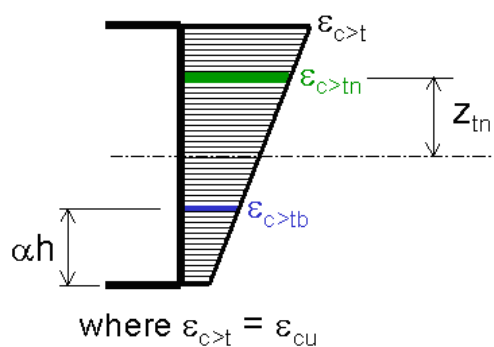
$$1 + \frac{f_{c>t} \cos^2 \psi_n}{\min(f_s, E_s \varepsilon_{c>tn} \cos 2\psi_n)}$$

where

$$n = (\theta_1 - \phi) \text{ or } (\theta_2 - \phi)$$

as appropriate.

Estimating principal compressive strain, $\varepsilon_{c>tn}$, at level of top reinforcement with procedure A



Minimum strain, $\varepsilon_{c<b}$, assumed to be $-\varepsilon_{c>b}$ for this calculation

Estimation of strain $\varepsilon_{c>tb}$ in **bottom** la

The formulae to calculate the optimum value of the tensile strain are given in [Appendix 2](#).

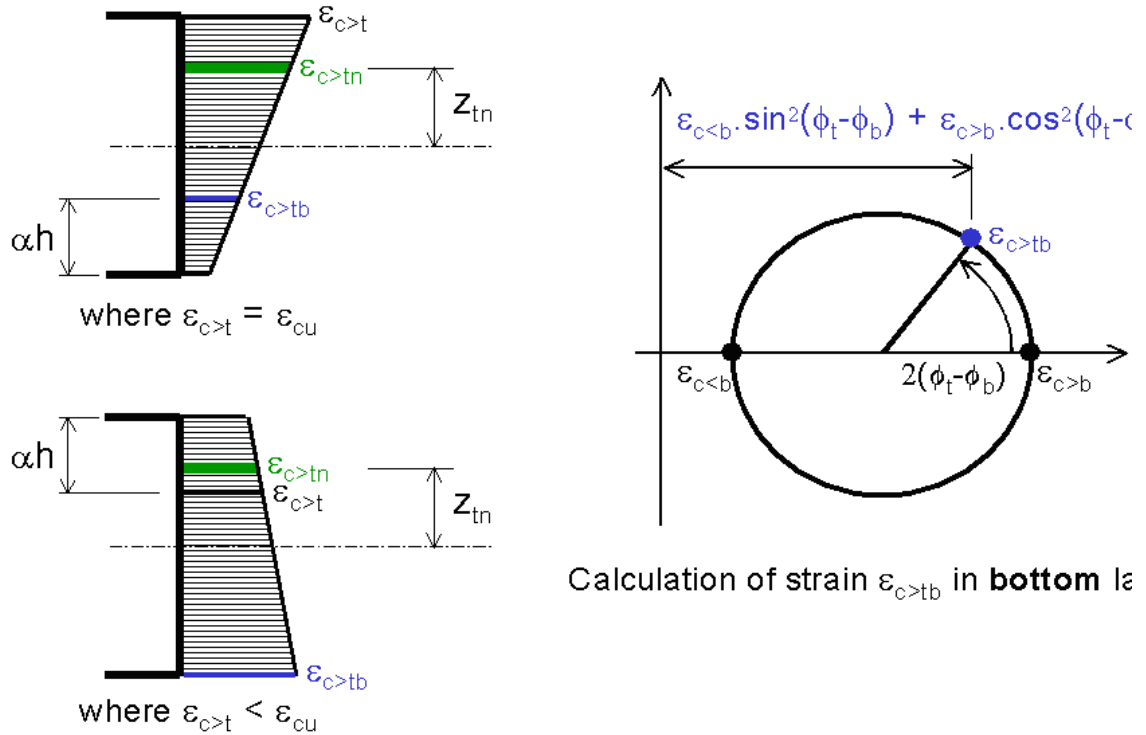
20.5.7.2 Procedure B

This procedure is adopted when strain compatibility is not required for tension reinforcement; the full design strength of the reinforcement is used.

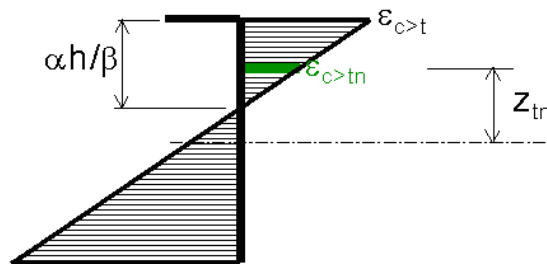
20.5.7.3 Procedure C

The first stage of this procedure is to calculate the principal compressive strain at the level of the reinforcement being considered. The direction of the principal strain is taken as constant within a layer but allowance is made for the different principal strain directions in the top and bottom layers.

Calculating principal compressive strain, $\epsilon_{c>tn}$, at level of top reinforcement with procedure C when no tension in bottom layer

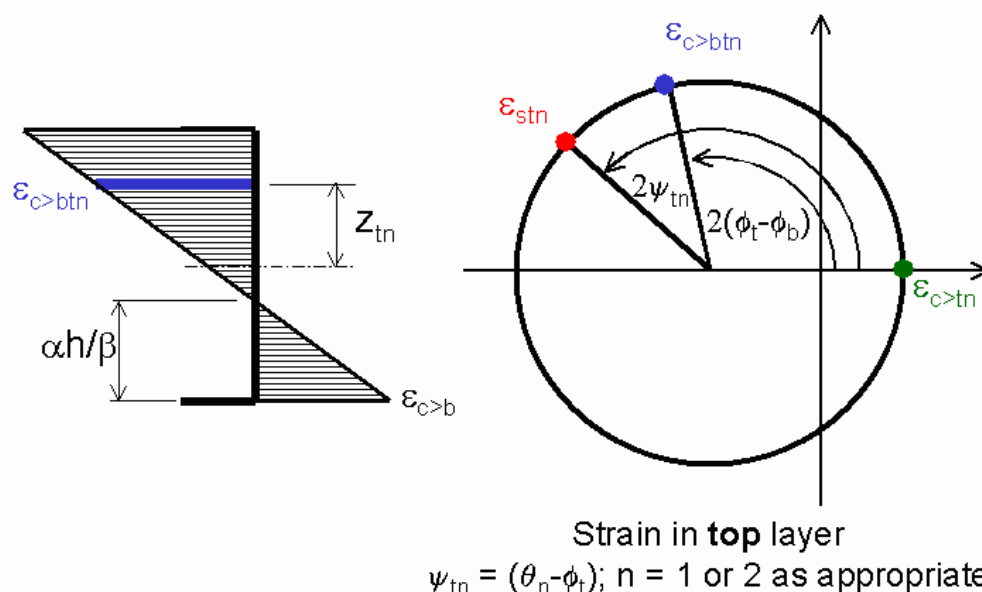


Calculating principal compressive strain, $\epsilon_{c>tn}$, at level of top reinforcement with procedure C when tension reinforcement required in bottom l



The second stage of procedure C is to calculate the tensile strain at the level of the reinforcement under consideration in the same direction as the principal compressive strain in the opposite face. Knowing both this and the principal compressive strain allows the strain in the direction of the reinforcement to be calculated.

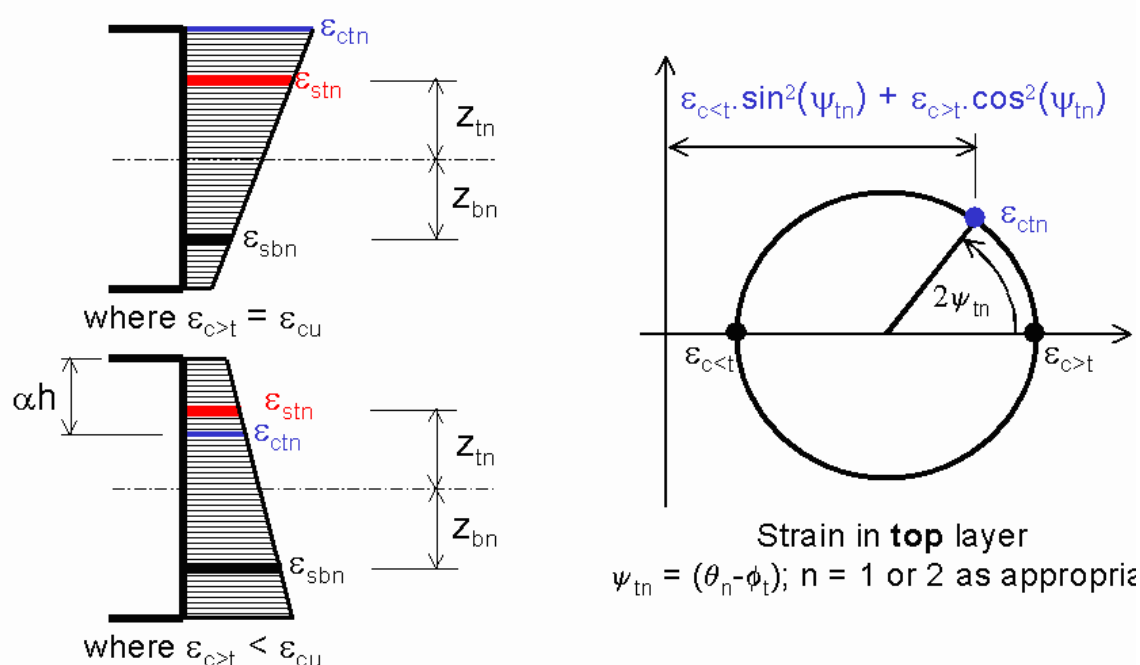
Calculating top reinforcement strain, ϵ_{stn} , with procedure C



20.5.7.4 Procedure D

The strain at the surface, or inner boundary as appropriate, of the layer under consideration is calculated in the direction of the reinforcement being considered. The strain in the reinforcement in the same direction but other face is known, enabling the strain in the reinforcement in the face under consideration to be calculated.

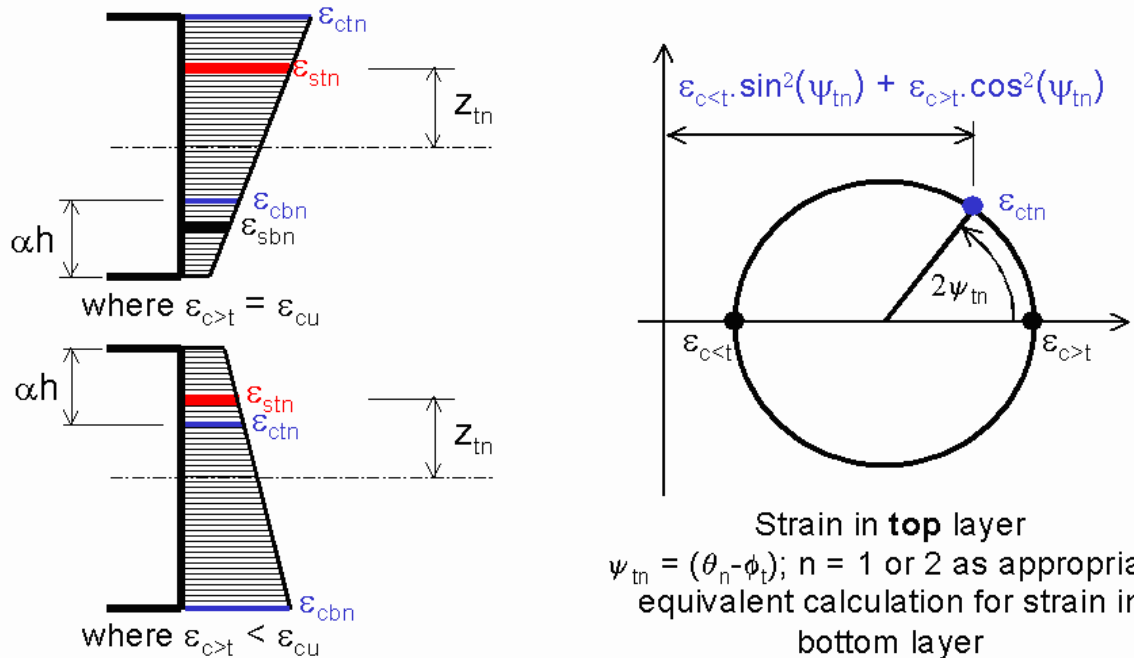
Calculating top reinforcement strain, ϵ_{stn} , with procedure D



20.5.7.5 Procedure E

The strains at the surface, or inner boundary as appropriate, of both layers are calculated in the direction of the reinforcement enabling the strains in the reinforcement to be calculated.

Calculating top reinforcement strain, ϵ_{stn} , with procedure E



20.5.8 Determination of stress in reinforcement

The stress in the reinforcement is calculated from the strain assuming that the steel behaves in elastically until the design strength is reached in either compression or tension, after which the strength is taken as the design strength. If the bar is to take compression, the calculated stress in the reinforcement is reduced by the layer's concrete stress in the reinforcement direction to allow for the displaced concrete.

20.5.9 Calculation of area of reinforcement

If the force and stress are of opposite signs then the calculation for this particular value of alpha is unsuccessful and the area per unit length is set the error value of half the section thickness. If the force required by in the reinforcement is of the same sign as the stress calculated in that steel, the area is calculated by dividing the forces by the stress; the area is limited to half the section thickness. The specified minimum area is used as appropriate. Finally, the four reinforcement areas are summed to give a total that is compared with the totals resulting from other layer thicknesses: the reinforcement areas with the smallest total are output by the program, with tensile reinforcement identified by negating its area.

20.5.10 Non-iterative technique

To speed up the calculation an option is available to use a non-iterative technique, available where the loading is primarily either in-plane or out-of-plane. This approach is likely to lead to slightly more conservative results. The user can choose to use this approach in appropriate situations and can specify a total area of reinforcement, as a percentage of the cross-sectional area, above which a rigorous, iterative solution is used.

This option can be enabled from the '[Analysis](#)' tab of the RC Slab Design Properties wizard.

Primarily in-plane

The applied loads are taken as primarily in-plane if the eccentricities of the moments equal the minimum eccentricity for each component for which there is a corresponding force. Where loads are primarily in-plane, α is taken as 0.5. If the maximum compressive stress in the layers exceeds its strength, the section is unreinforceable; otherwise, reinforcement areas are calculated.

Primarily out-of-plane

The applied loads are taken as primarily out-of-plane if the compressive orthogonal normal stresses and the magnitude of the in-plane shear stress are all less than $0.1 f_{cd}$. Tensile normal stresses do not prevent use of this technique. Where loads are primarily out-of-plane, the central layer is unstressed and the value of α is initially set to α_{lim} . If the maximum compressive stress in the compressive outer layers exceeds the strength, the section is unreinforceable. If compression reinforcement is required, the iterative solution is used. Otherwise, a reduced value of α is calculated so that the relatively more highly stressed of the top and bottom layers is just below its compressive strength, while remaining in equilibrium with the applied moments; limits on the change in α are imposed. The Oasys GSA software stresses are recalculated for this value of α , and a further revised value of α calculated. If the change in α exceeds 0.01, the stresses are recalculated again. Reinforcement areas are calculated for the final value of α .

20.6 Appendix 1

Summary of formulae for the calculation of stress to be taken by reinforcement in a layer

- Calculate principal stresses:

$$f_{a<} = \frac{(f_x + f_y)}{2} - \sqrt{\left[\frac{(f_x - f_y)^2}{4} + f_{xy}^2 \right]}$$

$$f_{a>} = \frac{(f_x + f_y)}{2} + \sqrt{\left[\frac{(f_x - f_y)^2}{4} + f_{xy}^2 \right]}$$

- Calculate strength of concrete:

$$f_{cd} = \max \left\{ f_{cd2}, f_{cd1} \left(1 - \min \left\{ 0, \frac{f_{a<}}{f_t} \right\}^2 \right) \right\}$$

- Check whether reinforcement is required:

If $f_{a<} \geq 0$ and $f_{a>} \leq f_{cd}$ then no reinforcement required

- Otherwise check whether solution possible:

$$a = -\sin(\theta_1 - \theta_2)^2$$

$$b = \frac{2\sin(\theta_1 - \theta_2)(f_x \sin\theta_2 \cos\theta_1 - f_y \sin\theta_1 \cos\theta_2) + f_{cd} \sin(\theta_1 - \theta_2)^2 + f_{xy}(\sin 2\theta_1 - \sin 2\theta_2)}{f_x \sin\theta_2^2 - f_y \cos\theta_2^2 + f_{xy} \sin 2\theta_2}$$

$$c = (f_x \sin\theta_2^2 - f_y \cos\theta_2^2 + f_{xy} \sin 2\theta_2)(f_{cd} - f_x - f_y) - f_{xy}^2 + f_x f_y$$

If $b^2 - 4ac < 0$ then layer is **unreinforceable**

Otherwise calculate:

General

$$f_{s1} = \frac{(b + \sqrt{b^2 - 4ac})}{2a}$$

$$f_{s2} = f_x + f_y - f_{cd} - f_{s1}$$

Tension reinforcement only

$$f_{s1T} = \frac{-\left[(f_x \cdot \sin \theta_2 + f_{xy} \cdot \cos \theta_2) \sin \theta_1 + (f_y \cdot \cos \theta_2 + f_{xy} \cdot \sin \theta_2) \cos \theta_1\right] + (f_x \cdot \sin \theta_2 + f_{xy} \cdot \cos \theta_2) \sin \theta_2 + (f_y \cdot \cos \theta_2 + f_{xy} \cdot \sin \theta_2) \cos \theta_2}{\sin^2(\theta_1 - \theta_2)}$$

$$f_{s2T} = \frac{-\left[(f_x \cdot \sin \theta_1 + f_{xy} \cdot \cos \theta_1) \sin \theta_2 + (f_y \cdot \cos \theta_1 + f_{xy} \cdot \sin \theta_1) \cos \theta_2\right] + (f_x \cdot \sin \theta_1 + f_{xy} \cdot \cos \theta_1) \sin \theta_1 + (f_y \cdot \cos \theta_1 + f_{xy} \cdot \sin \theta_1) \cos \theta_1}{\sin^2(\theta_1 - \theta_2)}$$

If $f_{s1T} > 0$, set $f_{s1T} = 0$ and calculate

$$f_{s2T} = \frac{f_x \cdot f_y - f_{xy}^2}{f_x \cdot \sin^2 \theta_1 + f_y \cdot \cos^2 \theta_1 + f_{xy} \cdot \sin 2\theta_1}$$

if denominator zero and θ_1 not equal to zero,

$$f_{s2T} = \frac{f_y}{\sin^2 \theta_1}$$

otherwise

$$f_{s2T} = \frac{f_x}{\cos^2 \theta_1}$$

If $f_{s2T} > 0$, set $f_{s1T} = 0$ and calculate

$$f_{s1T} = \frac{f_x \cdot f_y - f_{xy}^2}{f_x \cdot \sin^2 \theta_2 + f_y \cdot \cos^2 \theta_2 + f_{xy} \cdot \sin 2\theta_2}$$

if denominator zero and θ_2 not equal to zero,

$$f_{s1T} = \frac{f_y}{\sin^2 \theta_2}$$

otherwise

$$f_{s1T} = \frac{f_x}{\cos^2 \theta_2}$$

Compression reinforcement only

$$f_{s1C} = \frac{-\left| \left((f_x - f_{cd}) \cdot \sin \theta_2 + f_{xy} \cdot \cos \theta_2 \right) \sin \theta_1 + \left((f_y - f_{cd}) \cdot \cos \theta_2 + f_{xy} \cdot \sin \theta_2 \right) \cos \theta_1 \right| + \left| \left((f_x - f_{cd}) \cdot \sin \theta_2 + f_{xy} \cdot \cos \theta_2 \right) \sin \theta_2 + \left((f_y - f_{cd}) \cdot \cos \theta_2 + f_{xy} \cdot \sin \theta_2 \right) \cos \theta_2 \right|}{\sin^2(\theta_1 - \theta_2)}$$

$$f_{s2C} = \frac{-\left| \left((f_x - f_{cd}) \cdot \sin \theta_1 + f_{xy} \cdot \cos \theta_1 \right) \sin \theta_2 + \left((f_y - f_{cd}) \cdot \cos \theta_1 + f_{xy} \cdot \sin \theta_1 \right) \cos \theta_2 \right| + \left| \left((f_x - f_{cd}) \cdot \sin \theta_1 + f_{xy} \cdot \cos \theta_1 \right) \sin \theta_1 + \left((f_y - f_{cd}) \cdot \cos \theta_1 + f_{xy} \cdot \sin \theta_1 \right) \cos \theta_1 \right|}{\sin^2(\theta_1 - \theta_2)}$$

If $f_{s1C} < 0$, set $f_{s1C} = 0$ and calculate

$$f_{s2C} = \frac{(f_x - f_{cd}) \cdot (f_y - f_{cd}) - f_{xy}^2}{(f_x - f_{cd}) \cdot \sin^2 \theta_1 + (f_y - f_{cd}) \cdot \cos^2 \theta_1 + f_{xy} \cdot \sin 2\theta_1}$$

if denominator zero and θ_1 not equal to zero,

$$f_{s2C} = \frac{f_y - f_{cd}}{\sin^2 \theta_1}$$

otherwise

$$f_{s2C} = \frac{f_y - f_{cd}}{\cos^2 \theta_1}$$

If $f_{s2C} < 0$, set $f_{s2C} = 0$ and calculate

$$f_{s1C} = \frac{(f_x - f_{cd}) \cdot (f_y - f_{cd}) - f_{xy}^2}{(f_x - f_{cd}) \cdot \sin^2 \theta_2 + (f_y - f_{cd}) \cdot \cos^2 \theta_2 + f_{xy} \cdot \sin 2\theta_2}$$

if denominator zero and θ_2 not equal to zero,

$$f_{s1C} = \frac{f_y - f_{cd}}{\sin^2 \theta_2}$$

otherwise

$$f_{s1C} = \frac{f_y - f_{cd}}{\cos^2 \theta_2}$$

Calculate the principal stresses in the concrete and stress to be taken by the reinforcement:

If no reinforcement required:

$$f_{s1} = 0$$

$$f_{s2} = 0$$

$$f_{c<} = f_{a<}$$

$$f_{c>} = f_{a>}$$

Otherwise:

$$f_{c<} = 0$$

$$f_{c>} = f_{cd}$$

$$f_{c>T} = \frac{f_x + f_y - f_{s1T} - f_{s2T}}{2} + \frac{\sqrt{(f_x + f_y - f_{s1T} \cdot \cos^2 \theta_1 - f_{s2T} \cdot \cos^2 \theta_2)^2 + (2f_{xy} + f_{s1T} \cdot \sin 2\theta_1 + f_{s2T} \cdot \sin 2\theta_2)^2}}{2}$$

if $f_{c>T} < f_{cd}$ only tension reinforcement required and

$$f_{s1} = f_{s1T}$$

$$f_{s2} = f_{s2T}$$

$$f_{c>} = f_{c>T}$$

$$f_{c<C} = \frac{f_x + f_y - f_{s1C} - f_{s2C}}{2} + \frac{\sqrt{(f_x + f_y - f_{s1C} \cdot \cos^2 \theta_1 - f_{s2C} \cdot \cos^2 \theta_2)^2 + (2f_{xy} + f_{s1C} \cdot \sin 2\theta_1 + f_{s2C} \cdot \sin 2\theta_2)^2}}{2}$$

if $f_{c>C} > 0$ only compression reinforcement required and

$$f_{s1} = f_{s1C}$$

$$f_{s2} = f_{s2C}$$

$$f_{c>} = f_{c>C}$$

20.7 Appendix 2

Optimising strains when both tensile and compressive reinforcement is required in one face with tensile and compressive, or no, reinforcement required in the corresponding directions in the other face

- Estimate principal compressive strain at the level of compression reinforcement
- Calculate effective forces to be taken by reinforcement to allow for concrete compressive stresses displaced by reinforcement

If N_{s1} is compressive,

$$N_{s1e} = N_{s1} \left\{ 1 + \frac{f_{c>} \cdot \cos^2 \psi_1}{\min(f_{yd}, E_s \varepsilon_{cu} \cos 2\psi_1)} \right\}$$

where ψ_1 equals $(\theta_1 - \phi)$ for the layer under consideration.

Otherwise N_{s2} must be compressive and

$$N_{s2e} = N_{s2} \left\{ 1 + \frac{f_{c>} \cdot \cos^2 \psi_2}{\min(f_{yd}, E_s \varepsilon_{cu} \cos 2\psi_2)} \right\}$$

- Calculate the principal compressive strain at the level of the compression reinforcement, $e_c > n$
- Calculate optimum strains

$$\varepsilon_{s1} = \varepsilon_{c>n} (\sin^2 \psi_2 - \sin^2 \psi_1) N_{s1e} \frac{1 - \left| \frac{\sin \psi_1}{\sin \psi_2} \right| \sqrt{\frac{-N_{s2e}}{N_{s1e}}}}{N_{s1e} \sin^2 \psi_2 + N_{s2e} \sin^2 \psi_1}$$

if denominator is zero then

$$\varepsilon_{s1} = \varepsilon_{c>n} \frac{1 - \sin(\theta_1 - \phi)^2}{2 \sin(\theta_2 - \phi)^2}$$

but

$$-\frac{f_{yd}}{E_s} \leq \varepsilon_{s1} \leq \frac{f_{yd}}{E_s}$$

$$\varepsilon_{s2} = \varepsilon_{c>n} - (\varepsilon_{c>n} - \varepsilon_{s1}) \left(\frac{\sin \psi_2}{\sin \psi_1} \right)^2$$

if denominator is zero then

$$\varepsilon_{s2} = -\frac{f_{yd}}{E_s}$$

but

$$-\frac{f_{yd}}{E_s} \leq \varepsilon_{s2} \leq \frac{f_{yd}}{E_s}$$

Recalculate ε_{s1} since more efficient value may be possible if direction 2 has yielded

$$\varepsilon_{s1} = \varepsilon_{c>n} - (\varepsilon_{c>n} - \varepsilon_{s2}) \left(\frac{\sin \psi_1}{\sin \psi_2} \right)^2$$

if denominator is zero then

$$\varepsilon_{s1} = -\frac{f_{yd}}{E_s}$$

20.8 Appendix 3

Input and derived data required for calculations to various design codes

ACI318	AS3600	BS8110 & Concrete Society TR49	BS5400 Part 4	EC2: Part 1

Compulsory input values

Concrete strength

f'_c	f'_c	f_{cu}	f_{cu}	f_{ck}

Steel strength

f_y	f_{sy}	f_y	f_y	f_{yk}
Code parameters that can be overwritten				
Partial safety factor on concrete				
-	-	$\gamma_{mc} = 1.5$ [2.4.4.1]	$\gamma_{mc} = 1.5$ [4.3.3.3]	$\gamma_c = 1.5$ [2.4.2.4(1)]
Partial safety factor on steel				
-	-	$\gamma_{ms} = 1.05^\#$ [2.4.4.1]	$\gamma_{ms} = 1.15$ [4.3.3.3]	$\gamma_s = 1.15$ [2.4.2.4(1)]
Strength reduction factor for axial compression*				
$\phi = 0.7$ [9.3.2.2]	$\phi = 0.6$ [Table 2.3]	-	-	-
Strength reduction factor for axial tension and bending*				
$\phi = 0.9$ [9.3.2.1]	$\phi = 0.8$ [Table 2.3]			
Derived parameters that can be overwritten				
Uncracked concrete design strength for rectangular stress block: f_{cd1}				
$0.85f'_c$ [10.2.7.1]	$0.85f'_c$ [8.1.2.2]	$0.67 \frac{f_{cu}}{\gamma_{mc}}$ [Figure 3.3]	$0.60 \frac{f_{cu}}{\gamma_{mc}}$ [5.3.2.1(b)]	$f_{ck} \leq 50\text{MPa}$ $\frac{\alpha_{cc} f_{ck}}{\gamma_c}$ $f_{ck} > 50\text{MPa}$ $1.25 \left(1 - \frac{f_{ck}}{250} \right) \frac{\alpha_{cc} f_{ck}}{\gamma_c}$ α_{cc} is an NDP% [3.1.7(3)] ηf_{cd}
Cracked concrete design strength (equal to twice the upper limit on shear strength): f_{cd2}				
$\left(\frac{5}{3} \right) \sqrt{f'_c}$ f'_c in MPa	$0.4 \sqrt{f'_c}$ [11.5.3]	$\frac{2 \sqrt{f_{cu}}}{\gamma_{mc}^{0.55}}$ [3.4.5.2 & TR 3.1.4][5.3.3.3]	$\frac{\min \{ 11.875, 1.875 \sqrt{0.6 \left(1 - \frac{f_{ck}}{250} \right) \times \frac{f_{ck}}{\gamma_c}} \}}{\gamma_{mc}^{0.55}}$ [6.2.2(6)]	$v f_{cd}$
$20 \sqrt{f'_c}$ f'_c in psi [11.3.1.1 & 11.5.6.9]				
Concrete tensile design strength (used only to determine whether section cracked): f_t				
$\left(\frac{1}{3} \right) \sqrt{f'_c}$ f'_c in MPa	$0.4 \sqrt{f'_c}$ [6.1.1.3(a)]	$\frac{0.36 \sqrt{f_{cu}}}{\gamma_{mc}}$ [4.3.8.4]	$\frac{0.36 \sqrt{f_{cu}}}{\gamma_{mc}}$ [6.3.4.2]	$f_{ck} \leq 50\text{MPa}$ $\frac{\alpha_{ct} \times 0.21 f_{ck}^{\frac{2}{3}}}{\gamma_c}$ $f_{ck} > 50\text{MPa}$

$4\sqrt{f'_c}$ f'_c in psi [11.4.2.2]				$\frac{\alpha_{ct} \times 1.48 \ln[1.8 + f_c]}{\gamma_c}$ α_{ct} is an NDP% [3.1.7(3)] f_{ctd}
Compressive plateau concrete strain: ε_{trans}				
0.002 [assumed]	0.002 [assumed]	0.002 [assumed]	0.002 [assumed]	$f_{ck} \leq 50\text{MPa}$ 0.00175 $f_{ck} > 50\text{MPa}$ $0.00175 + 0.0005 \frac{f_{ck} - 50}{50}$ [Table 3.1] ε_{c3}
Maximum axial compressive concrete strain: ε_{cax}				
0.003 [10.2.3]	0.0025 [10.6.3(b)]	$f_{cu} \leq 60\text{MPa}$ 0.0035 $f_{cu} > 60\text{MPa}$ $0.0035 - 0.001 \left[\frac{(f_{cu} - 60)}{50} \right]$ [TR90 3.1.3]	0.0035 [5.3.2.1(b)]	$f_{ck} \leq 50\text{MPa}$ 0.00175 $f_{ck} > 50\text{MPa}$ $0.00175 + 0.0005 \frac{f_{ck} - 50}{50}$ [Table 3.1] ε_{c3}
Maximum flexural compressive concrete strain: ε_{cu}				
0.003 [10.2.3]	0.003 [8.1.2.1(d)]	$f_{cu} \leq 60\text{MPa}$ 0.0035 $f_{cu} > 60\text{MPa}$ $0.0035 - 0.001 \left[\frac{(f_{cu} - 60)}{50} \right]$ [TR90 3.1.3]	0.0035 [5.3.2.1(b)]	$f_{ck} \leq 50\text{MPa}$ 0.0035 $f_{ck} > 50\text{MPa}$ $0.0026 + 0.035 \left[\frac{(90 - f_{ck})}{100} \right]$ [Table 3.1] ε_{cu}
Proportion of depth to neutral axis over which constant stress acts: β				
$0.85 - \frac{0.05(f'_c - 30)}{7}$ f'_c in MPa $0.85 - 0.05(f'_c/1000 - 4)$ f'_c in psi but within limits 0.65 to 0.85 [10.2.7.3]	0.85-0.007(f'_c -28) but within limits 0.65 to 0.85 [8.1.2.2]	0.9 [Figure 3.3]	1.0 [5.3.2.1(b)]	$f_{ck} \leq 50\text{MPa}$ 0.8 $f_{ck} > 50\text{MPa}$ $0.8 - (f_{ck} - 50)/400$ [3.1.7(3)] 1

β_1

Maximum value of ratio of depth to neutral axis to effective depth in flexural situations: $(x'_d)_{\max}$

$\frac{1}{1 + 0.004/\varepsilon_{cu}}$	0.4 [8.1.3]	[upper limit]	$\frac{1}{1 + \varepsilon_s/\varepsilon_{cu}}$	[upper limit]
	$k_{u,\max}$	$(x'_d)_{\max}$		$(x'_d)_{\max}$

[10.3.5]

$(c'_d)_{\max}$

where

$$\varepsilon_s = 0.002 + \frac{f_y}{E_s \gamma_{ms}}$$

[5.3.2.1(d)]

$(x'_d)_{\max}$

Minimum value of ratio of depth to neutral axis to effective depth in flexural situations: $(x'_d)_{\min}$

0	0	$0.1/l_b$	0	0
$(c'_d)_{\min}$	$k_{u,\min}$	[3.4.4.1(e)]	$(x'_d)_{\min}$	$(x'_d)_{\min}$
		$(x'_d)_{\min}$		

Elastic modulus of steel: E_s

200GPa	200GPa	200GPa	200GPa	200GPa
[8.5.2]	[6.2.2(a)]	[Figure 2.2]	[Figure 2]	[3.2.7(4)]
			E_s	E_s

Design strength of reinforcement in tension: f_{yd}

f_y	f_{sy}	f_y/γ_{ms}	f_y/γ_{ms}	f_{yk}/γ_s
[10.2.4]	[8.1.5]	[Figure 2.2]	[Figure 2]	[3.2.7(2)]
				f_{yd}

Design strength of reinforcement in compression: f_{ydc}

f_y	f_{sy}	f_y/γ_{ms}	$\frac{f_y/\gamma_{ms}}{1 + \frac{f_y/\gamma_{ms}}{200}}$	f_{yk}/γ_s
[10.2.4]	[8.1.5]	[Figure 2.2]		[3.2.7(2)]
				f_{yd}
			[Figure 2]	

Maximum linear steel stress: f_{lim}

f_y	f_{sy}	f_y/γ_{ms}	$0.8f_y/\gamma_{ms}$	f_{yk}/γ_s
[10.2.4]	[8.1.5]	[Figure 2.2]	[Figure 2]	[3.2.7(2)]

Yield strain in tension: ε_{plas}

f_y/E_s	f_{sy}/E_s	$f_y/(\gamma_{ms} E_s)$	$f_y/(\gamma_{ms} E_s) + 0.002$	$f_{yk}/(\gamma_s E_s)$
[10.2.4]	[8.1.5]	[Figure 2.2]	[Figure 2]	[3.2.7(2)]

Yield strain in compression: ε_{plasc}

f_y/E_s	f_{sy}/E_s	$f_y/(\gamma_{ms} E_s)$	0.002	$f_{yk}/(\gamma_s E_s)$
[10.2.4]	[8.1.5]	[Figure 2.2]	[Figure 2]	

[3.2.7(2)]

Other parameters

Maximum concrete strength

-	$f'_c \leq 65\text{MPa}$ [6.1.1.1]	$f_{cu} \leq 100\text{MPa}$ [TR 1]	-	$f_{ck} \leq 90\text{MPa}$ [3.1.2(2)]

Maximum steel strength

-	$f_{sy} \leq 500\text{MPa}$ [6.2.1]	$f_y \leq 460\text{MPa}$ [Table 3.1]	-	$f_{yk} \leq 600\text{MPa}$ [32.2(3)]

Minimum eccentricity

0.10h [R10.3.6 & R10.3.7]	0.05h [10.1.2]	$\min\{h/20, 20\text{mm}\}$ [3.9.3.3]	0.05h [5.6.2]	$\max\{h/30, 20\text{mm}\}$ [6.1.4]

*Applied forces and moments are divided by the strength reduction factor to obtain design values for use within RCSlab. If the applied axial principal compressive stress exceeds 1% of the uncracked concrete strength, f_{cd1} , the lower values of ϕ are used.

#In Hong Kong, designs to BS8110 should use a value of 1.15 for γ_{ms} .

%NDP's are nationally determined parameters.

Part XXI

21 Bibliography

1. AISC Steel Design Guide Series 11, Floor Vibration Due to Human Activity, October 2003
2. ASCE 4-09, Seismic Analysis of Safety-Related Nuclear Structures and Commentary, ASCE 2010 (Draft)
3. ASCE 7-05, Loads for Buildings and Other Structures Seismic Loads, ASCE, 2005
4. ASCE 7-10, Loads for Buildings and Other Structures Seismic Loads, ASCE, 2010
5. Bathe K-J, Finite Element Procedures in Engineering Analysis, Prentice Hall, 1982
6. Bathe, K. J. and Dvorkin, E. N., A four-node plate bending element based on Mindlin/Reissner plate theory and a mixed interpolation, International Journal for Numerical Methods in Engineering, Vol 21, 367-383, 1985
7. Bell W.W, Matrices for Scientists and Engineers, Van Nostrand Reinhold, 1975
8. BS6841 - 1987, Measurement and evaluation of human exposure to whole-body mechanical vibration and repeated shock.
9. Structural Use of Concrete, BS 8110-1:1997, BSI, 1997
10. CEN, ENV 1998-1-1, Eurocode 8 – Design Provisions for Earthquake Resistance of Structures – Part 1-1: General Rules – Seismic actions and general requirements for structures, 1994.
11. Chopra A.K, Dynamics of Structures — Theory and Applications to Earthquake Engineering, Prentice Hall, 1995
12. Clough R.W & Penzien J, Dynamics of Structures, 2nd Edition, McGraw-Hill, 1993
13. Coates R.C, Coutie M.G & Kong F.K, Structural Analysis, Nelson, 2nd edition, 1980
14. Criswell H, Highway Spirals, Superelevation and Vertical Curves, County Surveyors' Society / The Carriers Publishing Co. 1969
15. Eurocode 8: Design of structures for earthquake resistance- Part 1: General rules, seismic actions and rules for buildings, EN1998-1, 2004, European Committee for Standardization
16. FEMA, Prestandard and Commentary for the Seismic Rehabilitation of Buildings, FEMA 356, ASCE/FEMA, 2000.
17. Ghali A. & Neville A.M, Structural Analysis, Chapman and Hall, 1978
18. Higham N. J., Accuracy and Stability of Numerical Algorithms, Second edition, SIAM, 2002.
19. Higham N. J. and Tisseur F, A Block Algorithm for Matrix 1-Norm Estimation, with an Application to 1-Norm Pseudospectra. SIAM Journal of Matrix Anal. Appl., 21(4): 1185-1201, 2000. <http://dx.doi.org/10.1137/S0895479899356080>
20. Horne M.R., Merchant W, The Stability of Frames, Pergamon 1965
21. Indian Standard – Criteria for Earthquake Resistant Design of Structures, IS 1893 (Part1) : 2002, Bureau of Indian Standards
22. International Conference of Building Officials, Uniform Building Code UBC-94, Whittier, CA, 1994
23. International Conference of Building Officials, Uniform Building Code UBC-97, Whittier, CA, 1997
24. International Conference of Building Officials, International Building Code 2000, Whittier, CA, 2000
25. Levy R & Spillers W.R, Analysis of Geometrically Nonlinear Structures, Chapman and Hall, 1994
26. Newmark N.M & Hall W.J, Earthquake Spectra and Design, Earthquake Engineering Research Institute, Berkeley, Calif., 1982
27. Ove Arup Partnership, Structural Guidance Notes, Volume 2, 1983

28. SI units and recommendations for the use of their multiples and of certain other units
BS5555:1993, ISO 1000:1992(E)
29. Smith, A.L., Hicks, S.J., & Devine, P.J. (2007) Design of Floors for Vibration: A New Approach, The Steel Construction Institute, P354
30. Steel Construction Institute & BCSA, Steelwork Design Guide to BS5950-1:2000 Volume 1
Section Properties Member Capacities 6th Edition (2001)
31. Timoshenko S.P & Woinowsky-Krieger S, Theory of Plates and Shells, McGraw Hill, 1964
32. Timoshenko S.P & Goodier J.N, Theory of Elasticity, McGraw Hill, 1970
33. Wilson E.L, Three Dimensional Dynamic Analysis of Structures — With Emphasis on
Earthquake Engineering, Computers and Structures Inc., 1997
34. Wilson E.L, der Kiureghian A & Bayo E, Earthquake Engineering and Structural Dynamics, Vol.
9, 187–194 (1981)
35. Willford, M.R. & Young, P. (2006) A Design Guide for Footfall Induced Vibration of Structures,
The Concrete Centre, CCIP-016.
36. BS6472 - 1992, Evaluation of human exposure to vibration in buildings (1 Hz to 80 Hz).
37. BS6472 - 2008, Guide to evaluation of human exposure to vibration in buildings, Part 1: Vibration
sources other than blasting
38. Kannan R., Hendry S., Higham N.J. and Tisseur F, Detecting the Causes of Ill-Conditioning in
Structural Finite Element Models, Computers and Structures, 133:79-84 2014. [http://
dx.doi.org/10.1016/j.compstruc.2013.11.014](http://dx.doi.org/10.1016/j.compstruc.2013.11.014)
39. AISC Steel Design Guide 11, second edition, Vibrations of Steel-Framed Structural Systems
Due to Human Activity, May 2016

Part XXII

22 Technical Notes

[Calculation of Torsion Constant](#)

[Calculation of Shear Areas](#)

[Non-symmetric Beam Sections](#)

[Torque Lines](#)

[2D element shape checks](#)

[Hourglassing](#)

[Grid loading](#)

22.1 Calculation of Torsion Constant

[Introduction](#)

[Saint Venant's Approximation](#)

[Rectangular Sections](#)

[Other sections](#)

[Approximating the torsion constant](#)

[Factored Values in Concrete](#)

22.1.1 Introduction

In structural elements capable of sustaining torsional moments it is necessary to define a constant to specify the “twisting stiffness”.

This term is referred to as the **Torsion Constant**. The following notes offer guidance on how to calculate the Torsion Constant. More detailed information may be found in the standard texts.

In the literature there is some confusion over the symbol used to represent the Torsion Constant. In many texts it is referred to as J . Elsewhere the symbol C is used in an effort to distinguish the Torsion Constant from the polar moment of inertia. In GSA and in these notes the symbol J is adopted.

If a circular bar of constant cross section and of length l is subjected to a constant torque T , the angle of twist θ between the ends is

$$\theta = \frac{T l}{G J}$$

where G is the shear modulus and J the polar moment of inertia.

When the cross section of the bar is non-circular, plane cross sections do not remain plane after deformation and warping will occur. Nevertheless the above equation can still be used with good accuracy, but J should be taken as the appropriate Torsion Constant (J) as defined below.

22.1.2 Saint Venant's Approximation

Saint Venant represented the torsion constant (J) of a solid section by a function relating the already known characteristic values of a cross section thus:

$$J = f(A, I)$$

$$= \frac{1}{K} \cdot \frac{A^4}{I}$$

For narrow rectangles ($b_{\max} \times b$) (i.e. thin plates):

$$J = \frac{1}{36} \cdot \frac{A^4}{I}$$

which reduces to

$$J = \frac{1}{3} b^3 \cdot b_{\max}$$

For circular sections

$$J = \frac{\pi^4}{2}$$

leading to

$$J = \frac{\pi^4}{2}$$

which is the polar moment of inertia.

It can be shown that for members composed of thin rectangles, the torsion constant is equal to the sum of the J -values of component rectangles, except when the section is closed or "hollow".

22.1.3 Rectangular Sections

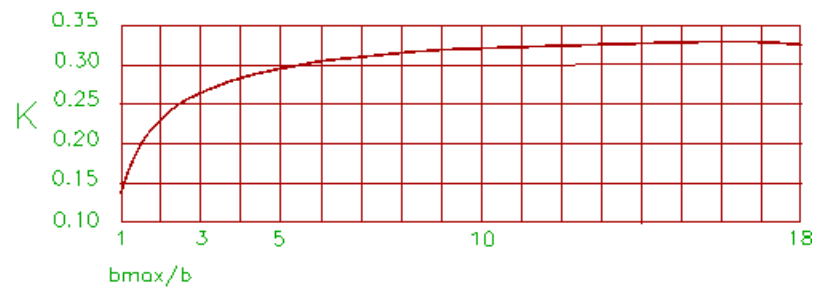
The Torsion constant J is given by:

$$J = K \cdot b^3 \cdot b_{\max}$$

where K is a constant depending on the ratio of $b_{\max}/b$, which can be read either from the table below. Linear interpolation may be used for intermediate values.

$b_{\max}/b$	K	$b_{\max}/b$	K	$b_{\max}/b$	K	$b_{\max}/b$	K
1.0	0.141	1.5	0.196	2.8	0.258	10.0	0.312
1.1	0.154	1.8	0.216	3.0	0.263	∞	0.333
1.2	0.166	2.0	0.229	4.0	0.281		
1.3	0.175	2.3	0.242	5.0	0.291		
1.4	0.186	2.5	0.249	7.5	0.305		

or from the graph



Note that K converges to $1/3$ for narrow rectangles.

Alternatively:

$$K = \frac{1}{3} \left[1 - 0.63 \frac{b}{b_{\max}} \left(1 - \frac{b^4}{12 b_{\max}^4} \right) \right]$$

generally, or if

$$\frac{b_{\max}}{b} \geq 2$$

then

$$\left(1 - \frac{b^4}{12 b_{\max}^4} \right) \approx 1$$

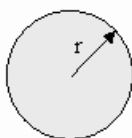
and so

$$K = \frac{1}{3} \left[1 - 0.63 \frac{b}{b_{\max}} \right]$$

22.1.4 Other sections

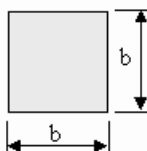
The following table from Ghali and Neville (1978) gives values of the torsion constant (J) for various cross-sectional shapes.

Section

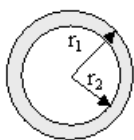


Torsion constant

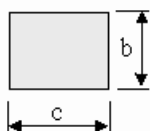
$$J = \frac{\pi r^4}{2}$$



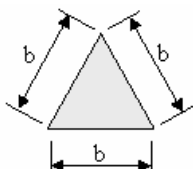
$$J = 0.1406b^4$$



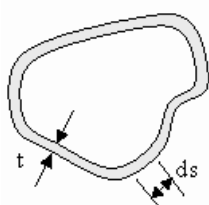
$$J = \frac{\pi(r_1^4 - r_2^4)}{2}$$



$$J = cb^3 \left[\frac{1}{3} - 0.21 \frac{b}{c} \left(1 - \frac{b^4}{12c^4} \right) \right]$$

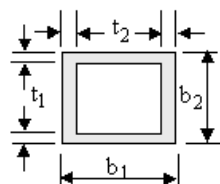


$$J = \frac{\sqrt{3}b^4}{80}$$

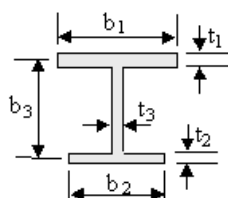


$$J = \frac{4a^2}{\oint \frac{ds}{t}}$$

where a is the area enclosed by a line through the centre of the thickness and the integral is carried out over the circumference.

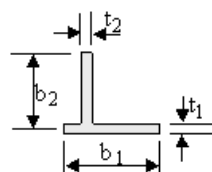


$$J = \frac{2t_1 \cdot t_2 (b_1 - t_2)^2 \cdot (b_2 - t_1)^2}{b_1 t_2 + b_2 t_1 - t_2^2 - t_1^2}$$



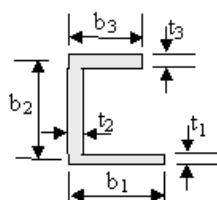
$$J = K_1 b_1 t_1^3 + K_2 b_2 t_2^3 + K_3 b_3 t_3^3 \text{ where } K \text{ is as defined for } \text{rectangular sections above}$$

$$J = \frac{b_1 t_1^3 + b_2 t_2^3 + b_3 t_3^3}{3} \text{ if } (t_i \ll b_i)$$



$$J = K_1 b_1 t_1^3 + K_2 b_2 t_2^3 \text{ where } K \text{ is as defined for } \text{rectangular sections above}$$

$$J = \frac{b_1 t_1^3 + b_2 t_2^3}{3} \text{ if } (t_i \ll b_i)$$



$$J = \sum K_i b_i t_i^3 \text{ where } K \text{ is as defined for } \text{rectangular sections above}$$

$$J = \frac{1}{3} \sum b_i t_i^3 \text{ if } (t_i \ll b_i)$$

All the above assumes that the material is **linear elastic**.

22.1.5 Approximating the torsion constant

There is no closed form solution for the J value of a general perimeter section. This calculation treats the perimeter section as an equivalent ellipse to calculate an approximate J value.

Note that this method has been depreciated in GSA with the introduction of a dedicated [torsion solver](#) for this purpose.

Calculate approximate torsion constant

For an ellipse the torsion constant with y axis radius a and z axis radius b is

$$J = \frac{\pi a^3 b^3}{a^2 + b^2}$$

To apply this we establish equivalent a and b values.

The I values for an elliptical section are

$$I_{yy} = \frac{\pi}{4} ab^3 \quad I_{zz} = \frac{\pi}{4} a^3 b$$

So if the I values are known for the section an equivalent aspect ratio can be calculated

$$\frac{a}{b} = \sqrt{\frac{I_{zz}}{I_{yy}}}$$

The area of an ellipse is

$$A = \pi ab$$

Substituting the a/b ratio in this equation gives

$$a = \left[\frac{A}{\pi} \sqrt{\frac{I_{zz}}{I_{yy}}} \right]^{\frac{1}{2}} \quad b = \left[\frac{A}{\pi} \sqrt{\frac{I_{yy}}{I_{zz}}} \right]^{\frac{1}{2}}$$

The calculation of an equivalent J values is then

1. calculate A, I_{yy} and I_{zz} for the perimeter
2. calculate a and b from equation above
3. calculate J for equivalent ellipse

Sections with voids can be treated as

$$J_{total} = J_{outer} - \sum J_{void}$$

Examples

For a square section, side w this gives a value of

$$J \approx \frac{w^4}{2\pi} = \frac{w^4}{6.28} \quad \left(cf \frac{w^4}{7.1} \right)$$

For a rectangle with sides w and t where w = 10 × t

$$J \approx \frac{100}{101} \frac{wt^3}{\pi} = 0.315wt^3 \quad (cf 0.312wt^3)$$

22.1.6 Factored Values in Concrete

In the case of concrete the Torsion Constant (J) needs to be modified for the following reasons:

- The above formulae do not consider cracking of the concrete.
- There is the practical difficulty of reinforcing for torsion.

For concrete, the actual value ranges from 0.15 to 0.5 of the theoretical (linear elastic) value. BS8110 Part 2 Clause 2.4.3 recommends that half the linear elastic value (calculated above) is used in an analysis.

22.2 Calculation of Shear Areas

In some cases, the shear deformation is very important, and the shear area must be found exactly. This Note provides such values for a range of sections, both solid and thin-walled.

[Thin-Walled Sections](#)

[Solid Sections](#)

[Formulae](#)

[Stress Factors](#)

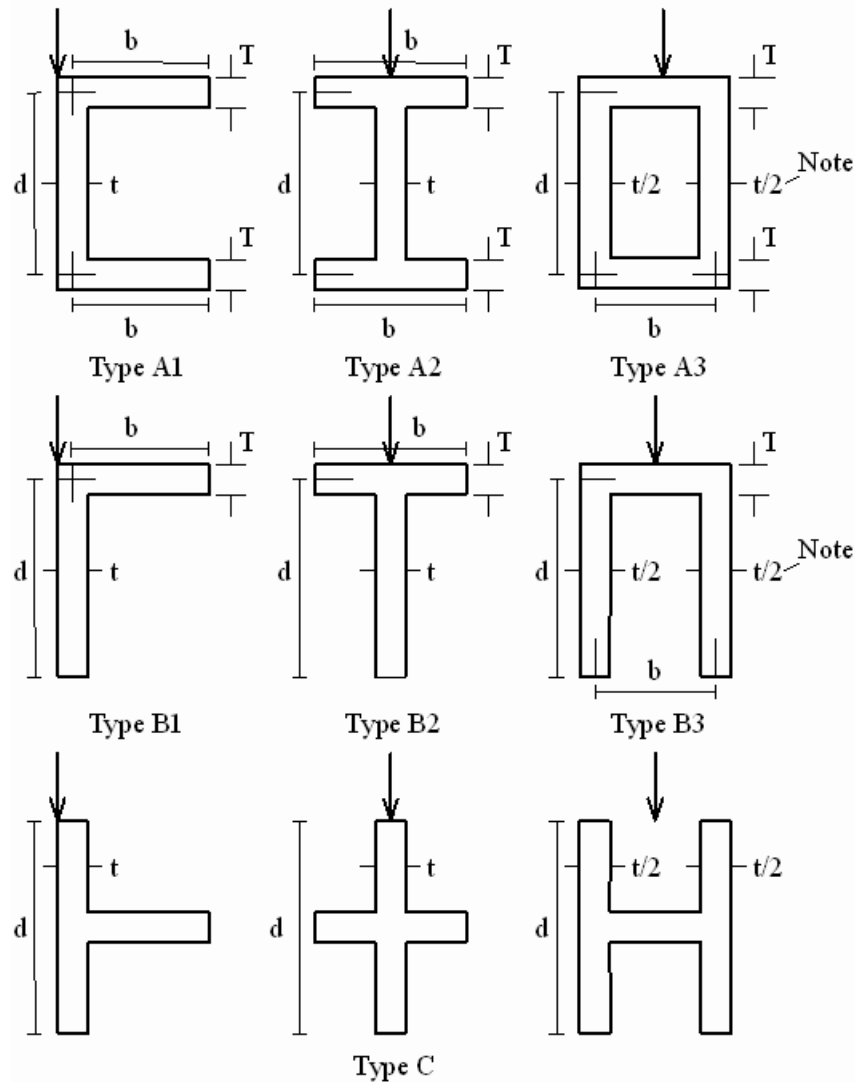
[Reference](#)

22.2.1 Thin-Walled Sections

The effect of shear deformation on the results of a structural analysis is usually negligible. Where it is more significant, it will usually suffice to make a simple approximation to the shear deformation area of members with a cross-section such as those shown in Fig 1. The usual approximation is, by analogy with a simple rectangular beam, to take $\frac{5}{6}$ or $\frac{11}{12}$ of the total web area (overall depth $\times$ web thickness).

For those rare structures where the shear deformation is very important it may be necessary to use a more exact value for the area. This Note gives formulae for F for the cross-sections of Fig.1 where

$$\text{shear deformation area} = Fdt/6$$



These were derived from the virtual work formula, shear deflection per unit length = $\int \frac{q^2}{Gt} ds$. Here q is the shear flow at any point at the middle of the wall thickness, the shear stress q/t is assumed constant across the wall thickness t , and the integration extends over the whole cross-section.

To see what the formulae mean in practice, they were applied to steel sections taken from the handbook with the results shown in Table 1. In a web or flange with varying thickness, t was assumed constant at its average value. The smaller values of F in a range correspond to cross-sections with squarer aspect ratios. Samples only of UBs, UCs and channels were taken.

It can be seen that, for sections with top and bottom flanges bending, as nature intended, in their strong direction, the usual approximation is satisfactory (although it should be noted that d is the distance between flange centres, not overall depth). For the more bizarre sections used in bending, values of F are seen to be lower than expected, especially when they are perversely bent in their weakest direction.

Table 1	
Bending in strong direction	
UB	$F = 5.72$ to 5.81

Joist	$F = 5.17 \text{ to } 5.78$
UBP	$F = 5.25 \text{ to } 5.28$
UC	$F = 5.28 \text{ to } 5.53$
SHS	$F = 5.0$
RHS	$F = 5.0 \text{ to } 5.49 \text{ (for } d/b = 1 \text{ to } 2)$
Channel	$F = 5.06 \text{ to } 5.60$
Angle	$F = 3.68 \text{ to } 4.62 \text{ (for } d/b = 1 \text{ to } 2)$
Tee from UB	$F = 4.78 \text{ to } 4.97$
Tee from UC	$F = 4.11 \text{ to } 4.35$
Cruciform	$F = 5$
Bending in weak direction	
Channel	$F = 2.12 \text{ to } 3.78$
RHS	$F = 4.02 \text{ to } 5.0 \text{ (for } d/b = 0.5 \text{ to } 1)$
Angle	$F = 2.55 \text{ to } 3.68 \text{ (for } d/b = 0.5 \text{ to } 1)$
I's, T's, & cruciforms	$F = 5.0$

With regard to calculating shear *stresses*, the exact distribution is not normally required, or even usable, because Codes of Practice base the shear strength on an allowable average shear stress calculated on the total net area Dt . However, the shear distribution is sometimes required to design welds or concrete stitches, and, since it was found in the process of deriving the formulae for F , formulae for the stress factors k_j and k_m are given. Here,

$$q_i = k_i \left(\frac{V}{d} \right)$$

is the shear flow in the web at the junction with the flange, and

$$q_m = k_m \left(\frac{V}{d} \right)$$

is the maximum shear flow in the web, in which V is the shear force at the section.

For circular annuli, assuming that the stress is constant across the wall thickness t , both the deflection and maximum stress can be obtained using a shear area of half the actual area, that is $\pi r t$, where r is the mean radius.

22.2.2 Solid Sections

For rectangular beams it is usually sufficiently accurate to take the shear area for deflection as $[\frac{5}{6}] bd$ where b is the breadth and d the depth of the section. The corresponding maximum shear stress is $[\frac{3}{2}] \frac{V}{bd}$. It should be noted however that for wide beams the maximum shear stress is underestimated by this formula: for a beam with an aspect ratio of 1 the maximum stress is 12.6% higher. (For a beam with an aspect ratio of 50 (for example a slab) the maximum stress is about 2000% higher but, as this is a Poisson's ratio effect, it is difficult to believe that this has any practical significance!¹)

For circular sections the shear area for deflections is:

$$\frac{6}{7 + \frac{v^2}{(1+v)^2}} \pi r^2$$

where v is Poisson's ratio and r the radius. The expression is very insensitive to the value of v . The maximum shear stress is given by:

$$\left[\frac{1.5 + v}{1 + v} \right] \frac{V}{\pi r^2}$$

which varies from $1.5V/\pi r^2$ for $v = 0$ to $1.33V/\pi r^2$ for $v = 0.5$ with $1.38V/\pi r^2$ for $v = 0.3$.

22.2.3 Formulae

$$\lambda \equiv \frac{dt}{bT} \quad \beta \equiv \frac{bt}{dT} \quad \phi \equiv \frac{d}{b}$$

$$\text{Shear Deformation Area} = F \frac{dt}{6}$$

Type A1

$$F = \frac{(6 + \lambda)^2}{4\beta + 6 + 2\lambda + 0.2\lambda^2}$$

Types A2 & A3

$$F = \frac{(6 + \lambda)^2}{\beta + 6 + 2\lambda + 0.2\lambda^2}$$

Special case of A3 with constant wall thickness so that $T = t$, $\lambda = 2\phi$, $\beta = 2/t_\phi$ gives:

$$F = \frac{10\phi(3 + \phi)^2}{5 + 15\phi + 10\phi^2 + 2\phi^3}$$

Type B1

$$F = \frac{(4 + \lambda)^2}{2\beta + 3.2 + 1.4\lambda + 0.2\lambda^2}$$

Special Case of B1 with $T = t$, $\lambda = \phi$, $\beta = 1/t_\phi$ gives:

$$F = \frac{5\phi(4 + \phi)^2}{(1 + \phi)(10 + 6\phi + \phi^2)}$$

Types B2 & B3

$$F = \frac{(4 + \lambda)^2}{0.5\beta + 3.2 + 1.4\lambda + 0.2\lambda^2}$$

Special Case of B3 with constant thickness $T = t/2$, $\lambda = 2\phi$, $\beta = 2/t_\phi$ gives:

$$F = \frac{20\phi(2 + \phi)^2}{5 + 16\phi + 14\phi^2 + 4\phi^3}$$

Type C

$$F = 5$$

22.2.4 Stress Factors

Types A1, A2, & A3

$$k_j = \frac{6}{6 + \lambda} \quad k_m = \left(\frac{3}{2}\right) \frac{4 + \lambda}{6 + \lambda}$$

Types B1, B2, & B3

$$k_j = \frac{6}{4 + \lambda} \quad k_m = \left(\frac{3}{2}\right) \frac{(2 + \lambda)^2}{(1 + \lambda)(4 + \lambda)}$$

22.2.5 Reference

(1) Timoshenko, S.P. and Goodier, J.N. Theory of elasticity. 3rd edition. McGraw-Hill, 1970.

This item was written by John Blanchard and Ian Feltham for Feedback Notes [an Ove Arup & Partners internal publication] (1992 NST/21) originally published in October 1992. Incorporates 1996NST/10 and is reproduced here with permission.

22.3 Non-symmetric Beam Sections

In a beam with a symmetric section the bending properties depend only on the I_{yy} and I_{zz} terms. If such a beam is loaded in the “y” or “z” axis the deflection is in the direction of the loading.

When the section is not symmetric and is loaded in the “y” or “z” direction there is a component of deflection orthogonal to the loading. This is because the bending properties depend on I_{yy} , I_{zz} and I_{yz} .

By rotating the section to principal axes this cross term can be omitted and if the beam is loaded in the “u” or “v” (principal) axis the deflection is in the direction of the loading. In this case the stiffness matrix for the element is calculated using the principal second moments of area and is then rotated into the element local axis system.

For a beam with a non-symmetric section the user must consider if the beam is restrained (so that deflections are constrained to be in the direction of the loading) or if it will act in isolation (resulting in deflections orthogonal to the loading).

If the beam is to act as constrained the user should ensure that “Ignore Iyz” is checked. In this case the I_{yy} and I_{zz} values are used and the I_{yz} value is discarded.

If the beam is to act in isolation the user should ensure that “Ignore Iyz” is unchecked. In this case the stiffness matrix for the element is calculated using the principal second moments of area and is then rotated into the element local axis system. In this case GSA assumes that the K_u value is the same as the K_y or K_z , — whichever of “y” and “z” is closest to the “u” axis, and likewise for the K_v value.

Where the user has specified section modifiers on the I or the K values there is no way to transfer

these modifiers to the “u” and “v” directions. In these situations “Ignore Iyz” must be checked.

All catalogue and standard sections except angles are symmetric. Explicit sections are assumed to be defined such that the principal and local axes coincide so there is no I_{yz} . Geometric (perimeter and line segment) sections are assumed to be non-symmetric.

22.4 Torce Lines

GSA gives the option of plotting “Torce lines”, which are similar to thrust lines but include the effects of torsion. They are equal for plane frames.

Good structural design requires a clear thinking head, well-presented information and some creative flair. At the detailed level designs progress by iterative evolution and the quality of the design depends on the effectiveness of the interaction between the designer and the information which describes the behaviour of the structure in the current cycle. In today’s world this information is almost always presented by a computer. Graphical plots of the parameters used in the numerical analyses are often ergonomically inefficient, meaning that although they contain the information they fail to transmit insight or understanding into the head of the designer. Better representations are needed.

For many applications **thrust lines** meet this need. They are useful where members are subjected to combined axial load and bending, particularly for compression members made of a material that takes no tension. In these cases the adequacy of the member can often be reasonably described by a limit on the eccentricity of the thrust line. Thrust lines are frequently used by the designers of masonry arches.

A thrust line is the locus along a member of points drawn at an eccentricity of

$$\frac{M}{F_x}$$

from the centroidal axis of the member, where M is the moment and F_x the axial force. It follows the line of action of the force carried by the member and two related points follow from this:

- No scale is needed when drawing a thrust line. It is not a diagram superimposed on a view of a structure. It occupies the position where it is shown.
- It is not subject to any sign conventions.

However the key feature of the thrust line which makes it so useful is that it is a complete description of the forces carried by the member. Because of this completeness property a designer can, with no loss of accuracy, substitute a thrust line, which he or she can visualise, for the combination of the three numbers representing the axial force, moment and shear, which remains stubbornly abstract.

In a computer model of a structure the thrust line is derived from the forces in a member but because it occupies the actual position of the force transmitted through the structure being modelled it follows that its position remains fixed in space even if the member which “supports” it is moved, providing the movement of the member does not change the force being transmitted. This can be referred to as the invariance property of thrust lines.

The above statements refer to thrust lines in two dimensions, as in a plane frame model. Thrust lines can also be drawn in three dimensions but unfortunately in shifting up a dimension they lose both their completeness and invariance properties and so they lose their usefulness to the designer.

The general state of force in a section of a member in three dimensions cannot be reduced to a single force, but it can be reduced to a combination of a force and a torque or, more completely, to any one of an infinite number of force/torque pairs. There is just one force/torque pair, hereafter referred to as the **torce**, for which the force axis and the torque axis are parallel. In three dimensions it is torce lines that have the completeness and invariance properties and so it is torce line which can be used by designers to visualise structural actions.

The position of a force line is derived from the forces in a section of a member as follows:

The resultant shear force, Q , and its angle to the z axis, α , are given by:

$$Q = \sqrt{F_y^2 + F_z^2}$$

$$\alpha = \tan^{-1} \frac{F_z}{F_y}$$

The angle β between the member axis and the thrust line is given by:

$$\beta = \tan^{-1} \frac{Q}{F_x}$$

The torque, T , of the force is resolved into moments about the three member axes. The component about the x axis is the torque M_x in the member. The other two components modify the bending components M_y and M_z .

Because the force components of the force (as opposed to the moment components) are the same as those on the member, the thrust line and the force are parallel.

Hence

$$T \cos \beta = M_x$$

Therefore

$$T = M_x \sec \beta$$

Eccentricities of the force, t_y and t_z , are given by:

$$t_y = -\frac{M_z - T \sin \beta \sin \alpha}{F_x}$$

$$t_z = -\frac{M_y - T \sin \beta \cos \alpha}{F_x}$$

Hence:

$$t_y = -\frac{M_z - M_x \cdot F_z / F_x}{F_x}$$

$$t_z = \frac{M_y - M_x \cdot F_y / F_x}{F_x}$$

How do forces behave? A force is a unique representation of the force state in a section of structure. Forces can be added or subtracted. There is always a unique result. Note that if two forces lie in the same plane their addition or subtraction is not necessarily in that plane. In three dimensions four forces on a body in equilibrium are necessarily coincident. The same is not true for forces. Consider four coincident forces in equilibrium. If one of the forces is translated without changing its direction then the body is subjected to a moment. Equilibrium can be restored by adjusting the forces in the other three forces. Hence equilibrium is achieved with four non-coincident forces.

Often it will be known that one or more of the forces have zero torque. For example gravity imposes zero torque on a body. How many torqueless forces are needed to restore the coincidence rule? The answer is four. This is the same as the four forces on a body rule. If just one of the four carries a torque it can be resisted by a combination of lateral shifts of the other three forces and

coincidence is lost. The usefulness of the concept of coincidence is limited to statements such as: The sum of two coincident forces, one of which is torqueless, is coincident with its components.

For the designer the lesson from this is that, unless a structure is conceived as being truly three dimensional, it is often better to analyse it as a two dimensional plane frame during the design evolution phase so the coincidence rule can be used to understand what is driving the magnitudes of the forces.

This item was written by Angus Low for Feedback Notes [an Ove Arup & Partners internal publication] (1999 NST/7) and is reproduced here with permission.

22.5 2D element shape checks

A number of element checks are carried out by GSA prior to a GSS analysis. Other analysis programs may have different limits but the same principles apply. For GSS the following warnings, severe warnings and errors are produced.

Triangle

warning

$$5 < R_{\max} < 15$$

$$15 < \theta_{\min} < 30$$

$$150 < \theta_{\max} < 165$$

severe warning

$$R_{\max} > 15$$

$$\theta_{\min} < 15$$

error

$$\theta_{\max} > 165$$

Quad

warning

$$5 < R_{\max} < 15$$

$$25 < \theta_{\min} < 45$$

$$135 < \theta_{\max} < 155$$

$$0.00001 < H_{\max} < 0.01$$

severe warning

$$R_{\max} > 15$$

$$\theta_{\min} < 25$$

$$H_{\max} > 0.01$$

error

$$\theta_{\max} > 155$$

where

$$R_{\max} = \text{longest side} / \text{shortest side}$$

$$\theta_{\min} = \text{minimum angle}$$

$$\theta_{\max} = \text{maximum angle}$$

$$H_{\max} = \text{distance out of the plane of the element of edge 1} / \text{longest side}$$

Notes:

The distance out of plane of edge 1 is calculated as

$$H_{\max} = \frac{\mathbf{n} \cdot (\mathbf{c}_2 - \mathbf{c}_1)}{s_{\max}}$$

where $\mathbf{n}$ is the element normal, $\mathbf{c}_1$ and $\mathbf{c}_2$ are the coordinates of the first and second corner nodes and $s_{\max}$ is the length of the longest side of the element.

Mid-side node locations not checked but should be approximately halfway along edge.

No check on ratio thickness/shortest side.

2D element geometry contour options

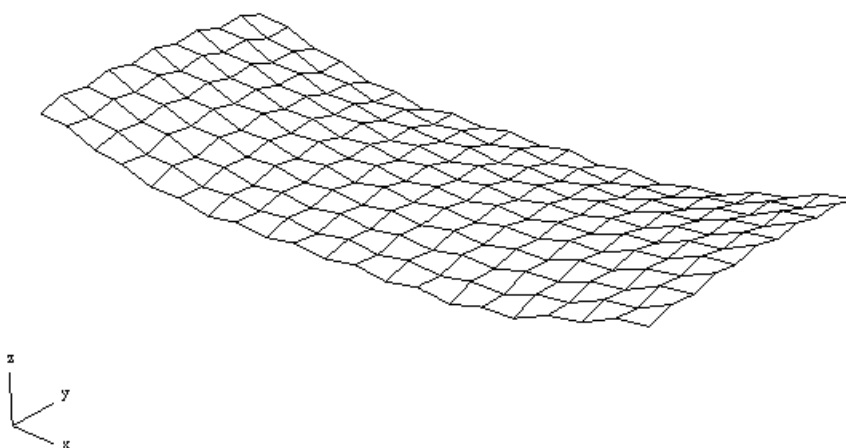
In the **Angle check** output:

- for Quad elements the value contoured is the largest ' $\text{abs}(\theta - 90)$ ' for the element.
- for Tri elements the value contoured is the largest ' $\text{abs}(\theta - 60)$ ' for the element.

In the **Warp check** output the value contoured is $H_{\max}$ as described above.

22.6 Hourglassing

When Quad 4 elements with the Mindlin formulation are used in bending it is possible to encounter hourglassing problems. This is a problem that arises with under-integrated elements where there are insufficient stiffness terms to fully represent the stiffness of the element. The problem is noticeable in the results by an hourglass pattern in the mesh as shown below.



This problem is avoided using the MITC formulation for Quad 4 elements. This formulation uses a separate interpolation method for the transverse shear strains and provides considerably greater stability than the original Mindlin method. The original Mindlin method is kept in GSA for compatibility with previous models although for new models the MITC formulation is recommended. For further details see [GSS Theory](#) and the [Advanced Solver Settings](#).

Alternatively when parabolic accuracy is required Quad 8 elements are recommended. These also formulate elements that are still stiff in all modes of deformation, even when under-integrated.

22.7 Equivalent Static & Accidental Torsion Loading

Many seismic codes have procedures for calculation of equivalent static or accidental torsion loads. GSA provides a method of calculating these loads.

The first stage in the calculation is to establish the nodal masses. This includes the mass of elements plus the additional mass derived from any loads. If a modal analysis has previously been carried out this information is picked up from that analysis task. For each storey we can calculate the storey mass M_s by summing the mass of all nodes in that storey.

If a response spectrum case has been selected the base shear, V , is extracted from that calculation, otherwise the base shear is calculated using the code equations (UBC, IBC or FEMA).

At this point different codes have different requirements. In UBC 1997, depending on the period, an additional force, F_p , is added to the top storey.

$$\begin{aligned}
 F_t &= 0 & T < 0.7s \\
 F_t &= 0.25 \cdot V & 0.07 \cdot T > 0.25s \\
 F_t &= 0.07 \cdot T \cdot V & \text{all other } T
 \end{aligned}$$

For IBC and FEMA an exponent, K , on the distribution function is required

$$\begin{aligned}
 K &= 1 & T < 0.5s \\
 K &= 2 & T > 2.5s \\
 K &= 1 + \frac{(T - 0.5)}{2} & \text{all other } T
 \end{aligned}$$

For other codes K is set to 1.

Equivalent Static

We then calculate the force to be applied to each storey, s , at a height h

$$F_s = \frac{M_s \cdot h_s^K}{\sum_{\text{storeys}} M_s \cdot h_s^K} (V - F_t)$$

And for the top storey

$$F_s = \frac{M_s \cdot h_s^K}{\sum_{\text{storeys}} M_s \cdot h_s^K} (V - F_t) + F_t$$

This storey force is then distributed to the nodes, n , in proportion to their mass

$$F_n = \frac{M_n}{M_s} F_s$$

Accidental Torsion

For the accidental torsion we calculate the storey masses as for the equivalent static and we calculate the centre of mass of each storey. Storey calculations are relative to the centre of mass. We also need the width of the storey which is calculated by the difference in the extreme coordinates in the direction of interest.

We have an offset, o , which is based on the width of the storey

The accidental torsion moment for the storey is $M_{zz,s}$ is then

$$M_{zz,s} = F_s \cdot o$$

This is then applied as forces to the nodes in the storey

$$\hat{F}_n = \frac{M_n}{M_s} F_s \cdot o_n$$

As well as a resulting torsion on the storey this may lead to a force

$$\hat{F} = \sum_{nodes} \hat{F}_n$$

So we correct for this by adjusting these forces by

$$\tilde{F}_n = \hat{F}_n - \frac{M_n}{M_s} \hat{F}$$

And sum the moment on the storey is

$$\tilde{M}_{zz,s} = \sum_{nodes} F_n \cdot o_n$$

Finally we adjust these force values so that we have the correct moment on the storey

$$F_n = \tilde{F}_n \cdot \frac{M_{zz,s}}{\tilde{M}_{zz,s}}$$

22.8 Grid loading

Grid loading is loading applied in space by means of a grid plane. There are three basic types of grid loading: point loads, line loads and area loads. The area loads can be subdivided into loads on the whole grid plane and those on a defined area bounded by a polyline defining a closed polygon. The panels to which the loads are applied can be one-way spanning, two-way spanning or multi-way spanning. The matrix of load type and span options are shown below.

Load Type	One-way spanning	Two-way spanning	Multi-way spanning
Point	✓	→	✓
Line	✓	→	✓
Area – Plane	✓	✓	✓
Area – Plane	✓	✓ or →	✓

Notes:

- Two-way spanning for point and line loads is always treated as multi-way spanning.
- Two-way spanning on planes uses the split-to-centroid option for three sided panels and “back-of-envelope” for four sided panels but uses the multi-way spanning algorithm for panels with more than four edges.
- Two-way spanning on rectangular and polygon areas is treated in the same way as plane loading where the whole panel is loaded, where the panel is partially loaded the panels is considered as multi-way spanning.

More:

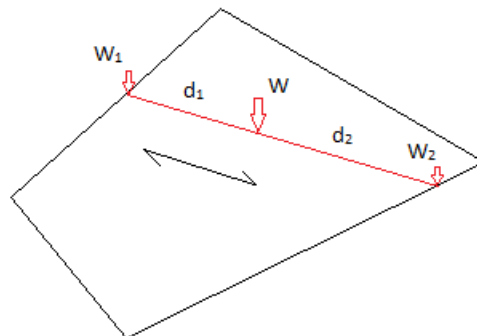
[One-way spanning](#)

[Two-way spanning](#)

[Multi-way spanning](#)

22.8.1 One-way spanning

A one-way spanning grid planes assume that load is transferred in the direction of the span. In the case of a point load the load is distributed between the opposite edges so that the system is in equilibrium.

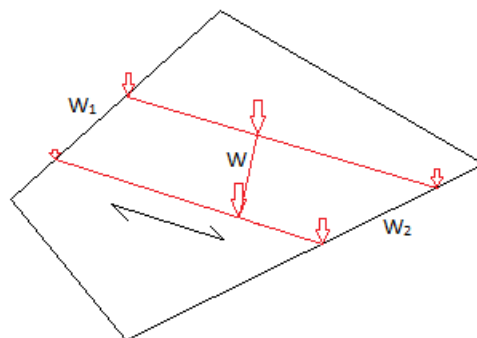


For the load W shown above the load is distributed to the opposite sides so that

$$W_1 = W \frac{d_2}{(d_1 + d_2)}$$

$$W_2 = W \frac{d_1}{(d_1 + d_2)}$$

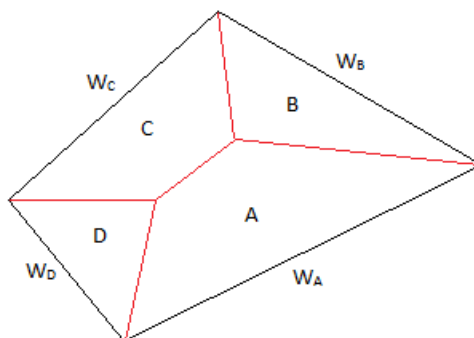
For a line load the same approach is taken but rather than distributing a point load to opposite faces the line load is transferred with the intensities at the ends related to the distance from the line to the edge elements



This extends in an obvious manner to area loads

22.8.2 Two-way spanning

The two-way spanning option applies only to triangular and quadrilateral panels loaded over the whole area of the panel. GSA checks to see if the two-way option can be applied and if not it will automatically use the multi-way algorithm.



The load in area A is applied as a tri-linear (or triangular) load W_A , the load in B is applied as a triangular (or tri-linear) load W_B , etc.

22.8.3 Multi-way spanning

The multi-way spanning tries to distribute load around the edge of the panel in a way that gives a realistic distribution and a set of loads in equilibrium with the applied load. Unlike the two-way spanning which assumes a load attributed to each side the multi-way spanning option makes assumption about how the load 'reaches' the edges of the panel.

There are several algorithm available for the load distribution the preferred option is plane 'approximation (corner)'

Legacy option – load distribution for a point load

The starting point for the distribution of load is to consider a circle of unit radius centred on the load point. The actual intensity at the edges is calculated by extrapolating from this point using a $1/r$ function where r is the distance from the load point to the edge of the panel.

We can satisfy these requirements with a distribution of the form

$$f = 1 + A \cos(2\theta + \phi)$$

We require that there is force and moment equilibrium. The form of these functions satisfies the moment equilibrium requirement and we can look for a solution for an arbitrary load. The term

$$\cos(2\theta + \phi)$$

takes account of the aspect ratio of the panel in determining the split between the long and short directions. If the panel is square we expect the A coefficient to be 0 and 1 for an infinitely long panel. To define the coefficient and phase angle we define a length direction for the panel and a width direction. The length is chosen to be in the direction of the sum or difference of the longest diagonals, whichever has the greater magnitude. The width direction is normal to the length direction. For a rectangular panel this accords with the normal definition of length and width.

We then define values as follows:

l_{max} – maximum dimension from point to panel edge in length direction

l_{min} – dimension from point to opposite edge in length direction

w_{max} – maximum dimension from point to panel edge in width direction

w_{min} – dimension from point to opposite edge in width direction

α – the angle of the width direction from the grid x axis

then we can calculate the coefficients from:

$$\begin{aligned}
 A &= 1 - \left| \frac{w_{min}}{l_{min}} \right| & \text{if } |w_{min}| < |l_{min}| \\
 &= 1 - \left| \frac{l_{min}}{w_{min}} \right| & \text{otherwise} \\
 \phi &= 2\alpha & \text{if } |w_{min}| < |l_{min}| \\
 &= 2\alpha + \pi & \text{otherwise}
 \end{aligned}$$

The load intensity at the edge of the panel is then calculated from the distance of the point on the edge from the load point.

$$p = \frac{f}{r}$$

If we consider a circle at unit radius from the grid point load we have a load intensity function of the form

$$f = 1 + A \cos(2\theta + \phi)$$

This must be mapped on to the surrounding elements. We can use the grid cell size, c , when establishing the size of the loaded patches on elements around the panel boundary.

The distribution of point loads can then be determined at a series of point that are a distance c apart along the elements on the boundary with a minimum of a start and end point on each boundary element. The number of segments for the load distribution can then be determined from the grid cell size and element length, l

$$n = \text{ceil}\left(\frac{l}{c}\right)$$

For the default value of grid load refinement and uniform sized square panels this will give four load patches along each side of the panel. The length of the patches is then

$$\Delta l = \frac{l}{n}$$

Consider two lines from the load point to the start and finish of the element segment. These will be at angles θ_0 and θ_1 . The load carried by this segment must then be

$$F_i = W \int_{\theta_0}^{\theta_1} f d\theta$$

where W is the load intensity. The angle at which this applies is determined from

$$M_i = W \int_{\theta_0}^{\theta_1} f \theta d\theta$$

$$\theta_F = \frac{M_i}{F_i}$$

If the vectors $\mathbf{v}_0$ and $\mathbf{v}_1$ are vectors from the load point to the ends of the beam segment then the

grid load F_i can be thought of as a point load along the vector $\mathbf{v}_F$, at an angle θ_F from $\mathbf{v}_0$.

The loading function contains terms of the form

$$\int \theta \cos(n\theta + \phi) d\theta$$

which integrates to give

$$\frac{\theta}{n} \sin(n\theta + \phi) + \frac{1}{n^2} \cos(n\theta + \phi)$$

The load F_i

$$\begin{aligned} \frac{F_i}{W} &= \int_{\theta_0}^{\theta_1} f d\theta \\ &= \int_{\theta_0}^{\theta_1} (1 + A \cos(2\theta + \phi)) d\theta \\ &= (\theta_1 - \theta_0) + \frac{A}{2} (\sin(2\theta_1 + \phi) - \sin(2\theta_0 + \phi)) \end{aligned}$$

and load moment M_i

$$\begin{aligned} \frac{M_i}{W} &= \int_{\theta_0}^{\theta_1} f\theta d\theta \\ &= \int_{\theta_0}^{\theta_1} (1 + A \cos(2\theta + \phi)) \theta d\theta \\ &= \frac{(\theta_1^2 - \theta_0^2)}{2} + \\ &\quad A \left[\left(\frac{\theta_1}{2} \sin(2\theta_1 + \phi) + \frac{1}{4} \cos(2\theta_1 + \phi) \right) - \left(\frac{\theta_0}{2} \sin(2\theta_0 + \phi) + \frac{1}{4} \cos(2\theta_0 + \phi) \right) \right] \end{aligned}$$

This point load must then be adjusted to allow for the distance of the beam from the load point. This can use a $1/r$ factor to preserve moment equilibrium. This “point” load can then be represented as a linearly varying patch load along the segment length. If the projection of the Load point on to the element is at position a along the segment and the ends are i and j then the equilibrium conditions require that

$$\begin{aligned} F_i &= \frac{(f_i + f_j)\Delta l}{2} \\ F_i a l &= \frac{f_i \Delta l^2}{2} + \frac{(f_j - f_i)\Delta l^2}{3} \end{aligned}$$

Knowing these equations can be solved for f_i and f_j .

$$\begin{aligned} f_j &= \frac{(6a - 2)F}{\Delta l} \\ f_i &= \frac{2F}{\Delta l} - f_j \end{aligned}$$

When a grid area load is applied in a multi-spanning panel the approach is to consider the loading to be represented as a grid of “point loads” distributed over the loaded area. The distributed load is then accounted for by summing all the “point loads”.

Having established the loading grid size then we can then work through the grid and determine if a cell is loaded or not.

Where a cell is bisected by the load boundary then the load intensity is reduced in proportion to the loaded area and adjustment is made for the position of the load. Where a cell is bisected by the structure boundary the point of application of the load must be moved so that the load is applied to the “structure” and not in “space”. In these cases the centroid of the trimmed loading grid cell is calculated and the (reduced) load on the whole cell applied at the recalculated centroid.

Legacy option – cells for distributed loads

In the legacy option a distributed load is handled by subdividing the panel is divided into a grid of cells. The load attributed to each cell is established and distributed with the final load being the integral of the cell point loads over the panel. The grid used needs to be fine enough to give an adequate representation of the load, so it needs to be based on the size of the panels that are loaded. The size and shape of the panels can vary significantly, so a robust way of determining the grid size is required.

For a square panel the load can be represented adequately by a 4×4 grid, but for a long thin panel the same grid would be unsuitable. The grid size is established as follows:

Calculate the area of the panel and set a representative panel dimension to be the square root of this. Then the grid size is this value divided by the grid refinement factor. This defaults to give typically 4 cells along the edge of a square panel. The user can adjust the grid refinement factor to a lower or higher value if required.

For a series of aspect ratios with a refinement factor of 4 the mesh densities are as follows.

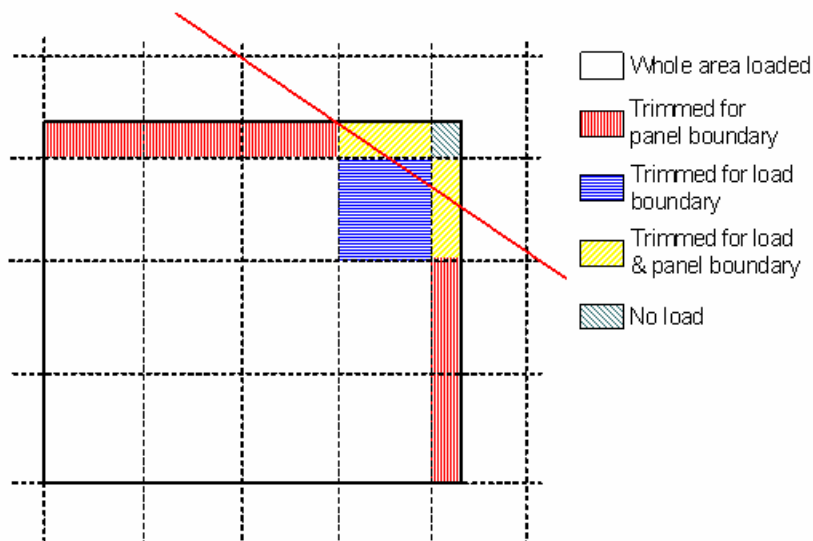
Aspect ratio	Mesh density
1 : 1	4×4
1 : 2	2.83×5.65
1 : 4	2×8
1 : 10	1.265×12.65
1 : 16	1×16

The calculation of the loading grid size can then be calculated on a panel by panel basis and the final size selected to give adequate representation on the smaller panels, with not being skewed unduly by a few very small panels. To ensure that this is the case the average (μ) and standard deviation (σ) of the individual panel loading grid sizes can be calculated and the loading grid size set to $\mu - \sigma$.

Legacy option – load distribution for a distributed load

Where a cell is bisected by a panel edge the load is applied to the panel at the centroid of the grid cell.

For the cases where this is too coarse the grid refinement factor can be increased.



[Panels](#) can be inclined with respect to the grid plane to which the grid load is applied. Also, grid area loads can be specified as projected. Both of these circumstances result in the specified load intensity being factored.

Inclined panels

For grid area loads specified as not 'projected' the load intensity depends on the panel area relative to the panel area projected on to the grid plane. If the direction of the normal to the grid plane is $\mathbf{n}_g$ and the direction of the normal to the panel is $\mathbf{n}_p$ then the load intensity is factored by

$$\frac{1}{\mathbf{n}_g \cdot \mathbf{n}_p}$$

- to result in the specified intensity being applied to the panel.

For panels that lie parallel to the grid plane this factor is unity.

Projected loads

For grid area loads specified as 'projected' the load intensity is reduced depending on the panel area projected on to the plane normal to the loading axis. If the direction of the load is $\mathbf{n}_l$ and the direction of the normal to the panel is $\mathbf{n}_p$ then the load intensity is factored by

$$\mathbf{n}_l \cdot \mathbf{n}_p$$

The inclination of the panel to the grid plane is ignored for loads specified as 'projected'.

Plane approximations option

The idea behind a plane approximation is that the load can be integrated first and then represented as a single point load. This does not give as fine a representation of the load distribution as the legacy method but as the calculations are much simpler it is much faster.

The load applied to the panel is integrated so that it can be represented as a point load.

$$W = \int_A w dA$$

$$x_w = \frac{1}{W} \int_A wx dA$$

$$y_w = \frac{1}{W} \int_A wy dA$$

The plane approximation assumes a load intensity function of the form

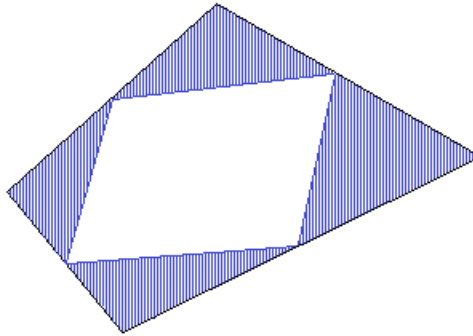
$$w = a + bx + cy$$

The intensity of the load is then calculated based on an assume uniform load along each edge with the intensity determined at the centre of the edge. The intensity function can then be calculated by solving

$$\begin{Bmatrix} \hat{w}_i \\ \hat{w}_i x_i \\ \hat{w}_i y_i \end{Bmatrix} = \begin{bmatrix} n & \sum_n x_i & \sum_n y_i \\ \sum_n (x_i - x_w) & \sum_n x_i (x_i - x_w) & \sum_n y_i (x_i - x_w) \\ \sum_n (y_i - y_w) & \sum_n x_i (y_i - y_w) & \sum_n y_i (y_i - y_w) \end{bmatrix} \begin{Bmatrix} a \\ b \\ c \end{Bmatrix}$$

where i refers to the i^{th} edge and n is the number of edges

If these coefficients yields negative intensities those edge are left unloaded and the coefficients recalculated. Where these conditions cannot be met (in corners, shown hatched below) the load is assumed to be uniform for a distance from the corner as required by equilibrium.



While this gives a load distribution that is in equilibrium with the applied load is is not realistic for panels with a high aspect ratio (ie long thin panels). In this case the intensities are adjusted so that more load is applied to the long edges (in effect the panel at the centre of long edges is practically one way spanning. To achieve this the load intensity is modified by so that

$$\hat{w}_i \rightarrow w_i \frac{r_i}{R}$$

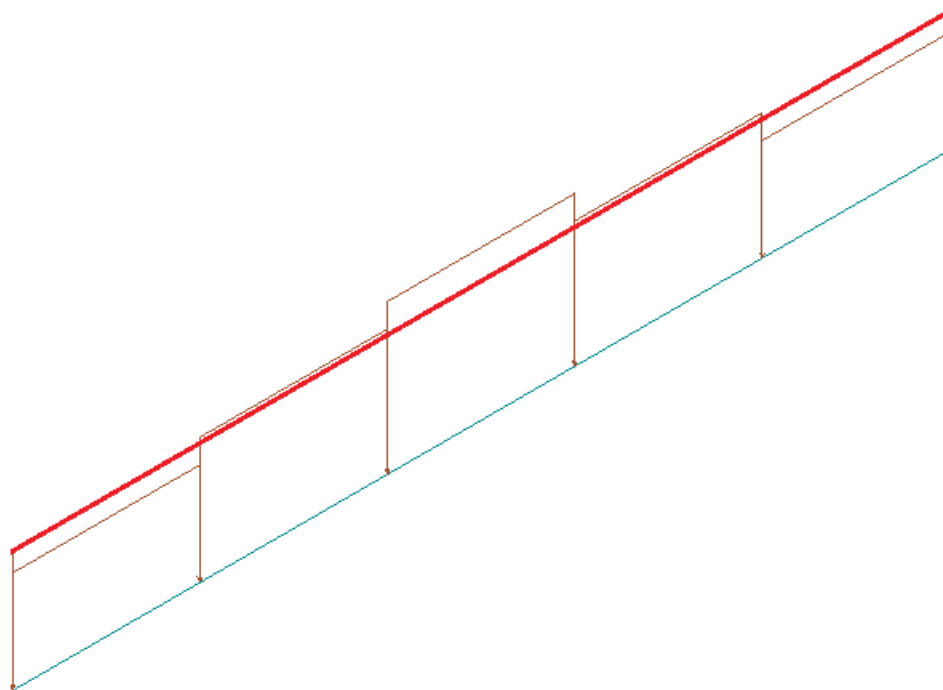
where r_i is distance from the centre of load to the mid-point of the edge and R is a representative dimension chosen so that

$$\sum_n \left(w_i \frac{r_i}{R} \right) = \sum_n \hat{w}_i$$

The disadvantage of this approach is that the load is calculated for each element along the edge so the load is concentrated at the middle of an edge when the edge is subdivided into a number of

elements while for a simple panel with one element per edge this does not happen, so the load distribution is model dependent.

To overcome this problem the smooth option was introduced. The smoothing option takes the plane distribution as its starting point. The set of uniform loads along each edge are replaced by a single linearly varying load then is in equilibrium with the applied loads.

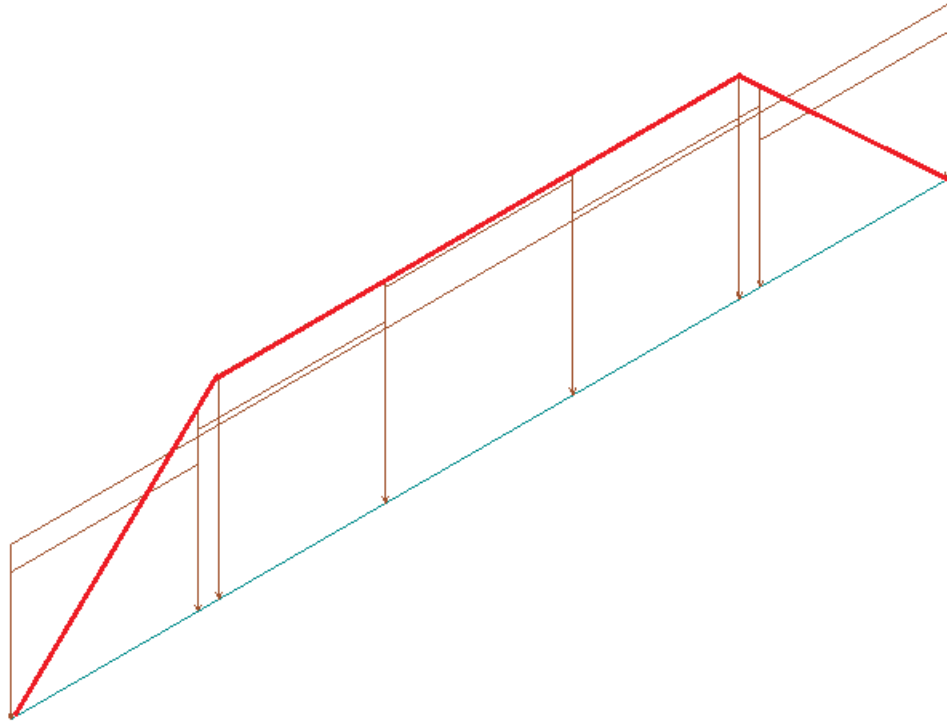


$$\sum_{edge} w_i = w_1 \left(1 - \frac{x_i}{l} \right) + w_2 \frac{x_i}{l}$$

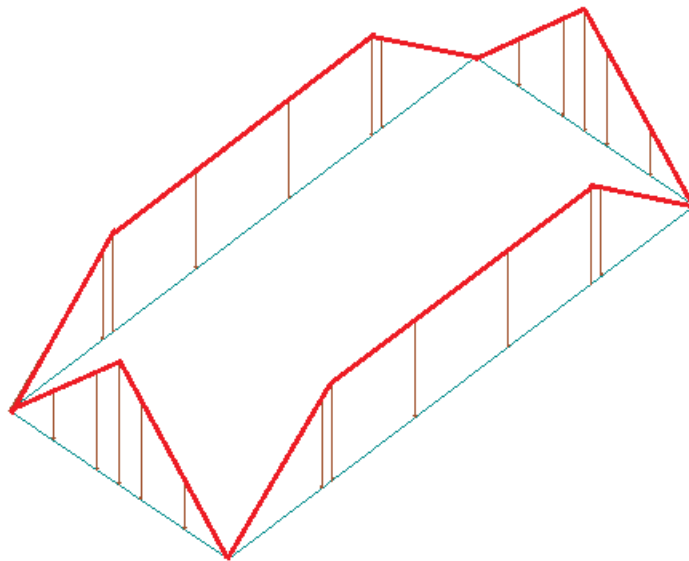
This removes the load distribution depending on the element density along the edges.

The final grid load option allows for corner effects. Where there is a 'sharp' corner the load intensity close to that corner is likely to be reduced. The adjustment for corner effects reduces until at 180 degrees there should be no adjustment.

With the corner option GSA finds the shortest edge and assumes that the extent of the corner effect is 1/2 of the minimum edge side. This approximation gives a load pattern familiar to engineers for rectangular panels, so seem intuitively right.



The linearly varying load is adjusted so that it is a tri-linear load. This can be done by considering equilibrium of force and moment for equivalent linear and tri-linear loads where the transition points for the tri-linear loads are determined directly from the panel geometry. This is shown diagrammatically below.



Discussion

Of the four grid load options in GSA for multi-way spanning grid planes the plane approximation with corner effects is recommended. This method is relatively fast and gives load patterns that are intuitive to a structural engineer. This typically gives slightly higher mid-span loads than the smoothing option, which is realistic. The basic plane approximation also gives higher mid-span loads when the edge is subdivided into multiple elements, but the load at mid-span can be too great and is always mesh dependent so generally this option is not preferred.

The legacy option is slow and is restricted in the type of panels it can load - so for example re-entrant panels cannot be loaded. This is retained only for compatibility with analysis in previous versions of GSA.

22.9 Patterned Load Analysis

Analytical Derivation

By the principle of superposition for linear elastic structural systems, the internal force in a section f can be calculated as

$$f_A = \iint_A I w dx dy \quad (1)$$

where, A is the floor area domain across the x - y plane, I is the influence surface function across the x - y plane, and w is an un-factored distributed load function varying across the x - y plane.

For the maximum internal force in a section $f_{A \max}$ resulted under a range of distributed load $w_{\max}$ and $w_{\min}$ can be calculated as

$$f_{A \max} = \iint_A I [p w_{\max} + (1-p) w_{\min}] dx dy \quad (2)$$

where, p is a binary function related to the influence surface I as

$$p = \begin{cases} 1 & I > 0 \\ 0 & I \leq 0 \end{cases} \quad (3)$$

And thus equation (2) can further be rewritten as

$$f_{A \max} = \iint_A I \left(\frac{w_{\max} + w_{\min}}{2} \right) dx dy + \iint_A I \left(\frac{w_{\max} - w_{\min}}{2} \right) dx dy \quad (4)$$

The floor area domain A can always be separated into a series of smaller and non-overlapping area a_i , which exclusively covers the entire area A . Assume the sign of I in each individually separated area a_i does not change, i.e. I is always positive or negative across the x - y plane within an area a_i , equation (4) can be expanded as

$$f_{A \max} = \iint_A I \left(\frac{w_{\max} + w_{\min}}{2} \right) dx dy + \sum_i \left| \iint_{a_i} I \left(\frac{w_{\max} - w_{\min}}{2} \right) dx dy \right| \quad (5)$$

which can be further simplified as an absolute sum function

$$f_{A \max} = f_{mean} + \sum_i |\Delta f_i| \quad (6)$$

where by definition

$$f_{mean} = \frac{1}{2} \iint_A I w_{max} dx dy + \frac{1}{2} \iint_A I w_{min} dx dy \quad (7)$$

$$\Delta f_i = \frac{1}{2} \iint_{a_i} I w_{max} dx dy - \frac{1}{2} \iint_{a_i} I w_{min} dx dy \quad (8)$$

And similarly, the minimum internal force in a section f_{Amin} can be derived as

$$f_{Amin} = f_{mean} - \sum_i |\Delta f_i| \quad (9)$$

In most situations, w_{max} and w_{min} differ only by a scalar factor, which is related to the load factor of safety in ultimate limit state design. Putting

$$w_{min} = s_{min} w \quad (10)$$

$$w_{max} = s_{max} w \quad (11)$$

equations (7) and (8) can be simplified as

$$f_{mean} = \frac{s_{max} + s_{min}}{2} \iint_A I w dx dy \quad (12)$$

$$\Delta f_i = \frac{s_{max} - s_{min}}{2} \iint_{a_i} I w dx dy \quad (13)$$

By comparing equations (12) and (13) to equation (1), it can be seen that f_{mean} can be evaluated directly from the analysis with all area fully loaded, and Δf_i can be evaluated directly from the analysis with load being only applied to the area a_i , which equations (12) and (13) can be further simplified as

$$f_{mean} = \frac{s_{max} + s_{min}}{2} f_A \quad (14)$$

$$\Delta f_i = \frac{s_{max} - s_{min}}{2} f_{a_i} \quad (15)$$

This item was written by Ir. Dr. Don Y.B. Ho of Ove Arup & Partners, Hong Kong Ltd and is reproduced here with permission.

Part XXIII

23 GSA Text (ASCII) File

The format of the text data file is a record by record file with fields separated by tabs or commas. Two different versions of this file can be used. The default file extension is .gwa. This reads/writes the file in an international format and is recommended where files have to be transferred between different countries. The alternative extension is .txt. This reads/writes files using the local settings for decimal points and list separators (see "Control Panel | Regional Settings") for ease of importing the files into spreadsheets.

The file consists of a series of records, each record consisting of a key word and a number of fields each separated with a tab. There may only be one record per line, but a record may be continued on more than one line. To spread a record over several lines a continuation marker (a \) should be placed in the next field position on that line. Comments can be included in the file on any line after a comment marker (an !) character. The \ should precede any comments and anything after the continuation marker will be treated as a comment. Both comments and continuation characters should be separated from the fields by a tab. Blank lines (no space or tab characters) are acceptable. A blank field is interpreted as a null string for string fields and a zero for numeric fields.

Each record is of the form:

KEY_WORD, value, value, value, value

Where a table is referred to this represents the following input

$n, x_1, y_1, \dots, x_n, y_n$

where n is the number of points, x_i is the value of the abscissa and y_i is the value of the ordinate.

Units

GSA data is stored in SI units both in GWB files and in the program. In text files, however, data units are set by UNIT_DATA records, defaulting to SI. The syntax of the UNIT_DATA record is described in the GSA Text (ASCII) File format section.

When reading a GWA file GSA registers the current unit for each unit type, defaulting to SI. Units may be adjusted several times in one GWA file. For example, for NODE records that precede any UNIT_DATA LENGTH records coordinates will be read as being in metres; after any UNIT_DATA LENGTH records coordinates in NODE records will be taken as being in the last specified unit of LENGTH.

For clarity, all length related values in GWA records use the LENGTH option and not DISP etc., regardless of the dimension option implemented elsewhere in the program.

When writing GWA files GSA offers the option to export in SI units or model units. When the option to export in model units is taken a set of UNIT_DATA records is written at the top of the file, specifying the current model units, and all data are written in those units.

When used from the COM interface(`GwaCommand`), the default units while reading the file are model units. It's worth noting that when using `GET GwaCommand`, GSA is actually writing a GWA file and while using `SET`, GSA is reading it in.

Names

Whenever records have a field for the object name it may be entered in double quotes "...". To specify that any characters forming part of the name are not field separators. GSA outputs names surrounded by quotes whenever a field separator character is part of the name.

String IDs

Most GSA objects can have an associated string ID (sID). In text files sIDs are assigned as follows:

- For sIDs to be recognised for any object a sID must first be assigned to the model by a SID record.
- Then a sID is defined for an object by appending a colon (':') to the keyword; the text following the

colon specifies the sID.

- Having assigned a model sID it is not necessary to assign sIDs to every object.
- sIDs may not be assigned to subsidiary records such as SEC_MOD records.

Examples of sID usage in GWA files:

SID, {Mesh:assigned}

- assigns the sID "{Mesh:assigned}" to the model.

NODE:{Mesh:1}, 8, 1.4, 0.0, 4.4, 0

- assigns the sID "{Mesh:1}" to node 8

Refer to [Program Fundamentals: — String IDs](#) for information on sID formatting.

Format

The format of the specific records is as follows:

23.1 Keywords

Details of the keywords are available in the gwa.html file in the GSA Docs folder.

Part XXIV

24 Miscellaneous Text Files

[Curve File](#)

[Tab and Comma Separated Files](#)

24.1 Curve File

Curve files are used to store tables of data, and are widely used in the LS-DYNA peripheral programs supplied by Oasys. A curve file is a text file with tabs, commas or spaces separating fields. The format of the file is as follows:

```
Title
X label <TYPE/FACTOR>
Y label <TYPE/FACTOR>
Curve identifier
x0, y1
x1, y1
...
CONTINUE
Title
X label <TYPE/FACTOR>
Y label <TYPE/FACTOR>
Curve identifier
x0, y1
x1, y1
...
```

By default TYPE is undefined and the FACTOR is 1. This is to allow backwards compatibility in that old programs reading curve files would not require any modification: the labels would just give some additional information. Newer versions would be able to read this information and give tighter control over the data.

More:

[Unit types](#)

[Example](#)

24.1.1 Unit types

The current options for TYPE are:

- UNDEFINED
- FORCE
- LENGTH
- MASS
- TIME
- TEMPERATURE
- DISP

- PRESSURE
- ACCEL
- WORK
- MOMENT
- STRESS
- FORCE_PER_LENGTH
- MOMENT_PER_LENGTH
- MOMENT_PER_ROTATION
- STRAIN
- AREA
- 1ST_MOMENT_AREA
- 2ND_MOMENT_AREA
- VOLUME
- PER_LENGTH
- PER_AREA
- PER_VOLUME
- CS_AREA
- CS_AREA_PER_LENGTH
- INERTIA
- DENSITY
- AREA_DENSITY
- WEIGHT_DENSITY
- MASS_PER_LENGTH
- PER_MASS
- VELOCITY
- ROTATION
- ANGULAR_VELOCITY
- ANGULAR_ACCELERATION
- FLOW_RATE
- FREQUENCY
- ANGULAR_FREQUENCY
- PERIOD
- THERMAL_EXPANSION
- TEMPERATURE_GRADIENT
- IMPULSE
- MOMENTUM
- ANGULAR_MOMENTUM

- DAMPING
- ENERGY
- ENERGY_DENSITY
- DEGREE
- PERCENT

The FACTOR is the value by which the data values should be multiplied to convert to SI units (e.g. 0.01 for cm).

24.1.2 Example

```

Test curve
Time <TIME>
Relative acceleration <ACCELERATION/0.001>
Reference point 1
0, 0
1, 1
2, 4
3, 9
4, 16

```

24.2 Tab and Comma Separated Files

These are text files that can be assembled in a spreadsheet or a text editor. The files can be either tab delimited (with the extension .txt) or comma delimited (with the extension .csv).

The text files are of the form

```

Title
abscissa name→ordinate name
abscissa type→ordinate type
x0→y0
x1→y1
...

```

where → is the delimiter either a “tab” or “comma”.

The axis types are as defined for [Curve Files](#).

Part XXV

25 Glossary

0D elements

Ground Spring and Mass are both 0D elements.

1D elements

Beam, Bar, Tie, Strut, Spring, Link and Cable are all 1D elements.

2D elements

Quad 4, Quad 8, Triangle 3 and Triangle 6 are all 2D elements.

AASHTO

Acronym for “American Association of State Highway and Transportation Officials”.

Alignment

An alignment defines a line in space, relative to a grid plane, from which lanes tracks or vehicle are positioned.

Allman-Cook

A method for the formulation of a 2D elements stiffness matrix. Method supersedes the Bilinear formulation. Applicable to formulating the stiffness of in-plane d.o.f. for Quad 4 elements only.

Analysis layer

The analysis layer is a view of elements in a Graphic View.

Applied Displacement

A degree of freedom where the displacement is known at the beginning of the analysis, rather than being solved for.

Axes triad

Axis sets are displayed in Graphic Views as colour coded triads. The red, green and blue lines represent the x, y and z axes respectively.

Bandwidth

The width of array required to store a matrix clustered around the diagonal, with the diagonal elements along the first column. A small bandwidth will be more efficient in the storage required.

Bar element

A particular type of finite element used to model elements designed to take only axial load, e.g. truss members.

Beam element

A particular type of finite element used to model beam, column and bracing structural elements. GSA uses a linear beam element with 2 nodes.

Beta angle

The angle used to define the orientation of beam elements in space and the local axis directions of 2D elements. Also called Orientation angle.

Bilinear

The name given to a function of two variables that is linear in both variable. Used by GSA to describe the original in-plane stiffness formulation of a linear 2D element. Formulation now superseded by Allman-Cook.

Buckling analysis

An analysis to determine the buckling response of a structure. This can be either an eigenvalue

analysis for linear structures or a non-linear buckling analysis.

Cable element

Cable elements are primarily used in fabric analysis. Cables are similar to tie elements but are defined by a stiffness and mass per unit length, rather than a cross section.

Click

Clicking is the action of pressing and immediately releasing the left mouse button.

Coincident nodes

Coincident nodes are two or more nodes that are positioned within a defined distance of each other.

Collapsing coincident nodes

Collapsing coincident nodes is the operation that results in all but the lowest numbered node in a set of coincident nodes to be deleted and all references to these deleted nodes to be changed to references to the remaining node.

Constraint

A constraint on a degree of freedom, such as an applied displacement, rigid link, repeat freedom or restraint.

Current grid

A reference to a grid plane. Coordinates are defined and reported in the current grid axes. The grid drawn in Graphic Views is defined by the grid layout of the current grid.

Cycle

This is a single time step in the dynamic relaxation solution process. During a cycle the model is assumed to be a set of linked nodal masses accelerating through space. At the end of each cycle the position, velocity and forces acting on the nodes are calculated, the resulting accelerations calculated, and the process repeated.

Degree of freedom

A freedom of a node to move in a particular direction this may be either a translational or rotational direction.

Design layer

The design layer is a view of members in a Graphic View.

Diagonal array

An array which has non-zero elements only on the diagonal.

Direction cosine array

The array relating local and global coordinate system, and used to transform vectors and tensors between different coordinate systems.

Displacement vector

The calculated displacements at all the degrees of freedom in the structure.

Drag

Dragging is the action of pressing the left mouse button down and moving the mouse before releasing the mouse button.

Drilling d.o.f.

The rotational degree of freedom (d.o.f) of a 2D element that rotates about the normal of the element plane. This 'zz' d.o.f is available for 2D elements specified with the [Allman-Cook](#) formulation.

Dummy element

An element that has its dummy attribute set. Dummy elements are ignored during analysis. Certain results can be output for dummy elements.

Dynamic analysis

An analysis to determine the dynamic response of a structure. A modal analysis gives the dynamic characteristics of the structure while a time-history analysis or response spectrum analysis gives the structural response.

Eigenvalue

Part of the solution of an eigenproblem. A dynamic analysis is an example of an eigenproblem and the natural frequencies correspond to the eigenvalues.

Element

An element is an entity that is analysed. An analysis model is made up of elements.

Engineering scale

An engineering scale is a scale of 1, 1.25, 2, 2.5 or 5 multiplied by a power of 10.

Entity

A node, element or member.

Eye distance

The eye distance in a Graphic View is the distance from the eye point to the object point.

Eye point

The eye point in a Graphic View is the position from which the model is being viewed.

Finite element

A mathematical representation of a piece of the structure located by nodes and contributing to the stiffness/inertia of the structure.

Finite element method

The method for analysis of structure by dividing the structure into a collection of connected finite elements.

Force vector

The collection of forces applied to an element or to the structure as a whole. Normally the force vector for the structure is known and we wish to calculate the displacements. Once the displacements at the nodes are calculated we then want to calculate the internal forces in the elements.

Form-finding

A process by which the natural shape of a fabric structure is established.

Front order

The order in which nodes and elements are merged into the structure stiffness matrix.

Full image

The full image is a Graphic View image that displays all aspects of the currently specified view.

Gaussian elimination

A method for solving large systems of linear equations by successively removing degrees of freedom.

Generic bar elements

Bars, ties and struts are all generic bar elements.

Geometric stiffness

The additional stiffness of an element that results from the load in the element.

Ghost image

The ghost image is a Graphic View image that displays a cuboid representing the global extents of the image. A triad representing the global directions is drawn at the object point. A dashed triad is drawn at the mid-point of the view.

Global axes

The global axes are the axis set to which all other axes relate. The datum axis set for the model.

Grid axes

The axes referred to by a grid plane. Commonly used to refer to the axes of the current grid.

Grid layout

Defines the distribution of grid points or grid lines as drawn in Graphic Views.

Grid plane

A grid plane is a way of defining a plane for loading etc by reference to an axis set and elevation.

Grid structure

A Grid structure is one modelled in the global XY (horizontal) plane. Global restraints are applied to the model to force it to deform only in the Z direction.

Hourglassing

Hourglassing arises with under-integrated elements where there are insufficient stiffness terms to fully represent the stiffness of the element and is noticeable in the results by an hourglass pattern in the mesh.

Increment

Used in non-linear, GsRelax analysis to define a single load stage. An increment represents the total imposed loads that are deemed to be present at the time of analysis. It is equivalent to an analysis case generated internally by the GsRelax Solver. For example with single increment analysis a single analysis case is created and all loads are applied at the start of analysis. Whereas with a 2-stage multiple increment analysis, the first increment could be an analysis case with half the load applied and the second increment is an analysis case with the total load applied.

Inertia matrix

Same as [mass matrix](#).

Influence analysis

An analysis that calculates the effect at a point due to load at a series of points along the structure.

Jacobi method

A method for calculating eigenvalues and eigenvectors used by GSA within the subspace iteration.

Joint

A joint is a constraint which links two nodes together in specified directions and is used for modelling features such as pinned connections.

Lagrange multiplier method

A technique that is used to impose constraints on a model; in particular applied displacements.

Legend

The legend is the panel of information that describes the content of a Graphic View. The legend may be displayed either on the Graphic View or in a message box.

Linear element

A finite element with a low order shape function and no mid-side nodes.

Linear shape function

2-D elements without mid-side nodes interpolate strains linearly between nodes. This can lead to significant stress and strain discontinuities across element boundaries.

Link element

A link element is a two node element that is rigid in the specified directions, and transfers forces and moments through the structure.

List field

This is a field that allows the user to enter a list. The syntax of the list is then checked before the field is accepted.

Lumped mass

A mass (and inertia) at a node. A lumped mass is treated as a one node element.

Mass matrix

The relationship between force and acceleration in a linear system. In finite element analysis we consider the inertia of the element as represented by the element mass matrix and the inertia of the whole structure as represented by the structure mass matrix.

Member

A member is an entity that is designed. Typically a member relates to one or more elements.

Mid-point

The mid-point is a 3D vector used to specify the pan in a Graphic View. The mid-point vector is the position of the mid-point of the picture relative to the object point in picture axes.

Mindlin

Mindlin-Reissner plate theory. In GSA we label our 2D element formulation that is based on Mindlin-Reissner plate theory concisely as the Mindlin formulation.

MITC

Mixed Interpolation of Tensoral Components. A method for the formulation of 2D finite elements. The MITC method differs from the Mindlin method in having superior predictive and stability properties.

Modal analysis

Modal analysis solves an eigenvalue problem either for a dynamic or buckling analysis of the structure.

Node

A point in space. Nodes are used to define the location of the elements.

Normalisation

A way of scaling a vector or matrix. For example the displacement vector for a node can be normalised, producing a displacement magnitude of one at that node.

Numeric field

A numeric field allows the user to enter a number. Depending on the context this will be either a real number (e.g. a coordinate) or a whole number (e.g. a load case).

Numeric format

The format used for the display of numbers. The choice is engineering, to a number of significant figures, decimal, to a number of decimal places, or scientific, to a number of significant figures.

Numeric/Percentage field

This is a field that allows the user to enter either a number or a percentage. These fields are used to allow the user to specify position in either an absolute sense (number e.g. 0.25) or a relative sense (percentage e.g. 10%).

Object point

The object point is the point about which rotations occur in a Graphic View. Also, in perspective views the position of the eye is measured from the object point.

Object to eye distance

See [eye distance](#).

Orientation angle

The angle used to define the orientation of beam elements in space and the local axis directions of 2D elements.

Orientation axes

The axes about which orientations occur in Graphic Views.

Orientation node

The node used to define the xy-plane for orientation of beam elements.

Panel

A panel is an area bounded by beam elements.

Parabolic element

A finite element with a higher order shape function — usually with mid-side nodes.

Parabolic shape function

2D elements with mid-side nodes (quad8 and tri6) interpolate strains using a fitted parabolic function. This provides more realistic stresses and strains than elements with linear shape functions.

Path

A path is the line across a structure followed by a lane, track or vehicle.

Picture axes

The picture axes in a Graphic View are x, left to right, y upwards and z out of the plane of the picture.

Plane structure

A Plane structure is one modelled in the global XZ (vertical) plane. Global restraints are applied to the model to force it to deform only in the XZ plane.

Prescribed displacement

See “Applied displacement” and “Settlement”.

Quad element

A linear or parabolic element used in two dimensional analyses with 4 sides and 4 or 8 nodes defining the element.

Reaction

Force which is generated at restraint.

Restraint

A degree of freedom that is fixed so that it is no longer free. Special cases of restraints are pins and encasté supports. (See also: Member Restraint and Restraint Property.)

Section definition axes

The axis system used for defining beam sections.

Selection field

This is a field that allows the user to choose from a list of options.

Selection/Numeric field

This is a field that allows the user to either choose from a list of options or enter a number (e.g. when specifying a material either a standard material can be selected or a user defined material specified).

Settlement

A degree of freedom which is restrained and given a fixed displacement prior to analysis.

Shear beam

A particular type of finite element similar to a beam element but taking account of shear deformations.

Skyline array

A storage scheme which only holds the entries in the stiffness or mass matrix which are on and above the diagonal. Its name derives from the outline of the array elements that are stored.

Space structure

A Space structure is one that may be modelled in 3 dimensions.

Spacer element

A spacer element is used to control the mesh during a form-finding analysis.

Spring element

An element which is defined by stiffnesses in each of the translational and rotational directions. Used to model springs and features such as elastic supports. A one node spring is considered to be a grounded spring.

Spring support

A spring support is a stiffness associated with a node. The spring support connects the node to ground (i.e. it restrains the model) but with a user defined stiffness.

Static analysis

A static analysis loads at how the structure responds to loads which do not vary with time.

Stiffness matrix

The relationship between force and displacement in a linear system. In finite element analysis we consider the stiffness of the element as represented by the element stiffness matrix and the stiffness of the whole structure as represented by the structure stiffness matrix.

Structure scale

The structure scale is the scale at which the structure is drawn in a Graphic View.

Strut element

A strut element is similar to a bar element in that it has axial stiffness only, but will only take compressive forces. Note that since this is a non-linear element, post-analysis combining of results is inappropriate.

Subspace iteration

A technique for solving for eigenvalues and eigenvectors iteration of large systems, by projecting the system of equations on to a smaller "subspace".

Text field

A text field allows the user to enter a string of text and is normally used for names.

Thread

Threads are a way for a computer program to fork (or split) itself into two or more simultaneously running tasks.

Tie element

A tie element is similar to a bar element in that it has axial stiffness only, but will only take tensile forces. Note that since this is a non-linear element, post-analysis combining of results is inappropriate.

Topology

The list of nodes which define an element or member.

Torque lines

Torque lines are generalised thrust lines for two or three dimensions. In 2D (plane frame analysis) they are synonymous with thrust lines (i.e. they show the line of action of axial load) but in three dimensions they can contain both torque and force components, hence the name.

Tree control

A tree control is a standard Windows control that offers options via expandable branches. Click on the “+” to expand a branch and on the “-” to collapse the branch. Double click on an item to invoke that item.

Sometimes multiple selection of items is possible. This is achieved by holding the control key down to continue selection or the shift key to include all items between the currently selected and the item Shift-Clicked.

Triangular element

A linear or parabolic element used in two dimensional analyses with 3 sides and 3 or 6 nodes defining the element.

Unattached node

An unattached node is one that is not referenced in the topology list of an element.

Variable UDL

A variable UDL is a uniformly distributed load whose intensity varies depending on the loaded length. This is used in bridge analysis.

Vehicle

A vehicle is defined as a list of axle positions and loadings along with an overall width for use in bridge analyses.

VUDL

See Variable UDL.

Wireframe image

The wireframe image is a Graphic View image that represents 1D elements as lines and 2D elements as outlines and displays nothing else, regardless of the current settings for the view.

X elevation axes

The X elevation axis set is a standard axis set that has its xy plane in the global YZ plane.

Y elevation axes

The Y elevation axis set is a standard axis set that has its xy plane in the global XZ plane.

Index

- * -

*= command
in table views 114

- / -

/= command
in table views 114

- + -

+= command
in table views 114

- < -

<displayed result>
in Object Viewer 112

- = -

= command
in table views 114

- - -

-- command
in table views 114

- = -

== command
in table views 114

- 1 -

1D Element Results Dialog 498
1D elements
pre-stress 222

- 2 -

2D derived force results
output options 527
2D derived stress results
output options 526
2D element displacements
output options 525
2D element edge loads
create dialog 374
definition dialog 374
2D element face loads
create dialog 373
definition dialog 373
2D element facet angle
change default 483
2D element loads
fundamentals 225
in non-linear analysis 226
load panel fundamentals 226
2D element pre-stress loads
create dialog 375
definition dialog 375
2D element stresses
output options 525
2D element thermal loads
create dialog 375
definition dialog 375
2D elements
2D Property Wizard 355
axis fundamentals 205
centre value results only 412
convert Quad4 to Quad8 165
forces 636
fundamentals 201
Generate 2D Elements Dialog (legacy option)
473
geometry checks 409
geometry contour options 760
Hourglassing 761
linear analysis introduction 58
linear to parabolic 165
load panels 226
loading data 272
mesh generation 44
modelling and loading introduction 56
moments 636

2D elements
 pre-stress 225
 property data 255
 results introduction 58
 shape 201
 shape checks 760
 split warped 461
 step by step 56
 stesses 636
 wall element modelling 56
 wall results 528
 wall stick direction 498
 2D polylines
 check for validity 339
 data 247
 definition dialog 339
 2D Property Wizard
 Load panels 357

- 3 -

3d modelling links 574
 3D PDF
 capture of graphic image 145

- A -

Abort command 144
 abs combination case operator 232
 accelerators 308
 Accidental torsion loads 761
 ACI 26
 Acknowledgements 32
 Active Column solver 407
 GSS theory 634
 ADC
 export options 570
 add elements sculpt tool 157
 add lines sculpt tool 158
 Add Nodes command 159
 add nodes sculpt tool 156
 Add String of 1D Elements command 161
 adornments
 apply to selection 136
 advanced features
 fundamentals 192
 preferences 485

Advanced Preferences Dialog 488
 Advanced Settings Dialog
 GsRelax 407
 Gss 407
 after analysis status 54
 AISC LRFD checker
 documentation 717
 AISC360 checker
 documentation 716
 Align 1D Elements command 168
 Align 2D Element Edges command 169
 Align 2D Element Z Axes command 168
 alignments
 data 283
 extrude geometry along 446
 Allman-Cook 409
 Analyse All command 55
 Analyse All Marked Tasks command 179
 Analyse command 178
 analysis
 Analysis Wizard 393
 bridge step by step 83
 bridge structures 87
 buckling step by step 64
 complex introduction 55
 creep 601
 dynamic step by step 60
 feature preferences 485
 footfall analysis 76
 harmonic analysis 73
 introduction to requesting 54
 introduction to static 54
 linear 2D element step by step 56
 linear static step by step 56
 long term 601
 non-linear step by step 66
 P-delta step by step 59
 periodic load excitation analysis 75
 piled-raft analysis 78
 raft analysis 78
 response spectrum step by step 72
 Ritz dynamic step by step 63
 seismic step by step 71
 status after completion 54
 step by step summary 55
 terminating 54
 threading 488

- analysis
 - Time history analysis 74
 - wizard introduction 55
- analysis cases
 - data 298
 - definition dialog 393
 - fundamentals 229
- analysis envelopes
 - Analysis Wizard 406
 - step by step 91
- analysis layer basics 88
- Analysis Layer in graphic views 141
- analysis options
 - Analysis Wizard 394
- Analysis Progress Dialog
 - GsRaft 413
- Analysis Specification Dialog 324
- analysis stages
 - colour elements by 424
 - creep 296
 - data 296
 - definition data 296
 - in graphic views 91
 - introduction to analysis 90
 - introduction to analysis results 90
 - long term analysis 296
 - modelling implications 88
 - property data 297
 - results introduction 90
 - step by step 87
- analysis tasks
 - data 298
 - fundamentals 230
- analysis tasks and cases
 - Analysis Wizard 181
 - copy and paste 181
 - working with 178
- Analysis Wizard
 - analysis envelopes 406
 - analysis options 394
 - buckling parameters 397
 - cases set up 406
 - footfall analysis parameters 403
 - form finding options 398
 - fundamentals 230
 - generic dynamic analysis parameters 402
 - GsBridge control 405
 - GsRaft control 404
 - GsRaft progress 405
 - GsRelax analysis progress information 399
 - GsRelax analysis task control 398
 - GsRelax member identification 399
 - harmonic analysis parameters 400
 - linear time history analysis parameters 401
 - mass calculation 397
 - modal dynamic parameters 395
 - non-linear static options 397
 - pseudo response spectrum analysis 400
 - response spectrum analysis 399
 - Ritz analysis parameters 396
 - static analysis cases 394
 - static p-delta 395
 - working with 181
- angle between nodes 111
- Angle check contour option 760
- Animate 142
- Animate command 142
- animation
 - Animation Settings Dialog 434
 - in graphics 142
 - settings for graphics 434
- Animation Settings command 142
- Annotate Selection command 136
- annotating graphic diagrams etc 136
- annotation of diagrams options 432
- ANSYS ASAS
 - export options 568
- Append GWA data 186
- applied displacements
 - create dialog 369
 - data 268
 - definition dialog 369
 - fundamentals 221
- Apply Adornments to Selection command 136
- Apply Defaults command 156
- Apply Defaults to Selection command 156
- Approximate Minimum Degree 407
- areas
 - copy / move graphically 165
 - Copy Areas Dialog 445
 - Create Regions per Grid Plane Dialog 459
 - definition dialog 335
 - labels in graphic views 423
 - Move Areas Dialog 445
 - working with 184

- AS 26
 - AS4100 checker
 - Buckling checks 713
 - documentation 711
 - Effective Section Properties 712
 - input data requirements 712
 - Local checks 712
 - ASAS
 - export options 568
 - ASCE 7 -05
 - ASCE7-05 spectrum 380
 - ASCII (GWA) file
 - syntax 776
 - assemblies 103
 - Check Assemblies command 237
 - data 299
 - definition dialog 391
 - fundamentals 237
 - labels in graphic views 423
 - Assisted Input Toolbar 302
 - auto views
 - details 177
 - AutoCAD
 - export options 568
 - import options 572
 - automatic constraints
 - fundamentals 220
 - AVI
 - saving image to 144
 - software interaction via 573
 - axes
 - 2D element fundamentals 205
 - cable fundamentals 204
 - coordinates, non-global definition fundamentals 197
 - Create User Axes Dialog 436
 - creating user graphically 158
 - cylindrical and spherical fundamentals 196
 - data 240
 - definition dialog 330
 - element fundamentals 203
 - global cylindrical 195
 - grid fundamentals 197
 - grid planes 247
 - grounded spring fundamentals 204
 - in output views 154
 - in table views 117
 - labels in graphic views 421
 - link fundamentals 205
 - mass fundamentals 204
 - Modify Axis Dialog 330
 - projected fundamentals 197
 - quad fundamentals 205
 - section fundamentals 205
 - sets fundamentals 195
 - spring fundamentals 204
 - steel member section axes 702
 - triangle fundamentals 205
 - use of fundamentals 196
 - vertical 195
 - x elevation 195
 - y elevation 195
 - Axes command
 - in graphic views 137
 - in output views 154
 - Axes Dialog 499
- ## - B -
- backup
 - set time interval 484
 - backups
 - working with 186
 - bar layout 263
 - bars
 - fundamentals 200
 - base shear
 - response spectrum 678
 - basic concepts in GSA 106
 - Batch Analysis command 179
 - Batch Bridge Analysis command 466
 - Batch Bridge Analysis Dialog 466
 - batch output file formats 484
 - batch printing views and view lists 178
 - beam cross strain energy density
 - output options 523
 - beam derived stresses
 - output options 519
 - beam displacements
 - output options 517
 - beam distortion loads
 - create dialog 371
 - definition dialog 371
 - beam elements
 - pre-stress 222

- beam forces
 - output options 518
 - beam loads
 - create dialog 370
 - definition dialog 370
 - fundamentals 222
 - in non-linear analysis 224
 - beam pre-stress loads
 - create dialog 371
 - definition dialog 371
 - beam sections
 - catalogue fundamentals 208
 - data 251
 - data in non-linear analysis 251
 - database fundamentals 213
 - definition axes 347
 - explicit fundamentals 212
 - fundamentals 205, 206
 - geometric fundamentals 210
 - Multiple Sections Dialog 353
 - naming convention fundamentals 207
 - Section Modifiers Dialog 352
 - standard fundamentals 208
 - wizard 347
 - beam strain energy density
 - output options 523
 - beam stresses
 - output options 519
 - beam thermal loads
 - create dialog 372
 - definition dialog 372
 - beams
 - axis fundamentals 203
 - concrete beam design property data 261
 - fundamentals 199
 - load data 269
 - loading data 269
 - RC beam design property data 261
 - RC Member Design Property Definition dialog 360
 - RC slab design property data 262
 - Steel Beam Design Property Definition dialog 358
 - steel design property data 260
 - Steel Restraint Property Definition dialog 359
 - bearings
 - data 265
 - beta angle
 - beam axis fundamentals 203
 - BIM 574, 585
 - black background in graphic views 435
 - blank in table views 116
 - bridge
 - analysis 87
 - bridge analysis
 - modelling implications 86
 - step by step 83
 - Bridge Loading Analysis Specification Dialog 328
 - Bridge Options Dialog 432
 - bridges
 - bridge loading data 285
 - data 283
 - Expand Bridge Loading Dialog 466
 - Broadcast Case List command 304
 - BS 26
 - BS5950 checker
 - angle struts 706
 - buckling checks 705
 - buckling interaction check 705
 - combined local check 704
 - documentation 704
 - input data requirements 704
 - limitations 707
 - local axial check 704
 - local bending check 704
 - local checks 704
 - local shear check 704
 - local torsion check 704
 - max plastic/elastic ratio 706
 - specialised input data 706
 - buckling analysis
 - step by step 64
 - buckling analysis results
 - output options 515
 - buckling parameters
 - Analysis Wizard 397
- C -**
- cables
 - axis fundamentals 204
 - definition dialog 357
 - fundamentals 201
 - property data 257
 - theory 657

- CAD files
 - use for constructing a model 48
- calculation sheet layout 484
- case description
 - definition dialog 390
- case descriptions 299
- cases
 - analysis case data 298
 - Analysis Case Definition Dialog 393
 - analysis case fundamentals 229
 - analysis task data 298
 - basics 108
 - Case Description 390
 - combination case data 298
 - combination case fundamentals 231
 - Combination Cases dialog 390
 - data 297
 - fundamentals 229
 - load case data 297
 - load case fundamentals 229
 - Load Case Titles dialog 389
 - output by 153
 - selecting for graphic views 135
 - selecting for output 152
 - syntax of combination descriptions 232
- cases set up
 - Analysis Wizard 406
- catalogue sections
 - fundamentals 208
- CEAP Modal Damping Dialog 467
- cell operators
 - in table views 114
- Chart
 - Axis Style 148
 - Background Style 149
 - Curve Style 148
 - Point Style 149
 - View Style 147
- chart views
 - chart view settings 452
 - context menu 145
 - Footfall analysis 454
 - Forces on 2D Element Cut 456
 - Generic periodic load analysis 454
 - Harmonic analysis 453
 - Linear time history analysis 454
 - Modal analysis details 453
 - non-linear analysis 453
 - saved chart settings 457
- Check Assemblies command 237
- Check Data command 179
- Check for completeness
 - entity lists 462
- Chinese Standards
 - GB 50011-20012 spectrum 383
- CIMsteel
 - export options 569
 - import options 573
- CIMsteel Step files
 - use for constructing a model 48
- Clear Colour Modifications command 137
- clipboard
 - copy graphic image to 143
 - copy output to 449
- Codes
 - Design 26
- coexistent values in envelopes
 - specifying 451
- coexistent values in output view envelopes 152
- coincidence tolerance
 - changing 481
- coincident elements
 - deleting graphically 128
 - highlighting in graphics 140
 - selecting 128
- coincident nodes
 - collapsing graphically 160
 - deleting graphically 128
 - highlighting in graphics 140
 - selecting 128
- Collapse Coincident Nodes command 160
- colour
 - dark background 435
 - in graphics 137
 - in tables 118
 - overrides in graphics 435
- colour elements by property etc. 424
- columns
 - RC Member Design Property Definition dialog 360
- COM Export Functions 533
- COM interface 533
 - COM C++ example 557
 - COM export functions 534
 - COM VBA example 557

- COM units 776
- COM version 533, 558
- combination cases
 - Append Analysis Case 391
 - Append Combination Case 391
 - Append Envelope 391
 - Append List Case 391
 - data 298
 - definition dialog 390
 - fundamentals 231
 - syntax of descriptions 232
- Combine model 186
- command file (GWC)
 - run by 559
- command line interface 533
- Command Toolbar 307
- company name
 - setting 484
- Comparing models 186
- complex analysis
 - introduction 55
- Concrete
 - Design Codes 26
- concrete beam
 - dialog 497
 - export 497
- concrete beam design 497
- concrete beam design properties
 - data 261
- concrete member bar layouts
 - Definition dialog 365
- concrete member design properties
 - Definition dialog 360
- concrete properties
 - Member Wizard: Concrete Properties 344
- concrete slab design properties
 - data 262
 - Definition dialog 361
- Condense Cases option 304
- condition number 596, 624
- conditioning 596
- conflicting constraints
 - fundamentals 220
- Conjugate Gradient solver 407
 - GSS theory 634
- Connect 1D Elements command 162
- Connect 1D Elements Dialog 437
- Connect Lines command 162
- connecting 1D elements graphically 162
- connecting lines graphically 162
- consistent mass 608
- Constraint equations 599
 - data 266
 - definition dialog 368
 - fundamentals 219
- Constraints
 - automatic fundamentals 220
 - axes fundamentals 198
 - conflicting fundamentals 220
 - constraint equation data 266
 - constraint equation fundamentals 219
 - constraint equations 599
 - creating rigid graphically 170
 - fundamentals 216
 - joint data 265
 - joint fundamentals 219
 - joints 598
 - rigid constraint data 264
 - rigid constraints 598
 - rigid fundamentals 218
 - tied interface data 266
 - tied interface fundamentals 219
- constructing a model
 - CAD and Step files 48
 - introduction 43
- construction (analysis) stages
 - in graphic views 91
 - introduction to analysis 90
 - introduction to analysis results 90
 - modelling implications 88
 - results introduction 90
 - step by step 87
- construction grid
 - in graphic views 127
- contour barrel widths in graphic views 435
- Contour Settings command 134
- Contour Settings Dialog 427
- Contour Values Dialog 427
- contours
 - apply to selection 136
- contours on graphic views 134
- constraints
 - data 263
- Convergence

- Convergence
 - Jacobi 408
 - subspace iteration 408
- Convert Nodes to Grid command 241
- Convert Nodes to Standard command 241
- Convert Orientation Node to Angle command 173
- coordinates
 - axes, non-global definition fundamentals 197
 - grid fundamentals 197
 - rounding graphically 160
- copy
 - Copy Areas Dialog 445
 - Copy Elements Dialog 445
 - Copy Lines Dialog 445
 - Copy Members Dialog 445
 - Copy Nodes Dialog 445
 - Copy Regions Dialog 445
 - from Gateway 110
 - graphic image to clipboard 143
 - nodes, elements, members, lines, areas, regions graphically 165
 - Object Viewer 111
 - output to clipboard 449
- copy and paste analysis tasks and cases 181
- Copy command 143, 154
 - in Graphic Views 130
- Copy Selection command 165
- copying
 - cutting and pasting in table views 115
- copying between table views and spreadsheets 117
- Create 1D Elements from Lines command 161
- Create 2D Edge Load command 171
- Create 2D Face Load command 171
- Create 2D Pre-stress command 171
- Create 2D Thermal Load command 171
- Create a New Raft Model command 464
- Create a New Raft Model Dialog 464
- Create Applied Displacement command 171
- Create Beam Distortion command 171
- Create Beam Load command 171
- Create Beam Thermal Load command 171
- Create CEAP Modal Damping File command 467
- Create Facade List commands 475
- Create Footfall Response Data command 470
- Create Gravity Load command 171
- Create Grid Area Load command 172
- Create Grid Line Load command 172
- Create Grid Plane command 159
- Create Grid Point Load command 172
- Create Joints command 170
- Create Lines from 1D Elements command 160
- Create Load Panels on Surface dialog 475
- Create Mass Elements from Element Mass command 173
- Create Mass Elements from Grid Loads command 477
- Create Mass Elements from Member Mass command 173
- Create Mass from Loads command 476
- Create Mass from Loads Dialog 476
- Create Members from Elements command 459
- Create Members from Elements Dialog 459
- Create New Model from Deformed Geometry command 460
- Create New Model from Deformed Geometry Dialog 460
- Create Node Load command 171
- Create Patterned Load Combinations dialog 476
- Create Patterned Loads dialog 475
- Create Pre-stress command 171
- Create Regions per Grid Plane command 459
- Create Regions per Grid Plane Dialog 459
- Create Rigid Constraints command 170
- create rigid membranes from storeys 186
- Create Settlement command 171
- Create Storey Grid Planes command 185
- Create User Axes command 158
- Create User Axes Dialog 436
- creep
 - analysis 601
 - analysis stages 296
- cross strain energy density
 - output options 523
- Cross Strain Energy Density Results command 523
- CSA 26
- CSV
 - batch saving saved views to 178
- CSV (GWA) file
 - syntax 776
- CSV file
 - export options 564
 - import options 572

- current grid
 - definition dialog 331
 - fundamentals 197
 - in graphic views 127
 - current polyline in Graphic Views 131
 - Cursor Mode Toolbar 305
 - cursor modes
 - in Graphic Views fundamentals 237
 - Curve Data Selection Dialog 491
 - curves
 - alignment data 283
 - extrude geometry along alignment 446
 - Modify Curve Dialog 491
 - custom links
 - fundamentals 218
 - property data 256
 - cut
 - Forces on 2D Element Cut 456
 - Cut section forces 545
 - Assemblies 299
 - Cut Section Forces command 457
 - Cut Section Forces Dialog 457
 - cutting
 - copying and pasting in table views 115
 - cutting 1D elements graphically 162
 - cutting lines graphically 162
 - cylindrical axes fundamentals 196
- D -**
- damping
 - CEAP Modal Damping Dialog 467
 - damping table 280
 - modal damping 601
 - theory 648
 - Damping Coefficient and Soil Stiffness Dialog
 - GsRaft 413
 - Damping Coefficient Dialog
 - GsRaft 414
 - damping table
 - data 280
 - dark background in graphic views 435
 - data
 - delete from Gateway 109
 - extents in output views 153
 - generation basics 108
 - Manage Data Dialog 470
 - selecting what to output 152
 - data defaults
 - in graphics (sculpting) 156
 - Data Defaults Dialog 156
 - Data Generation Wizard 314
 - Data Management
 - comparing models 186
 - create grid planes from storeys 185
 - create rigid membranes from storeys 186
 - create storey grid planes 185
 - import GWA data 186
 - tools 185
 - Data Options Toolbar 304
 - data preferences 482
 - data scaling unreliable message 121
 - Decrease Precision command
 - in graphic views 137
 - in output views 154
 - default view settings
 - details 175
 - introduction 175
 - defaults
 - data in graphics (sculpting) 156
 - in table views 114
 - Define Current Grid command 127
 - defining current grid
 - in graphic views 127
 - definition axes
 - fundamentals 197
 - definition dialogs
 - 2D element edge loads 374
 - 2D element face loads 373
 - 2D element pre-stress loads 375
 - 2D element thermal loads 375
 - 2D polylines 339
 - 2D Property Wizard 355
 - Analysis Case 393
 - applied displacements 369
 - areas 335
 - assemblies 391
 - axes 330
 - beam distortion loads 371
 - beam loads 370
 - beam pre-stress loads 371
 - beam thermal loads 372
 - cables 357
 - constraint equations 368

definition dialogs

- Current Grid 331
- Element Wizard 341
- Environmental Impact Wizard 330
- generalised restraints 366
- gravity loads 377
- grid area loads 376
- grid layouts 333
- grid line loads 376
- grid lines 333
- grid nodes 339
- grid planes 331
- grid point loads 376
- joints 367
- lines 334
- links 357
- masses 355
- Material Wizard 345
- members 343, 344
- Multiple Sections Dialog 353
- Node Grid Settings 340
- node loads 369
- Node Mesh Attribute 341
- Node Stiffness 340
- nodes 339
- Non-linear Spring Curve 355
- polylines 339
- RC member bar layouts 365
- RC member design properties 360
- RC slab design properties 361
- regions 336
- Response Spectrum Wizard 378
- rigid constraints 366
- Section Modifiers Dialog 352
- Section Wizard 347
- settlements 369
- spacers 358
- Spring Property Wizard 353
- steel beam design properties 358
- steel restraint properties 359
- tied interfaces 368
- Variable UDL Wizard 388

deformation

- shear effects 608, 753

Deformation Settings command 135

Deformation Settings Dialog 426

deformed geometry

Create New Model from Deformed Geometry Dialog 460

Deformed Image command 135

deformed image on graphic views 135

degrees of freedom

theory 588

delete

data from Gateway 109

Delete All Polyline command 131

Delete All Results command 55

Delete command 116, 170

Delete Displayed Loading command 173

Delete Element command 141

Delete Element Loading command 141

delete in table views 116

Delete Member command 141

Delete Nodal Loading command 141

Delete Node command 141

Delete Polyline Point command 131

Delete Results From Files command 187

deleting blank elements

data management 185

deleting blank nodes

data management 185

deleting loads

data management 185

deleting nodes and elements by data management 470

deleting results 55

derived stress output

2D elements 526

beams 519

descriptions

cases 299

design

property data 260

steel design results step by step 97

steel design step by step 93

steel design tools step by step 97

steel member restraints step by step 96

Design Code Specification Dialog 327

Design Codes 26

design features

preferences 485

design layer basics 88

Design Layer in graphic views 141

design properties

- design properties
 - RC Member Bar Layouts Definition dialog 365
 - RC Member Design Property Definition dialog 360
 - RC Slab Design Property Definition dialog 361
- design section information
 - fundamentals 213
- Diagram Settings command 133
- diagrams
 - annotation options 432
 - apply to selection 136
 - change scale on change of case switch 480
 - Diagram Settings Dialog 430
 - simple/detailed & draw in plane of picture switches 430
- diagrams on graphic views 133
- dialogs
 - 1D Element Results 498
 - 2D Element Edge Load Definition 374
 - 2D Element Face Load Definition 373
 - 2D Element Pre-stress Load Definition 375
 - 2D Element Thermal Load Definition 375
 - 2D Polyline Definition 339
 - 2D Property Wizard 355
 - Advanced Preferences 488
 - Analysis Case Definition 393
 - Analysis Specification 324
 - Analysis Wizard 393
 - Animation Settings 434
 - Append Analysis Case 391
 - Append Combination Case 391
 - Append Envelope 391
 - Append List Case 391
 - Applied Displacement Definition 369
 - Area Definition 335
 - Axes 499
 - Axis Definition 330
 - Batch Bridge Analysis 466
 - Beam Distortion Definition 371
 - Beam Load Definition 370
 - Beam Pre-stress Definition 371
 - Beam Thermal Load Definition 372
 - Bridge Loading Analysis Specification 328
 - Bridge Options 432
 - Cable Definition 357
 - CEAP Modal Damping 467
 - Combination Cases 390
 - command 468
 - concrete beam 497
 - Connect 1D Elements 437
 - Constraint Equation Definition 368
 - Contour Settings 427
 - Contour Values 427
 - Copy Areas 445
 - Copy Elements 445
 - Copy Lines 445
 - Copy Members 445
 - Copy Nodes 445
 - Copy Regions 445
 - create 2D element edge load 374
 - create 2D element face load 373
 - create 2D element pre-stress load 375
 - create 2D element thermal load 375
 - Create a New Raft Model 464
 - create applied displacement 369
 - create beam distortion load 371
 - create beam load 370
 - create beam pre-stress load 371
 - create beam thermal load 372
 - create gravity load 377
 - create grid loading 376
 - Create Mass from Loads 476
 - Create Members from Elements 459
 - create node load 369
 - Create Regions per Grid Plane 459
 - create settlement 369
 - Create User Axes 436
 - Current Grid Definition 331
 - Curve Data Selection 491
 - Cut Section Forces 457
 - Data Defaults 156
 - Data Generation Wizard 314
 - Deformation Settings 426
 - Design Code Specification 327
 - Diagram Settings 430
 - Disconnect Elements 444
 - Display Methods 424
 - DXF Export Options 495
 - DXF Import Options 495
 - Element Wizard 341
 - Environmental Impact Specification 329
 - Environmental Impact Wizard 330
 - Equivalent Static / Accidental Torsion Load 468
 - Expand Bridge Loading 466
 - Expand Gravity Loading 462
 - Expand Grid Loading 462

dialogs

- Export Member Input Data to CSV 498
- Extrude 446
- Find 489
- Flex 447
- Footfall Response Data 470
- Further Options 432
- Further Options (output) 451
- Generalised Restraint Definition 366
- Generate 2D Elements (legacy option) 473
- Generate Static Vehicle Load 464
- Go To 491
- Graphic Fonts and Styles 435
- Graphic Settings Wizard 417
- Gravity Load Definition 377
- Grid Area Load Definition 376
- Grid Layout Definition 333
- Grid Line Definition 333
- Grid Line Load Definition 376
- Grid Node Definition 339
- Grid Plane Definition 331
- Grid Point Load Definition 376
- GsRaft Analysis Progress 413
- GsRaft Damping Coefficient 414
- GsRaft Damping Coefficient and Soil Stiffness 413
- GsRaft Displacement Residual 414
- GsRelax Advanced Settings 407
- Gss Advanced Settings 407
- GWA Import Options 491
- History 500
- Joint Definition 367
- Labels and Display Methods 420
- Lighting Settings 138
- Line Definition 334
- Link Property Definition 357
- Load Case Titles 389
- Manage Data 470
- Manage User Modules 472
- Mass Definition 355
- Material Wizard 345
- Member Definition 343
- Member Restraint Dialog 360
- Modify 490
- Modify Axis 330
- Modify Curve 491
- Modify Element Offsets 443
- Modify Element Releases 443
- Modify Elements 442
- Modify Members 444
- Modify Nodes 437
- Move Areas 445
- Move Elements 445
- Move Lines 445
- Move Members 445
- Move Nodes 445
- Move Regions 445
- Multiple Sections Dialog 353
- NASTRAN Export Options 496
- New Model Wizard 313
- Node Definition 339
- Node Grid Settings Definition 340
- Node Load Definition 369
- Node Mesh Attribute Definition 341
- Node Stiffness Definition 340
- Non-linear Spring Curve 355
- Numeric Format 499
- Open 107
- OpenSees Export Options 496
- Optimise Path Loading 465
- Orientation Settings 434
- Output Settings Wizard 449
- Path Loading 389
- Polyline Definition 339
- Preferences 479
- Preferred Folders 487
- Preferred Section 487
- Preferred Units 486
- RC beam 497
- RC Member Bar Layouts Definition 365
- RC Member Design Property Definition 360
- RC Slab Design Property Definition 361
- Refine Quad Elements 440
- Region Definition 336
- Replace 490
- Response Spectrum Wizard 378
- Rigid Constraint Definition 366
- Saved Image Settings 486
- Section Modifiers Dialog 352
- Section Wizard 347
- Settlement Definition 369
- Spacer Definition 358
- Split 1D Elements 438
- Split Quad Elements 439
- Split Tri Elements 441
- Spring Property Wizard 353

- dialogs
 - Steel Beam Design Property Definition 358
 - Steel Restraint Property Definition 359
 - Storey Displacements and Forces 456, 463
 - Storey Masses 470
 - Tied Interface Definition 368
 - Titles 323
 - Tolerances 326
 - Translucency Settings 139
 - Unwrap Settings 139
 - Variable UDL Wizard 388
 - View List 414
 - Welcome to GSA 313
 - Wheel Patch Definition 467
 - Disconnect Elements command 169
 - Disconnect Elements Dialog 444
 - disconnecting elements graphically 169
 - Displacement Residual Dialog
 - GsRaft 414
 - displacements
 - applied displacement data 268
 - applied fundamentals 221
 - intermediate results 498
 - output options 516
 - Storey Displacements and Forces Dialog 456, 463
 - Display Excluded Entities Faint command 125
 - Display Favourites Toolbar 307
 - Display Methods Dialog 424
 - display methods in graphic views 133
 - displayed result
 - in Object Viewer 112
 - distance between nodes 111
 - distortion loads
 - data 271
 - Double size of diagrams command 135
 - drag nodes sculpt tool 157
 - Drag on node - Collapse command 157
 - Drag on node - Co-locate command 157
 - Drag on node - Ignore command 157
 - Drag on node - Joint command 157
 - Draw Grid command 127
 - Draw Selection command 125
 - drawing part of the model 125
 - drilling degrees of freedom 409
 - dummy elements
 - draw 424
 - grid loading 183
 - DXF
 - saving image to 143
 - software interaction via 573
 - DXF Export Options Dialog 495
 - DXF Import Options Dialog 495
 - dynamic analysis
 - effective mass 601
 - modal mass 601
 - modal stiffness 601
 - participation factor 601
 - scaling of modes 469
 - step by step 60
 - dynamic analysis results
 - output options 515
 - dynamic load
 - dynamic load factor 281
 - load curve 281
 - dynamic load factor
 - data 281
 - dynamic response
 - data 279
 - Dynamics
 - results 601
- E -**
- early binding 558
 - EC3 checker
 - buckling checks 708
 - buckling interaction check 708
 - combined local check 707
 - documentation 707
 - effective net area factor 708
 - input data requirements 707
 - limitations 711
 - local axial check 707
 - local bending check 707
 - local checks 707
 - local shear check 707
 - local torsion check 707
 - National Annexes 708
 - specialised input data 708
 - edge loads
 - data 274
 - specification 324
 - edges of elements

- edges of elements
 - highlighting in graphics 140
- Edit Design Property command 141
- Edit Element command 141
- Edit Material command 141
- Edit Member command 141
- Edit Node command 141
- Edit Property command 141
- Edit Restraint Property command 141
- Edit Section Property command 141
- Edit text File command 187
- editing
 - tables 113
- editing models in graphics 155
- effective lengths
 - steel members 97
- effective mass
 - by part 467
 - dynamic analysis 601
- eigenvalue 102
 - introduction to buckling analysis 64
- eigenvalue buckling analysis 64
 - modelling implications 64
- eigenvalue buckling results
 - introduction 65
- Element
 - 2D element release 343
 - Beam release 342
 - End stiffness 342
- element properties
 - Object Viewer 111
- Element Properties command 141
- element user modules 187
- Element User Modules command 472
- Element Wizard
 - Beam end stiffness 342
 - Beam releases 342
 - Releases 342
- elements
 - 2D fundamentals 201
 - 2D mesh generation 44
 - 2D shape 201
 - add/modify one by one sculpting 156
 - adding graphically introduction 161
 - adding string of 1D graphically 161
 - axis fundamentals 203
 - bar fundamentals 200
 - beam fundamentals 199
 - cable fundamentals 201
 - colour in graphics 137
 - Connect 1D Elements Dialog 437
 - connecting 1D graphically 162
 - convert Quad4 to Quad8 165
 - convert Tri3 to Tri6 165
 - copy / move graphically 165
 - Copy Elements Dialog 445
 - Create Members from Elements Dialog 459
 - creating from lines graphically 161
 - data 242
 - deleting blank data management 185
 - deleting by data management 470
 - deleting graphically 170
 - Disconnect Elements Dialog 444
 - disconnecting graphically 169
 - display methods 424
 - dummy 242
 - Extrude Dialog 446
 - extrude graphically 166
 - flipping graphically 168
 - fundamentals 199
 - Generate 2D Elements Dialog (legacy option) 473
 - grounded spring fundamentals 200
 - highlighting coincident in graphics 140
 - highlighting edges in graphics 140
 - joining 1D graphically 164
 - labels in graphic views 422
 - length of selection 111
 - link fundamentals 201, 218
 - mass fundamentals 200
 - mass of selection 111
 - Modify beam release 443
 - Modify Element Offsets Dialog 443
 - Modify Element Releases Dialog 443
 - Modify Elements Dialog 442
 - Modify end stiffness 443
 - modifying graphically 169
 - Move Elements Dialog 445
 - offset fundamentals 218
 - orientation node, convert to angle 173
 - quad fundamentals 202
 - Refine Quad Elements Dialog 440
 - renumbering 470
 - renumbering data management 185
 - selecting for graphic views 125

- elements
 - selecting for output 152
 - shrinking in graphics 137
 - spacer fundamentals 201
 - spinning 2D graphically 169
 - Split 1D Elements Dialog 438
 - Split Quad Elements Dialog 439
 - Split Tri Elements Dialog 441
 - splitting 1D and 2D graphically 162
 - spring fundamentals 200
 - strut fundamentals 200
 - surface area of selection 111
 - tie fundamentals 200
 - triangle fundamentals 202
 - wizard 341
- End stiffness
 - Beam element 342
 - Element Wizard 342
- Engineering Scale command 121
- engineering scale for structure
 - default switch 417
- Entities
 - check for completeness 462
 - selecting for loading 228
- envelope subject component
 - specifying 451
- enveloping
 - fundamentals 231
 - operator fundamentals 232
 - optimising fundamentals 232
 - permutations 232
 - stored step by step 91
 - syntax of combination descriptions 232
- enveloping in output views 152
- environmental impact
 - specification dialog 329
 - step by step 92
 - wizard 330
- Environmental Impact Specification Dialog 329
- Equivalent Static / Accidental Torsion Load dialog 468
- Equivalent static loads 761
- error messages
 - limit numbers 484
- error norm
 - output options 514
- Eurocode 26
 - Eurocode 8 - 1994 spectrum 382
 - Eurocode 8 - 2003 spectrum 381
- Evaluating expressions 188
- Expand Bridge Loading command 466
- Expand Bridge Loading Dialog 466
- Expand Gravity Loading command 462
- Expand Gravity Loading Dialog 462
- Expand Grid Loading command 462
- Expand Grid Loading Dialog 462
- explicit sections
 - fundamentals 212
- export
 - ADC 570
 - ANSYS ASAS 568
 - AutoCAD 568
 - CIMsteel 569
 - DXF Export Options Dialog 495
 - footfall response 571
 - GSA 6.x 565
 - GWA 564
 - input data 564
 - LS-DYNA 565
 - NASTRAN 567
 - NASTRAN Export Options Dialog 496
 - OpenSees 568
 - OpenSees Export Options Dialog 496
 - output views 571
 - Pdisp 570
 - Revit 574
 - SAP2000 566
 - steel members 571
- Export command 117
- Export Member Input Data to CSV Dialog 498
- export options 564
- exporting
 - from table views 115
- Expressions 188
 - Map beam loads on 1D elements 171
 - Map node loads on nodes 171
 - Map pressures on 2D elements 171
- extents of output
 - specifying 451
- Extract a sub-model 477
- extrude
 - nodes and elements graphically 166
- Extrude Dialog 446
- Extrude Selection command 166

Eye Distance In command 124
 Eye Distance Out command 124

- F -

fabric
 theory 656
 fabric analysis
 modelling implications 69
 fabric analysis and results 69
 fabric structures
 step by step 69
 Facade list 475
 face loads
 data 273
 facet angle of 2D elements
 change default 483
 features
 advanced fundamentals 192
 preferences 485
 FEMA
 FEMA 356 spectrum 382
 field types
 in tables 113
 file backups
 working with 186
 file format basics 107
 file locations
 change default 484
 find
 in table views 116
 Find command 116, 130
 Find Dialog 489
 find in Graphic Views 130
 Flex Dialog 447
 Flex Selection command 166
 flexing lines of nodes graphically 166
 Flip Elements command 168
 folders
 Preferred Folders Dialog 487
 font
 in graphic views 141
 in table views 117
 fonts
 Graphic Fonts and Styles Dialog 435
 footfall analysis
 chart view 454

 modelling 77
 results 77
 step by step 76
 theory 687
 footfall analysis parameters
 Analysis Wizard 403
 footfall response
 export options 571
 Footfall Response Data Dialog 470
 force density
 1D data 259
 2D data 259
 force density form-finding 71
 forces
 2D elements 636
 Cut Section Forces Dialog 457
 intermediate results 498
 Storey Displacements and Forces Dialog 456, 463
 Forces on 2D Element Cut command 456
 form finding options
 Analysis Wizard 398
 format
 Numeric Format Dialog 499
 format of numeric output
 change default 483
 formats
 numeric fundamentals 238
 formatting of numeric output 154
 form-finding
 by force density method 71
 by soap film method 70
 modelling implications of analysis 70
 property data 258
 structures step by step 70
 Freeze command 144
 front ordering 407
 fundamentals
 constraint equation 219
 tied interface 219
 Further Options Dialog 432
 Further Options Dialog (output) 451

- G -

Gateway
 copy and paste 110

- Gateway
 - working with 108
- Gateway command 108
- GB 50011-2001
 - GB 50011-2001 spectrum 383
- generalised restraints
 - data 263
 - definition dialog 366
 - fundamentals 217
- Generate
 - Orthogonal Frame 461
- Generate 2D Elements Dialog (legacy option) 473
- Generate data basics 108
- Generate Static Vehicle Load command 464
- Generate Static Vehicle Load Dialog 464
- Generate User Modules command 187
- generating an imperfect geometry 64
- generating models
 - 2D element meshes 44
 - introduction 44
- Generation Wizard
 - Data 314
- generic dynamic analysis parameters
 - Analysis Wizard 402
- Generic periodic load analysis
 - chart view 454
- geomentric sections
 - fundamentals 210
- geometric entities
 - adding graphically introduction 160
 - working with 184
- Geometric stiffness
 - pre-load 412
- geometric stiffness options 413
- geometry
 - Create New Model from Deformed Geometry 460
 - generating imperfect 64
- Getting started with GSA 34
- go to
 - in table views 116
- Go To command 114, 116
- Go to Design Property Data command 141
- Go To Dialog 491
- Go to Element Data command 141
- Go to Material Data command 141
- Go to Member Data command 141
- Go to Node Data command 141
- Go to Property Data command 141
- Go to Restraint Property Data command 141
- Go to Section Property Data command 141
- Graft model 186
- Graphic Display Toolbar 306
- Graphic Fonts and Styles Dialog 435
- Graphic Settings command 119
- graphic views
 - adjusting object point 123
 - adornments 132
 - advanced orientation 123
 - animation 142
 - annotating diagrams etc 136
 - axes 137
 - axes label settings 421
 - case list 135
 - colour in 137
 - colour overrides 435
 - contour barrel widths 435
 - contour settings 427
 - contour values 427
 - contours 134
 - current grid 127
 - cursor mode fundamentals 237
 - dark background 435
 - defining current grid 127
 - deformation settings 426
 - deformed image 135
 - diagram settings 430
 - diagrams 133
 - display method settings 424
 - display methods 133
 - double-clicking 119
 - drawing part of the model 125
 - element label settings 422
 - entity list 125
 - formatting of numbers 137
 - highlighting coincident elements 140
 - highlighting coincident nodes 140
 - highlighting edges of elements 140
 - image basics 120
 - labels 132
 - layer 141
 - lighting settings 138
 - node label settings 421
 - object point 123
 - Object Viewer 111

- graphic views
 - opacity 139
 - orientation about non-global axes 124
 - output options 142
 - polylines in 131
 - preferences 480
 - printing from 142
 - properties 111
 - rendering quality 488
 - report results in Object Viewer 112
 - resetting adornments 140
 - resetting entities 125
 - right click menus 141
 - save to file preferred format 484
 - scaling diagrams etc 135
 - scaling for printing 123
 - scaling the image and zooming 121
 - sculpting (editing) models in 155
 - sculpting preferences 481
 - selection sets 128
 - settings 119
 - Settings Wizard 417
 - shrinking elements in 137
 - storey label settings 421
 - surface shading 138
 - symbol sizes 435
 - threading 488
 - translucency settings 139
 - units 137
 - unreliable data scaling 121
 - unwrap settings 139
 - wizard 119
 - working with 118
- gravity
 - data 279
 - load fundamentals 228
- gravity loads
 - create dialog 377
 - definition dialog 377
- grid
 - in graphic views 127
 - in table views 117
- grid area loads
 - data 278
 - definition dialog 376
- grid axes
 - fundamentals 197
- Grid command 150
- grid coordinates
 - fundamentals 197
- grid layout
 - fundamentals 198
- grid layouts
 - definition dialog 333
 - disabling 331
- grid line loads
 - data 277
 - definition dialog 376
- grid lines
 - definition dialog 333
- Grid Loading 763
 - create dialog 376
 - create graphically 172
 - data 276
 - dummy elements 183
 - fundamentals 227
 - graphical display 184
 - re-entrant panels 183
 - working with 182
- grid nodes
 - data 241
 - definition dialog 339
 - settings definition dialog 340
- Grid output view
 - opening 150
- grid planes 183
 - creating graphically 159
 - data 247
 - definition dialog 331
 - fundamentals 198
 - graphical display 184
 - grid load expansion algorithms 198
 - simplified tributary areas 198
 - storeys 247
 - tolerance data 326
 - working with 182
- grid point loads
 - data 276
 - definition dialog 376
- grid points
 - definition 333
 - disabling 331
- Grid Window command 150
- grids
 - Expand Grid Loading Dialog 462

- Ground View command 120
 - grounded springs
 - axis fundamentals 204
 - fundamentals 200
 - grounded springs v. spring supports
 - fundamentals 213
 - group
 - output by 153
 - GSA
 - basic concepts 106
 - where do I start? 34
 - working with the program 106
 - GSA 6.x
 - export options 565
 - GSA Dialog
 - Welcome to 313
 - GSA Toolbar 303
 - GsaElement 537, 555
 - GsaEntity 537, 555
 - GsaNode 537, 555
 - GsaResults 555
 - GsaSection 537, 555
 - GsBridge control
 - Analysis Wizard 405
 - GsRaft Analysis Progress Dialog 413
 - GsRaft control
 - Analysis Wizard 404
 - GsRaft Damping Coefficient and Soil Stiffness Dialog 413
 - GsRaft Damping Coefficient Dialog 414
 - GsRaft Displacement Residual Dialog 414
 - GsRaft progress
 - Analysis Wizard 405
 - GsRelax Advanced Settings Dialog 407
 - GsRelax analysis progress information
 - Analysis Wizard 399
 - GsRelax analysis task control
 - Analysis Wizard 398
 - GsRelax member identification
 - Analysis Wizard 399
 - GsRelax solver
 - theory 648
 - GsRevit 574
 - Gss Advanced Settings Dialog 407
 - GWA
 - append data in Gateway 109
 - copy/paste in Gateway 109
 - cut/copy/paste in table views 115
 - graft data in Gateway 109
 - numeric precision 483
 - GWA file
 - export options 564
 - import options 572
 - string IDs (sID) 776
 - syntax 776
 - units 776
 - GWA Import Options Dialog 491
 - GWA, TXT, CSV 564, 572
 - GwaCommand units 776
 - GWC command file
 - run by 559
- ## - H -
- Halve size of diagrams command 135
 - harmonic analysis
 - chart view 453
 - modelling 74
 - results 74
 - step by step 73
 - theory 683
 - harmonic analysis parameters
 - Analysis Wizard 400
 - Hide Areas command 125
 - Hide Contours command 134
 - Hide Diagrams command 133
 - Hide Elements command 125
 - Hide Labels command 132
 - Hide Lines command 125
 - Hide Members command 125
 - Hide Regions command 125
 - Highlight Coincident Elements command 140
 - Highlight Coincident Nodes command 140
 - Highlight Edges command 140
 - history
 - Dialog 500
 - functionality 500
 - prompt for history note when saving 484
 - HKSUOS checker
 - Buckling checks 714
 - documentation 713
 - Effective Section Properties 713
 - input data requirements 713
 - Local checks 714

Hong Kong 26
 Hourglassing 761
 How do I get started? 34
 HTML
 batch saving saved views to 178

- I -

IBC
 IBC 2000 spectrum 383
 IBC 2006 (ASCE 7-05) spectrum 380
 IBC 2009 (ASCE 7-05) spectrum 380
 ignore Iyz
 section fundamentals 205
 ill conditioned 594, 596
 ill conditioning 102, 594, 596
 ill-conditioning 102
 image
 orientation of 120
 imperfect geometry
 generating 64
 import
 AutoCAD 572
 CIMsteel 573
 CSV 572
 DXF Import Options Dialog 495
 GWA 572
 GWA Import Options Dialog 491
 Pdisp 573
 Revit 574
 TXT 572
 Import GWA data 186
 import options 571
 Increase Precision command
 in graphic views 137
 in output views 154
 Indian Standards
 IS 1893 - 2002 spectrum 384
 individual member buckling
 theory 665
 Inertia details 603
 initial pre-load 412
 input data
 export options 564
 Input data only, open 107
 input preferences 482
 Insert command 116

insert in table views 116
 interesting points on beams 518
 intermediate beam forces
 output options 518
 intermediate displacements and forces 498
 introduction to analysis 54
 introduction to constructing a model 43
 introduction to generating models 44
 introduction to sculpting models graphically 48
 Introduction to GSA 34
 Invert Selection command 128, 136
 IRS 26
 IS 26
 IS 1893
 IS 1893 - 2002 spectrum 384
 IS800 checker
 documentation 717
 Isometric command 120
 Italian
 Ordinanza PCM 3274 spectrum 385

- J -

Jacobi iteration
 convergence 408
 Join String of 1D Elements command 164
 Join String of Lines command 164
 joining strings of 1D elements graphically 164
 joining strings of lines graphically 164
 Joints 598
 adding by disconnecting elements graphically 169
 creating graphically 170
 data 265
 definition dialog 367
 fundamentals 219
 Jp 413
 JPEG
 batch saving saved views to 178
 resolution 143
 saving image to 143
 software interaction via 573

- K -

keyboard accelerators 308
 keyboard shortcuts 308

keystrokes in graphics
 changing sensitivity 480
 Ky 753
 Kz 753

- L -

labels
 apply to selection 136
 Labels and Display Methods command 132, 133
 Labels and Display Methods Dialog 420
 Labels in graphic views 132
 late binding 558
 layer basics 88
 layout
 grid fundamentals 198
 legend panel in graphics
 changing 417
 length of element selection 111
 lighting settings
 in graphics 138
 Lighting Settings command 138
 linear 2D element analysis
 step by step 56
 linear equations 596
 linear static 596
 linear static analysis
 step by step 56
 Linear time history analysis
 chart view 454
 modelling 75
 linear time history analysis parameters
 Analysis Wizard 401
 lines
 connecting graphically 162
 copy / move graphically 165
 Copy Lines Dialog 445
 Create Regions per Grid Plane Dialog 459
 creating from 1D elements graphically 160
 definition dialog 334
 joining graphically 164
 labels in graphic views 423
 Move Lines Dialog 445
 rationalize lines 461
 working with 184
 links
 axis fundamentals 205

definition dialog 357
 fundamentals 201, 218
 property data 256

Lists

check for completeness 462
 data 299
 effective mass 467
 embedded fundamentals 214
 syntax fundamentals 214

Lists Toolbar 304

load cases

data 297
 deleting by data management 470
 fundamentals 229
 renumbering 470
 titles dialog 389

load curve

data 281

load panels

2D element fundamentals 226
 2D Property Wizard 357
 create on surface 475
 draw edges 424

loading

2D Element Edge Load Definition dialog 374
 2D Element Face Load Definition dialog 373
 2D Element Pre-stress Load Definition dialog 375
 2D Element Thermal Load Definition dialog 375
 Applied Displacement Definition dialog 369
 Beam Distortion Definition dialog 371
 Beam Load Definition dialog 370
 Beam Pre-stress Definition dialog 371
 Beam Thermal Load Definition dialog 372
 create 2D element edge load dialog 374
 create 2D element face load dialog 373
 create 2D element pre-stress load dialog 375
 create 2D element thermal load dialog 375
 create applied displacement dialog 369
 create beam distortion load dialog 371
 create beam load dialog 370
 create beam pre-stress load dialog 371
 create beam thermal load dialog 372
 create element load graphically 171
 create gravity load dialog 377
 create grid load graphically 172
 create grid loading dialog 376
 create nodal load graphically 171

loading

- create node load dialog 369
- create settlement dialog 369
- delete from Gateway 109
- Gravity Load Definition dialog 377
- grid Definition dialog 376
- map loading graphically 171
- Node Load Definition dialog 369
- Settlement Definition dialog 369
- working with grid 182

loads

- 2D element fundamentals 225
- 2D element loading data 272
- applied displacement data 268
- beam fundamentals 222
- beam load data 269
- beam loading data 269
- beam pre-stress load data 270
- case basics 108
- Create Mass from Loads Dialog 476
- deleting data management 185
- deleting graphically 173
- distortion load data 271
- edge load data 274
- Expand Bridge Loading Dialog 466
- Expand Gravity Loading Dialog 462
- Expand Grid Loading Dialog 462
- face load data 273
- Generate Static Vehicle Load Dialog 464
- gravity data 279
- gravity fundamentals 228
- grid area loads data 278
- grid line loads data 277
- grid loading data 276
- grid loading fundamentals 227
- grid point loads data 276
- names in tables, preference 482
- nodal loading data 267
- node load data 267
- node load fundamentals 221
- Optimise Path Loading Dialog 465
- pre-stress 2D load data 274
- settlement data 268
- thermal 2D load data 275
- thermal beam load data 272

local axes

- constraint axes fundamentals 198
- element fundamentals 203

long term analysis 601

LS-DYNA

export options 565

lumped mass 608

- M -

Manage Data command 470

Manage Data Dialog 470

Manage Tasks command 179

Manage User Modules command 472

Manage User Modules Dialog 472

manual backup

working with 186

Manual Backup command 186

Map Beam Loads on 1D Elements command 171

Map beam loads on beam elements

Expressions 188

Map Face Loads on 2D Elements command 171

Map node loads on nodes

Expressions 188

Map Node Loads on Nodes command 171

Map pressures on 2D elements

Expressions 188

mapping file 586

margin around image

changing 480

mark for analysis

working with 178

Mark Task For Analysis command 179

marking views and view lists for printing and saving to file 178

Mass 236

create from grid loads 477

Create Mass from Loads Dialog 476

results 601

mass calculation

Analysis Wizard 397

Mass details 603

mass of element selection 111

masses

axis fundamentals 204

definition dialog 355

fundamentals 200

property data 255

Storeys 470

material properties

- material properties
 - data 248
- materials
 - standard data 248
 - user-defined data 248
 - wizard 345
- max combination case operator 232
- measure distance between nodes 111
- member properties
 - Object Viewer 111
- Member Properties command 141
- Member Restraints 708
 - Member Restraint Dialog 360
 - steel restraint description 700
- member user modules 187
- Member User Modules command 472
- members
 - concrete properties 344
 - copy / move graphically 165
 - Copy Members Dialog 445
 - Create Members from Elements Dialog 459
 - data 245
 - definition dialog 343
 - display methods 424
 - labels in graphic views 422
 - length of selection 111
 - mass of selection 111
 - Modify Members Dialog 444
 - Move Members Dialog 445
 - offsets 344
 - orientation node, convert to angle 173
 - releases 344
 - selecting for graphic views 125
 - selecting for output 152
 - steel design step by step 93
 - steel properties 344
 - straightness tolerance 326
 - surface area of selection 111
- mesh generation
 - 2D elements 44
 - modelling regions 46
 - tied interfaces in 334
 - tips on modelling 47
- messages in report
 - limit numbers 484
- metafiles
 - saving image to 143
- methods
 - Display Methods Dialog 424
- min combination case operator 232
- Mindlin 409, 612
- Miscellaneous
 - deleting results from files 187
 - edit text file 187
 - file backups 186
 - unlocking files 186
- MITC 409, 612, 616
- Modal analysis
 - normalisation of modal results 408
 - Sturm check 408
- Modal analysis details
 - chart view 453
- modal analysis results
 - output options 515
- modal damping
 - CEAP Modal Damping Dialog 467
 - dynamic analysis 601
- modal dynamic analysis 60
 - modelling implications 61
 - theory 592
- modal dynamic parameters
 - Analysis Wizard 395
- modal dynamic results
 - introduction 62
- modal dynamics
 - introduction to analysis 61
- modal mass
 - dynamic analysis 601
- modal P-delta analysis
 - step by step 62
- modal P-delta dynamic analysis
 - modelling implications 62
 - non-linear elements 62
- modal p-delta results
 - introduction 62
- modal stiffness
 - dynamic analysis 601
- model
 - Create a New Raft Model Dialog 464
 - units fundamentals 194
- Model comparison 186
- model construction
 - introduction 43
- model data generation basics 108

- model stability 102
 - model stability analysis 594
 - Model Wizard
 - New 313
 - modelling
 - 2D elements 56
 - footfall analysis 77
 - for analysis (construction) stages 88
 - for bridge analysis 86
 - for eigenvalue buckling analysis 64
 - for fabric analysis 69
 - for form-finding analysis 70
 - for non-linear buckling analysis 66
 - for non-linear static analysis 68
 - for P-delta analysis 59
 - for RC slab reinforcement design 98
 - for seismic analysis 72
 - for steel design 93
 - harmonic analysis 74
 - linear time history analysis 75
 - modal dynamic analysis 61
 - modal dynamic P-delta analysis 62
 - periodic load excitation analysis 75
 - wall elements 56
 - models
 - Create New Model from Deformed Geometry Dialog 460
 - generating 2D element meshes 44
 - generating introduction 44
 - sculpting graphically introduction 48
 - modify
 - in table views 116
 - Modify Colour command 118
 - Modify Colour of Selection command 137
 - Modify Colour of This command 137
 - Modify command 116
 - Modify Curve Dialog 491
 - Modify Dialog 490
 - Modify Element Offsets Dialog 443
 - Modify Element Releases Dialog 443
 - Modify Elements command 169
 - Modify Elements Dialog 442
 - modify elements sculpt tool 158
 - Modify end stiffness
 - Beam element 443
 - Modify Linear 2D to Parabolic command 165
 - Modify Members Dialog 444
 - Modify Nodes command 159
 - Modify Nodes Dialog 437
 - modify nodes sculpt tool 157
 - Modify release
 - Beam element 443
 - moments
 - 2D elements 636
 - mouse sensitivity preferences
 - changing 480
 - move
 - Move Areas Dialog 445
 - Move Elements Dialog 445
 - Move Lines Dialog 445
 - Move Members Dialog 445
 - Move Nodes Dialog 445
 - Move Regions Dialog 445
 - nodes by expression graphically 166
 - nodes, elements, members, lines, areas, regions graphically 165
 - Move Selection command 165
 - multi-page table views 112
- ## - N -
- names
 - reporting data references by name 451
 - reporting entity names in results output 451
 - naming convention for sections
 - fundamentals 207
 - NASTRAN
 - export options 567
 - NASTRAN Export Options Dialog 496
 - New Analysis Task command 179
 - New command 108
 - New Graphic View (main thread) command 118
 - New Graphic View command 118
 - New Model basics 108
 - New Model Wizard 313
 - new models
 - Create New Model from Deformed Geometry Dialog 460
 - New Output View command 149, 154
 - New users introduction 34
 - Next Case command 304
 - Next Display command 304
 - nodal user modules 187
 - Nodal User Modules command 472

- node loads
 - create dialog 369
 - definition dialog 369
 - fundamentals 221
 - node properties
 - Object Viewer 111
 - Node Properties command 141
 - NodeConnectedEnt 539, 545
 - nodes
 - add more than one graphically 159
 - add/modify/drag one by one sculpting 156
 - adding graphically 159
 - collapsing coincident graphically 160
 - coordinates, non-global definition fundamentals 197
 - copy / move graphically 165
 - Copy Nodes Dialog 445
 - data 241
 - definition dialog 339
 - deleting blank data management 185
 - deleting by data management 470
 - deleting graphically 170
 - draw shrunk 424
 - Extrude Dialog 446
 - extrude graphically 166
 - Flex Dialog 447
 - flexing lines graphically 166
 - highlighting coincident in graphics 140
 - labels in graphic views 421
 - load data 267
 - loading data 267
 - Modify Nodes Dialog 437
 - modifying attributes graphically 159
 - Move Nodes Dialog 445
 - Node Grid Settings Definition dialog 340
 - Node Mesh Attribute Definition dialog 341
 - Node Stiffness Definition dialog 340
 - renumbering 470
 - renumbering data management 185
 - rounding coordinates graphically 160
 - selecting for graphic views 125
 - selecting for output 152
 - shrink 424
 - straightening lines graphically 168
 - transform graphically 166
 - non-linear
 - combination fundamentals 231
 - non-linear analysis
 - chart view 453
 - section data in 251
 - step by step 66
 - non-linear buckling analysis 65
 - modelling implications 66
 - non-linear buckling results
 - introduction 66
 - non-linear convergence 411
 - non-linear elements
 - modal P-delta dynamic analysis 62
 - non-linear springs
 - curves data 254
 - non-linear static
 - introduction to analysis 68
 - non-linear static analysis 67, 68
 - modelling implications 68
 - non-linear static options
 - Analysis Wizard 397
 - non-linear static results
 - introduction 67
 - non-stiff degrees of freedom 409
 - Normalisation of modal results 408
 - numeric format
 - in preferred and saved views 177
 - Numeric Format command
 - in graphic views 137
 - in output views 154
 - Numeric Format Dialog 499
 - numeric formats
 - fundamentals 238
 - numeric output format
 - change default 483
- O -
- Object Viewer
 - working with 110
 - Object Viewer command 110
 - offsets
 - draw marks 424
 - element fundamentals 218
 - Member Wizard: Releases and Offsets 344
 - members 344
 - Modify Element Offsets Dialog 443
 - Omit From Display cursor mode 125
 - opacity
 - in graphics 139

- Open command 107
- open command file (GWC) 559
- OpenSees
 - export options 568
- OpenSees Export Options Dialog 496
- operations
 - in table views 114
- operators
 - abs 232
 - combination case descriptions 232
 - in table views 114
 - max 232
 - min 232
 - sign 232
 - signabs 232
 - srss 232
- Optimise Path Loading command 465
- Optimise Path Loading Dialog 465
- optimising enveloping
 - fundamentals 232
- options
 - Bridge Options Dialog 432
 - DXF Export Options Dialog 495
 - DXF Import Options Dialog 495
 - Further Options Dialog 432
 - GWA Import Options Dialog 491
 - NASTRAN Export Options Dialog 496
 - OpenSees Export Options Dialog 496
- Orient About Grid Axes command 124
- orientation
 - element fundamentals 203
- orientation angle
 - beam axis fundamentals 203
- orientation axes 124
- orientation node
 - beam axis fundamentals 203
- orientation of graphic image
 - adjusting object point 123
 - advanced 123
 - non-global axes 124
 - object point 123
- Orientation Settings Dialog 434
- Orientation Toolbar 305
- Orthogonal Frame
 - Generate 461
- output
 - case and entity lists 152
 - change default numeric format 483
 - enveloping 152
 - grid view 150
 - of data on a selection set 152
 - of the graphic image 142
 - tabbed tables 150
- Output command 154
- output options
 - 2D element derived force results 527
 - 2D element derived stress results 526
 - 2D element displacements 525
 - 2D element stresses 525
 - beam cross strain energy density 523
 - beam derived stresses 519
 - beam displacements 517
 - beam force sign convention 518
 - beam forces 518
 - beam strain energy density 523
 - beam stresses 519
 - buckling analysis results 515
 - cross strain energy density 523
 - displacements 516
 - dynamic analysis results 515
 - dynamic error norm 514
 - error norm 514
 - intermediate beam forces 518
 - modal analysis results 515
 - modal error norm 514
 - options 502
 - reactions 516
 - SED 523
 - static analysis results 515
 - static error norm 514
 - strain energy density 523
 - stress averaging 530
 - torce lines 524
- output preferences 483
- Output Settings command 150
- Output view
 - change spacing 150
- output views
 - axes 154
 - by case/property/group 153
 - data extents 153
 - export options 571
 - formatting of numbers 154
 - from Gateway 109
 - printing from views 154

output views
 save to file preferred format 484
 selecting what to output 152
 settings 150
 Settings Wizard 449
 summary 153
 threading 488
 to spreadsheets 154
 totals 153
 units 153
 wizard 150
 working with 149
 overturning moment
 response spectrum 678

- P -

Page Setup command 142
 output 154
 painter 119
 Pan Down command 122
 Pan Left command 122
 Pan Right command 122
 Pan Up command 122
 panels
 draw 424
 graphical display 184
 panning the graphic image 121
 parallel direct solver
 GSS theory 634
 parallel solver
 GSS theory 634
 participation factor
 dynamic analysis 601
 paste
 from Gateway 110
 Paste Append command 491
 Paste command
 in Graphic Views 130
 Paste Special command 491
 pasting
 in table views 115
 Path Loading dialog 389
 paths
 data 284
 Optimise Path Loading Dialog 465
 patterned load
 Create Load Panels on Surface dialog 475
 Create Patterned Load Combinations dialog 476
 Create Patterned Loads dialog 475
 fundamentals 228
 technical note 773
 Pause command 144
 P-delta analysis
 modelling implications 59
 step by step 59
 P-delta introduction to analysis 59
 P-delta modal dynamic analysis
 step by step 62
 P-delta results
 introduction 60
 P-delta Ritz analysis
 step by step 63
 PDF
 3D capture of graphic image 145
 Pdisp
 export options 570
 import options 573
 periodic load analysis
 theory 685
 periodic load excitation analysis
 modelling 75
 results 76
 step by step 75
 permutations
 fundamentals 232
 Pick Colour of This command 137
 pick tolerance
 changing 481
 piled-raft
 introduction to analysis 80
 piled-raft analysis
 step by step 78
 Pin Display option 304
 select grid plane to change displayed elements 127
 Plan command 120
 PNG
 batch saving saved views to 178
 resolution 143
 saving image to 143
 software interaction via 573
 Polyline Properties command 131

- polylines
 - check for validity 339
 - data 247
 - definition dialog 339
 - in Graphic Views 131
 - length 131
 - selecting within 128
- post analysis status 54
- post-processing
 - assemblies 237
- precision 102, 594
- preferences
 - basic concept 106
 - user fundamentals 191
- Preferences command 106
- Preferences Dialog 479
- Preferred Folders Dialog 487
- Preferred Section Dialog 487
- preferred units
 - fundamentals 193
- Preferred Units Dialog 486
- preferred views
 - batch printing 178
 - batch saving to file 178
 - details 176
 - introduction 175
 - units and numeric format 177
- pre-load 412
- pre-stress
 - 2D elements 225
 - beam elements 222
- pre-stress 2D loads
 - data 274
- pre-stress beam loads
 - data 270
- Previous Case command 304
- Previous Display command 304
- principal axes
 - section fundamentals 205
- print
 - Object Viewer 111
 - page setup 484
- print by case
 - changing 417
 - image file 143
 - printing 142
- Print command 142
- output 154
- Print Preview command 142
 - output 154
- printing
 - views and view lists remotely 178
- printing from graphic views 142
- printing from output views 154
- printing view lists 177
- programming interface 533
- Progress Dialog
 - GsRaft Analysis 413, 414
- projected axes
 - fundamentals 197
- properties
 - 2D element property data 255
 - 2D Property Wizard 355
 - analysis stage property data 297
 - beam section data 251
 - cable property data 257
 - concrete beam design property data 261
 - concrete slab design property data 262
 - data 253
 - design property data 260
 - force density 1D property data 259
 - force density 2D property data 259
 - form-finding property data 258
 - link property data 256
 - mass property data 255
 - material data 248
 - Non-linear Spring Curve 355
 - non-linear spring curves data 254
 - Object Viewer 111
 - output by 153
 - RC beam design property data 261
 - RC member bar layout 263
 - RC Member Bar Layouts Definition dialog 365
 - RC Member Design Property Definition dialog 360
 - RC slab design property data 262
 - RC Slab Design Property Definition dialog 361
 - restraint property data 261
 - soap film 1D property data 259
 - soap film 2D property data 259
 - spacer property data 257
 - spring curves data 254
 - spring matrices data 254
 - spring property data 253
 - Spring Property Wizard 353

properties
 steel beam design property data 260
 Steel Beam Design Property Definition dialog 358
 steel restraint property data 261
 Steel Restraint Property Definition dialog 359
 property tables
 changing number of points 482
 Pseudo response spectrum analysis
 dynamic details 400
 Pseudo response spectrum parameters
 Analysis Wizard 400

- Q -

quads
 axis fundamentals 205
 fundamentals 202
 property data 255
 Refine Quad Elements Dialog 440
 Split Quad Elements Dialog 439
 Quick Delete command 170

- R -

raft
 Create a New Raft Model Dialog 464
 introduction to analysis 79
 raft analysis
 step by step 78
 raft analysis data requirements 78
 raft analysis notes 82
 raft data 282
 raft results
 introduction 82
 raft solution method 79
 Randomized load vector
 Ritz analysis 409
 rationalise sections 585
 Rationalize Lines command 461
 RC beam
 dialog 497
 export 497
 RC beam design 497
 RC beam design properties
 data 261
 RC design

 patterned load 773
 RC member bar layout
 data 263
 RC Member Bar Layouts Definition dialog 365
 beams 365
 circular/elliptical columns 365
 definition 365
 perimeter 366
 rectangular columns 365
 RC Member Design Property Definition dialog 360
 RC slab
 response cases 72
 RC slab design properties
 data 262
 RC Slab Design Property Definition dialog 361
 analysis 364
 basic properties 361
 concrete 362
 reinforcement 363
 RC slab reinforcement design
 modelling implications 98
 step by step 98
 RC slab reinforcement design results
 introduction 98
 negative areas 98
 reactions
 error norm 516
 output options 516
 Record command 144
 Recorder Toolbar 308
 recording a moving graphic image 144
 redo
 summary 173
 Redo command 173
 Redo View command 174
 Re-entrant panels
 grid loading 183
 Refine Mesh at Nodes command 163
 Refine Quad Elements command 163
 Refine Quad Elements Dialog 440
 Regenerate command 121
 regions
 copy / move graphically 165
 Copy Regions Dialog 445
 Create Regions per Grid Plane Dialog 459
 definition dialog 336
 labels in graphic views 423

- regions
 - modelling for mesh generation 46
 - Move Regions Dialog 445
 - tips on modelling for mesh generation 47
 - working with 184
- Release
 - 2D element 343
 - Beam element 342
 - Element Wizard 342
- releases
 - Member Wizard: Releases and Offsets 344
 - members 344
 - Modify Element Releases Dialog 443
- renumbering elements
 - data management 185
- renumbering nodes
 - data management 185
- renumbering nodes and elements 470
- repeated freedoms
 - data 265
- replace
 - in table views 116
- Replace command 116
- Replace Dialog 490
- report
 - in Object Viewer 112
- Requesting analysis
 - introduction 54
- Rescale Data command 135
- Reset Display Adornments command 140
- Reset Pan command 122
- Reset Preferences command 109
- Reset To All Entities command 140, 152, 304
- resetting graphics display adornments 140
- resetting graphics entities 125
- Residual Dialog
 - GsRaft Displacement 414
- response
 - dynamic response data 279
 - Footfall Response Data Dialog 470
- response spectrum
 - analysis 678
 - ASCE 7-05 380
 - base shear 678
 - Eurocode 8 - 1994 382
 - Eurocode 8 - 2003 381
 - FEMA 356 382
 - GB 50011-2001 383
 - IBC 2000 383
 - IBC 2006 380
 - IBC 2009 380
 - IS 1893 - 2002 384
 - Ordinanza PCM 3274 385
 - overturning moment 678
 - type 379
 - UBC 1994 385
 - UBC 1997 386
 - user defined 387
 - wizard 379
- Response spectrum analysis
 - combination method 399
 - dynamic details 400
 - rigorous or fast 399
 - spectrum 399
 - step by step 72
- Response spectrum parameters
 - Analysis Wizard 399
- response spectrums
 - data 280
 - wizard 378
- Restore Manual Backup command 186
- Restore Timed Backup command 186
- restraint
 - steel restraint properties 702
 - steel restraint properties step by step 95
- restraint properties
 - data 261
 - Member Restraint Dialog 360
 - step by step 95
- restraint property
 - dialog 95
- restraints
 - fundamentals 217
 - generalised restraint data 263
 - steel member restraint description 700
- result
 - steel design results step by step 97
- Result selection 412
- results 545
 - change output view type 150
 - deleting 55
 - dynamics 601
 - footfall analysis 77
 - harmonic analysis 74
 - mass 601

- results 545
 - periodic load excitation analysis 76
 - steel design 702
 - time history analysis 75
 - Revit link
 - export from Revit to GSA 574
 - import to Revit from GSA 577
 - introduction 574
 - Model suitability for BIM Coordination 574, 576, 577, 585, 586
 - recommended practice 585
 - registration 586
 - update Revit from GSA 576
 - right click menu 141
 - right click menus in graphic views 141
 - Rigid constraints 598
 - data 264
 - definition dialog 366
 - fundamentals 218
 - Rigorous or fast
 - response spectrum analysis 399
 - Ritz Analysis
 - randomized load vector 409
 - Ritz analysis parameters
 - Analysis Wizard 396
 - Ritz dynamic analysis
 - step by step 63
 - Ritz p_delta results
 - introduction 63
 - Ritz P-delta analysis
 - step by step 63
 - Rotate About Selected Node command 123
 - Round Coordinates command 160
 - rounding coordinates graphically 160
 - run by command file (GWC) 559
- S -**
- SANS10162 checker
 - buckling checks 716
 - buckling interaction check 716
 - documentation 715
 - Effective Section Properties 715
 - input data requirements 715
 - limitations 716
 - local checks 715
 - SAP2000
 - export options 566
 - Save As Preferred View command 176
 - Save command 107
 - Save Default View Settings command 175
 - Save DXF command 143
 - Save Graphic View command 177
 - Save JPEG command 143
 - Save Output View command 177
 - Save PNG command 143
 - Save Selection As 2D Polyline command 132
 - Save Selection As List command 130
 - Save WMF command 143
 - Saved Image Settings Dialog 486
 - saved view
 - save to file preferred format 484
 - Saved Views
 - batch printing 178
 - batch saving to file 178
 - cut section force chart view 457
 - details 177
 - introduction 175
 - management 178
 - units and numeric format 177
 - saving the graphic image to file 143
 - saving to file
 - views and view lists remotely 178
 - scale
 - diagrams change scale on change of case switch 480
 - structure to engineering scale default switch 417
 - Scale to fit command 121
 - scaling
 - graphic diagrams etc 135
 - printed graphic image 123
 - the graphic image 121
 - Scaling of modal results 469
 - sculpt
 - adding elements introduction 161
 - adding geometric entities introduction 160
 - adding string of 1D elements 161
 - collapsing coincident nodes 160
 - connecting 1D elements 162
 - connecting lines 162
 - copy / move nodes, elements, members, lines, areas, regions 165
 - create element loading 171
 - create grid loading 172

sculpt

- create joints 170
- create nodal loading 171
- create rigid constraints 170
- creating 1D elements from lines 161
- creating lines from 1D elements 160
- data defaults in 156
- delete elements 170
- delete nodes 170
- deleting loading 173
- disconnecting elements 169
- extrude nodes and elements 166
- flex lines of nodes 166
- flipping elements 168
- geometry cursor modes 156
- grid planes 159
- joining lines 164
- joining strings of 1D elements 164
- map loading on elements 171
- map loading on nodes 171
- models graphically introduction 48
- models in graphics 155
- modify node attributes 159
- modifying elements 169
- nodes adding more than one 159
- nodes/elements one by one 156
- rounding nodal coordinates 160
- spinning 2D elements 169
- splitting 1D and 2D elements 162
- straightening lines of nodes 168
- transform nodes 166
- user axes 158

sculpt tool

- add elements 157
- add lines 158
- add nodes 156
- drag nodes 157
- modify elements 158
- modify nodes 157

Sculpt Toolbar 306

sculpting

- change default 481
- changing coincidence tolerance 481
- changing highlighting of selection 481
- changing pick tolerance 481
- changing reference point for selection 481
- changing straightness tolerance 481
- changing tolerances 481

preferences 481

section definition axes

- fundamentals 205
- wizard 347

Section Display command 133

section mapping, Revit

- mappings 586

sections

- axis fundamentals 205
- catalogue fundamentals 208
- Cut Section Forces Dialog 457
- data 251
- data in non-linear analysis 251
- database fundamentals 213
- definition axes 347
- explicit fundamentals 212
- fundamentals 205, 206
- geometric fundamentals 210
- Multiple Sections Dialog 353
- naming convention fundamentals 207
- Preferred Section Dialog 487
- Section Modifiers Dialog 352
- sorting into vertical and non-vertical 194, 205, 206, 207, 208, 210, 212, 213, 251, 347, 352, 353, 457, 487, 585
- standard fundamentals 208
- units fundamentals 194
- wizard 347

SED

- output options 523

Seismic

- Accidental torsion loads 761
- Equivalent static loads 761

Seismic analysis

- Equivalent Static / Accidental Torsion Load Dialog 468
- modelling implications 72
- scaling of modes 469
- step by step 71

Select 2D Polyline command 132

Select All command 115, 128, 136

Select By Criteria command 128

Select By List command 128

Select By Name command 128

Select Closest To Vertical 1D Elements command 128

Select command 115

Select Highest Coincident command 128

- Select List command 130
- Select None command 115, 128, 136
- Select Out-Of-Range command 128
- Select Overlapping command 128
- Select Polyline command 131
- Select String command
 - elements and members 128
 - for polyline points 131
- Selected results only, open 107
- selecting cells in table views 115
- selecting entities for loading 228
- selecting entities in sculpt
 - changing highlighting 481
 - changing references 481
 - copy expanded list 481
 - forming selection sets 128
 - selecting within polylines 128
- selection
 - Curve Data Selection Dialog 491
- selection properties
 - Object Viewer 111
- Selection Properties command 111
- Set Area Defaults as This command 156
- Set Current Grid to This
 - in graphic views 127
- Set Current Grid to This command 141
- Set Element Defaults as This command 156
- Set Line Defaults as This command 156
- Set Member Defaults as This command 156
- Set Node Defaults as This command 156
- Set Region Defaults as This command 156
- sets
 - selection in graphic views 128
- settings
 - Animation Settings Dialog 434
 - Contour Settings Dialog 427
 - Contour Values Dialog 427
 - Deformation Settings Dialog 426
 - Diagram Settings Dialog 430
 - Graphic view 119
 - lighting settings 138
 - Orientation Settings Dialog 434
 - Output Settings Wizard 449
 - Output view 150
 - Saved Image Settings Dialog 486
 - translucency settings 139
 - unwrap settings 139
- Settings Dialog
 - GsRelax Advanced 407
 - Gss Advanced 407
- settlements
 - create dialog 369
 - data 268
 - definition dialog 369
 - fundamentals 218
- shade surfaces
 - changing 417
- Shade Surfaces command 138
- shading surfaces
 - in graphics 138
- shape checks for 2D elements 760
- shear area 608, 753
- shear deformation 608, 753
- shear modulus
 - theory 657
- Shine Light From Here command 138
- shortcuts 308
- show in Graphic Views 130
- Shrink command 137
- shrinking elements
 - in graphics 137
- sID Manipulation 541
- sIDs 534
 - COM export functions 541
 - convention 238
 - format 238
 - fundamentals 238
 - GWA 776
- sign combination case operator 232
- signabs combination case operator 232
- simple static analysis
 - introduction to 54
- simplified tributary areas
 - grid plane 198
- single page table views 112
- size columns to fit
 - in table views 117
- size rows to fit
 - in table views 117
- Skew View command 120
- slab reinforcement design
 - modelling implications 98
 - step by step 98
- slab reinforcement design for slabs

- slab reinforcement design for slabs
 - step by step 98
- slab reinforcement design results
 - introduction 98
 - negative areas 98
- slabs
 - concrete slab design property data 262
- sliding joints
 - data 265
- Snap - Align with Grid Axis command, adding elements 157
- Snap - Align with Grid Axis command, adding lines 158
- Snap - Centre command, adding elements 157
- Snap - Centre command, adding lines 158
- Snap - Perpendicular command, adding elements 157
- Snap - Perpendicular command, adding lines 158
- Snap to Grid Points command 127
- soap film
 - 1D data 259
 - 2D data 259
- soap film form-finding 70
- soil
 - interaction data 282
 - zones and profiles display 424
- soil interaction
 - data 282
- Soil Stiffness Dialog
 - GsRaft Damping Coefficient and 413
- solid sections in graphic views 133
- solver options
 - Analysis Wizard 394
- solver preferences 482
- spacers
 - definition dialog 358
 - fundamentals 201
 - property data 257
 - theory 657
- Sparse Direct solver 407
 - GSS theory 634
- Sparse parallel direct solver
 - GSS theory 634
- specification
 - Analysis Specification Dialog 324
 - Bridge Loading Analysis Specification Dialog 328
 - Design Code Specification Dialog 327
 - Environmental Impact Specification Dialog 329
 - Output Settings Wizard 449
 - unit data 325
- spectrums
 - response spectrum data 280
- spherical axes fundamentals 196
- Spin 2D Elements command 169
- Split 1D Elements command 163
- Split 1D Elements Dialog 438
- Split Quad Elements command 163
- Split Quad Elements Dialog 439
- Split Quad to Tri elements command 163
- Split Tri Elements command 163
- Split Tri Elements Dialog 441
- Split Warped 2D Elements command 461
- splitting elements 162
- spreadsheets
 - copying to and from table views 117
 - data from output views 154
 - Fully Populate Fields output view option 154
 - Output All Data per Entity on a Single Row output view option 154
 - specifying output format for 451
 - use for constructing a model 48
- spring supports v. grounded springs
 - fundamentals 213
- springs
 - axis fundamentals 204
 - curves data 254
 - fundamentals 200
 - grounded fundamentals 200
 - matrices data 254
 - Non-linear Spring Curve Definition 355
 - property data 253
 - Spring Property Wizard 353
- srss combination case operator 232
- srss combination cases
 - fundamentals 232
- stability 102, 594
- stability analysis 594
- staged analysis
 - step by step 87
- stages
 - analysis stage data 296
 - definition data 296
 - property data 297
- standard materials

- standard materials
 - data 248
- standard sections
 - fundamentals 208
- Standard Toolbar 303
- starting to construct a model 43
- starting to generate a model 44
- starting to sculpt a model graphically 48
- Starting to use GSA 34
- startup
 - display empty graphic view 484
 - initial views 484
 - show welcome dialog 484
- static analysis 596
 - introduction to 54
 - non-linear 67, 68
 - step by step 56
 - theory 591
- static analysis cases
 - Analysis Wizard 394
- static analysis results
 - output options 515
- static p-delta
 - Analysis Wizard 395
- status
 - post analysis 54
- steel
 - Design Codes 26
 - member restraint description 700
 - restraint properties 702
- steel beam design properties
 - data 260
- Steel Beam Design Property Definition dialog 358
- steel design
 - modelling implications 93
 - restraint properties step by step 95
 - results step by step 97
 - step by step 93
 - tools step by step 97
- steel design results
 - effective lengths 97
 - introduction 97
- steel member
 - design results step by step 97
- steel member restraints
 - description 700
 - step by step 96
- steel members
 - export options 571
 - restraint description 700
 - restraint properties 702
 - section axes 702
- steel properties
 - Member Wizard: Steel Properties 344
- steel restraint properties
 - data 261
 - step by step 95
- steel restraint property
 - dialog 95
- Steel Restraint Property Definition dialog 359
- steel restraints
 - Member Restraint Dialog 360
- step by step
 - analysis (construction) stages 87
 - analysis envelopes 91
 - analysis summary 55
 - bridge analysis 83
 - buckling analysis 64
 - dynamic analysis 60
 - environmental impact 92
 - footfall analysis 76
 - harmonic analysis 73
 - linear 2D element analysis 56
 - linear static analysis 56
 - modal P-delta analysis 62
 - non-linear analysis 66
 - P-delta analysis 59
 - periodic load excitation analysis 75
 - piled-raft analysis 78
 - raft analysis 78
 - RC slab reinforcement design 98
 - response spectrum analysis 72
 - Ritz analysis 63
 - Ritz P-delta analysis 63
 - seismic analysis 71
 - steel design 93
 - Time history analysis 74
 - Wave loading 98
- Step by step guide
 - introduction 34
- Step files
 - use for constructing a model 48
- stick direction of walls 498
- stiffness matrix 102, 594
- Stiffness options

- Stiffness options
 - GSS 407
 - Stop 54, 142
 - Stop command 142, 144
 - stopping analysis 54
 - Storey Displacements and Forces command 456, 463
 - Storey Masses command 470
 - storey views
 - details 177
 - storeys
 - create grid planes 185
 - Displacements and Forces command 456, 463
 - Displacements and Forces Dialog 456, 463
 - grid planes 247
 - Storey Displacements and Forces Dialog 456, 463
 - Straighten Spacing Evenly command 168
 - Straighten Spacing Proportionally command 168
 - straightening lines of nodes graphically 168
 - straightness tolerance
 - changing 481
 - strain energy density
 - output options 523
 - stress averaging
 - output options 530
 - stress output
 - 2D derived stresses 526
 - 2D elements 525
 - beam derived stresses 519
 - beams 519
 - stresses
 - 2D elements 636
 - string IDs
 - COM export functions 534
 - convention 238
 - format 238
 - fundamentals 238
 - GWA 776
 - structure
 - default to engineering scale switch 417
 - Structure comparison 186
 - struts
 - fundamentals 200
 - Sturm check 408
 - styles
 - Graphic Fonts and Styles Dialog 435
 - in graphic views 141
 - subject component in output view envelopes 152
 - Sub-model 477
 - Subspace iteration
 - convergence 408
 - summary output 153
 - surface area of element selection 111
 - surface shading
 - changing 417
 - in graphics 138
 - sustainability 92
 - Switch Layer in graphic views 141
 - symbol sizes in graphic views 435
 - syntax
 - combination case descriptions 232
- T -**
- Tab Separate Tables command 150
 - table views
 - basic operations 114
 - cell operators 114
 - colour in 118
 - copying cutting and pasting in 115
 - copying to and from spreadsheets 117
 - data display in 117
 - defaults 114
 - delete blank and insert in 116
 - find replace go to and modify in 116
 - from Gateway 109
 - navigation and editing 114
 - Object Viewer 111
 - operators 114
 - properties 111
 - selecting cells in 115
 - single and multi-page 112
 - working with views 112
 - tabular data entry and editing 113
 - task view
 - working with 179
 - task view right-click menu
 - working with 179
 - tasks
 - Analysis Wizard 181
 - copy and paste 181
 - fundamentals 230
 - working with 178

- tasks (analysis) and cases
 - fundamentals 229
 - terminating an analysis 54
 - Text (GWA) file
 - syntax 776
 - theory
 - constraint equations 599
 - footfall analysis 687
 - harmonic analysis 683
 - joints 598
 - linear time history analysis 686
 - periodic load analysis 685
 - rigid constraints 598
 - thermal 2D loads
 - data 275
 - thermal beam loads
 - data 272
 - Tied interface
 - Theory 599
 - tied interfaces
 - data 266
 - definition dialog 368
 - fundamentals 219
 - in mesh generation 334
 - ties
 - fundamentals 200
 - time history analysis
 - results 75
 - step by step 74
 - theory 686
 - Time step control 412
 - timed backup
 - set interval 484
 - working with 186
 - Time-history analysis 412
 - Titles Dialog 323
 - tolerances
 - changing in sculpting 481
 - preferred model 482
 - Tolerances Dialog 326
 - toolbars 302
 - Assisted Input 302
 - Command 307
 - Cursor Mode 305
 - Data Options 304
 - Display Favourites 307
 - Graphic Display 306
 - GSA 303
 - Lists 304
 - Orientation 305
 - Recorder 308
 - Sculpt 306
 - Standard 303
 - Toolbars command 107
 - tools
 - create grid planes from storeys 185
 - create rigid membranes from storeys 186
 - create storey grid planes 185
 - Generate Orthogonal Frame 461
 - steel design step by step 97
 - torque lines
 - output options 524
 - torsion constant 350, 748
 - totals in output 153
 - touch gestures
 - generally 189
 - in Graphic Views 189
 - touch screen 189
 - Transform geometry
 - Expressions 188
 - Transform Geometry command 166
 - translucency
 - in graphics 139
 - Translucency Settings command 139
 - triangles
 - axis fundamentals 205
 - fundamentals 202
 - property data 255
 - Split Tri Elements Dialog 441
 - trimming 1D elements graphically 162
 - trimming lines graphically 162
 - TXT
 - batch saving saved views to 178
 - TXT file
 - export options 564
 - import options 572
- ## - U -
- UBC
 - UBC 1994 spectrum 385
 - UBC 1997 spectrum 386
 - undo
 - enable/disable undo, preference 488

- undo
 - number of undo steps, preference 482
 - summary 173
 - undo analyses, preference 482
 - undoing edits 173
 - undoing views 174
- Undo command 173
- Undo View command 174
- units
 - change default 482
 - for output 153
 - fundamentals 192
 - GWA 776
 - in model fundamentals 194
 - in preferred and saved views 177
 - in sections fundamentals 194
 - in table views 117
 - in views fundamentals 194
 - preferred fundamentals 193
 - Preferred Units Dialog 486
 - specification data 325
 - unit set fundamentals 193
- Units command
 - in graphic views 137
 - in output views 153
- Unlock File command 186
- unreliable data scaling 121
- unwrap mode
 - in graphics 139
- Unwrap Settings command 139
- user axes
 - Create User Axes Dialog 436
 - creating graphically 158
- user modules 187
- user preferences
 - fundamentals 191
- user-defined materials
 - data 248
- utilization
 - steel design results step by step 97

- V -

- variable UDLs
 - wizard 388
- VBA interface 533
- Vdisp data 282

- vehicles
 - data 285
 - Generate Static Vehicle Load Dialog 464
- video recording 144
- View List Dialog 414
- view lists
 - definition dialog 414
 - details 177
 - introduction 175
- view settings 175
- View settings painter
 - Graphic view 119
 - Output view 150
- views
 - auto view details 177
 - batch printing 178
 - batch saving to file 178
 - default view settings details 175
 - from Gateway 109
 - management 178
 - management of savedpreferred 178
 - preferred view details 176
 - saved and preferred 175
 - saved view details 177
 - storey view details 177
 - units fundamentals 194
- Visual Basic interface 533
- volume clipping 125

- W -

- wall elements
 - equivalent beam direction 498
 - limitations 58
 - modelling 56
 - results 528
 - stick direction for equivalent beam results 498
- Warp check contour option 760
- warped 2D elements
 - split 461
- Wave loading
 - step by step 98
- Weight 236
- Welcome to GSA Dialog 313
- Wheel Patch Definition command 467
- Wheel Patch Definition Dialog 467
- Where do I start? 34

- window placement shortcuts 308
- windows metafiles
 - saving image to 143, 573
- wizard 113
 - Graphic view 119
 - Output view 150
- Wizard command 119, 150
- wizards
 - 2D Property 355
 - Analysis Wizard 393
 - analysis wizard fundamentals 230
 - Data Generation 314
 - element 341
 - environmental impact 330
 - Graphic Settings 417
 - in tables 113
 - material 345
 - New Model 313
 - Output Settings 449
 - Response Spectrum 378
 - section 347
 - Spring Property 353
 - Variable UDL 388
- WMF
 - batch saving saved views to 178
 - saving image to 143
- working with GSA 106

- X -

- X Elevation command 120

- Y -

- Y Elevation command 120

- Z -

- Zoom In command 122
- Zoom Out command 122
- zooming the graphic image 121

Endnotes 2... (after index)

Back Cover