



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 1985–2020

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: January 2020

Table of Contents

Part I About GSA	15
1 Overview	15
2 GSA Analysis Features	15
3 GSA Design Features	16
4 GSA Program Features	17
5 Documentation	18
6 Design Codes	18
Steel	18
Concrete	20
7 Validation	22
8 Acknowledgements	22
Part II Getting Started	27
Part III Working with GSA	29
1 Dockable Panes	30
Explorer Panes	31
Output Panes	33
Graphics Palettes	34
Entity Panes	36
2 Table Views	38
Single and Multi-page Tables	38
Field Types	39
Cell Operators	39
Basic Operations in Tables	40
Cutting, Copying and Pasting in tables	41
Delete and Blank Records in Tables	41
Find, Replace, Go To and Modify in Tables	42
Copying To and From Spreadsheets	43
Adjusting Data Display	43
Colour In Tables	43
3 Output Views	44
Output Settings	44
Output View Table Format	45
Selecting Data to Output	46
Case and Entity Lists	46
Outputting for a Selection Set of Entities	46
Enveloping	47
Data Extents	47
Output Summary	47
Output By Case, By Property, By Group	47
Printing from Output Views	48
Interacting with Spreadsheets	48
4 Graphic Views	48

Changing the Content of a Graphics View	49
Basic Orientation of the Image	49
Scaling the Image and Zooming	50
Advanced orientation of the image	53
Identifying What Is To Be Draw n	53
Graphical Representation of Entities	55
Current Grid	56
Selection Sets	57
Polylines in Graphic View s	60
Adornments	61
Shrinking Elements	66
Colour In Graphic View s	67
Shading Surfaces	67
Translucency	68
Unw rap Graphics	68
Highlighting Element Edges	69
Highlight Coincident Nodes	70
Highlight Coincident Elements	70
Resetting the Display	70
Sw itch Layer	70
Right-click Menus	70
Graphic Fonts and Styles	71
Animation	71
Printing from Graphic View s	72
Cursor Modes in Graphic View s	72
Copying the Graphic Image to the Clipboard	74
5 Chart Views	74
Chart Menus	74
Chart Styles	75
6 Analysis	77
Elements	78
Analysis Types	83
Running Analyses	84
Summary	85
7 Sets and Lists	86
Lists and Embedded Lists	86
8 Axes, Grids and Grid Planes	89
Axis Sets	89
Use of Axis Sets	91
Projected Axes	92
Grid Axes and the Current Grid	92
Constraint Axes	93
Element and Member Axes	93
Grid Lines, Planes and Surfaces	93
9 Saved Views and Preferred Views	94
Default View Settings	94
Preferred View s	94
Saved View s	95
Auto View s	96
Units and Numeric Format	96
View Lists	96
Batch Printing and Saving of View s	97
10 Analysis Tasks and Cases	97

Task View	97
Task View Right-click Menu	98
Tasks, Cases and the Analysis Wizard	99
Copy and Paste Tasks and Cases	100
Cases and Tasks	101
11 Grid Planes and Grid Loading	108
Specifying Grid Planes	108
Specifying Grid Loading	108
Displaying Grid Planes and Grid Loading	108
12 Sections	109
Section Profile	110
Design Parameters	110
13 Geometric Entities and 2D Element Meshes	111
Modelling Regions for Mesh Generation	114
Modelling Tips	115
14 Constraints	115
Element Offsets	116
Link Elements and Rigid Constraints	116
Conflicting Constraints	117
Automatic Constraints	117
15 Miscellaneous	117
Unlock File	117
File Backups	117
Delete Results from Files	118
User Modules	118
View List	119
Evaluating Expressions	120
Touch Gestures	121
16 Toolbars and Keyboard Accelerators (shortcuts)	121
Toolbars	122
Keyboard Accelerators (shortcuts)	129
Mouse accelerators (shortcuts)	132
17 Patterned Load	133
18 Mass and Weight	133
19 String IDs	134
20 Members and Element Meshes	135
Part IV GSA Data	138
1 Titles	138
2 Specification	138
Units Specification	138
General Specification	141
Tolerances	142
Design Specification	143
Raft Analysis Specification	143
Bridge Loading Analysis Specification	144
Environmental Impact Specification	145
3 Geometry	145
Axes	145
Grid Lines	147

Grid Planes	148
Grid Surfaces	149
Polylines	150
Lines	151
Areas	152
Regions	152
4 Materials	154
Material Grades	155
Analysis Materials	161
Material Curves	164
Material Matrices	165
5 Properties : Section	165
Section Profile	167
Geometric Section Properties	178
Reference Point	180
Point Voids	180
Steel Design	180
Reinforcement	180
Section Modifiers	185
Section Design Pools	186
6 Properties : General	186
Spring Properties	186
Mass Properties	189
2D Element Properties	189
3D Element Properties	193
Link Properties	194
Cable Properties	194
Damper Properties	195
Spacer Properties	195
External Matrix	196
Environmental Impact Wizard	197
7 Properties : Form-finding	197
Force Density 1D	197
Soap Film 1D	198
Force Density 2D	198
Soap Film 2D	198
8 Properties : Concrete Slab	199
Reinforcement	199
Calculation	199
Strength Reduction Factors	200
9 Nodes	201
10 Members	204
Definition	204
Design Properties	206
Miscellaneous	207
1D Member Axes	207
Restraints at Points	209
Restraints at Spans	209
11 Blocks	209
12 Member End Conditions	210
Steel Member Restraint Conditions	210
13 Elements	212

Element Types	215
Element Axes	217
14 Assemblies	220
15 Constraints	222
Generalised Restraints	222
Rigid Constraints	223
Joints	224
Constraint Equations	225
Tied Interfaces	226
16 Loading	227
Load Case Specification	229
Nodal Loading	229
Beam Loading	232
2D Element Loading	238
3D Element Loading	244
Grid Loading	246
Gravity	250
17 Dynamic Response	251
Response Spectra	251
Frequency Damping	261
Mode Damping	261
Load Curve	262
Dynamic Load Factor	262
Frequency Weighting Curve	263
18 Raft	263
Soil	263
Raft Interaction	267
Pile Interaction	268
19 Bridge	268
Alignments	268
Paths	269
Vehicles	270
Bridge VUDL	271
Influence Effects	272
Path Loading	274
Bridge Loading	275
20 Analysis Stages	281
Stage Definition	282
Stage Properties	282
Stage Materials	283
21 Tasks and Cases	283
Analysis Case Definition	283
Analysis Wizard	283
Advanced Solver Settings	300
Soil-Structure Analysis Progress	308
Combination Cases	308
Design Wizard	309
22 General Data	310
Lists	310
Case Descriptions	311
Part V Analysis in GSA	313

1 Linear Static Analysis	313
P-delta Analysis	313
Linear 2D Element Analysis	314
2 Nonlinear Static Analysis	316
Modelling Implications	317
Ties and Struts	317
Dynamic Relaxation	318
Explicit Nonlinear Static Analysis	319
Analysis of Fabric Structures	319
Form-Finding Analysis	320
3 Dynamic Analysis	321
Modal Dynamic Analysis	321
Ritz Dynamic Analysis	323
Explicit Dynamic Analysis	324
Dynamic Response Analysis	325
4 Buckling Analysis	328
Eigenvalue Buckling Analysis	328
Nonlinear Buckling Analysis	330
5 Dynamic Relaxation Analysis	330
Nonlinear Analysis Options	331
Dynamic Relaxation Convergence & Damping	335
6 Model Stability Analysis	338
7 Seismic Analysis	339
Equivalent Static Procedures	340
Response Spectrum Analysis	341
8 Bridge Analysis	345
Modelling Implications	347
Delete Grid Loading Tool	348
Automatic Path Generation	348
Analysis	349
9 Raft & Piled-raft Analysis	350
Data Requirements	350
Solution Method	351
Raft Analysis Steps	352
Piled-raft Analysis Steps	353
Results	354
Notes	354
10 Analysis Stages	355
Modelling Implications	355
Analysis and Design Layers	356
Analysis of Stages	358
Results	358
Stages and Graphic Views	359
11 Analysis Envelopes	360
12 Environmental Impact	360
13 Wave loading	361
14 Using Assemblies in Analysis	364
15 LS-DYNA Analysis	365
16 RC Slab Reinforcement Design	365

17 Results	366
Static Analysis Results	366
Modal Analysis Results	367
Displacements	368
Reactions	368
Soil Contact Bearing Pressure	368
Beam Element Results	369
2D Element Results	376
3D Element Results	382
18 Displaying Data and Results	383
Model Data Display Options	383
Load Data Display Options	386
Results Display Options	388
Bridge Data Display Options	393
Bridge Results Display Options	393
Analysis Stage Data Display Options	394
User Modules Display Options	394
Analysis Diagnostics: Error norm	394
Numeric formats	395
Part VI Design in GSA	397
1 Steel Design	397
Generating Steel Member Restraints	397
Member Results	398
Steel Checks to AISC LRFD	399
Steel Checks to AISC 360-05	400
Steel Checks to AS4100:1998	400
Steel Checks to BS5950-1:2000	402
Steel Checks to EN1993	405
EN1993 National Annexes	406
Steel Checks to HKSUOS:05	409
Steel Checks to IS800	411
Steel Checks to SANS 10162-1:2011	412
Results	414
Tools	414
2 Member Results	415
Part VII GSA Tools	417
1 Welcome to GSA dialog	417
2 Undo and Redo	417
3 New Model Wizard	419
Titles	419
Structure	419
Properties	420
4 Data Generation Wizard	420
Structure Types	420
Portal Frame	420
Grid	421
Orthogonal Frame	422
Truss	423
Vierendeel	423
Pratt Truss	424

Pitched Portal	425
Roof Truss	425
2D Element Orthogonal Mesh	426
2D Element Skew Mesh	426
2D Element Elliptical Mesh	427
3D Element Orthogonal Mesh	428
Generate	428
5 Preferences & Settings	428
Preferences	428
Miscellaneous Preferences	433
6 Sculpting	435
Sculpt Dialogs	436
Sculpt Geometry Cursor Modes	445
Creating User Axes Graphically	448
Creating Grid Planes Graphically	448
Adding Nodes Graphically	449
Modifying Nodes Graphically	449
Collapsing Coincident Nodes	449
Rounding Nodal Coordinates Graphically	450
Adding Geometric Entities Graphically	450
Adding Elements Graphically	451
Connecting 1D Elements Graphically	451
Splitting Elements Graphically	452
Joining 1D Elements Graphically	456
Convert Orientation Node to Angle	456
Modifying 2D Elements From Linear To Parabolic	457
Modifying 2D Elements from Parabolic to Linear	457
Moving and Copying Entities Graphically	457
Extruding Nodes and Elements Graphically	458
Transforming Nodes Graphically	458
Flexing Lines of Nodes Graphically	458
Straightening Lines of Nodes Graphically	460
Flipping Elements Graphically	460
Spinning 2D Elements Graphically	461
Modifying Elements Graphically	461
Disconnecting Elements Graphically	461
Deleting Nodes and Elements Graphically	462
Creating Rigid Constraints Graphically	462
Creating Joints Graphically	462
Creating Nodal Loading Graphically	463
Creating Element Loading Graphically	463
Creating Grid Loading Graphically	464
Deleting Loading Graphically	465
7 File Tools	465
Import GWA Data	465
Footfall Response Data	465
8 Output View Dialogs	466
Wizard: Output Settings	466
Output Wizard: Further Options	468
9 Graphic View Dialogs	469
Wizard: Graphic Settings	469
Labels and Display Methods	472
Deformation Settings	479
Contour Settings	480

Diagram Settings	483
Bridge Options	485
Further Options	486
Animation Settings	487
Orientation Settings	488
Graphic Fonts and Styles	489
10 Chart View Dialogs	489
Non-linear Analysis	490
Modal Analysis Details	490
Harmonic Analysis	490
Periodic Load Analysis	491
Linear Time-history Analysis	491
Footfall Analysis	491
Storey Displacements and Forces	492
Forces on 2D Element Cut	493
11 Model Tools	493
Manage Data	493
Current Grid Definition	495
Grid Layout Definition	496
History	496
Coordination	497
Section Tools	501
Checking Tools	502
Manipulate Model	503
Mesh Generation	506
Contact Constraints	508
Create Grid Lines	509
Building Modelling	510
Raft Modelling	513
Bridge Modelling	513
Dynamics	517
Seismic	518
Offshore	521
Manage User Modules	521
Miscellaneous Tools	522
12 Miscellaneous Dialogs	524
Find	524
Replace	524
Modify	525
Go To	525
Modify Curve	525
Curve Data Selection	526
GWA Import Options	526
CAD Export Options	527
CAD Import Options	527
DXF Export Options (legacy option)	530
DXF Import Options (legacy option)	530
Nastran Export Options	531
OpenSees Export Options	531
ADC AdBeam Export	532
AdSec Export	532
Export Member Input Data to CSV	533
1D Element Results	533
Numeric Format	534

Axes	534
Part VIII GSA Text Files	536
1 Text Data File Format	536
2 Reinforcement Template File	537
3 Curve File	541
4 Tab and Comma Separated Files	542
Part IX API and Command Line Interface	544
1 GSA API	544
2 Command File	544
Part X Interaction with Other Software	549
1 Analysis in LS-DYNA	549
2 Export options	551
Input data	551
SAP2000	551
Nastran	552
OpenSees	553
AutoCAD	553
PDisp	554
ADC	554
AdSec	554
Steel Member(s)	555
Footfall Response	555
Text output	555
3 Import options	555
Text files	555
AutoCAD	556
PDisp	557
4 Links With 3D Modelling Packages	557
Revit	557
IFC	570
Part XI Bibliography	573
Part XII Glossary	576

Part I

1 About GSA

1.1 Overview

Oasys 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 10 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 (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
- Explicit 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 on surfaces representing floor or walls etc. In addition to forces and moment applied to the structure, load types include gravity, prestress, thermal and distortion loads.
- Restraints and constraints can apply to non-global and user defined axis directions as well to global axis directions
- Displacements and forces are calculated at automatically determined intermediate points along 1D 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 10 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 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 model to AdBeam for design. For slab design the required areas of steel are calculated and displayed as graphical or text output as requested by the user, based on the analysis results.

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

Output

Output can be in the form of tables showing the numerical results, graphics showing diagrams or contours and charts showing plots of one variable against another for some analysis results

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 be 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 Documentation

In addition to the help file there is further documentation on the use of GSA.

- [Getting Started](#)
- [Theory](#)
- [Slab Design](#)
- [Concrete Code Reference](#)
- [API Reference](#)
- [IFC Reference](#)
- [Text File Format](#)

GSA makes use of third party software. Details can be found [here](#).

1.6 Design Codes

1.6.1 Steel

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

Code	Title	Cou ntry	Dat e	Other version s	Notes
AISC360-10	Specification for Structural Steel Buildings, American Institute of Steel Construction	USA	2010	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	USA			<i>Design checks of I, H, RHS, Tee, Channel and CHS including local capacity and</i>

					<i>member buckling checks (non-sway frame only).</i>
AS4100-1998	Standards Australia	Australia	1998	8	<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	Canada	2009		
BS5950-1:2000	Structural use of steelwork in building. Code of practice for design - Rolled and welded sections, British Standards Institution	UK	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>
EN 1993-1-1+2005	Eurocode 3: Design of steel structures, European Committee for Standardisation	Europe	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: Design of steel structures - Fire, European Committee for Standardisation	Europe	2005		<i>National Annexe for UK</i>
HKSUOS-11	Code of Practice for the Structural Use of Steel, Building Authority Hong Kong	Hong Kong	2011	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 in Steel – Code of Practice, Bureau of Indian Standards	India	2007	1984	<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	South Africa	2011		<i>Design checks of I, H, RHS, CHS, Tee, Channel and Angle sections including local capacity and member buckling checks.</i>
Stress Check					<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 of</i>

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

1.6.2 Concrete

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

Code	Title	Country	Date	Other versions
ACI318	Building Code Requirements for Structural Concrete (ACI318-14)	USA	2014	2011, 2008, 2005, 2002
ACI318M	Building Code Requirements for Structural Concrete (ACI318M-14) (metric version)	USA	2014	2011, 2008, 2005, 2002
AS3600	Australian Standard Concrete Structures 2009	Australia	2018	2001, 2009
AS5100	Australian Standard Bridge Design	Australia	2017	
BS5400-4	Steel, concrete and composite bridges – Code of practice for design of concrete bridges	UK	1990	IAN70/06
BS8110-1	Structural Use of Concrete Part 1. Code of practice for design and construction (Incorporating Amendments Nos. 1, 2 and 3)	UK	2005	1997, 1985
BS EN 1992-1-1	<i>Eurocode 2-1</i>	UK	2004	PD6687:2006
BS EN 1992-2	<i>Eurocode 2-2</i>	UK	2005	
CAN CSA A23.3	Design of Concrete Structures	Canada	2014	2004
CAN CSA S6	Canadian Highway Bridge Design Code	Canada	2014	
CYS EN 1992-1-1	<i>Eurocode 2-1-1</i>	Cyprus	2004	

DIN EN 1992-1-1	<i>Eurocode 2-1-1</i>	Germ any	2004
DIN EN 1992-2	<i>Eurocode 2-2</i>	Germ any	2010
DS/EN 1992-1-1	<i>Eurocode 2-1-1</i>	Denm ark	2004
DS/EN 1992-2	<i>Eurocode 2-2</i>	Denm ark	2005
EN1992-1-1	Eurocode 2: Design of concrete structures – Part 1-1: General rules and rules for buildings		2004
EN 1992-2	Eurocode 2: Design of concrete structures. Concrete bridges - Design and detailing rules		2005
Hong Kong Code of Practice	Code of Practice for the Structural Use of Concrete	Hong Kong	2013 2004 (AMD 2007), 2004, 1987
Hong Kong Structures Design Manual	Structures Design Manual for Highways and Railways	Hong Kong	2013 2002
IRC:112	Code of Practice for Concrete Road Bridges	India	2011
IRS Concrete Bridge Code	Code of Practice for Plain, Reinforced & Prestressed Concrete for General Bridge Construction	India	1997
I.S. EN 1992-1-1	<i>Eurocode 2-1-1</i>	Ireland	2004
I.S. EN 1992-2	<i>Eurocode 2-2</i>	Ireland	2005
IS 456	Plain and reinforced concrete – Code of Practice	India	2000
NEN EN 1992-1-1	<i>Eurocode 2-1-1</i>	Netherlands	2004
NEN EN 1992-2	<i>Eurocode 2-2</i>	Netherlands	2011

NF EN 1992-1-1	<i>Eurocode 2-1-1</i>	France	2005
NF EN 1992-2	<i>Eurocode 2-2</i>	France	2006
NS-EN 1992-1-1	<i>Eurocode 2-1-1</i>	Norway	2004
PN-EN 1992-1-1	<i>Eurocode 2-1-1</i>	Poland	2008
SFS-EN1992-1-1	<i>Eurocode 2-1-1</i>	Finland	
UNE-EN 1992-1-1	<i>Eurocode 2-1-1</i>	Spain	2010 /2013
UNE-EN 1992-2	<i>Eurocode 2-2</i>	Spain	2010
UNI EN 1992-1-1	<i>Eurocode 2-1-1</i>	Italy	2004
UNI EN 1992-2	<i>Eurocode 2-2</i>	Italy	2006

1.7 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. 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 the procedures set out in this QMS. A certificate is produced to confirm all testing has been completed in accordance with the QMS, and this can be provided on request.

1.8 Acknowledgements

Arup

Many engineers have contributed to the development of GSA. In particular the dynamic relaxation solver is based on the work of Alistair Day, the P- and modal buckling options in GSA were developed in conjunction with Pat Dallard, the RC Slab options have been developed by Ian Feltham, the footfall analysis options by Peter Young and Mike Willford and the pattern loading by Don Ho.

Third Party Components

Oasys programs make use of the following third party components

- LDL Version 1.1
- AMD Version 1.1
- Resizeable dialogs
- Delaunay triangulation
- CM2 MeshTools
- LAPACK

See below for copyright and licensing details.

LDL

Copyright, and License:

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

LDL License:

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.

Availability:

www.cise.ufl.edu/research/spares/ldlamd/

AMD

Copyright, and License:

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

AMD License:

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.

Availability:

www.cise.ufl.edu/research/spares/amd/

Resizable dialogs

Copyright, and License:

Resizable dialogs, Copyright (c) 2000-2001 by Paolo Messina. All Rights Reserved.

www.codeproject.com/Articles/1175/ResizableLib

Delaunay triangulation

Copyright, and License:

Copyright (c) 2015 Simon Zeni (simonzeni@gmail.com)

Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

<https://github.com/Bl4ckb0ne/delaunay-triangulation>

CM2 MeshTools

Copyright, and License:

CM2 MeshTools by Computing Objects. All Rights Reserved.

<http://www.computing-objects.com/content/view/4/14/lang.en/>

LAPACK

Copyright, and License:

Copyright (c) 1992-2011 The University of Tennessee and The University of Tennessee Research Foundation. All rights reserved.

Copyright (c) 2000-2011 The University of California Berkeley. All rights reserved.

Copyright (c) 2006-2011 The University of Colorado Denver. All rights reserved.

COPYRIGHT

Additional copyrights may follow

HEADER

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

- Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
- Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer listed in this license in the documentation and/or other materials provided with the distribution.
- Neither the name of the copyright holders nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

The copyright holders provide no reassurances that the source code provided does not infringe any patent, copyright, or any other intellectual property rights of third parties. The copyright holders disclaim any liability to any recipient for claims brought against recipient by any third party for infringement of that parties intellectual property rights.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

<http://www.netlib.org/lapack/>

Part II

2 Getting Started

GSA is a program for structural analysis, and it is assumed the user has a knowledge of structural engineering concepts. The GSA help is aimed at explaining the features and use of the program but not the structural concepts expressed in the program. A new user to GSA should refer to the [Getting Started](#) manual which gives a worked example on the use of GSA.

Data Modules

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.

Data Generation

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.

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](#)".

Spreadsheets & CAD

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.

BIM

Models assembled in BIM packages can be imported into GSA either via an Industry Foundation Classes (IFC) file or from the GSA/Revit link.

Part III

3 Working with GSA

This chapter describes how to use the GSA interface

Wizards

The [New Model Wizard](#) and [Data Generation Wizard](#) provide a quick way of generating simple structural forms.

The New Model Wizard runs when you start a new document 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 GSA Tools section.

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 Analysis Tasks and Cases section.

Preferences

Preferences are set in the [Preferences Dialog](#) available from the “Tools | Preferences” (Ctrl+F7) menu command.

Toolbar and dockable pane selection and positioning and window sizes are 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](#)

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 Explorer Panes.

Toolbars and dockable panes

Toolbars and dockable panes can be controlled from the “View | Toolbars and Dockable Panes” menu command or the GSA toolbar. Toolbars can be customised using the dropdown arrow at the end of the bar. With the exception of the [Lists toolbar](#) all toolbars can be either docked (attached to the main frame) or floating. Toolbars are covered in detail in the [Toolbars and Keyboard Accelerators](#) section. documentation.

File formats

The default file type for GSA is a GWB file. There is also an editable text file available (GWA) which can be used when taking data to an earlier version of GSA. The text file format is given in [GSA Text \(ASCII\) Data File Format](#)

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](#).

Views

[Explorer Panes](#) – access to Table Views, Output Views, Saved Views and Preferred Views.

[Output Panes](#) – display properties; display results for individual nodes, elements and members and access data defaults; display reports of the analysis process

[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.

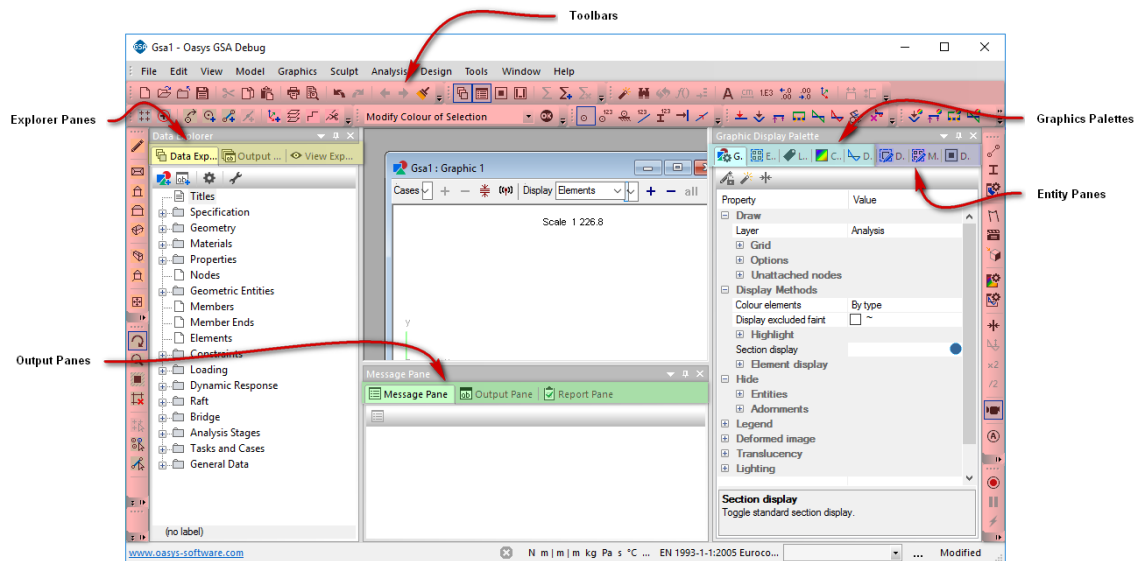
3.1 Dockable Panes

GSA uses dockable panes to control the model. These fall into four categories:

- [Explorer Panes](#) offer tree controls that give access to input data modules (typically [Table Views](#)), [Output Views](#) and [Saved Views](#).
- [Output Panes](#) display text messages, results and task reports.
- [Graphics Palettes](#) give direct control of [Graphic Views](#).
- [Entity Panes](#) provide tools for editing entities.

Their default positions are shown below.

Dockable panes can be moved, floated and closed. When floating they are displayed on top of any other view and they can be dragged outside the GSA window. They can be docked along any edge of the GSA window, with any other dockable panes. When docked the part of the screen that is available for other views excludes that occupied by the dockable panes. The default position can be recalled using the 'View | Reset Toolbars and Dockable Panes' menu command. All dockable panes can all be toggled on and off using the 'View | Show or Hide Dockable Panes' menu command (or  button on the [GSA toolbar](#)), and individual dockable panes can be opened and closed from the commands in the 'View | Toolbars and Dockable Panes' sub-menu.



3.1.1 Explorer Panes

The Explorer Panes are a collection of [Dockable Panes](#):

- [Data Explorer pane](#) – gives access to all input data modules
- [Output Explorer pane](#) – gives access to all Output Views that are available for the model.
- [View Explorer pane](#) – gives access to all Saved Graphic Views and Saved Output Views.

Double clicking on an item will open the appropriate view.

The Explorer Panes are [Dockable Panes](#). The content of the Explorer Panes always relates to the current model.

Data Explorer pane

The Data Explorer pane organises the data into several categories.

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.

Output Explorer pane

The Output Explorer pane 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.

View Explorer pane

The View Explorer pane 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.

3.1.2 Output Panes

The Output Panes are a collection of [Dockable Panes](#):

- [Message Pane](#) – displays properties for a selected item.
- [Result Pane](#) – displays results for a selected item.
- [Report Pane](#) – displays task logs (i.e. a history of Report View logs).

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

Message Pane

The Message Pane displays properties for a selected item or a set of selected items. It is also where messages from various operations are displayed.

In Graphic Views:

- clicking on a node, element or member causes the properties for that item to be displayed in the Message Pane, regardless of the current cursor mode.
- selecting items and then using the “Graphics | Selection Properties” command displays properties relating to the current selection.

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 Table Views:

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.

Result Pane

The Result Pane displays results for a selected item. This is done as follows:

1. Open the Result Pane by selecting 'View | Toolbars and Dockable Panes | Result Pane' command (or clicking the Result Pane button toolbar button if available).
2. Select the required output option from the drop-down list at the top of the Result Pane. Only those options currently available for the model are displayed.
3. Select the required case or cases in the Cases list at the top of a Graphic View.
4. In the Graphic View, click on a node, element or member to display their results.

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 Result Pane. 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.

Report Pane

The Report Pane displays a history of all logs generated by GSA tasks, such as analysis and design. For example, during an analysis the Report View displays the analysis log as it happens; upon completion the report log stored with the model and displayed in the Report Pane.

The contents of the Report Pane can be deleted by using the Delete Report Logs button on the Report Pane toolbar.

3.1.3 Graphics Palettes

The Graphics Palettes are a collection of [Dockable Panes](#) that define what is displayed in the Graphics view.

- [Graphics Display Palette](#) – provides quick access to frequently used Graphic View settings.
- [Entity Display Palette](#) – provides a convenient way of specifying the entities that are to be displayed.
- [Label Palette](#) – provides a convenient way of setting labels.
- [Contour Palette](#) – provides a convenient way of setting contours.
- [Diagram Palette](#) – provides a convenient way of setting diagrams.

Each Graphics Palette includes a toolbar offering options that relate to the palette. All Graphics Palette toolbars offer the **Lock Image Update** command. When Lock Image Update is set no updating of the image in the current Graphic View occurs; unsetting Lock Image Update causes the image to be regenerated picking up all the changes to the graphic settings that have been applied. Rotating, zooming or panning the view unsets Lock Image Update. This feature is useful for large models which take a

significant time to update the view.

Graphics Display Palette

The Graphics Display Palette provides quick access to a frequently used subset of Graphic View settings.

The Graphics Display Palette toolbar offers:

- **Lock image update** – (see above)
- **Wizard** – opens the [Graphics Settings](#) dialog.
- **Reset display adornments** – resets the Graphics Settings to the current defaults.

Right-clicking on a field displays a context menu offering the settings dialog for that field, where appropriate.

Entity Display Palette

The Entity Display Palette provides a convenient way of specifying the entities that are to be displayed in the current Graphic View in terms of saved lists, grid surfaces, properties, etc. The categories are offered as branches on a tree control. The selected item is applied to the Display List for the current Graphic View.

The Entity Display Palette toolbar offers:

- **Lock image update** – (see above)
- **Highlight mode** – when set the entities selected in the tree control are highlighted in the Graphic View.
- **Check boxes** – when set the tree control is displayed with check boxes against each item to allow multiple selection.
- **Reset to all entities** – resets the Display List to to 'all' and clears all Volumes.

Right-clicking on an item displays a context menu offering options to open the definition dialog, to select all entities with that item set, and to display the properties for the item in the Message Pane, where appropriate.

Label Palette

The Label Palette provides a convenient way of switching label options on and off in the current Graphic View.

The Label Palette toolbar offers:

- **Lock image update** – (see above)
- **Hide Labels** – when set all labels are removed from the display in the current Graphic View, without adjusting the labelling settings. Unsetting 'Hide Labels' re-displays the labels.
- **Reset labels** – resets all labelling settings to default.
- **Labels and Display Methods** – opens the [Labels and Display Methods](#) dialog.
- **Apply Adornments to Selection** – when set labels are applied to the current selection only.

Contour Palette

The Contour Palette provides a convenient way of setting contours in the current Graphic View.

The Contour Palette toolbar offers:

- **Lock image update** – (see above)
- **Hide Contours** – when set contours are removed from the display in the current Graphic View, without adjusting the contouring settings. Unsetting 'Hide Contours' re-displays the contours.
- **Reset contours** – resets contour settings.
- **Layer combo** – sets the output layer for 2D element stress contours.
- **Contour settings** – opens the [Contour Settings](#) dialog.
- **Apply Adornments to Selection** – when set contours are applied to the current selection only.

Diagram Palette

The Diagram Palette provides a convenient way of setting diagrams in the current Graphic View.

The Diagram Palette toolbar offers:

- **Lock image update** – (see above)
- **Hide Diagrams** – when set diagrams are removed from the display in the current Graphic View, without adjusting the diagram settings. Unsetting 'Hide Diagrams' re-displays the diagrams.
- **Reset diagrams** – resets diagram settings.
- **Layer combo** – sets the output layer for 2D element stress diagrams.
- **Diagram settings** – opens the [Diagram Settings](#) dialog.
- **Apply Adornments to Selection** – when set diagrams are applied to the current selection only.


3.1.4 Entity Panes

The Entity Panes are a collection of [Dockable Panes](#) that allow you to view and modify the default and current settings of entities.

- [Define Entity pane](#) – Settings for the current entity are displayed and can be edited within the pane.
- [Modify Entities pane](#) – Settings for an entity type can be specified or grabbed from the current selection. Checked settings can then be applied to the current selection or be painted onto entities in a Graphic View.
- [Data Defaults pane](#) – Default entity settings are displayed and can be edited.

Define Entity pane

The Define Entity pane is configured and populated according to the current 'definition entity'. The 'definition entity' is set by clicking on the entity in a Graphic View, and is identified in Graphic Views by a cyan circle. Double-clicking on an entity both sets the entity as the 'definition entity' and brings the Define Entity pane to the fore; though note that if the Define Entity pane is hidden then double-clicking opens the definition dialog for the entity.

The entity settings can be edited within the pane, or you can use the Wizard button  at the top of the pane to open the respective definition dialog. Settings changed within the pane are applied directly to the entity upon terminating the edit.

Right-clicking on the pane displays a context menu offering options to open the definition dialog for the 'definition entity' and to open other definition dialogs relating to the entity, where appropriate. E.g. When the 'definition entity' is an element the context menu offers options to open the element definition dialog and the definition dialogs for the property, the analysis material, and the material grade that relate to the element.

Modify Entities pane

The Modify Entities pane is configured for the entity type that relates to the current selection mode (or the last selection mode used if the current cursor mode is not a selection mode). To the left of every setting is a check box that 'activates' the setting. 'Active' settings can be applied to existing entities by either applying to the current selection or by painting them on entities.

A toolbar at the top of the Modify Entities pane




... provides various functionality, as described below.

Right-clicking on the pane displays a context menu offering options to open other definition dialogs relating to the entity, where appropriate.



Assigning settings in the Modify Entities pane

As well as typing in values into fields the following methods may be used to assign settings in the Modify Entities pane.


- Import Data Default settings by clicking the 'Default properties' button . All settings are set to default.
- [Select](#) a single entity. All settings are assigned to the settings of the selected entity and are activated.
- Select several entities. Only those settings that are common to the selected entities are assigned and activated.

Modifying entities


Existing entities may be modified by applying the active settings in the Modify Entities pane as follows.

- When in a selection mode, click the 'OK' button  to apply the 'active' settings to the current selection.
- When in the 'Modify Entities Sculpt Tool' , apply the 'active' settings to entities by selecting them in a Graphics View.

Applying settings to defaults

Click the 'Apply to defaults' button  to apply the active settings to Data Defaults.

Selecting entities by active settings

Click the 'Select by Properties' button  to select all entities that have the active settings assigned. E.g. if for elements the property field is active and is set to 3 and the dummy field is active and is checked then 'Select by Properties' will select all dummy elements with property 3 assigned.

Note that in this search process 'name' fields search for substrings, so selecting for all entities named 'fred' will select those named 'alfred' and those named 'frederick'.

Data Defaults pane

The Data Defaults pane displays and allows editing of default settings for entities. Default settings are applied when new entities are created in GSA.

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. Default settings can also be assigned from the [Modify Entities pane](#).

The current default settings can be applied to:

- a single entity by right-clicking on the entity and selecting the “Apply Defaults” command, or
- several entities by [selecting](#) the entities and giving the “Sculpt | Apply Defaults to Selection” command.

Note that:

- The data defaults are also used by Table Views and may be edited from there.
- 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.

3.2 Table Views

Most input data required by GSA can be entered in tables. Tables in GSA are similar to spreadsheets, but there are a number of significant differences. These are highlighted in the sections below.

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.

For many of the tables there is the option of defining the data in a Wizard. The wizard is available from the Wizard button in the [Data Options toolbar](#), or from the table context menu or with the shortcut Ctrl+W. 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.

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 that would be invalidated by the changes.

The details of the data entered in each table are covered in the [Data in GSA](#) section.

3.2.1 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.

3.2.2 Field Types

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

- **Text** – for names and labels.
- **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).
- **Button** – for access to [wizards](#).

3.2.3 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.
-----	-------------------------------

<code>--f</code>	to subtract f from the copied value.
<code>*=f</code>	to multiply the copied value by f.
<code>/=f</code>	to divide the copied value by f.

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

3.2.4 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.

Selection

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.

3.2.5 Cutting, Copying 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.

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 blocks of selected data to the clipboard as tab delimited text. 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 (held in the clipboard in tab delimited format) are inserted if acceptable, see below.

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.

3.2.6 Delete and Blank Records 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.

3.2.7 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).

3.2.8 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.

3.2.9 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.

3.2.10 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.

3.3 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** menu 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.

You can also open Output Views via the Output explorer. This opens the selected Output View with the default output specification. Note that the default output specification is “All” cases and the first 25 nodes, elements or members, so opening an Output View via the Output explorer 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 View Explorer pane](#) for accessing these.

3.3.1 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.

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](#).

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](#).

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.

3.3.2 Output View Table Format

Tables of output may be formatted by displaying the output in a standard Output View or a Grid Output View. Output that is for inspection, presentation or printing is best formatted with a standard Output View, while the Grid Output View is offered to facilitate the copying of output to spreadsheets.

New Output Views are opened in the standard Output View 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.

Standard Output Views

Generally standard Output Views are best for the presentation or printing of output.

Advantages:

- Table header of the "active" table remains at the top of the window
- Output options may be specified directly.
- Output format is optimised for printing / saving as PDF
- Table formatting preserved well to spreadsheets if entire rows are selected and copied into a spreadsheet

Disadvantages:

- Individual columns may not be selected and copied to a spreadsheet

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.

3.3.3 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 explorer.

Refer to the [Output Settings dialog box](#) documentation for more details.

3.3.4 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 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.

3.3.5 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.

3.3.6 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 [Enveloping in GSA](#).

3.3.7 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.

3.3.8 Output Summary

Most tables can be summarised in terms of the maximum and minimum values encountered in the table for each component. The maximum and minimum values are displayed in bold; the coexistent components reported in the summary are the same as the coexistent values reported in the table, even when the case being reported is an enveloping case.

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.

3.3.9 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.

3.3.10 Printing from Output Views

The output may be printed by giving the **File | Print** 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.

3.3.11 Interacting with Spreadsheets

The spreadsheet format option is offered to assist in the interaction between Output Views and spreadsheets. Spreadsheet format is selected in the [Output Wizard: Further Options](#).

Spreadsheet format 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 if you want to target specific columns. Otherwise, copying from a Standard Output View would work as well. 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. 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 “Output | Export Output” menu command writes the output as currently displayed to file in either tab or comma delimited text format.

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.

3.4 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.

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 | Miscellaneous View Options | New Graphic View** (main thread) 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 View Explorer pane](#) for accessing these.

3.4.1 Changing the Content of a Graphics View

The content of a Graphic View can be set either in:

- The [Graphics Palettes](#). Refer to the [Dockable Panes](#) section for details on accessing these.
- 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.
 - using the “Wizard” command on the [Graphics Display Palette](#) or [Data Options toolbar](#).
 - Double-clicking on the background of a Graphic View.

New Graphic Views use the [default view settings](#) unless there is a [Saved Graphic View](#) named “startup” (case insensitive) in which case these settings are used.

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. 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.

3.4.2 Basic Orientation of the Image

3.4.2.1 Standard Views

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.

The following commands (and their associated accelerator keys) are available from the **Graphics | Orientation | Standard Views** 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.

3.4.2.2 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](#).

3.4.2.3 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.

3.4.3 Scaling the Image and Zooming

3.4.3.1 Scaling to Fit

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.

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.

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.

3.4.3.2 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 $\frac{1}{16}$ of the picture width.
- **Pan Left** (Shift+Lft). Adjusts the mid-point to result in the picture shifting right by $\frac{1}{16}$ of the picture width.
- **Pan Up** (Shift+Up). Adjusts the mid-point to result in the picture shifting down by $\frac{1}{16}$ of the picture height.
- **Pan Down** (Shift+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.

3.4.3.3 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](#).

3.4.4 Advanced orientation of the image

3.4.4.1 Adjusting the Object Point

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.

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.

3.4.4.2 Orthographic and Perspective Projections

The view may be toggled between orthographic and perspective projections using the **Graphics | Orientation | Perspective Proj.** (Ctrl+Alt+P) 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.

3.4.4.3 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.

3.4.5 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.


More:

[Volume clipping](#)

[Entity lists](#)

3.4.5.1 Volume Clipping

To identify the entities included in a volume,

- Set the [cursor mode](#) to “Volume” .
- Drag the mouse to specify a rectangle (elements within the volume are **included**; when Ctrl-dragging the mouse, elements within the volume are **excluded**).
- Use Ctrl+mouse wheel to specify the near and far extents of the volume. Ctrl+rotate moves the **near** plane of the volume; Shift+rotate moves the **far** plane of the volume; holding the mouse wheel down while rotating refines the increment of movement.

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. 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.

3.4.5.2 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.)

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.

3.4.6 Graphical Representation of Entities

3.4.6.1 Default Representation of Entities

There are various ways in which nodes, elements, members and geometric entities are represented in Graphic Views.

Note that all surface colours are affected by the current [shading](#) settings, and all colours and symbol sizes are affected by the current [Graphic Fonts and Styles](#) settings.

Note that the colours of entities stated below can be overridden by [specifying the colour of entities](#) explicitly.

Nodes

Nodes that are attached to elements, members or lines are drawn only if the ‘dots at nodes’ [label option](#) is switched on; unattached nodes are drawn only if the respective ‘draw unattached nodes’ [display method](#) is switched on.

Standard nodes are drawn as circles (or diamonds, on close inspection). Grid nodes are drawn as inverted triangles.

Unattached nodes are drawn red. Attached nodes are drawn black, except when part of a region (see ‘Regions’ below), else when attached to an area (see ‘Areas’ below).

Elements and Members

The colour of elements and members depends on the *colour elements*’ [display method](#) setting:

- By type – each element type is assigned a distinct colour. When 2D elements are displayed ‘filled’ and ‘solid’ the top (+ve z) surface is drawn light grey and the bottom surface is drawn dark grey.
- By property – each element property is assigned a unique colour. Elements are coloured according to their property.
- By material – each material is assigned a unique colour. Elements are coloured according to their material.
- By initial stage – each analysis stage is assigned a unique colour. Elements are coloured according to their initial analysis stage.

Lines

Line entities are drawn black, except when part of a region (see ‘Regions’ below).

Areas

Areas, when not filled, and nodes referenced by areas are drawn burgundy, except when part of a region

(see 'Regions' below).

Regions

Areas, lines and nodes associated with a region are coloured teal.

3.4.6.2 Specifying Colour of entities

The colour of entities displayed in Graphic Views can be defined explicitly, as follows:

Colours can be assigned to the following object types:

- Nodes
- Elements and Members
- Materials
- Properties
- Polylines
- Analysis Stages

Displaying colours

When the view is set to colour elements *by property*, or *by material* or *by initial stage*, elements (or members) are displayed in the colour of the property, material or stage, respectively.

Otherwise entities are displayed in the colour of the entity.

If the colour is set to 'undefined' (the default) default colours are assumed.

Setting colours

The colours of objects are assigned using the **Modify Colour of Selection** command.

If the view is set to colour elements 'by property', or 'by material' or 'by initial stage' the colours of the properties, materials or stages of the selected elements (or members), respectively, are modified.

Otherwise the colours of the selected entities are modified.

Choosing colours

Colours are chosen in the Select Colour dialog; either by selecting from a list of preset colours or by picking in a Colour dialog.

The colour may be set to 'undefined'.

The Select Colour dialog remembers the 'last selected colour'.

The 'last selected colour' can be set by picking the currently assigned colour of an object, by using the right-click 'Pick Colour of This' command.

The colour of all objects in the model can be reset to 'undefined' by using the 'Clear Colour Modifications' command.

3.4.7 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.

3.4.7.1 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.

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.

The grid can be switched on and off by:

- the **Sculpt | Draw Grid** (Ctrl+Alt+G) command (also available on the [Sculpt toolbar](#))
- the **Draw Grid** option in the [Graphics Display Palette](#)

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.

3.4.7.2 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.

3.4.8 Selection Sets

3.4.8.1 The Model 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.

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.

3.4.8.2 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.

3.4.8.3 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** deselects the identified items; otherwise the previous selection is not emptied.
- **Shift+Clicking** or **Shift+Dragging** selects the identified items; the previous selection is not emptied.

Right-clicking on an entity displays a [right-click menu](#) that includes the option to select that entity. 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 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).

- *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.

Note also the scope for interaction with [lists](#) and the [clipboard](#).

3.4.8.4 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 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 List** are also available on the Graphic View right-click menu when in a selection mode.

3.4.8.5 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](#).

3.4.8.6 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.

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.

3.4.9 Polylines in Graphic Views

In GSA a polyline is a sequence of points in 3D space, which may optionally be specified in 2D, and which may optionally be related to a grid plane. There is a [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 with respect to its grid plane axes or, if the grid plane is undefined, 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.

3.4.9.1 Use of Polylines

Polylines may be used for:

- Creating an area for grid loading
- Specifying a selection set

Selection sets may be [formed](#) by selecting entities within the current polyline.

Several [sculpt commands](#) require a polyline as input, including [Create Grid Loading](#) and the (legacy) generate 2D elements commands.

Areas can be created from polylines, either by right-clicking on a displayed polyline or by right-clicking when the current cursor mode is set to Polyline. New nodes and lines are created as necessary.

Polylines can include voids. The void is defined by having an anticlockwise set of points around the outer perimeter which connect to a clockwise set of point to define a void before connecting back to the outer perimeter.

When in the Polyline cursor mode the properties of the polyline can be obtained using the **Graphics / Selection Properties** 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.

3.4.9.2 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** when the grid is not drawn adds a point to the polyline at that position on the picture plane; i.e. on the plane normal to the line of sight through the object point.
- **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.

3.4.9.3 Interaction of Graphic Polylines with 2D Polylines module

When in the Polyline cursor mode the **Edit | Save Selection As Polyline** command saves the currently displayed polyline to the Polylines module, giving the option to overwrite an existing polyline or create a new one.

Also, when in the Polyline cursor mode the **Edit | Select Polyline** menu command allows an existing polyline to be displayed as the current polyline.

On giving the **Save Selection As Polyline** or **Select 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 Polyline** and **Select Polyline** are also available on the Graphic View right-click menu when in the Polyline cursor mode.

3.4.10 Adornments

3.4.10.1 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 [Labels toolbar](#).

Note that [resetting the display](#) reverts to the labels set in the [default view settings](#).

Refer to the [Displaying Data and Results section](#) 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.

3.4.10.2 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.

3.4.10.3 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** 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 [Diagrams 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 [Displaying Data and Results section](#) 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](#)

3.4.10.4 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** 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 [Displaying Data and Results section](#) 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 shown 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](#)

3.4.10.5 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](#)

3.4.10.6 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.

3.4.10.7 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.

3.4.10.8 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 [Contours](#) and [Diagrams](#) toolbars. 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.

3.4.10.9 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.
4. Note that the **Edit / Select All** (Ctrl+A), **Edit / Select None** and **Edit / Invert Selection** 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*.

3.4.10.10 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 [Displaying Data and Results section](#) 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](#).

3.4.11 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](#).

3.4.12 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 pool* explicit colours are ignored.
- when colours are *by analysis material* then the colour assigned to the analysis material is used.
- when colours are *by material type* explicit colours are ignored.
- when colours are *by material grade* then the colour assigned to the material grade 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** 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** 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](#).

3.4.13 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](#).

3.4.14 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](#).

3.4.15 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' = 0. The conical unwrap mode unwraps splitting at 'unwrap axes' = 180 to result in the apex of the cone at 0,0,0 and 'unwrap axes' = 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.

3.4.16 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.

3.4.17 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](#).

3.4.18 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](#).

3.4.19 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.

3.4.20 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.

3.4.21 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.

3.4.22 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.

3.4.23 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** 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.

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](#).

3.4.24 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.)

3.4.25 Cursor Modes in Graphic Views

There are various modes of operation of the cursor in Graphic Views. The following summarises the scope of these cursor modes.

Rotate cursor mode

In general, left mouse button behaviour replicates right mouse button behaviour. Additional features include:

Set centre of rotation

- Ctrl + left click

Reset centre of rotation

- Shift + left click

Zoom cursor mode

Incremental zoom

- Left click (zoom in)
- Ctrl + left click (zoom out)

Zoom to selection

- Left button drag

Volume cursor mode

New volume

- Left button drag (inclusive volume)
- Ctrl + left button drag (exclusive volume)

Adjust last volume depth

- Ctrl + wheel rotate (adjust near plane)
- Ctrl + Shift wheel rotate (adjust far plane)

Delete last volume

- Left click

Delete all volumes

- Ctrl + left click

Select cursor modes

New entity selection

- Left click
- Left button drag

Add to existing selection

- Shift + left click
- Shift + left button drag

Deselect entities

- Ctrl + left click
- Ctrl + left button drag

Polyline cursor mode

Add vertex

- Left click

Drag vertex

- Left button drag

Sculpt geometry cursor modes

Add node/point on new entity

- Left button click

Drag node on current grid plane

- Left button drag

3.4.26 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 Chart Views

There are some results in GSA which are presented as charts, where results are plotted against another parameter, for example, nodal displacements against time from time history analysis. 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, for example, copying curve data on the chart view from right click menu.

3.5.1 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 displays 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.
- **Coordinates of Nearest Point** – the tooltip displays 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 for the nearest curve to be switched on/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.

Curve Style

When near to a curve there is an option to edit the style of that curve. This allows the colour, line style and weight, symbols etc. to be modified for that curve.

Save Image

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.

3.5.2 Chart Styles

3.5.2.1 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
- **Symbols** - the symbols on the individual curves

Height and Width

This controls the height allowed for the header (at the top of the view) and the width of the legend (on the right). 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.

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.

Bar width

For a bar graph this defines the width of the individual bars

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.

3.5.2.2 Axis Styles

This allows the display of the axes to be adjusted. Similar option exist for both abscissa and ordinate.

Label: the text that is displayed on the axis; this can be displayed in bold or italic. 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>.

Annotation: check to include the values on the axis.

Major tick marks/Minor tick marks: check to include on the axis.

Autoscale: scale chart to fit the data (default). When **Autoscale** is selected there is an option to include the zero value in the autoscale calculation.

User defined: specify explicit scale

Axis Type: display as **Linear** or **Logarithmic** axes (depending on the data in the curves)

Axis Position: position either on the zero line or at either extremity of the axis.

3.5.2.3 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 option, 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 adjust,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 be selected for one curve this can the be applied to all with **Set all symbols to this size**.

Hide

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.

3.5.2.4 Point Style

In some charts it is possible and useful to identify particular points. The can be either reference points to particular data point, 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 for all to be switched on or off.

3.6 Analysis

Different element types and analysis types are available in GSA. The types of analysis may be limited or modified by the elements selected.

Analysis Task

Analysis in GSA is organised in tasks. So a linear static analysis or a modal dynamic analysis would each be a single task.

Analysis Case

Each analysis task consists of one or more analysis cases. For static analysis the analysis cases are closely associated with load cases, while for a modal dynamic analysis the analysis cases are associated with modes and frequencies.

3.6.1 Elements

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
0D Element Types				
Grounded Spring	1	Similar to a 2 noded Spring (see below) but one end 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 by the specification of a mass scalar, and the 2nd moments of inertia of the body about its three principal axes (whose orientation is specified using any user axis or the global axes).	Mass can also be present in a model from the "create mass from loading" option in the wizard, and from other elements' self masses.	
Grounded Damper	1	Similar to a 2 noded Damper (see below), but one end is attached to the "ground" and so no node is specified.		
1D 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	Release not allowed.	Grounded spring element

		global axes.	
Bar	2	Models a bar or beam with axial, stiffness only – no bending stiffness.	Releases are not relevant
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.
Strut	2	Compression-only element. Stiffness depends on section area, Young's modulus and length.	No moment stiffness. Analysis is cases involving models containing these elements

			<p>nts cannot be combi ned, becau se they are non- linear. Treate d as bar eleme nt for modal analys is</p>
Tie	2	Tension-only element. Stiffness depends on section area, Young's modulus and length.	<p>No Bar momeeleme nt nt, stiffne Beam ss. eleme Analys nt is cases involvi ng model s contai ning these eleme nts cannot be combi ned, becau se they are non- linear. Treate d as bar eleme nt for modal analys is</p>

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 Rigid link has a slave node
Cable	2	Tension only elements. Connect individual cable elements together into a cable by giving them the same property number.	Tie element , Space 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
2D Element Types			
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 strain accurately due to insufficient degrees of

			freedom.
Tri6	6	Parabolic shape functions	2-D element property types

3D Element Types

Brick8	8	Linear shape functions, two types of Brick8 elements available, one is with finite size and one is for modeling infinite boundary
--------	---	---

2D Element Property Types

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

Type	Stiff degrees of freedom per node	Limitations	Comments
Plane stress	2: In-plane translation only. (x,y)		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)		Similar to a shell element but excluding in-plane effects
Flat shell	5: All except rotation about local z axis. (x,y,z,xx,yy) 6: All (x,y,z,xx,yy,zz) for Allman-Cook formulation		Most commonly used 2D element for modelling slabs and walls.
Curved shell	6: All (x,y,z,xx,yy,zz)	Not currently supported by GSA, 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

3.6.2 Analysis Types

The most common analysis in GSA is a linear static analysis. If a model is set up and loading defined a linear static analysis can be initiated using the 'analyse' (sigma) button.

Other analysis types need to be set up using the analysis wizard to specify the particular parameters to control that analysis.

Analysis Options

The main analysis options in GSA are:

- Static analysis – [linear static](#), static P-delta or [nonlinear static](#)
- Dynamic analysis – [modal dynamic](#), [Ritz analysis](#), explicit time-history analysis and response analysis. Both modal and Ritz option can be linear or can include P-delta effects ([modal P-delta](#) and [Ritz P-delta](#)). The response analyses require a modal or Ritz analysis on which to base the response analysis which can be response spectrum analysis, linear time history analysis, harmonic analysis, footfall induced vibration analysis or periodic excitation analysis.
- Buckling analysis – [modal buckling](#) and [component buckling](#).
- Raft analysis – [soil-structure interaction](#) (using [PDisp](#))
- [Bridge](#) analysis
- [Envelope](#) analysis – a method of calculating results for combination cases with many permutations, Envelope analysis records the permutation number for the maximum and minimum requested values rather than relying on the result calculation each time they are requested.
- [Mass](#) analysis – to determine the mass of the structure and the breakdown of mass.
- [Form finding](#) analysis – for use with fabric and cable element models to determine the structural form. Analogous to hanging chain model.
- Model stability analysis – an analysis of the stiffness matrix to identify potential ill-conditioning problems.

Features & Limitations

Static – A direct solution of the linear analysis problem (except where non-linear elements such as tie, strut, cable and some spring elements are used). Result cases can be combined post-analysis. Ignores non-linear effects except for strut and ties.

Modal dynamic – Calculates the dynamic characteristics of the structure (natural frequencies and mode shapes) using eigenanalysis, based on required number of eigenpairs. This is an iterative solution considering linear behaviour only – struts, ties and cables are modelled as bars.

Ritz – Calculates approximate natural frequencies and mode shapes using force-dependent Ritz vectors. This considers only linear behaviour – struts, ties and cables are treated as bars.

Buckling – Calculates the buckling characteristics (load factors and mode shapes) using eigenanalysis. This is an iterative solution considering only linear behaviour – 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.

Static P-delta – similar to static analysis but the stiffness matrix is modified using geometric stiffness (from the same or another analysis case).

Modal dynamic P-delta – similar to modal dynamic but stiffness is modified using geometric stiffness (from a defined load case).

Ritz P-delta – similar to Ritz, but stiffness is modified using geometric stiffness (from a defined load case).

Explicit time history – a non-linear time history analysis using explicit time integration.

Nonlinear static – applies load and uses dynamic relaxation to iterate until nodal forces are in an equilibrium state. It won't find all possible results (e.g. snap through buckling may lead to more than one valid solution). Generally this will be slower than a direct solution and may not always converge. Currently does not handle brick elements.

Component buckling – allows determination of the effective length of a component (/member), taking into consideration the restraint afforded by the rest of the structure.

Response spectrum – earthquake response based on statistical characterisation of seismic event. Based on modal dynamic or Ritz analysis results.

Harmonic – response to a steady state sinusoidal excitation. Based on modal dynamic or Ritz analysis results.

Linear time history – responses of structures subjected to dynamic loads or base acceleration. Linear response only, i.e. large deformation and material plasticity are ignored. Based on modal dynamic or Ritz analysis results.

Footfall – dynamic response analysis of structures subjected to the actions of human footfall. Based on modal dynamic or Ritz analysis results.

Periodic excitation – similar to footfall but requires the user to define the periodic excitation functions.

Raft – soil-structure analysis of raft and piled-raft analysis. Superseded by modelling using brick elements.

Bridge – bridge load optimisation with automatic path generation and influence analysis. Lane by lane, and carriageway optimisation. As the number of analysis cases in the bridge analysis task depends on the bridge specification and influence effects etc. all bridge analysis related data are locked once there is a bridge analysis task.

Envelope – batch envelope generation and storage of envelope results. An alternative to the traditional 'on-the-fly' envelopes.

3.6.3 Running Analyses

Analysis – simple static



The quickest way to run analyses is to use 'Analyse All' on the menu or the [GSA toolbar](#). This runs a simple static analysis or the [analysis tasks](#) that have been set up in the [Analysis Wizard](#). You can also use the "Analysis | Analyse All" menu command.

Individual analysis cases can be run via the [Analysis Tasks window in the Data Explorer](#) or the Analysis task manager (available from the "Analysis | Manage tasks" menu command).

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 [Working with the Program – 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.

Termination



The analysis may be aborted at any stage during the analysis by using the 'Stop' button on the status bar or by closing the Report View.

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'.

Deleting results



The results may be deleted using 'Analysis | Erase All Results' on the menu or 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.

Other analysis types & analysis wizard



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 'Analysis | New Analysis Task' on the menu or the [Data Options toolbar](#).

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.

3.6.4 Summary

Analyse

The default analysis option in GSA is a linear static analysis. When this option is selected a linear static analysis task is created and an analysis case is created corresponding to each load case defined in the model. The results of the analysis are stored in analysis cases. For a linear static analysis these will be global results, displacements, forces and moments, etc. corresponding to the applied loads. Modal analysis also produces displacements, etc. but in this case these are natural frequencies (or load factors) and mode shapes.

Analysis cases are set up automatically for a simple static analysis and but may also be set up via the [Analysis Wizard](#). The more advanced analysis options are always defined in 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.

3.7 Sets and Lists

There are many instances in GSA where there is a need to work with collections of nodes, elements, cases, etc. 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 Graphic Views](#).

Lists can either be [explicit or embedded](#).

3.7.1 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.

A list can be defined for the following entities:

- **Node**

- **Element**
- **Member**
- **Case**
- **Line**
- **Area**
- **Region**
- **Polyline**
- **undefined**

List Definition

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** – 2D property (area)

- **PV** – 3D property (volume)
- **PL** – link property
- **PR** – rigid property
- **PG** – spacer property (geodesic)
- **PC** – cable property (chain)
- **PD** – damper property
- **DCS** – RC slab design property
- **G** – group
- **M** – analysis material
- **MS** – steel grade
- **MC** – concrete grade
- **MP** – FRP grade
- **DMR** – reinforcement grade

Member lists

- **E** – member (optional)
- **P** – property
- **PB** – beam property
- **G** – group
- **M** – analysis material
- **MS** – steel grade
- **MC** – concrete grade
- **MP** – FRP grade
- **DMR** – reinforcement grade
- **L** – member pool

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:

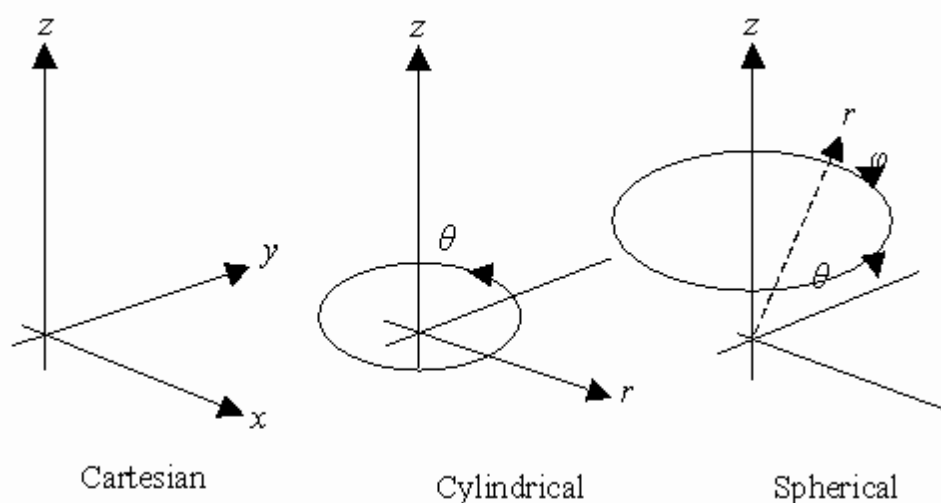
```
(a and b) or c or not d
((a and b) or c) or not d
((a and b) or c) or (not d)
(((a and b) or c) or (not d))
```

3.8 Axes, Grids and Grid Planes

3.8.1 Axis Sets

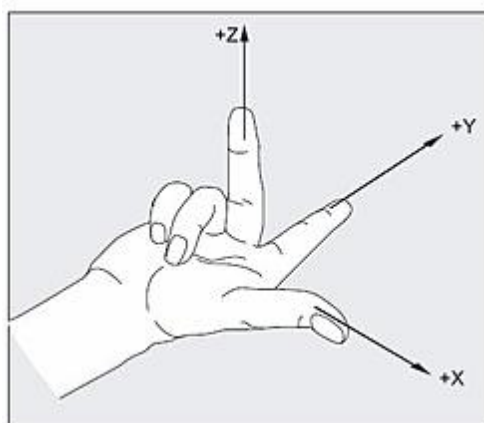
Axes

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 (x, y, z), cylindrical (r, θ , z), or spherical (r, θ , ϕ). Spherical axis follow the physics (ISO) convention: radial distance r, polar angle θ (theta), and azimuthal angle ϕ (phi).

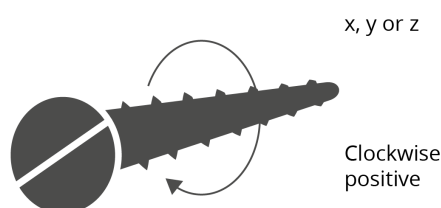


The basic axis system is the Global Cartesian axis system, normally referred to as the **Global** axis system. All other axis systems are located relative to the Global axis system. Global axis directions are generally denoted X , Y and Z to distinguish from other, more general axis directions which use x , y and z .

All the axes systems in GSA are right handed axes systems.



Rotations about the axes follow the right hand screw rule



Predefined Axes

There are a number of predefined axes systems:

- X elevation – to give an xy plane for the X elevation (origin at global origin, x in -Y, y in Z and z in -X).
- Y elevation – to give an xy plane for the Y elevation (origin at global origin, x in X, y in Z and z in -Y)
- Vertical – to give a vertically aligned axis set (origin at global origin, x in Z, y in X and z in Y)
- Global cylindrical – origin at global origin, z in Z and $\theta = 0$ in X

User Defined Axes

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 (\mathbf{x}_a) direction and the xy plane (\mathbf{y}_p) of the axis is defined by any vector lying in the xy plane. The axis set is then constructed from these vectors.

$$\mathbf{x} = \frac{\mathbf{x}_a}{|\mathbf{x}_a|}$$

$$\mathbf{z} = \frac{\mathbf{x} \times \mathbf{y}_p}{|\mathbf{x} \times \mathbf{y}_p|}$$

$$\mathbf{y} = \mathbf{z} \times \mathbf{x}$$

3.8.2 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
- orientation of assemblies
- 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:	yes
Restraints	yes
Spring Supports	yes
Joints (in GSS)	yes
Joints (in GsRelax)	no
Rigid Constraints	no
Constraint Equations	no

Link Elements	no
Settlements	yes
Grid Planes	no
Spring Properties	yes
Mass Properties	yes
2D Element Properties	yes
Node Loads	yes
Applied Displacements	yes
Beam Loads	no
2D Element Loads	no
Current Grid Axes	yes
Output Axes	yes (but data dependent)

3.8.3 Projected Axes

When working with 2D elements it is often desirable to inspect results transformed into user defined directions in the plane of the element. Standard or user defined axis sets may be specified as output axes for this purpose.

So as to arrive at an axis set that lies in the plane of the element, GSA 'projects' the xy plane of the specified axis set on to the element. The method applied for projecting the axes is the same as for calculating '2D element axes defined by axis set', as described in the [2D element axes](#) section.

3.8.4 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 **Model | Define Current Grid** 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.

3.8.5 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 with x in the radial direction, y in the hoop direction and z unchanged. The x, y and z directions of this local axis are used.

3.8.6 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.

3.8.7 Grid Lines, Planes and Surfaces

There are three grid related modules:

- Grid lines - defining a grid of lines (typically in the global xy plane).
- Grid planes - defining flat planes (typically in the global xy plane).
- Grid surfaces - defining how grid loading should be related to grid planes.

Grid Lines

Grid lines are used to set out the basic geometry of typical building structures. The grid lines are typically aligned with global x and y axes, but can be in a user defined axis to model skewed structures. Where a circular geometry is required the grid lines can be defined as circular arcs.

Nodes can be defined as grid nodes in which case the node location is tied to a grid point defined by the intersection of grid lines.

Grid Planes

A grid plane is a planar surface, defined as being parallel to the xy plane of a specified axis set. (See [Grid axes and the current grid](#) above.) The orientation of the plane is determined by the axis - typically global, giving a horizontal plane; but vertical grid planes can be defined by selecting a different axis system. The location of the grid plane is then fixed by the elevation of the grid plane which defined the z coordinate of the grid plane axis at which the plane is located.

Horizontal grid planes can be identified as storeys for storey related calculations. A storey tolerance is used to define what distance above and below the plane are considered to be associated with the plane.

Grid Surfaces

Grid surfaces are used to define how grid loading will be interpreted. A grid surface is located on a grid plane and defines which elements are considered for grid loading and if these should be 1D or 2D elements. The set may be further refined by use of the element list. Only elements that both lie within the grid surface tolerance of the grid surface and are in the element list are included. The other parameters define how the load should be distributed from the grid location to the analytic elements.

Grid loading is applied to a grid surface associated with a grid planes, positioned in grid plane coordinates.

Bridge alignments are defined with respect to grid planes.

3.9 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.

Options to rename and delete views and to reset default settings to GSA default are available on right-click menus for the [View Explorer pane](#).

3.9.1 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 [View Explorer pane](#).

3.9.2 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 [View Explorer pane](#). 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 are present in the model.

3.9.3 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 [View Explorer pane](#). 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.

3.9.4 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.

3.9.5 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.

3.9.6 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 or saved to file in sequence and for that list to be preserved to allow the same list of views to be printed or saved to file 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, save to file, rename and delete view lists are accessible from the [View Explorer pane](#).

To create a new view list right click **View Lists** in the [View Explorer pane](#). 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 and selecting **Edit "<view list name>"**. Both of these open the [View Lists dialog](#).

See also [Batch Printing and Saving of Views](#).

3.9.7 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 [View Explorer pane](#).
- from the **View / Batch View Options** menu or
- remotely (i.e. without running GSA interactively) by [COM Automation](#) or by [Command File](#).

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 [View Explorer pane](#) 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.

3.10 Analysis Tasks and Cases

3.10.1 Task View

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.

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
- **post-analysis** – the task has been analysed and the model contains results

Each task can also be enabled or disabled.

When a static analysis of each load case is requested by simply giving the **Analysis / Analyse Enabled Tasks** command (or **Analyse Enabled** 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.

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 **Enabled** option several tasks may be identified as being ready for analysis. This allows for a key subset of tasks to be run.

See also the Getting Started section on [Running Analyses](#).

3.10.2 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.
- Paste as New Task – pasting a new task (or tasks), appending to the existing tasks (see [Copy and Paste Tasks and Cases](#))
- Enabled – toggle the enabled status of tasks
- Enable All – re-enable all tasks
- Analyse Enabled Tasks – starts the analysis of all marked tasks

- Erase Results for Enabled Tasks
- Delete All Tasks
- Manage Tasks – opens a dialog which allows tasks to be quickly marked for analysis and for deletion of the task/task results.
- Terminate Analysis
- 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:

- Open – 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 – copy the selected task (see [Copy and Paste Tasks and Cases](#))
- Analyse
- Erase results – erases the results from the selected task and any associated tasks.
- Delete – deletes the selected task and any associated tasks.
- Rename – equivalent to in-place editing of the task name.
- Properties – to display a summary of this task in the [Message Pane](#).
- Check Data for Task

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

3.10.3 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 appends 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.

3.10.4 Copy and Paste Tasks and Cases

Data explorer

Data can be copied and pasted from the [Data Explorer pane](#), 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 Data explorer 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 Data explorer 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:

$ANAL \rightarrow 1 \rightarrow \text{dead} \rightarrow 1 \rightarrow L1 \rightarrow 0$

$ANAL \rightarrow 2 \rightarrow \text{imposed} \rightarrow 1 \rightarrow L2 \rightarrow 0$

$ANAL \rightarrow 3 \rightarrow \text{dead+imposed} \rightarrow 1 \rightarrow 1.4L1 + 1.6L2 \rightarrow 0$

... or a modal analysis:

$ANAL \rightarrow 1 \rightarrow \text{mode 1} \rightarrow 1 \rightarrow M1 \rightarrow 0$

$ANAL \rightarrow 2 \rightarrow \text{mode 2} \rightarrow 1 \rightarrow M2 \rightarrow 0$

$ANAL \rightarrow 3 \rightarrow \text{mode 3} \rightarrow 1 \rightarrow M3 \rightarrow 0$

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.

3.10.5 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.

3.10.5.1 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.

3.10.5.2 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**load cases**

A1

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](#).

3.10.5.3 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.

3.10.5.4 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

$$a_1 A_1 + a_2 A_2 + \dots$$

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.

3.10.5.5 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.

3.10.5.6 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. $0.0 \times$ (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.

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
$C3 = A1 \text{ to } A3$	$1(p1) -2(p3)$
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.

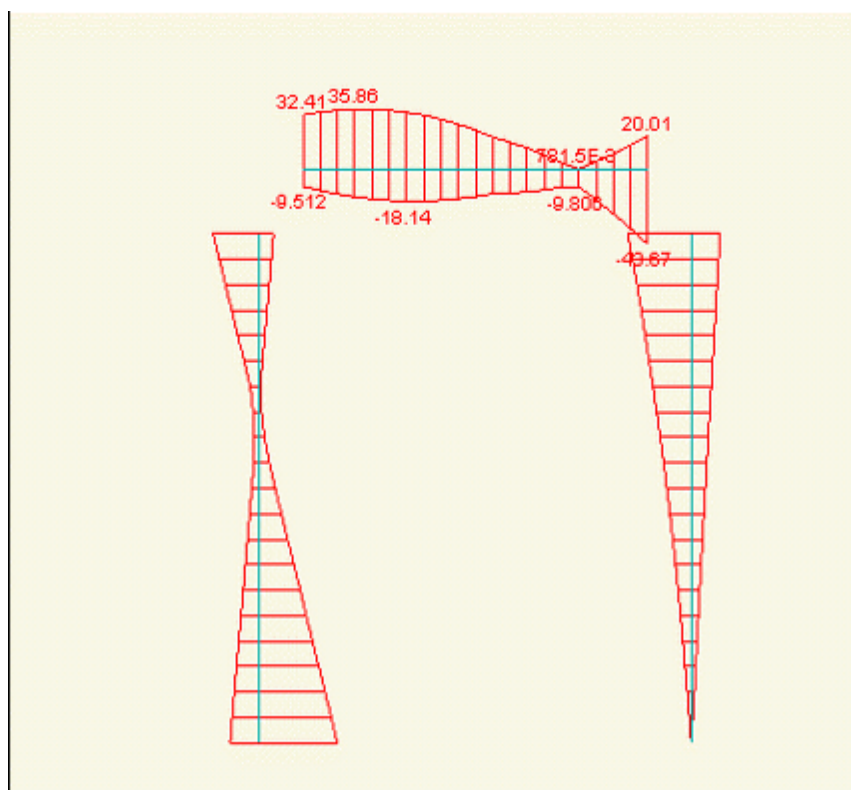
Combination examples

Examples of valid combination case descriptions:

Case	Description	Permutations
C1	$1.4A1 + 1.6A2 - A3$	(Not an envelope)
C2	$1.4A1 \text{ or } 1.6A2 \text{ or } A3$	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	(C6max or C6min) + (C7max or C7min)	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.



3.11 Grid Planes and Grid Loading

3.11.1 Specifying Grid Planes

Grid planes lie in a plane parallel to the 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.

3.11.2 Specifying Grid Loading

Grid loading is applied to grid surfaces which lie on a grid planes. Any number of grid loads can be applied to a grid surface. Note that the grid surface element list can be used to exclude elements that lie in the plane from consideration 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](#).

3.11.3 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.

3.12 Sections

The properties of a beam (or related) element are determined by the section shape and material. These are defined through a section definition and separate material definitions. In the most general case a section can be comprise of several component parts and may vary in along the length of the element. The variation along the length can be either a variation of the section dimensions, or in the case of a concrete section a variation in the reinforcement.

At present GSA does not support multiple components but does allow for varying section dimensions along the length of the section.

Sections are assigned a member type, with the basic options being beam or column. For analysis this makes little difference but there are significant differences for design, for example the reinforcement in a beam is quite different from the reinforcement in a column. When a member is defined in the GSA model the type of the member is inferred from the section member type.

The material associated with a section component is defined in two parts: a material type and a material grade, with an optional analysis material. Choosing a material type give the options of selecting a suitable material grade. So for a steel material type the steel grades are offered, and for a concrete material type the grades appropriate to the current design code are offered. For other material type various grade are also offered but mainly as a means of initialising an analysis material.

If a material type and grade are chosen the relevant properties are copied into the GSA data structure. A subsequent change to the design code will not update or delete these definitions. If updated properties are required a new grade should be chosen after changing the design code.

In the case where a material type and material grade have been selected the material properties for analysis can be inferred from the grade, so the analysis material can be set to 'from Grade' to use grade derived values. The properties picked up from the grade are:

E – elastic modulus

– Poisson's ratio

G – shear modulus

– density

– thermal expansivity

For many analyses these properties are sufficient, but if the analysis properties need to be modified, for example an elasto-plastic material, a separate analysis material can be defined.

The shape and dimensions of a section are defined with a [section profile](#). For steel and concrete sections additional [design parameters](#) can be specified.

Sections can have associated environmental parameters, such as embodied energy and CO₂. These can either be read from the environmental database or alternatively entered directly for the section. If no values are defined for the section the material values are used.

3.12.1 Section Profile

The profile defines the shape of the section.

Profile

There are three basic ways of defining this:

Catalogue – this gives the option to choose a section from the different categories in various section tables. This is intended for steel sections. For a catalogue section the profile description will always start CAT.

Standard shapes – for most concrete sections a standard section will be used. The required shape is chosen and the defining dimensions given to construct the section profile. For a standard shape the profile description will always start STD followed by a characters denoting the section shape, e.g. R for rectangular or CHS for a circular hollow section.

Geometric definition – for a section that cannot be selected from a catalogue or described by a standard shape there is an option to define the geometry more directly, either as a perimeter or as a series of line segments. The most commonly used of these is the perimeter option where the profile is defined by a series of (y,z) points. This option also allows standard bridge beams to be imported stored as perimeter sections. For a geometric section the profile will always start GEO.

Once the basic section is defined there are options to manipulate the basic definition by rotating, reflecting and translating the profile.

Reference point

By default the reference point is taken as the geometric centroid of the section. In some cases it is convenient to work with a different reference point. There are two options for modifying the reference point. The first uses the extremes of the section: top/bottom and left/right. If further adjustment s required an additional (y,z) offset can be specified directly

Tendon positions

Tendon positions are holes in the section are distinct from voids. A rectangular hollow section would be a section with a void. A tendon position hole is treated as a point area, so is only suitable for small or zero area holes (positions) such tendon positions in bridge beams.

Bridge beams

GSA has a database of precast concrete bridge beams. These are considered as a special case of perimeter sections. Rather than defining the perimeter directly the section is read from the database. In addition to the perimeter, the database holds the positions for tendons. These are also imported as point voids (holes with zero area). The user can then place tendons at the required positions in the section, based on the point voids.

3.12.2 Design Parameters

The available design parameters depend on the material type.

Steel

Parameters used in the steel design such as section net:gross area ratio, and maximum plastic:elastic ratio are defined here.

Concrete

A full description of the section requires details of the reinforcement. For a column the main reinforcement is assumed to be the same along the length of the column, so only one definition of the reinforcement is required. For a beam the reinforcement is assumed to be different at the ends from the middle, so three reinforcement sets are required. The simplest way to define reinforcement is to use the reinforcement templates. These provide a straightforward way to define the reinforcement for most commonly used arrangements. In situations where this does not provide sufficient flexibility the general option can be used which allow individual bars, or various type of lines of bars to be defined, and includes the option for pre-stressed bars.

When the template option is used the cover, links (stirrups) and aggregate are required to determine the position of the bars. All bars, and positioning checks are based on the currently selected design code.

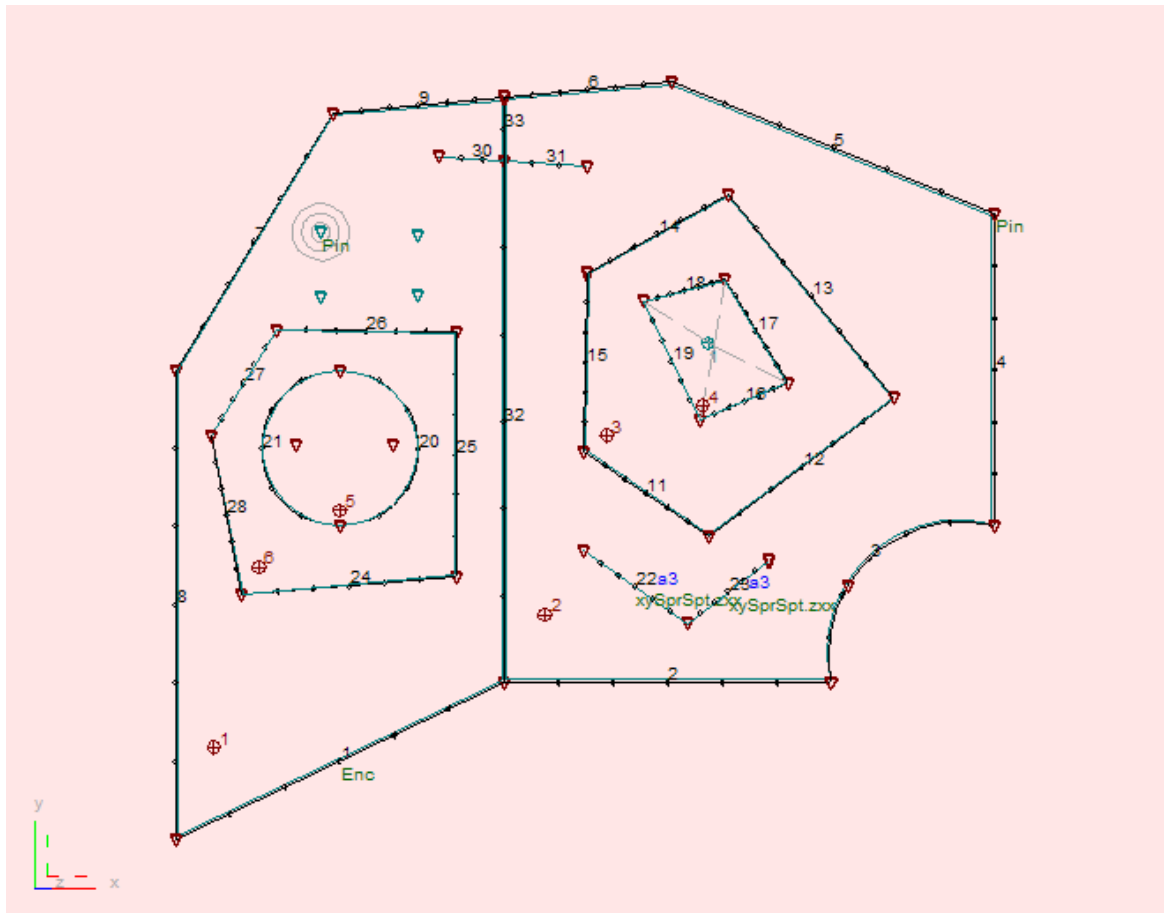
For situations where bars are spread across a large section, requiring a large number of bars to be positioned, there is an option for 'smeared' reinforcement. This assumes the reinforcement is spread evenly across the section.

3.13 Geometric Entities and 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:

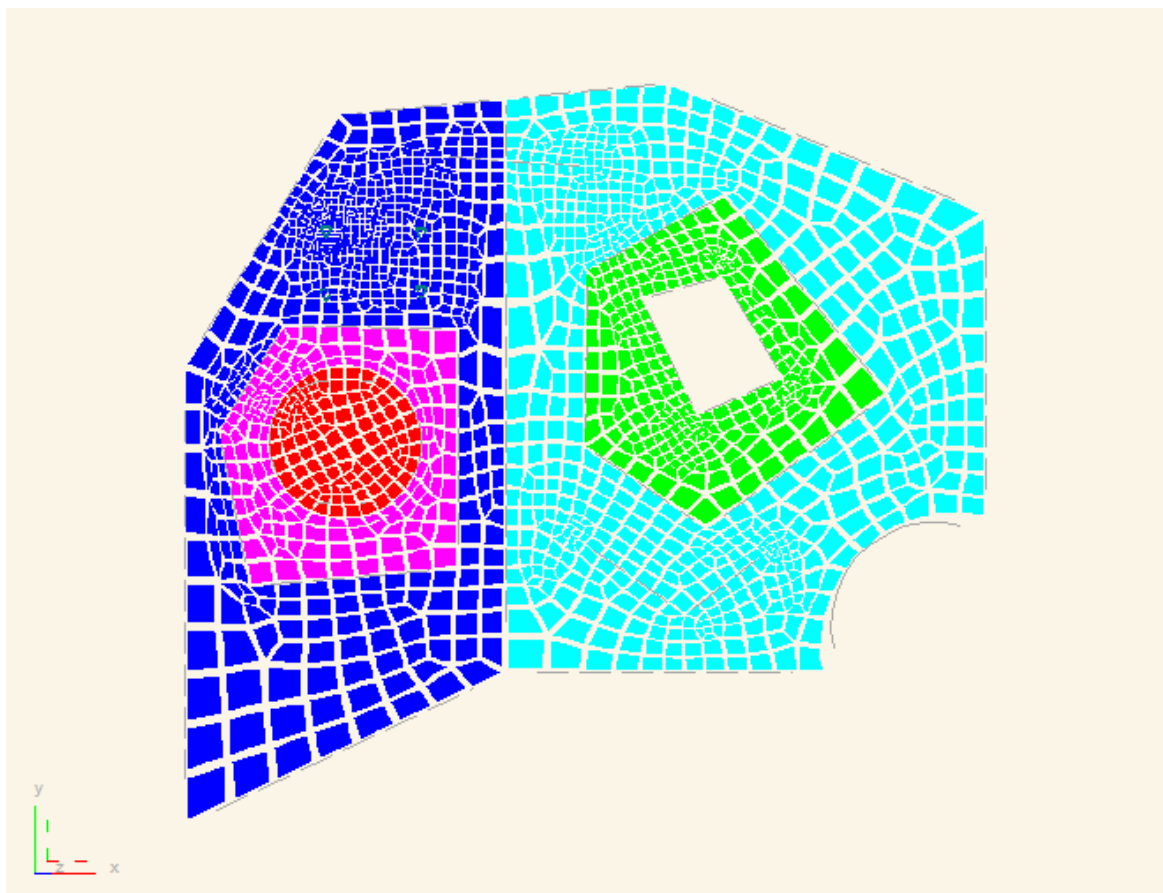
- The region is made up of two adjacent areas, [areas](#) 1 and 2. (The area selection points are represented by maroon crossed circles inside the first corner of the area.)
- These areas each have different 2D property references. Generated elements will be assigned the property reference associated 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 'Model | 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 'Model | 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 'Model | 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](#)

3.13.1 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:

- [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](#) packages, [Revit](#) and [IFC](#).

3.13.2 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.
- Think about the number of elements you want along a line. 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 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'.
- 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'.
- 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.

3.14 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. The general case of a constraint can be described by a constraint equation of the form

$$u_{s,i} = \sum_j \alpha_j u_{m,j} + \beta$$

The types of constraints consist in GSA are:

- [Restraints and generalised restraints](#)
- [Settlements](#)
- [Element offsets](#)
- [Link elements and rigid constraints](#)

- [Joints](#)
- [Constraint equations](#)
- [Tied interfaces](#)

These are discussed in more detail below.

It is important to avoid [conflicting constraints](#). This is where there is more than a single constraint on a particular degree of freedom. For example one degree of freedom on a node participating in both a joint and a rigid constraint. Note that the constraints act on degrees of freedom so a node with a joint constraining the z direction and a rigid constraint constraining the x, y and zz degrees of freedom, does not cause any conflict.

There are two situations where GSA temporarily adds constraints to the model during an analysis. With 2D elements [automatic constraints](#) can be added to account for the non-stiff degrees of freedom. The other is when the user chooses additional restraint in a modal dynamic analysis.

3.14.1 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.

3.14.2 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.

3.14.3 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.
- in a linear analysis a master degree of freedom may not be a slave degree of freedom to another constraint. (Does not apply to nonlinear analysis.)

3.14.4 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.

3.15 Miscellaneous

3.15.1 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.

3.15.2 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.

3.15.3 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.

3.15.4 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 the [Text File Format](#) document 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.

3.15.5 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 [View Explorer pane](#).

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

The views in the View List are printed in the order in which they appear in the list.

Save to file

The views in the View List are saved to file in the order in which they appear in the list. The format of the saved file is determined by the respective 'batch output file format' setting in the [Miscellaneous Preferences dialog](#).

3.15.6 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. $1000*\sin(x) + y^2$

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)
asin(x)	inverse sine (in radians)
acos(x)	inverse cosine (in radians)
atan(x)	inverse tangent (in radians)
atan(y,x)	inverse tangent of (y/x) (in radians)
sinh(x)	hyperbolic sine
cosh(x)	hyperbolic cosine
tanh(x)	hyperbolic tangent

<code>asinh(x)</code>	inverse hyperbolic sine
<code>acosh(x)</code>	inverse hyperbolic cosine
<code>atanh(x)</code>	inverse hyperbolic tangent
<code>radians(x)</code>	conversion of degrees to radians
<code>degrees(x)</code>	conversion of radians to degrees
<code>floor(x)</code>	round a number down to integer value
<code>ceil(x)</code>	round a number up to integer value
<code>sign(x)</code>	return sign of number
<code>max(x,y)</code>	maximum of two numbers
<code>min(x,y)</code>	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.

3.15.7 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)

3.16 Toolbars and Keyboard Accelerators (shortcuts)

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 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 and Dockable Panes" menu command. Toolbars can be customised by clicking the dropdown at the right or lower end of the toolbar and selecting "Add and Remove buttons" followed by the toolbar name.

[Standard](#)

[GSA](#)

[Data Options](#)

[Sculpt](#)

[Command](#)

[Labels](#)

[Diagrams](#)[Contours](#)[Orientation](#)[Cursor Mode](#)[Graphic Display](#)[Lists](#)

[Keyboard Accelerators](#) (keyboard shortcuts) give a quick route to a selection of commands. They cannot be customised.

3.16.1 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 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 and Dockable Panes" menu command. Toolbars can be customised by clicking the dropdown at the right or lower end of the toolbar and selecting "Add and Remove buttons" followed by the toolbar name.

[Standard](#)[GSA](#)[Data Options](#)[Sculpt](#)[Command](#)[Labels](#)[Diagrams](#)[Contours](#)[Orientation](#)[Cursor Mode](#)[Graphic Display](#)[Lists](#)

3.16.1.1 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

3.16.1.2 GSA



The GSA toolbar provides access to some of the main GSA options:

[Data Explorer pane](#) – show or hide the Data Explorer pane

[Message Pane](#) – show or hide the Message Pane

[Data Defaults pane](#) – show or hide the Data Defaults pane

[Show or Hide Dockable Panes](#) – Toggle on and off all dockable panes

[Analyse](#) – analyse the model

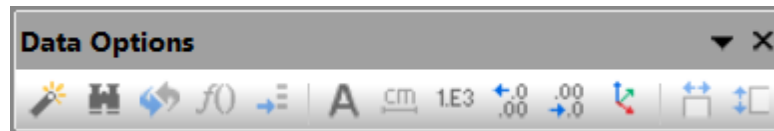
[New Analysis Task](#) – Set up a new analysis task using the wizard

[Erase Results](#) – delete all the results and optionally the analysis cases

[New Design Task](#) – Set up a new design task using the wizard

[Toolbar Options](#) – Use the dropdown arrow at the end of the bar to add or remove buttons for additional commands

3.16.1.3 Data Options



The Data Options toolbar gives access to various methods for modify the way data can be viewed or edited in [Table Views](#), [Graphic Views](#) and [Output Views](#):

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

Fonts and Styles – 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

3.16.1.4 Sculpt



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

[Drag Nodes Sculpt Tool](#) – use the cursor to drag nodes

[Add Nodes Sculpt Tool](#) – use the cursor to add nodes

[Add Elements Sculpt Tool](#) – use the cursor to add elements and nodes, if required

[Add Lines Sculpt Tool](#) – use the cursor to add lines

[Modify Entities Sculpt Tool](#) – use the cursor to modify nodes

[Create User Axes](#) – edit the user axes

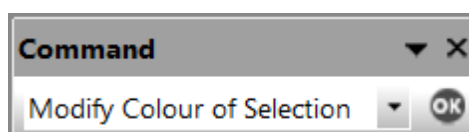
[Copy Selection](#) – move the current selection or copy to create new nodes and elements

[Add String of 1D Elements](#) – add a string of elements linking the selected nodes

[Split 1D Elements](#) – split the selected 1D elements

[Toolbar Options](#) – Use the dropdown arrow at the end of the bar to add or remove buttons for additional commands

3.16.1.5 Command

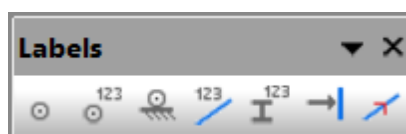


The Command toolbar is associated with the [sculpting](#) options:

Command – the sculpt command to be carried out when Execute Sculpt Command is pressed

Execute command – invoke the selected Sculpt Command

3.16.1.6 Labels



The Labels toolbar gives quick access to a number of commonly used [label](#) options:

Label node dots – display dots at the nodes

Label node IDs – display the node numbers

Label restraints – label the restraints at nodes

Label element IDs – display the element numbers

Label section shapes – display the element section shapes

Label element releases – display the element releases

Label element x axis – display the element x axes

[Toolbar Options](#) – Use the dropdown arrow at the end of the bar to add or remove buttons for additional label options

3.16.1.7 Diagrams



The Diagrams toolbar gives quick access to a number of commonly used [diagram](#) options:

All Load diagrams – display diagram of “All Loads”

[U] displacement diagrams – display diagram of “Displacements”

Fx, Fy and Fz reaction diagrams – display diagram of “Reaction Forces”

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})

2D principal stress diagrams – display diagram of principal 2D element stress

[Apply Adornments to Selection](#) – when set, apply labels, diagrams and contours to the current selection set

[Toolbar Options](#) – Use the dropdown arrow at the end of the bar to add or remove buttons for additional diagram options

3.16.1.8 Contours



The Contours toolbar gives quick access to a number of commonly used [contour](#) options:

[U] displacement contours – contour “Displacements”

[F] reaction contours – contour “Reaction Forces”

Fx axial force contours – contour “Axial Force” (F_x)

Fz shear force contours – contour “Shear Force” (F_z)

Myy bending moment contours – contour “Bending Moment” (M_{yy})

2D moment, |Mx| + |Mxy| contours – contour “2D Moment” ($|M_x| + |M_{xy}|$)

2D moment, |My| + |Myx| contours – contour “2D Moment” ($|M_y| + |M_{yx}|$)

[Toolbar Options](#) – Use the dropdown arrow at the end of the bar to add or remove buttons for additional contour options

3.16.1.9 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

[Toolbar Options](#) – Use the dropdown arrow at the end of the bar to add or remove buttons for additional commands

3.16.1.10 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

3.16.1.11 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

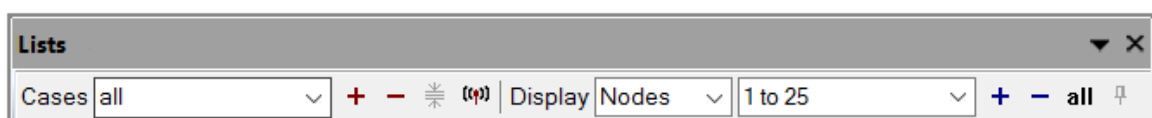
[Rescale Data](#) – rescales the data for the current diagram

[Double Size of Diagram](#) – increases the size of the diagrams

[Halve Size of Diagram](#) – reduces the size of the diagrams

[Toolbar Options](#) – Use the dropdown arrow at the end of the bar to add or remove buttons for additional commands

3.16.1.12 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 View](#) 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](#) – trim down the cases in the drop list to those where there is data

Broadcast case list – Set this case list in all Graphic Views and Output Views

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.

Pin Display – When pinned selecting a grid plane of elements also sets that as the current grid plane, and vice versa

3.16.2 Keyboard Accelerators (shortcuts)

Key	Action
Alt+0	Data Explorer pane
Alt+1	New Graphic View
Alt+2	New Output View
Alt+3	Message Pane
Alt+4	Data Defaults pane
A	Select for annotation
Ctrl+A	Select All
Ctrl+Alt+A	Reset to All Entities
B G	Colour by group
B L	Colour by pool
B M	Colour by material
B P	Colour by property
B R	Colour by material grade
B S	Colour by initial stage
B T	Colour by type
B Y	Colour by material 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 Entities 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
Ctrl+Alt+H	Shine light from here
I	Isometric
K	Skew View
L	Polyline
Ctrl+Alt+Shft+L	Add Lines Sculpt Tool
Ctrl+Alt+L	Legend display
ME	Select edges
MF	Select faces
MT	Select topologies
MW	Select whole entity
Ctrl+M	Modify
Ctrl+Alt+M	Move Selection
Ctrl+N	New
Ctrl+Alt+Shft+N	Add Nodes Sculpt Tool
Ctrl+Shft+N	Drag Nodes Sculpt Tool
O	Omit from display
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
Ctrl+Alt+W	Define current grid
X	X elevation
Ctrl+X	Cut
Y	Y elevation
Ctrl+Y	Redo
Ctrl+Alt+Y	Redo View
Z	Zoom
Ctrl+Z	Undo
Ctrl+Alt+Z	Undo View
F1	Help
Shift+F1	Context help
F2	Highlight; Rename task/case
F4	Execute sculpt command
F5	Regenerate Graphic View
Ctrl+F7	Preferences
F11	Full screen
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; Previous case
Ctrl+Page Up	Previous display

Page Down	Scroll down ; Next case
Ctrl+Page Down	Next display
↑Up	Row Up ; Up 7.5°
Shift+↑Up	Pan Up
Ctrl+↑Up	Zoom In
Alt+↑Up	Fast rotate up
←Lft	Column Left ; Left 7.5°
Shift+←Lft	Pan Left
Ctrl+←Lft	Eye Distance Out
Alt+←Lft	Fast rotate left
→Rt	Column Right ; Right 7.5°
Shift+→Rt	Pan Right
Ctrl+→Rt	Eye Distance In
Alt+→Rt	Fast rotate right
↓Dn	Row Down ; Down 7.5°
Shift+↓Dn	Pan Down
Ctrl+↓Dn	Zoom Out
Alt+↓Dn	Fast rotate down

3.16.3 Mouse accelerators (shortcuts)

The following mouse functions apply to all cursor modes. Functionality for specific cursor modes is summarised in the [cursor modes](#) article.

Rotate

- Right button drag

Zoom

- Wheel rotate
- Ctrl + right button drag up/down
- Ctrl + mouse wheel drag up/down

Zoom extents

- Ctrl + Shift + left click

Pan

- Mouse wheel drag
- Shift + right button drag

Adjust lens length (perspective views only)

- Ctrl + right button drag left/right

- Ctrl + mouse wheel drag left/right

3.17 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 **Model | Building Modelling** 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 modal 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.

3.18 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 × length × 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.

3.19 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.

3.20 Members and Element Meshes

2D and 3D meshes are generated from 2D and 3D [members](#)

Adding members interactively

To add a member in the graphical view

1. Ensure that
 - a. The Grid Layout is set to the desired spacing
 - b. The correct Grid Plane is set
 - c. The grid display is on (Draw Grid or Ctrl+Alt+G)
 - d. The graphical display is set to the Design Layer
2. Use the Add Entities sculpt tool (Ctrl+Shift+Alt+E)
 - a. Set the member type in the Properties pane
 - b. Click on the graphical view to place the member nodes
 - i. for 1D Members, hold down the Ctrl key to add a chain of members. Release Ctrl before defining the last node
 - ii. for 2D Members, finish by either clicking back on the first node or press Return

Nodes and lines can be added interactively to existing 2D members:

1. Select the member in the graphical view
2. Click the Add Entities sculpt tool
 - a. Ensure that the member type in the Properties pane matches the 2D member
 - b. To add a void: hold down the Shift keys while adding the corner points. Click back onto the first node to close the void definition.

c. To add a line: hold down the Ctrl key while adding the lines. Press Esc to exit.

d. To add nodes: hold the Ctrl + Shift keys while adding the nodes. Press Esc to exit.

Meshing tips

Here are some tips on modelling regions for mesh generation:

- Model the member to achieve the desired mesh; this is not necessarily exactly the same as a physical representation.
- Try to avoid placing internal nodes or the ends of internal elements close to member boundaries as this will produce very small elements. Nodes on the member boundary are OK.
- Model adjacent areas by having the members reference the same nodes.
- The number of elements generated along an edge will increase automatically as necessary to achieve a feasible mesh and to reduce large discrepancies in the mesh.
- 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 member is causing difficulties for the mesh generator then consider splitting the member into several adjoining members.

Part IV

4 GSA 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 dialogs and tables can be accessed from the 'Model | Data Modules' pull down menu, or from the Data explorer. Most tables have wizards which can be accessed from the 'Wizard' button on the 'Data Options' toolbar, the 'Wizard' option on the table context menu or Ctrl+W.

4.1 Titles

The titles dialog can be accessed from the Data explorer and from the "Model | Data Modules | Titles" menu command.

This sets the title information for the model.

The title entries 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.

Definition

Job Number, Initials

The job number, and initials are alphanumeric strings used to identify the job and the creator of the model. This is not used in the analysis.

Job Title, Subtitle & Calculation Heading

The title and subtitle of the job and a description specific to this particular model.

Notes

Notes on the model can be stored here. This is a text field that can be appended to as required.

Location information

The location information is used to set some country and site specific information. For most GSA model this can be ignored. The angle to north is used in some tools to identify the different faces of a building in terms of north, south, east and west. The currency used the standard ISO 4217 three-letter currency abbreviations.

4.2 Specification

4.2.1 Units Specification

GSA stores all data in SI unit, but allows the user to specify the units in which his model is defined. These become the base units for working, however these can be changed at any time.

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.

This can be accessed from the Data explorer, from the 'Model | Data Modules | Specification | Units Specification' menu command, from the [General Specification](#) 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 [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.

Units

Force

Used for force and force derived units such as moment, etc., but not for stress.

Length – large, small and section

Large length units are used for length and length derived units such as area. *Small* length units are used for data that are typically small such as displacements and cross-sectional dimensions. *Section* length units are typically used for cross-sectional properties.

Large length units are also used in units derived from length, such as moment.

Mass

Used for mass and mass derived units such as inertia, etc.

Time

Used for time and time derived units, such as frequency, etc.

Temperature

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 distinct from force and length. These units are also used for stress related quantities like the elastic modulus.

Strain

Strain is dimensionless however it is convenient to be able to work with strain units such as μ .

Velocity

Used for velocity and velocity derived 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.

Angle

It is convenient to be able to choose between degrees and radians for angle measures.

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.

Notes

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.

4.2.1.1 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, tf
- **Length - large** – m, cm, mm, ft, in
- **Length - small (displacement)** – m, cm, mm, ft, in
- **Length - sections** – m, cm, mm, ft, in
- **Mass** – kg, t, kt, g, lb, Ton, slug, kip.s²/in, kip.s²/ft, lbf.s²/in, lbf.s²/ft, kip
- **Time** – s, ms, min, h, d
- **Temperature** – °C, K, °F
- **Stress** – Pa, kPa, MPa, GPa, N/m², N/mm², kip/in², psi, psf, ksi
- **Strain** – , % , m , μ
- **Velocity** – m/s, cm/s, mm/s, ft/s, in/s, km/h, mph
- **Acceleration** – m/s², cm/s², mm/s², ft/s², in/s², g, %g, milli-g, Gal
- **Energy** – J, KJ, MJ, GJ, kWh, in.lbf, ft.lbf, cal, Btu

Also set 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.

4.2.1.2 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.

4.2.2 General Specification

This can be accessed from the Data explorer or from the “Model | Data Modules | Specification | General Specification” menu command. The analysis specification gives information about the analysis model as a whole.

Definition

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 stress** – 2D analysis option with plane stress conditions.
- **Plane strain** – 2D analysis option with plane strain conditions.
- **Axisymmetric** – 2D analysis option with axisymmetric conditions.

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. In previous versions of GSA there were also 'Plane' and 'Grid' structure types. These are no longer supported but it is straightforward to set up the appropriate global restraints.

Edge Load Specification

It can be convenient to describe edge loads as pressures or as force per unit length.

Tolerances

There are a number of options in GSA that depend on setting appropriate tolerances that apply to the whole model.

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.

Currency

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.

4.2.3 Tolerances

Definition**General**

The *node coincidence* is the tolerance within which nodes are assumed to be coincident. The *grid plane* is the tolerance with which entities are considered to belong to a grid plane. 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. *Element plane* is the angle within which elements are considered planar.

Grid Loading Tolerances

The *grid edge straightness* is used when identifying panels on a grid surface (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.

The *grid loading edge tolerance* is used 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.

The *grid plane tolerance* is used to determine if entities are 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.)

The *grid loading refinement tolerance* is used where a [grid surface](#) is multi-way spanning in 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

The *maximum difference of element orientation* angle is the acceptable difference of the orientation angles of all the elements constitute a member. The *maximum distance from the centre line* is the allowable distance of deviation of the intermediate nodes of a member from the centre line. The *offset distance from curved member* controls the difference between the arc and the chord for curved members.

Form-finding

The *bar spacer leg length* is 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.)

4.2.4 Design Specification

The design codes for both steel and concrete design, and for loading.

Definition

Steel Design

This is the code that will be used for steel member utilisation checks and design. Internal restraint rules can be established for the automatic generation of member restraints. A default steel grade can also be selected.

Concrete Design

The code that will be used for concrete beam and slab design. Superseded codes can be included and default concrete and rebar models selected for the chosen design code.

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.

Predefined [reinforcement templates](#) can be defined. These are then offered to simplify the definition of reinforcement in the section.

Load Design

Specified the design code used when generating load combinations.

4.2.5 Raft Analysis Specification

When modelling a raft supported on soil some information is needed to clarify how the soil is considered.

Definition

Rigid Boundary Level

The soil layers are defined by a top soil level. This defines the level at which the ground can be considered rigid. Typically the rock level.

Maximum Allowable E Ratio

In a soil the E value typically varies with depth through a soil profile. The E ratio determines how many sub-layers are needed within a single soil layer in the profile. The larger the ratio the fewer the number of sub layers and the coarser the soil analysis.

Mindlin Method

The Mindlin analysis method opens up a number of options:

- **Use weighted average** – In this method, the average Young's modulus and Poisson's ratio are calculated between the centroid of each load and the displacement point at which the horizontal displacement is required.
- **Use greatest stiffness** – In this method, the maximum Young's modulus that occurs between the centroid of each load and the displacement point is used for the calculations. The Poisson's ratio is set to that of the stratum where the greatest E is found.
- **Use stiffness at displacement point** – In this method, the stiffness at the displacement point is used for calculating horizontal displacements. The soil properties between the load and the displacement points are disregarded. This method should be used with caution.

Boussinesq Method

A Boussinesq analysis requires that a global Poisson's ratio for the soil.

4.2.6 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 dialogue 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) 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.
- **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 lanes 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/HK 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.

Note: As the number of analysis cases in the bridge analysis task depends on the bridge specification, the editing of bridge analysis specification will be disabled if there is a bridge analysis task.

4.2.7 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.

4.3 Geometry

4.3.1 Axes

This feature enables new user-defined axes systems to be specified e.g. to specify an inclined roller support, to assist in interpretation of results, or to specify the Construction Grid. Axes systems may be Cartesian, cylindrical or spherical.

Definition

A user axes set is created by specifying the following:

Name

Used as a convenient way of identifying an axis.

Type

Axes can be [Cartesian, cylindrical or spherical](#).

Origin

The location of this axis system in the global axis system.

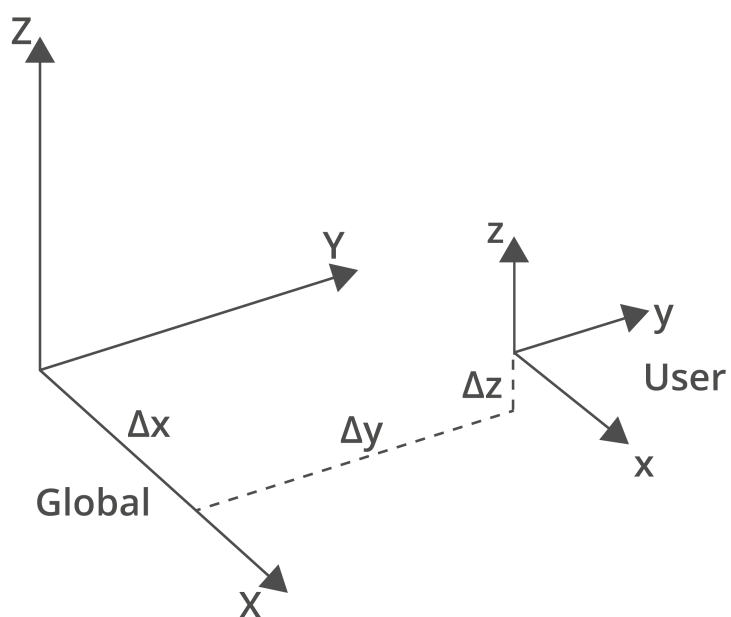
X Vector

A vector, defined in the global axis system, in the direction of the local x-axis.

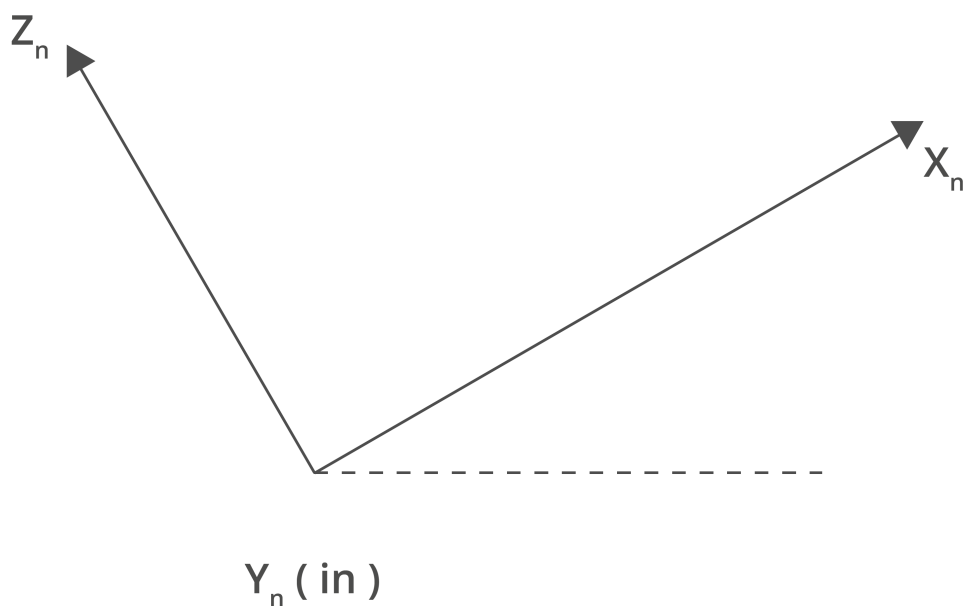
XY Vector

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.

Modify Axis

The modify axis dialog is used to modify an axis set. This dialog can be activated from the Axis Definition dialog 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.

See also [Working with the Program – Axis Sets](#).

4.3.2 Grid Lines

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.

4.3.3 Grid Planes

A grid plane defines the geometry of a surface. The load behaviour of the grid plane is defined by a [grid surface](#). The Grid Plane data table is available in the Data Explorer. The Grid Plane Definition 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](#).

Refer to [Use of Grid Planes and Grid Surfaces](#) for a general description of grid planes and their use.

Grid Plane**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.

Grid plane tolerance

The grid plane tolerance is used to determine whether an element lies in the grid plane.

Storey Tolerance below/above

Nodes are associated with the storey if they lie within these tolerances of the elevation. The different tolerance above and below are to allow for situations where there may be a step in the floor levels but these are to be considered as a single storey at the specified elevation. If the tolerance is set to "auto" then the nodes are associated with the nearest storey.

4.3.4 Grid Surfaces

A grid surface is required in order to apply grid loads to the structure. The grid surface details how the grid load is transferred on to the structure.

Definition

Name

A label for the grid surface. The name is shown when grid surfaces have to be selected.

Grid plane

The grid plane that the grid surface is defined to use.

Element list

Elements included in the grid surface. In some situations only some of the elements are to be considered to be part of the grid surface (e.g. secondary bracing) so the element list can be used to specify these.

Element type

Elements type of a grid surface specifies the type of elements that are to be considered as part of the grid surface. It can be 1D or 2D.

Grid load tolerance

This defines a volume (above and below the actual grid plane) within which elements are considered as candidates for the grid plane. It is used in conjunction with the element list to determine whether an element lies in the grid surface. A floor may use a relatively small tolerance while a roof structure may use a much larger tolerance.

Span

The span defines the way in which loading on the grid surface will be transferred from space to the surrounding elements.

- **One-way** – the load is transferred in the specified span direction. The direction is the angle to the x-direction of the axis defining the grid plane.
- **Two-way** – an area load on a complete panel. The load is expanded according to the specified algorithm and distributed in a pattern resembling the back of an envelope. The possible expansion algorithms are
 - **Legacy** – Legacy option: loads assumed to ripple out to edges of panel (does not work on re-entrant panels)
 - **Plane** – Uses a plane approximation to distribute the load and adjusts for aspect ratio (load distribution depends on model refinement)
 - **Smooth (plane)** – Uses a plane approximation to distribute the load; adjusts for aspect ratio and smooths loads along edges
 - **Corner (plane)** – Uses a plane approximation to distribute the load; adjusts for aspect ratio and smooths loads along edges, adjusting for corner effects (recommended)

- **Simplified tributary areas** option. Give simple distribution on fully loaded triangular and quadrilateral panels as shown below

Grid load expansion algorithms

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 does not work for re-entrant panels, e.g. L shaped areas.

The plane approximation options treats 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 uniformly distributed loads on the individual elements and aggregates 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.

4.3.5 Polyline

Polyline are used mainly in conjunction with grid loading to define lines or area that are loaded. Polyline are 2D entities that are interpreted with respect to the x-y plane of a grid plane.

Definition

Name

The name used to refer to the polyline.

Grid plane

Polyline can either exist in 3D space or can be associated with a grid plane

Description

Polyline are defined by a series of x-y or x-y-z coordinates. These are entered as a description of the form:

$$(x_1, Y_1) (x_2, Y_2) (x_3, Y_3)$$

or

$$(x_1, Y_1, z_1) (x_2, Y_2, z_2) (x_3, Y_3, z_3)$$

A 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 Polyline dialog.

Invalid polylines are treated as errors in the pre-analysis data checks.

The polyline definition dialog is used to define or modify a polyline. This dialog can be activated from the 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.

4.3.6 Lines

Lines are the entities that connect nodes either as straight lines or arcs to form the edge of areas.

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'.

Topology

The nodes defining the line – the third node is only required for arcs.

Radius

Only used when the type is "Arc defined by radius and point" to define the arc radius.

Mesh Generation

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:

- Region element size
- Element size
- Number of elements

Elements Size and Number of Elements

When mesh by 'element size' is selected: specify the desired element sizes at the start and end of the line.

When mesh by 'number of elements' is selected: specify the desired number of elements to be generated along the line.

Tie Meshes

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.

4.3.7 Areas

Areas are the basic entity that are meshed with 2D elements

Definition

Name

The name is used only as a label for the area.

Type

Choose from:

- Void
- Solid

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.

4.3.8 Regions

Regions are the entity passed to the meshing engine. This collects grid nodes, lines and areas for meshing.

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 be 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. The 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. The 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 an 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.

4.4 Materials

Different aspects of GSA require different material definitions, though there is a significant overlap in the properties used. To accommodate this GSA has two material definitions associated with a section – a material grade and an [analysis material](#).

The basic definition of a material in GSA is a material type and a material grade. This defines the material properties used in design.

For analysis a set of material properties are required which depend on the material model chosen. The most commonly used material model - elastic isotropic - can be constructed from the material grade properties. In this case the analysis material definition can be skipped and the analysis material set to 'from Grade'.

Separately, but related are [reinforcement grades](#) used in the definition of reinforcement for a section.

4.4.1 Material Grades

4.4.1.1 Steel

For steel design a steel material grade must be defined. Properties used in the steel design are taken from the steel material grade and not from the analysis material. The Grade Wizard should normally be used to define the steel grade properties.

Definition

The detailed properties are defined here:

Name

The name is only used as a convenient way of identifying a material.

Standard Grades

The standard grades allows selection from among a predefined set of properties.

Basic Properties

The following basic properties are specified:

- Yield strength (f_y)
- Ultimate strength (f_u)
- Failure strain (ϵ_{max})
- Elastic modulus (E)
- Shear modulus (G)
- Density ()
- Thermal expansivity ()
- Poisson's ratio ()

Section Design Properties

The following section design properties are specified:

- Strength (taken from basic properties)
- Material curve (one of linear, elastic-plastic, strain hardening, Park, explicit)
- Strain at plateau stress
- Ultimate strain

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 current environmental values for the material, whether global or overridden, are reported.

4.4.1.2 Concrete

For concrete design a concrete material grade must be defined. Properties used in the concrete section and slab design are taken from the concrete material grade and not from the analysis material. The Grade Wizard should normally be used to define the concrete grade properties.

Definition**Name**

The name is only used as a convenient way of identifying a material.

Standard Grades

The standard grades allows selection from among a predefined set of properties.

Basic Properties

The following basic properties are specified:

- Strength (f_{ck}) (subscript varies with design code)
- Density ()
- Failure strain (ϵ_{max})
- Elastic modulus (E)
- Shear modulus (G)
- Thermal expansivity ()
- Poisson's ratio ()

Section Design Properties

The following section design properties properties are specified:

- Strength (taken from basic properties)
- Confining stress (used by confined concrete models only)
- Serviceability state concrete curves for tension and compression
- Strains at peak and maximum stress
- Ultimate state concrete curves for tension and compression
- Strains at plateau and ultimate stress

Slab Design Properties

The following section design properties properties are specified:

- Strength (f_{ck}) (taken from basic properties)
- Uncracked design strength (f_{cd})
- Cracked design strength (f_{cdc})
- Tensile design strength (f_{ctd})
- Plateau strain (ϵ_{tran}) (note: this not the same as the plateau strain for section design)
- Failure strain (ϵ_{max})
- Maximum axial strain (ϵ_{ax})
- Constant stress depth ()

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 current environmental values for the material, whether global or overridden, are reported.

4.4.1.3 FRP

For design including FRP an FRP material grade must be defined. The Grade Wizard should normally be used to define the FRP grade properties.

Definition

Name

The name is only used as a convenient way of identifying a material.

Standard Grades

The standard grades allows selection from among a predefined set of properties.

Basic Properties

The following basic properties are specified:

- Strength
- Elastic modulus (E)
- Bond strain limit
- Density ()
- Poisson's ratio ()
- Thermal expansivity ()

Section Design Properties

The following section design properties are specified:

- Strength (taken from basic properties)
- Material curve
- Bond parameters
- Strains at plateau and ultimate stress

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 current environmental values for the material, whether global or overridden, are reported.

4.4.1.4 Aluminium

The Grade Wizard should normally be used to define the aluminium grade properties.

Definition

Name

The name is only used as a convenient way of identifying a material.

Standard Grades

The standard grades allows selection from among a predefined set of properties.

Basic Properties

The following basic properties are specified:

- Elastic modulus (E)
- Poisson's ratio ()
- Density ()
- Thermal expansivity ()

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 current environmental values for the material, whether global or overridden, are reported.

4.4.1.5 Timber

The Grade Wizard should normally be used to define the timber grade properties.

Definition

Name

The name is only used as a convenient way of identifying a material.

Standard Grades

The standard grades allows selection from among a predefined set of properties.

Basic Properties

The following basic properties are specified:

- Strength (f_{mk})
- Elastic modulus (E)
- Poisson's ratio ()
- Density ()
- Thermal expansivity ()

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 current environmental values for the material, whether global or overridden, are reported.

4.4.1.6 Glass

The Grade Wizard should normally be used to define the glass grade properties.

Definition

Name

The name is only used as a convenient way of identifying a material.

Standard Grades

The standard grades allows selection from among a predefined set of properties.

Basic Properties

The following basic properties are specified:

- Elastic modulus (E)
- Poisson's ratio ()
- Density ()
- Thermal expansivity ()

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 current environmental values for the material, whether global or overridden, are reported.

4.4.1.7 Fabric

The Grade Wizard should normally be used to define the fabric grade properties.

Definition

Name

The name is only used as a convenient way of identifying a material.

Standard Grades

The standard grades allows selection from among a predefined set of properties.

Basic Properties

The following basic properties are specified:

- Elastic moduli (E_{warp} , E_{fill})
- Poisson's ratio warp-fill ()
- Shear modulus (G)
- Density ()

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 current environmental values for the material, whether global or overridden, are reported.

4.4.1.8 Reinforcement

For reinforced concrete design concrete and reinforcement material grades must be defined. Properties used in the concrete section and slab design are taken from the concrete and reinforcement material grades and not from the analysis material. The Grade Wizard should normally be used to define the reinforcement grade properties.

Definition

Name and Type

The name is only used as a convenient way of identifying a material. The type allows selection between normal steel reinforcement, prestress tendons and FRP bars.

Standard Grades

The standard grades allows selection from among a predefined set of properties.

General Properties

The following basic properties are specified:

- Strength (f_y)
- Material partial safety factor (γ_m) (for selected design codes)
- Failure strain (ϵ_{max})
- Elastic modulus (E)
- Hardening parameter (k)
- Density (ρ)
- Thermal expansivity (α)
- Ductility (One of low, normal or high: this is a function of failure strain.)

Section Design Properties

The following section design properties are specified:

- Strength (taken from basic properties)
- Material curve
- Strains – at plateau stress, ultimate strain
- Bond parameters (for selected design codes)

Slab Design Properties

The following section design properties are specified:

- Strength and design strengths (f_{yk}) (taken from basic properties)
- Stress limits
- Strain limits

Miscellaneous Properties

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 current environmental values for the material, whether global or overridden, are reported.

The cost allocates a unit cost to the reinforcement material.

4.4.2 Analysis Materials

All elements that refer to [Beam Sections](#), [2D Properties](#) or [3D Properties](#) require an analysis material. This can either be an explicitly defined material or an implied material from a material grade. The latter are always assumed to be elastic isotropic materials.

The Material Wizard should normally be used to define the analysis material properties.

Explicit analysis materials

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

Young's modulus, Poisson's ratios, Shear Modulus

Elastic isotropic material - Young's modulus, Poisson's 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

$$E_x, E_y, E_z$$

$$\begin{array}{cccc}
 & * & & * \\
 yz' & & zy' & & zx' & & & xz \\
 & & & & & & & \\
 G_{xy} & & & & & & &
 \end{array}$$

* 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's 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's 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.

Initialize Analysis Material from Design Material

This command is available from the wizard. You can set the material properties either from materials in a catalogue or from material grades in the model.

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](#).

This page defines the properties used for fabric analysis. Normal materials go to a separate page.

Allow Compression

This box should be unchecked if the fabric is to allowed to behave like a true fabric and not allow compression.

4.4.3 Material Curves

In most analysis no material curve needs to be defined, but for LS-DYNA analysis and for non-linear springs material curves must be defined.

Definition**Name**

The name is only used as a convenient way of identifying a material (or spring) curve.

Type

The type defines the spring is translational or rotational.

Description

A table defining the force/displacement, moment/rotation, stress/strain relationship for a non-linear material or spring.

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 material curve, it must be in ascending order. When the data table is used in analysis, if abscissa value is beyond the definition range, the ordinate value will be taken as zero, ordinate value beyond defined range is not allowed and it will not occur in the analysis.

Diagram

The diagram shows the curve of the non-linear spring defined in the data table.

4.4.4 Material 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.

Definition

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.

4.5 Properties : Section

All beam type elements refer to a section defining the geometrical properties (e.g. area, second moment of area) referenced by the element or member through the property number. The set of sections in the model form the [Section Library](#). Individual sections in the section library can have [Section Analysis Properties](#) that are modified for analysis, e.g. reducing the J value so that the element doesn't attract torsion. Sections are then grouped in [Section Pools](#).

Basic definition

The basic definition of a section includes:

Name and Colour

These are used to identify sections and show elements or members which refer to these.

Material

The material definition breaks into three parts. The first part is the material type. This is one of:

- Steel – gives option to assign to various steel design properties to the section
- Concrete – gives option to assign reinforcement information to the section
- FRP – allows a set of properties for various grades
- Glass – allows a set of properties for various grades
- Aluminium – allows a set of properties for various grades
- Timber – allows a set of properties for various grades
- <undefined> – not associated with any grade

The grade defines the actual material design properties. These depend on the material type. New grades can be initialized from the standard grades for the current design code. The final material attribute is the analysis material. This defines a set of elastic properties. If a grade is specified it is possible to refer to this for the analysis material (from Grade). In this case the required analysis properties are inferred from the grade.

Profile, Reference Point and Point Voids

The [profile](#) defines the shape and geometry of the section. These can be either catalogue (steel) sections, standard section shapes, or shapes defined by geometry (perimeter, or line segments). By default the reference point is at the centroid of the section. In this case an axial force applied at the centroid results in

no bending. In some cases it can be convenient to change the reference point. The other change to the section geometry is to add point voids. This allows (small) ducts to be included in the section. This is also used in conjunction with bridge beams to identify locations for prestress tendons – in this case the voids are assigned a zero area.

Rate

This associates a unit cost against sections.

Modified

This is a flag to indicate that there are section modifiers associated with this section.

Pool

The section can be assigned to a pool. This groups sections for design.

Section Properties

These are the actual section properties of the chosen section, so for example a 400x200mm rectangular section would have an area of 0.08m².

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 400x200mm 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 0.16m².

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 0.16m² but the mass and gravity loads would be based on either an area of 0.08m² or 0.16m².

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.

Description

This gives a description of the section from which the section properties can be determined. The description is explained in the "[Section Profile](#)" 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 vv axes respectively.

J

The torsion constant for the section; note this is not the polar moment of inertia unless the section is circular.

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 [Shear Area]/[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.

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.

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.

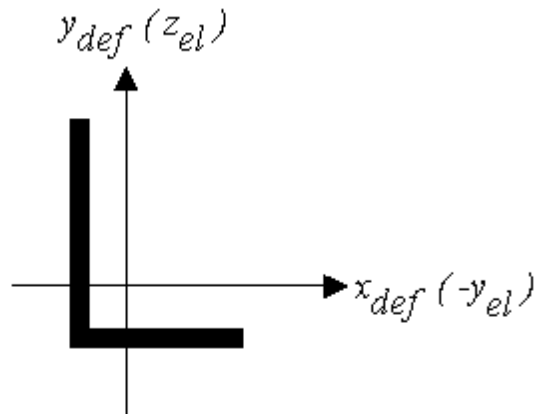
Shear action is only modelled elastically and the shear centre is assumed to be at the section centroid.

4.5.1 Section Profile

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.

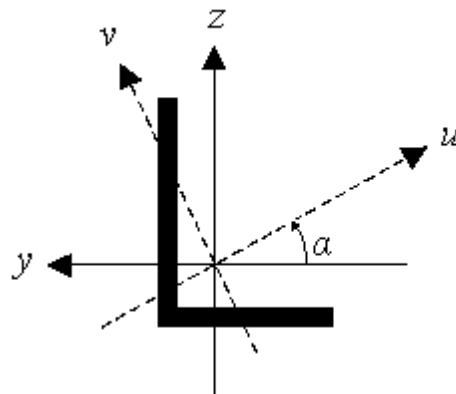
The first step is to select the basic [section definition](#) – catalogue, or standard shape, etc. Depending on this choice the profile wizard will load either [catalogue sections](#), [standard sections](#), or geometric sections ([perimeter sections](#), or [fabricated and line segment sections](#)). Finally the [defined section](#) properties are displayed and an option is given to convert the section to a perimeter.

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.

4.5.1.1 Definition

Definition

Name

The name is a label used to identify a particular section.

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.

The page that follows this will depend on the selection made at this stage.

Section Definition Axis System

The axis along a beam element in GSA is the x axis. Following this assumption the cross section is defined in a (y,z) axis system.

Section for tapered elements/members

If the section is referred to by tapered.elements/members, check box "Section tapers along element length" should be checked on the section wizard, then two different sizes of the section can be defined and the elements/members refer to this section will be tapered. Catalogue section, Fabricated section, Line segment section and Point area section cannot be used by tapered elements/members, i.e. check box "Section tapers along element length" will be always unchecked and only one section size can be defined

4.5.1.2 Catalogue Sections

The catalogue sections are stored in a section database, installed with GSA. This provides an interface to that database.

Definition

A catalogue section is defined by

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.

Transform Section

The basic section may not be in the required orientation, so there is an option to rotate and translate sections. By default this is not offered.

Catalogues

Catalogues cover sections from the UK, Europe, USA, Canada, Australia, Japan, China, India, Korea, Russia and South Africa.

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
```

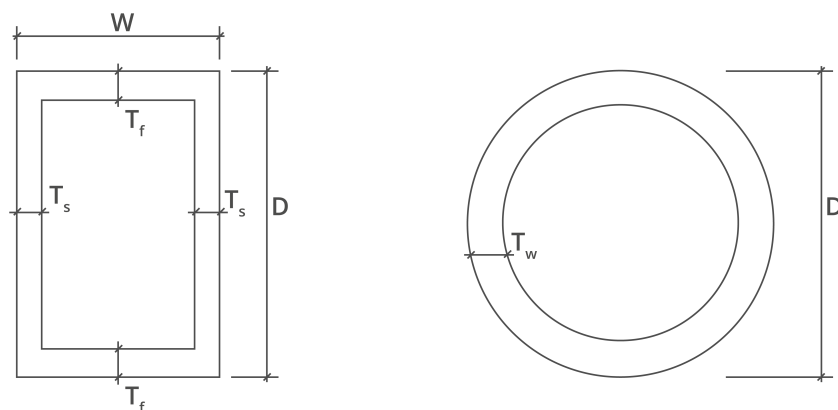
The date field is normally used only to identify superseded sections, so normally the section description would be abbreviated to

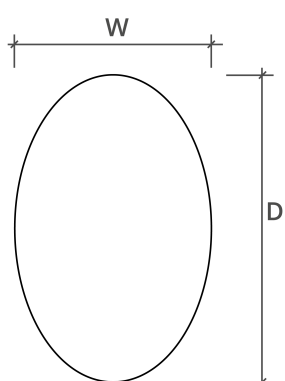
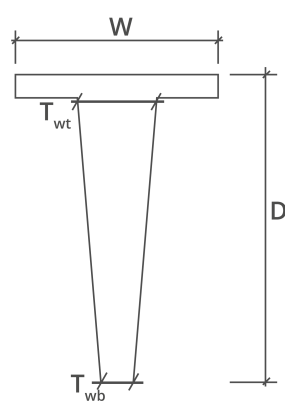
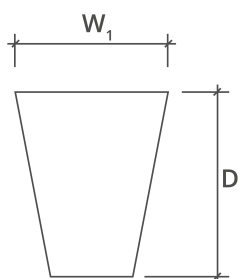
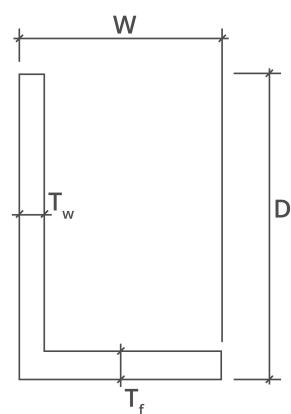
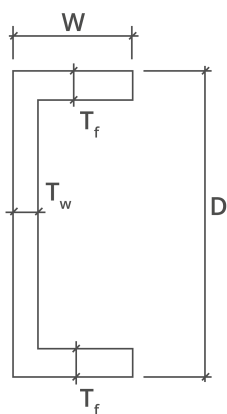
```
CAT UB 457x152x82
```

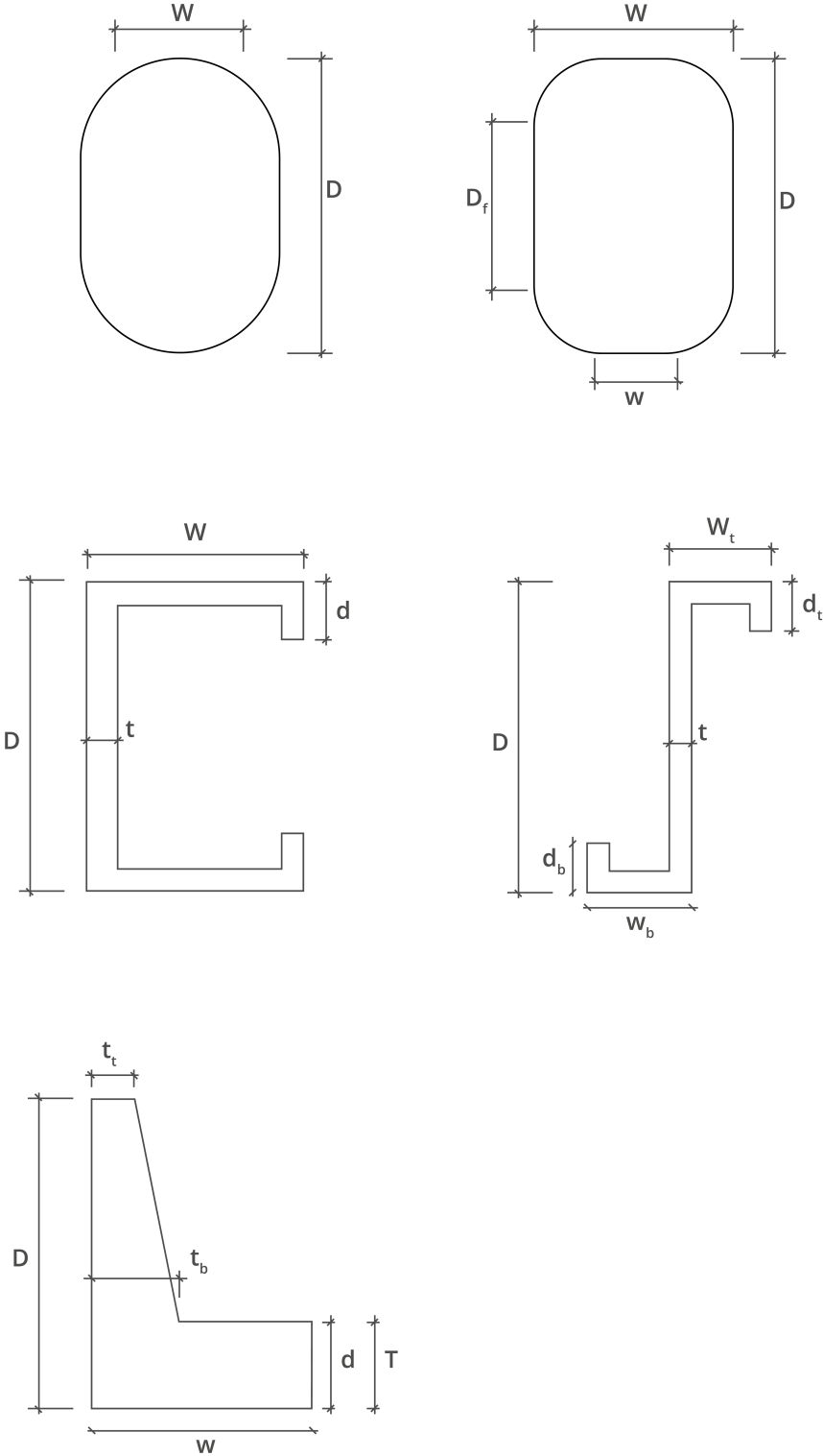
4.5.1.3 Standard Sections

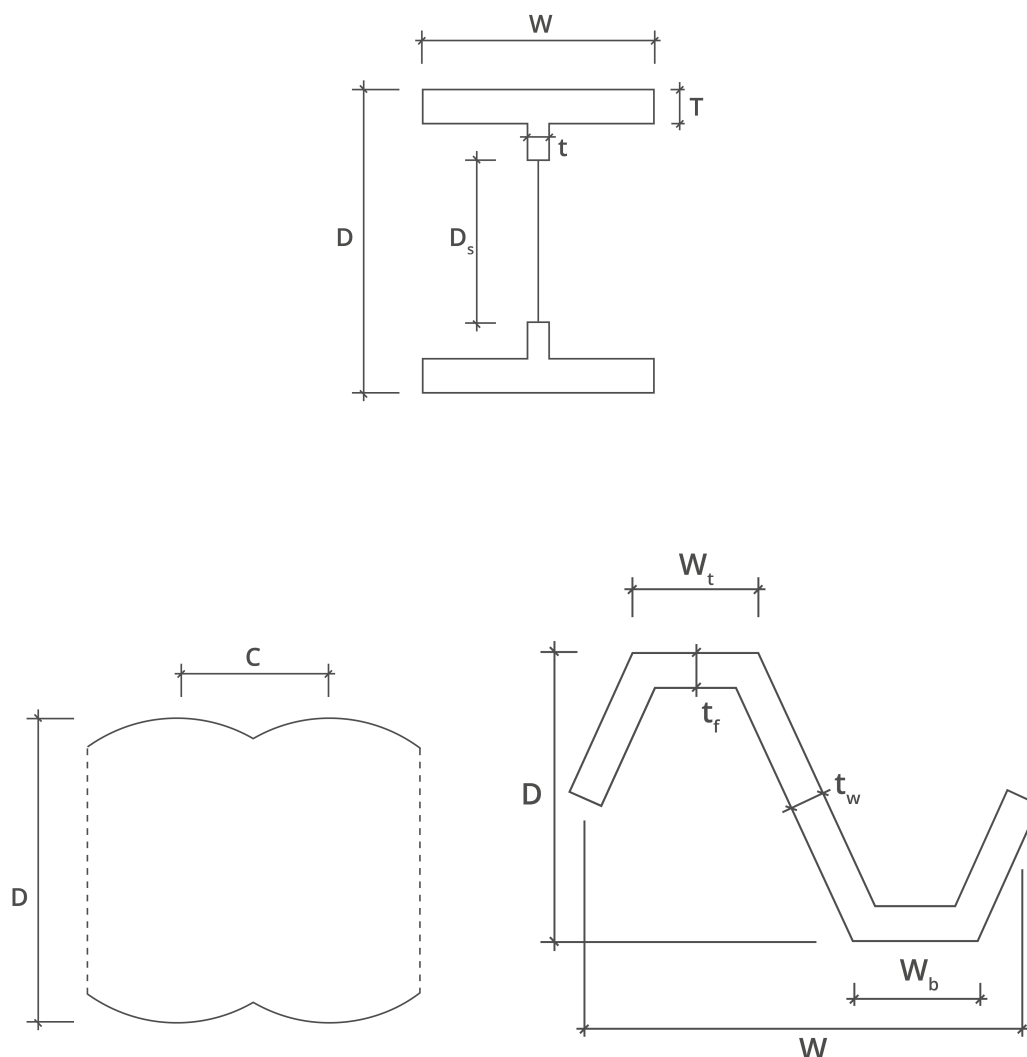
Section dimensions

Standard sections can be rectangular, circular, I, channel, T or angles, etc. 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.

Transform Section

The basic section may not be in the required orientation, so there is an option to rotate and translate sections. By default this is not offered.

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:

Section shape	Shape code	Dimensions
rectangular	R	d, b
rectangular hollow	RHS	d, b, t_w , t_f

circular	C	d
circular hollow	CHS	d, t
I section	I	d, b, t_w , t_f
tee	T	d, b, t_w , t_f
channel	CH	d, b, t_w , t_f
angle	A	d, b, t_w , t_f
taper	TR	d, b_t , b_b
ellipse	E	d, b, k
general I section	GI	d, b_t , b_b , t_w , t_{ft} , t_{fb}
taper T section	TT	d, b, t_{wt} , t_{wb} , t_f
taper angle section	TA	d, b, t_{wt} , t_{wb} , t_f
recto-circular	RC	d, b
recto-ellipse	RE	d, b, d_f , b_f , k
taper I section	TI	d, b, b_t , b_b , t_w , t_{wb} , t_{ft} , t_{fb}
secant pile section	SP	d, c, n
secant pile wall	SPW	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
```

4.5.1.4 Perimeter Sections

Perimeter 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.

Import Options

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. These are stored as perimeter sections in GSA, but have associated tendon positions stored as point voids along with the section geometry.

The shear factors, K_y and K_z are also set as zero.

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.

Perimeter 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.

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 [Section Profile](#) 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)
```

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](#)

4.5.1.5 Fabricated and Line Segment Sections

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.

Line Segment Sections

There are two types of geometric section: perimeter and line segment. Both are described in terms of 2D vertices. 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	(y, z)	M(y z)
Line to position	(y, z)	L(y z)
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 [Section Profile](#) 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 line segment section definition is of the form:

```
GEO L T(10) M(5|5) L(195|5) L(195|395) L(5|395) L(5|5)
```

[Geometric section properties](#)

4.5.1.6 Section Definition

This page summarizes the section definition and allows extra data to be specified.

Definition

Properties

This shows the values of the section properties as used in analysis (i.e. with modification factors applied). Depending on the stiffness calculation option the modifiers will apply to either the yz or uv values.

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.

More

The 'More' option displays a more comprehensive list of (unmodified) section property values.

4.5.2 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.

$$\begin{aligned}A &= \frac{1}{2} \sum (xy' - x'y) \\A_x &= \frac{1}{6} \sum (xy' - x'y)(y - y') \\A_y &= \frac{1}{6} \sum (xy' - x'y)(x - x') \\I_{ox} &= \frac{1}{12} \sum (xy' - x'y)(y^2 - yy' + y'^2) \\I_{oy} &= \frac{1}{12} \sum (xy' - x'y)(x^2 - xx' + x'^2) \\I_{oxy} &= \frac{1}{24} \sum (xy' - x'y)(2xy + x'y + xy' + 2x'y')\end{aligned}$$

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.

4.5.3 Reference Point

The reference point is the point where axial loads on a section are assumed to apply. By default this is the centroid, meaning that an axial loads produces no bending. However in some situations it is better to assume loads are applied with an eccentricity. One way to achieve this is to apply an offset to the section.

Definition

Basic Definition

The basic reference point definition allows the reference point on the section – the point at which axial loads are assumed to apply to be moved from the centroid. This means that axial loads will introduce bending into the section.

Offset

The basic reference point definition can then be further adjusted by applying an offset specified in terms of a vector.

4.5.4 Point Voids

There are a number of situation where it is useful to introduce (small) voids into the section. If these are small they can be considered as a having an area concentrated at a point.

4.5.5 Steel Design

In order to use a section in steel design, a number of steel specific parameters are required. These may vary from one steel design code to another.

Definition

The section can be one of:

- Cold formed
- Hot rolled/finished
- Welded

The other parameters related to various ratios and reduction factors associated with the section and used in the design.

4.5.6 Reinforcement

For analysis the reinforcement in concrete section is normally ignored, However for design or strength calculations the reinforcement is important.

Associated with the definition of longitudinal reinforcement are a number of other aspects of reinforcement. In particular [links](#) (stirrups), [cover](#) and aggregate. These are all used to position the bars for [template reinforcement](#). For complete control of bar positioning there is also the option of [general reinforcement](#) with allows bars and groups of bars to be placed at any location in the section. A further reinforcement option allows for the reinforcement to be considered as [smeared](#) across the section.

The final reinforcement can be any combination of template, general and smeared reinforcement.

Template reinforcement covers the most commonly used reinforcement arrangements for the selected member types. The options available will be limited by the member type.

Template reinforcement is available for [beams](#), columns - [circular](#) and [rectangular](#), [slab/walls](#), [ribbed slabs](#) and [perimeters](#).

4.5.6.1 Template Definition

Depending on the section member type different options are possible for template reinforcement. The main reinforcement applied to beam, circular columns and ribbed slabs.

Definition

Beams

In beam sections bars are specified at the top and bottom of the section. These can be individual bars or bundles of bars of specified diameter. The bars can be in one or more rows.

Columns

In circular column sections bars are specified in rings. This is the outer reinforcement. For circular hollow sections there is also the option to add reinforcement at the inner face. This can be individual bars or bundles of bars of specified diameter. One or more layers of bars can be specified at each face.

In rectangular column sections bars are specified as particular arrangements of bars.

Slabs/walls

In slab sections reinforcement is placed in layers at the top and bottom of the section.

Ribbed Slabs

Ribbed slab sections are considered to have a beam component and a slab component. The main reinforcement related to the beam part of the section. The specification of the reinforcement in beam part of a ribbed slab is the same as for a beam element.

Perimeters

In perimeter sections reinforcement is placed in layers around the perimeter of the section.

4.5.6.2 Template Main

The main reinforcement applied to beam, circular columns and ribbed slabs.

Definition

The number of bars is specified, or none if no bars at the location. These can either be single bars or bundles of bars of specified diameter. The bars can be in one or more rows or rings. By default GSA will choose a spacing between bars in layer or rings, but the user can override this.

Beams

In beam sections bars are specified at the top and bottom of the section.

Circular Columns

In circular column sections bars are specified in rings. This is the outer reinforcement. For circular hollow sections there is also the option to add reinforcement at the inner face.

Ribbed Slabs

Ribbed slab sections are considered to have a beam component and a slab component. The main reinforcement related to the beam part of the section. The specification of the reinforcement in beam part of a ribbed slab is the same as for a beam element.

4.5.6.3 Template Side

For many beams bars are introduced into the sides of the beam to control cracking. Side reinforcement allows side bars to be included in the section.

Definition

The side reinforcement can be specified either in terms of the number of bars or the maximum spacing of the bars, and a bar diameter.

4.5.6.4 Template Rectangular Column

The arrangement of bars in rectangular columns is different from the other types of member. In this case the bars are chosen from a set of predefined bar arrangements.

Definition

The basic classification is based on the number of bars. Once the number of bars is selected a varying number of arrangements are available to choose from.

The bars can either be all the same size or different bar sizes can be selected for the corner bars from the side bars.

4.5.6.5 Template Slab/Wall

Slab reinforcement is specified for both the longitudinal and transverse directions. This is split into top and bottom reinforcement.

Definition

Include Bars

The reinforcement can be excluded from either the top or the bottom of the section. If reinforcement is included it can be either single bars or bundles of bars.

Bar Size and Spacing

Separate bar sizes and spacing are specified for the top and bottom of the section. The reinforcement can be in a single layer or multiple layers if required.

4.5.6.6 Template Slab Layers

Reinforcement for slabs is defined in two orthogonal directions.

Definition

The outer layer of reinforcement is specified for both the top and bottom surfaces. The longitudinal direction along the direction of the member. The transverse direction is in the 'plane of the section'.

4.5.6.7 Template Perimeter

The perimeter option places bars around the perimeter of the section.

Definition

Both bar sizes and spacing are selected, so that bars can be placed around the perimeter of the section. These can be single bars or bundles of two, three or four bars, in one or more layers. The bars in inner layers are offset orthogonal to the local section surface where possible.

4.5.6.8 Transverse Allowance

The positioning of the reinforcement may need to be adjusted for transverse reinforcement, requiring the bars to be located further from the surface.

Definition

The transverse allowance can be specified for each face separately. The transverse allowance works in conjunction with the links to determine the location of the reinforcement.

4.5.6.9 General Reinforcement

General reinforcement allows positioning of bars or groups of bars in the section. These positions are fixed and do not update if the section profile changes.

Definition

Bar Details

The bar details allows selection of the bars. This gives a choice of bar size, and the option to bundle bars. A reinforcement grade can be selected and a prestress included. The start and end extents of the bar along the length of the member can be specified as a percentage.

Bar Positions

The next stage is to position the bars or bar bundles in the section. The section extents give extreme coordinates for bar positions. The positioning options are:

- Single – a single bar or bundle is placed at the specified location in the section.
- Line – a line of bars or bundles is placed between the start and end positions in the section.
- Arc – an arc of bars or bundles is placed between the start and end positions on an arc passing through the point defining the arc. The number of bars can either be specified directly or by giving a maximum spacing for the bars
- Circle – a circle of bars or bundles is placed on a circle determined from the centre and a point on the circle. The first bar is placed at the specified point and the remaining bars spaced around the perimeter.
- Perimeter – the bars are positioned by setting a cover. The bars are then spaced around the perimeter ensuring that the maximum spacing is not exceeded. For more than one row the bars in subsequent rows are, where possible, offset from those in the outer perimeter.
- Face – the face option allows bars to be placed relative to the selected face on the section with a selected cover. The start and end points are offsets from the start and end of the face.

For the line, arc, circle and perimeter options the number of bars can either be specified directly or by giving a maximum spacing for the bars.

4.5.6.10 Smeared Reinforcement

For complex shaped or large sections there can be many bars spread fairly evenly across the section. In such cases a simplified approach can be used which smears the reinforcement across the section.

Definition

Reinforcement

The reinforcement is specified as an area fraction, e.g. 2% reinforcement, spread across the section. Bars are not required but the reinforcement grade must be specified.

Crack Check Information

This gives information for future calculation of crack widths, etc. which are code dependent and a function of the bar size and spacing.

4.5.6.11 Links

Links or stirrups are used to determine the position of the main reinforcement in the section. The details of this depend on link material and the diameter of the links.

Definition

Link Bars

The link materials and bar sizes for links / stirrups.

Link Arrangement

Depending on the member type different arrangements of links are possible. This can cover the link shapes, where the links are included, and number of link legs.

Link Zones

The member is considered to be assigned three zones. Two end zones and a middle zone. Different link spacing can be assigned to these three zones.

4.5.6.12 Cover

The cover must be specified in order to position bars for template reinforcement.

Definition

Cover Type

In most cases the cover will be uniform on all faces, but variable cover can be specified as required. Code cover options are provided for some of the design codes which allows the cover to be selected based on the code specified parameters such as exposure.

Environmental Cover

The environmental cover is the cover required to protect the bars. This depends on the degree of exposure the section will experience.

Outer Bar Zone

In some cases the positioning of the bars need to be adjusted for bars which are running in the orthogonal direction. This allows this to be considered when positioning the bars.

Position of bars

For bars with diameter d , if the cover is denoted c , the links size l , the outer bar zone z , then the distance to the centre of the bars from the surface of the section s , is

$$s = c + \max(l, z) + \frac{d}{2}$$

4.5.6.13 Tendons at Point Voids

When a bridge beam is selected a set of (zero area) point voids are created at the positions where the tendons can be placed. This allows the tendons to be positioned at these points.

Definition

Tendon Grade and Strand

A combination of tendon grade and stand diameter should be selected. Normally there is some relation between these two parameters, but this is not enforced by GSA. The tendon grade does not prevent the selection of normal bars.

Point and Tendon

The point list details the point voids in the section. Those that are to be allocated tendons are selected and the '>' option moves these to the tendon list.

4.5.7 Section Modifiers

Normally the section properties are derived from the section geometry, but in some cases it is convenient to modify some of the parameters.

Definition

Bending Axes

The section can be considered to bend about either the local (y & z) axes or the principal (u & v) axes. So for an asymmetric section restrained by a slab it may be preferable to ignore the I_{yz} term and just have bending about the y & z axes.

Analysis Reference Point

In defining a section the reference point may be adjusted, but this can be overridden for the analysis if required.

Modifiers

Modifiers can be applied to any of the section properties. The modification can be

by – values modified by a factor

to – values modified to a value

Of significance is the special case of modifying the K_y & K_z values to 0. In this case beam elements are considered as simple beams, rather than shear beams.

Stress Calculation

Applying modifiers to the section means that the stress calculation is not straightforward, so there are

options:

- Don't calculate
- Use unmodified properties
- Use modified properties

Miscellaneous

The **clear modifiers** option returns the properties to standard properties derived from the section.

The **simple beams** option sets the shear factor to zero so that shear stiffness terms are not included in the calculation.

The **multiple sections** option is mainly intended for situations where there are several physical sections modelled as a single section property but where the properties are simply a multiple of the base properties, for example a hanger which may comprise two identical bars which act together but don't interact with each other.

4.5.8 Section Design Pools

The design pools are a way to group sections for design parameter to sections. The way in which design pools are used for steel and concrete (RC) members are quite different.

RC member design is not available at present.

For steel members there is a [steel section pool](#).

4.5.8.1 Steel Section Pools

The steel section pool allows section to be grouped for design purposes. In steel design the process is one of choosing a suitable section. If a member refers to a particular pool, it can end up with any of the sections included in that pool after the design process.

The actual pool referenced by a member will depend on the member type and material type.

Definition

Name

Pools are given a name to make it easier to work with them.

Sections

The Sections are a list of section that are included in the pool. This attribute is set for the individual sections.

4.6 Properties : General

4.6.1 Spring Properties

Springs are a general type of element which can be used to model simple spring but also more sophisticated types of behaviour. The use of [spring supports and ground springs](#) is discussed in more detail elsewhere. The two simplest and most robust types of spring are the axial and rotational springs which have only an axial or rotational stiffness respectively. The more general spring types can violate equilibrium conditions so use be used with care. General non linear springs require [material curves](#) to

define the load deflection characteristics. A completely general linear ground spring uses a [material matrix](#) to define the stiffness, but this matrix must be positive definite.

Definition

Name

The name is only used as a convenient way of identifying a spring property.

Axis

The axis used in the absence of other information to orientate a spring element. If the spring has two nodes and a non-zero length the orientation of the spring follows the rules for a beam element. If the spring is a ground spring element or the length is zero the axis is used. This may be global, vertical or a user defined axis. The x, y and z directions of the spring will follow those of the axis system.

Type

The type defines the spring is one of:

- Axial – single translational degree of freedom
- Torsional – single rotational degree of freedom
- General – six degrees of freedom, linear or non-linear
- Matrix – six degrees of freedom referring to a full 6×6 stiffness matrix
- Tension only – single translational degree of freedom, only active in tension
- Compression only – single translational degree of freedom, only active on compression
- Connector – six degree of freedom coupling different parts
- Lock-up – single translational degree of freedom, active when the spring reaches the lock up limit
- Gap – single translational degree of freedom, active when the gap closes
- Friction – three translational degrees of freedom, active when a gap closes, and generates a transverse friction force

Material Curve and Stiffness

If it is linear spring or tension only or compression only springs, springs stiffness can be defined here directly. Most spring types only require a single stiffness, but the general spring has six degree of freedom and six spring stiffness need to be defined, if type is friction, three friction coefficients need to be defined. If spring is nonlinear, the reference to the nonlinear spring curve (material curve) need to be defined here and no stiffness definitions are needed, the relationships of force-displacement and moment-rotation of the spring are defined by the referred nonlinear spring curve.

Material Matrix

For a spring defined by a matrix the stiffness characteristics are defined in a material matrix. This allows for definition of the 6×6 stiffness matrix. This can only be referenced by ground spring elements.

Lock-up

Lock up elements have no stiffness until the element locks up at the specified displacements. At this point the element behaves as a spring with the defined stiffness. The lock up for tension and compression are both defined as positive values.

Friction Coefficient

For a friction spring the transverse (shear) forces are limited to the by the friction coefficient \times the normal force.

Damping Ratio

The damping ratio is used during a dynamic analysis to calculate an estimate of the modal damping ratio.

Spring Properties

Spring elements define both conventional 'springs', but also other 1D elements which are not defined as beams or bars. The characteristics of these are described below.

- Tension only – uniaxial nonlinear spring element. This element have no stiffness under compressive loading and creates penetration without any resistance.
- Compression only – uniaxial nonlinear spring element. This element have no stiffness when subjected tensile loading/displacements. Under tensile loading this element will create gap.
- Connector – six degree of freedom linear spring element coupling two different parts. Program automatically calculate the stiffness for this element using total stiffness at connecting nodes.
- Lock-up – uniaxial nonlinear spring element, active when the spring displacement reaches the lock up limit. Similar to the Connector element, program automatically calculate the stiffness for this element using total stiffness at connecting nodes. This stiffness can be overridden by providing stiffness greater than zero in the element property definition.
- Gap – special type of compression only spring element with initial gap. Initial gap is calculated from the element nodal positions. This element will have zero stiffness in the analysis while the displacements are within the initial gap or element is subjected to tensile loading. Program automatically calculate the stiffness for this element using total stiffness at connecting nodes. By default program assigns very high stiffness. This stiffness can be overridden by providing stiffness greater than zero in the element property definition.
- Friction – Gap element with two additional translational degrees of freedom. When axial stiffness is active, this element will generate frictional resistance in transverse directions. This element will have zero stiffness in the analysis while the axial displacements are within the initial gap or element is subjected to tensile loading. Program automatically calculate the stiffness for this element using total stiffness at connecting nodes. By default program assigns very high stiffness. This stiffness can be overridden by providing stiffness greater than zero in the element property definition.

Tips for Convergence

Compression only and Gap elements

- Usually Compression only elements subjected to tensile loading requires large number of iteration to converge, if the stiffness is high.
- To identify the elements, which creates the convergence issue, start the analysis using very low axial stiffness and check for the convergence.
- Verify the results and check if the elements are subjected to tensile loading or creates gap then leave the stiffness of those elements as it is or reduce even further to increase the convergence.
- If the results of the spring elements indicates large penetration beyond acceptable limit, then slowly increase the axial stiffness for those elements until acceptable results obtained.
- In case, the non-convergence is due to rigid body motion between the two bodies replace the elements with friction elements.

Tension only elements

- Similar to Compression only elements, use very low stiffness and check for the convergence by running the analysis.
- If the results creates large gap beyond acceptable limit, increase the axial stiffness of the elements.

Friction Element

- This type of elements have the convergence issues either when they are subjected tensile loading or when these elements are subjected lateral movement.
- To avoid convergence issues due to tensile loading on the element follow similar steps defined for Compression only elements.
- To avoid non-convergence due to lateral movement, start providing transverse stiffness equal to 1/10th stiffness in normal direction or less. Increase the stiffness gradually for the elements subjected to compressive loadings.

4.6.2 Mass Properties

A mass element concentrates mass and inertia at a single point. In a static analysis (linear or non-linear), the only effect of a mass element is to give a loading based on $m \times g$ when gravity loads are applied to the structure. In a dynamic analysis both mass and inertia are used in the construction of the mass matrix

Definition**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 a positive definite matrix so 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.

4.6.3 2D Element Properties

There are three different classes of 2D elements. The basic 2D element is the normal finite elements used for structural applications included in-plane and/or out-of-plane behaviour. There is also a wall element which is defined like a normal finite element but is expanded internally in the solution to get higher accuracy, at the expense of more calculation effort. The final class is the load panel element which is a non-structural element used to apply load to beam elements.

Property Type**Name**

The name is only used as a convenient way of identifying a 2D element property.

Color

Define the color of the elements that use this property when element coloring by property on graphic view

Element 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** – plane effects only (no out of plane strain), it is only active/used when the Structure Type is "Plane strain"
- **Axisymmetric** – in plane effects only (the out of plane direction is the hoop direction), it is only active/used when the Structure Type is "Axisymmetric"
- **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, Curved shell cannot be used in GSA analysis at present, it can be used to build a GSA model and export to other software to do the analysis
- **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 large object such as a wall between two floors.
- **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.

Axis

Users may assign an axis to be used to define the orientation of 2D elements. If the axis is set to "Topological" the orientation of the element will be determined by the first two nodes of the elements and the orientation angle. If an axis (global or user defined) is assigned here then that axis will be projected on to the element plane to determine its [orientation](#).

Material

The material definition breaks into three parts. The first part is the material type. it is one of:

- Steel
- Concrete - if it is concrete, then concrete slab property can be set for RC slab design
- FRP
- Aluminium
- Timber
- Glass
- Fabric
- <undefined> – not associated with any grade

The second part is material grade index that this 2D property refers to. The material grade with actual material design properties is defined in material grade table. If material type is assigned and material grade is defined, analysis can use this material grade as the analysis material when "from Grade" is used

in "Analysis" column.

The third part is the Analysis material index that this 2D property refers to. The analysis material property can be defined in Analysis Material table. If analysis material is set to "from Grade", then the material properties defined by the material grade will be used in the analysis .

LS-DYNA Composite

When setting up a model for an LS-DYNA analysis it is possible to set up a composite material by defining a series of layers with a thickness and analysis material assigned to each layer. This cannot be used in a GSA analysis and this can only be done through the wizard..

Concrete slab property

The Concrete slab property is used in concrete slab reinforcement area calculations

Thickness

This specifies the thickness of the elements. A uniform thickness is assumed for the elements that use this property. No thickness definition is needed if material type is Fabric, also a unit thickness is assumed for plane strain and axisymmetric structure types, so thickness definition is not required.

Reference Surface

The reference surface is a virtual plane that the in-plane forces are assumed to act in a 2D element. If the reference surface is the top or bottom surface, in-plane loads will also generate a bending moment. The reference surface setting is only used by shell elements. There are two parameters to define the reference surface, "Surface" defines top, middle or bottom of the 2D elements as the reference surface, "Offset" defines an extra offset from the selected top, middle or bottom surface. If offset is zero, then the selected top, middle or bottom surface will be the final reference surface.

Property Modifiers

There are four factors, three for stiffness modifications and one for volume modification. The three stiffness modifiers are for in-plane stiffness, bending stiffness and through thickness shear stiffness, the volume modifier is for modifying the volume that is used to calculate the mass and self-weight of the elements. By using the property modifiers, the 2D element properties can be different from those calculated by the given thickness which may be useful to deal with situations such as cracked concrete slabs (reduced bending stiffness due to 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 modifiers can be an absolute values or a percentage of the calculated values from the given thickness. For Plane Strain, Axisymmetric or Fabric elements, these modifications are not relevant so they are not enabled.

The stresses in 2D elements are calculated using unmodified thickness.

Composite slab

This allows definition of a composite slab based on steel decking and concrete, this is a tool to help to setup the appropriate modifiers for the composite slab, this can only be done through the wizard.

Hollow slab

This simplify the definition of hollow slab for the analysis as this tool helps to setup the appropriate modifiers for the hollow slab, this is also done through the wizard.

Additional Mass

Normally the gravity loads on a 2D element are calculated from the density and thickness or modified volume. Fabric elements have no thickness but it may still be desirable to assign a mass, or where a slab is modeled which has a non-structural screed that adds mass. Both of these situations are provided for by the additional mass per unit area. This mass is in addition to the mass implied by the thickness and

density.

Load Panel Definition

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.

4.6.3.1 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.

Definition

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:

$$\tilde{D} = vD + (1 - v)\sqrt[3]{6D^2t - 12Dt^2 + 8t^3}$$

4.6.3.2 Composite Slab Definition

A composite slab consists of profiled decking and concrete. The properties are adjusted to account for the reduced effective thickness of slab when decking is introduced. The modifiers for the 2D properties are calculated to allow for the modified geometry and the combined steel and concrete properties.

Definition

Decking option

The decking can either be used defined where the thickness and dimensions defining the profile should be specified, or selected from a catalogue of standard decking profiles. The decking can be considered at 0° or 90° to the longitudinal direction. At 0° the concrete in the trough is assumed to contribute to the stiffness while at 90° it is ignored.

Profile

The profile description is either a reference to the selected standard profile e.g. CAT RLSD Ribdeck AL (0.9) or a string of the form 'STD D 50 0.9 300 120 140 10 40' where the numeric values related to the depth, thickness and profile dimensions.

4.6.3.3 LS-DYNA Composite Shell

When creating a model for analysis in LS-DYNA it is useful to be able to define a composite shell. This is defined in terms of layers of different materials

Definition

The composite shell is defined from the bottom up in layers. Each layer is defined by an analysis material, a thickness and an optional angle for orthotropic materials. The total thickness of the shell is determined from the sum of the layer thicknesses.

4.6.4 3D Element Properties

This defines the type and properties ascribed to each 3D Element Property Number. Properties can be edited using the "3D Property Definition".

Definition**Name**

The name is only used as a convenient way of identifying a 3D element property.

Element Type

Whether it is a solid or infinite element (infinite elements are used to model the surrounding areas in soil modelling).

Axis

Users may assign an axis to be used to define the orientation of 3D elements. If the axis is set to local the orientation of the element will be determined from the topology (point 1 and point 2 will be taken to define the x axis). 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 Analysis

The material definition breaks into three parts. The first part is the material type. This is one of:

- Steel – gives option to assign to various steel design properties to the section
- Concrete – gives option to assign reinforcement information to the section
- FRP – allows a set of properties for various grades
- Glass – allows a set of properties for various grades
- Aluminium – allows a set of properties for various grades
- Timber – allows a set of properties for various grades
- <undefined> – not associated with any grade

The grade defines the actual material design properties. These depend on the material type. New grades can be initialized from the standard grades for the current design code. The final material attribute is the analysis material. This defines a set of elastic properties. If a grade is specified it is possible to refer to this for the analysis material (from Grade). In this case the required analysis properties are inferred from

the grade.

4.6.5 Link Properties

A link is unlike other elements in that it does not have stiffness, instead it created a constraint condition between the two ends of the element. The behaviour of a link element is like a rigid constraint with just two nodes. Degrees of freedom that are not linked have no connection at all.

Definition

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

Constraints

More details are given in [Link elements and rigid constraints](#). Care should be taken to avoid [conflicting constraints](#).

4.6.6 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.

Definition

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. Used in the same way 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.

4.6.7 Damper Properties

Damper elements are only considered in the explicit non-linear solver. Including damper elements in a modal dynamic analysis gives rise to a completely different problem with complex results. This is not considered in GSA.

Definition**Name**

The name is only used as a convenient way of identifying a damper property.

Axis

The axis used in the absence of other information to orientate a damper element. If the damper has two nodes and a non-zero length the orientation of the damper follows the rules for a beam element. If the damper is a ground damper element or the length is zero the axis is used. This may be global, vertical or a user defined axis. The x, y and z directions of the damper will follow those of the axis system.

Type

The type defines the damper as one of:

- Axial
- Torsional
- General

Dashpot constant

The relevant dashpot constant must be supplied. x for an axial damper, xx for a torsional damper and all six for a general damper.

4.6.8 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.

Spacer elements are only used in form-finding analysis. They are ignored in all other solutions.

Definition

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.

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,

$$\frac{l_{i+1}}{l_i} = r$$

where r is the Spacer leg ratio ($i = 1, 2, 3 \dots$).

4.6.9 External Matrix

In some instances a stiffness matrix may have been determined outside GSA. This can then be included directly onto the structure matrices during the analysis

Definition

Node and Direction

A node and direction identifies a degree of freedom (row or column in the structure matrices), so this identifies the stiffness and mass to associate with a particular degree of freedom.

Stiffness and Mass

There is the stiffness and mass associated with the defined degree of freedom. If only static analysis is required then the mass terms can be ignored.

Note

The terms defined for stiffness and mass should not lead to non-positive definite matrices.

4.6.10 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 grades to specify values for a particular user material.

Definition

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.

4.7 Properties : Form-finding

Form-finding is an analysis which seeks to determine the shape of a lightweight structure under a set of boundary constraints. During a form-finding analysis with the solver 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.

There are two types of form finding properties: [force density 1D](#) and [force density 2D](#) for 1D and 2D elements respectively and [soap film 1D](#) and [soap film 2D](#) for 1D and 2D elements respectively.

4.7.1 Force Density 1D**Definition****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.

4.7.2 Soap Film 1D

Definition

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 prestressing 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.

4.7.3 Force Density 2D

Definition

Name

The name is only used as a convenient way of identifying a form-finding property.

Force density

The final axial stress () and the final area (A) of the 2D element that uses this property will satisfy the following equation

$$= A \times FD$$

where FD is the force density.

4.7.4 Soap Film 2D

Definition

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.

4.8 Properties : Concrete Slab

4.8.1 Reinforcement

There are two aspects to the reinforcement - the reinforcement material and the reinforcement layout.

Definition**Name**

The name is used only as a label for the slab design property.

Grades

The grade of reinforcement. This refers to a material entry in the [reinforcement grades](#) module.

Bar Direction

The bars are aligned in two directions A and B, relative to the element x axis. Normally these will be orthogonal, but the angle between the two directions can be adjusted to suit.

Axis to Surface Distance

These distances locate the position of the bars from the top and bottom of the slab. As no bar size is specified these are distances to the centre of the reinforcement.

Minimum Areas of Reinforcement

The minimum area of reinforcement to be provided in the top or bottom face. A minimum area of compression reinforcement for each face in each direction can also be specified – either according to code or a user specified value.

4.8.2 Calculation

A number of parameters which control the calculation can be adjusted.

Definition**Solution Method**

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.

Design Thickness

By default the thickness used is the thickness defined in 2D element property for the analysis. If this proves difficult to reinforce the slab, the thickness can be overridden as a trial thickness to see if a solution can be found without the need to change 2D element property thickness and redo the analysis. Once a suitable thickness is worked out, it can be used as the slab thickness in 2D element property, in this case, a re-analysis should be carried out after modifying the analysis (2D element property) thickness.

Eccentricity

The 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 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.

Two-way spanning modifier, normally 0 should be used for one-way spanning slab and 50 to 100% for two-way spanning slab, see details for how to work out the two-way spanning modifier on RC slab theory manual.

Stress/strain limits

The value of x/d_{max} will normally be that specified in the design code but can be overridden if required. The design strain limit (ϵ_{su}) limits the allowable strain in the section. This may be a function of both the concrete and reinforcement material. The crack rotation limit is specified in some of the codes, but is normally 90°.

4.8.3 Strength Reduction Factors

Design codes have one of two methods for introducing a factor of safety: either applying a factor to the material properties or a reduction to the calculated strength

Strength Reduction Factors

For codes (ACI and Australian codes) which implement a strength reduction factors there is a ϕ factor for in-plane compression and a ψ factor for in-plane tension & bending. Depending on the forces/moment on the section the ϕ value will vary between the limits for compression and tension. The actual reduction value used will depend on the actual strain plane across the slab.

Note

Typical compression and axial tension & bending reduction values are 0.65 and 0.9 respectively.

4.9 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 all attributes of nodes.

Basic Definition

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.

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](#).

The grid node definition dialog is an extension of the node definition dialog, where **extensions are highlighted**. Activating the definition dialog from the Nodes table view or Graphic View, as described above, for a node that is a grid node will open the grid node definition dialog. The grid node definition dialog can also be activated from the [Geometric Entities view](#).

The units are displayed in the top right corner of the dialog.

Name

The name is used only as a label for the node. (default: blank)

Grid coordinates

Grid coordinates are only available for grid nodes. Coordinates are specified with respect to the specified datum on the specified grid plane.

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.

Coordinates

The location of the node in the global axis system or, in the case of grid nodes, 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 each node.

Mesh generation

Mesh generation is Only available for grid nodes.

The following parameters apply to [2D element mesh generation](#):

Element edge length

The desired element edge length around an internal node of a region. (Ignored for nodes that are not internal nodes.)

Radius of influence

The radius around an internal node that the element edge length has influence. (Ignored for nodes that are not internal nodes.)

Generate column rigidity

(not yet available)

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.

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.

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.

Spring Supports

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.

See also:

[Axis Sets](#)

[Definition Axes](#)

[Generalised Restraints](#)

[Conflicting constraints](#)

4.10 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 by a [member end condition](#). Members should not be confused with elements, which are used for analysis. A member is comprised of one or several elements.

Members cannot be used for the 2D element structure types.

Each member is assigned a property number, a pool number and a group number. The property number is used to assign the section and material information in the beam section table. The pool number is used to define a set of section variants to choose from during design. The group number can be used for data management.

Steel Design sizes sections against the criteria set out in a Design Task. This is achieved by changing the section property reference in the member during design. Post-design, these references must be updated in the corresponding elements before carrying out further analysis.

Members, as opposed to elements, are suitable for coordination and round-tripping with BIM software programs. In BIM oriented workflows, the design layer containing members interacts with both BIM models as well as the GSA analysis model.

This wizard can be activated from the Members table view using the wizard button or from a Graphic View by selecting an member and then 'Edit Member' from the right-click menu. The basic [definition](#) includes geometry and topology. This leads to separate pages for end conditions and effective lengths for steel and concrete members [design properties](#). Finally [offsets](#) can be assigned.

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.

4.10.1 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](#).

Definition

Name

An optional identifier for the member.

Type

Members are assigned a type. This embraces both one dimensional (1D) and two dimensional (2D) members:

1D – Generic 1D, Beam, Column, 1D void cutter

2D – Generic 2D, Slab, Wall, 2D void cutter

The 'void cutter' member types can be used to create voids in intersecting or coplanar 2D members.

Property

These are the properties that define the type, shape and dimensions of the section or 2D properties. 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.

Topology

The geometry of the member is defined through the topology and an orientation angle. The syntax for member topology is:

- an ordered list of nodes
- node number preceded by A for arc through node

For 2D members there are options:

- V(node list) for a void
- P(node list) for points to be included
- L(node list) for lines to be included
- A(node list) for additional solid areas to be included

So a simple beam member between nodes 21 and 22 would have topology '21 22'. For a curved passing through node 23 the topology becomes '21 A23 22'.

For a simple slab defined by nodes 41, 42, 43 and 44 the topology is '41 42 43 44'. If there is a void defined by nodes 51, 52 and 53 the syntax extends to become '41 42 43 44 V(51 52 53)'. If a line between nodes 61 and 62 is to be defined and points at nodes 71, 72 and 73, the topology extends to '41 42 43 44 V(51 52 53) L(61 62) P(71 72 73)'.

For GSA 9 members defined with a radius the member is transformed to one with a third node defining the arc.

Orientation node and angle

The orientation of 1D members is controlled by an orientation angle and an orientation node. The orientation node along with the end nodes defines the xy plane of the member.

For 1D members the orientation angle defines the rotation of the members about its axis from the default orientation, so for example setting an I beam to work as an H beam. For 2D members the orientation angle relates to a rotation about an out-of-plane axis from the local x direction defined by the properties. The angle is measured in degrees.

Member end conditions

This specifies the member conditions that apply at the ends of a 1D member.

Meshing

When creating elements from members GSA needs to know how to create element meshes. Typically the meshing will take account of other members, but the members can be meshed independently or excluded altogether. The resulting mesh will be influenced by the target mesh size for the resulting elements.

Tapered 1D member

An 1D member can be tapered from one end to the next if the member refers to a section that includes two different size sections. See [section definition](#) on how to define section for tapered members.

4.10.2 Design Properties

This page is available for 1D members when the member type is steel or concrete and is defines the properties relating to design from the interaction of the member with the rest of the structure. Not all the fields are relevant for concrete members.

Definition

End conditions

Member ends allow specification of restraint conditions at the end of members. Member ends are defined in the [Member Ends](#) table.

Effective Lengths

The options to determine effective lengths are

- Automatic – uses only the [member end conditions](#) and the geometry of the structure to define the effective lengths.
- Define the internal restraint on the member – both point restraints (where another member is connected) and span restraints (such as a slab on the top flange of a beam)
- Specify the effective lengths directly

The 'All Spans' drop down provides a shortcut to set the restraint condition for all spans along the member. The Advanced option allows restraints along individual [spans](#) to be specified.

The 'All Points' drop down provides a shortcut to set the restraint condition for all points along the member. The Advanced option allows restraints along at [points](#) to be specified.

Destabilizing Load (Steel)

This is the height at which the load is assumed to act which can have a destabilizing effect on the member. By default this is relative to the shear centre, but can be relative to the top or bottom flanges.

Reinforcement (Concrete)

The members can be assigned reinforcement. This is defined in conjunction with the section. The 'show' option gives a chart displaying the member and section including the reinforcement.

Note

GSA automatically detects and registers points and spans based on the nodes and elements along the member. When a Design Task is executed, GSA automatically runs the [Generate Member Internal Restraints](#) tool to assign point and span restraints. If the 'All Spans' member restraint is set to advanced the spans restraint conditions can be specified for individual spans. Likewise if the 'All Points' member restraint is set to advanced the spans restraint conditions can be specified for individual positions along the member.

4.10.3 Miscellaneous

Where a member may have offsets these are defined here.

Members also support 'Automatic axial offset'. This option, when checked, applies offsets automatically based on the geometry of the member, and that of the member(s) that connect to it. Automatic offsets are only applied when the member has an I, rectangular, or a circular section profile. The value of the offset is set to half the section depth (or width, as applicable) of the connecting member section when the connecting angle is 90° (allowing for a tolerance).

Definition

Time line & exposure

The member can be assigned timing information. This includes the time the member is created and removed (a removal time of zero means it is not removed). For concrete members there is also the option to specify the start of drying and the age at which it is loaded. The exposed faces can also be specified.

Fire resistance

The required fire resistance can be specified as 30, 60, 90, 120, 180 or 240 minutes.

Offsets for transfer to elements

Allows offsets to be specified at the nodes. Offsets are in local axis directions. Two kinds of offset are defined

- **axial offsets** – which are in the axial direction towards the centre of the member, and
- **transverse offsets** – which apply to the member as a whole.

Automatic axial offsets will apply an offset to the end of a beam based on the shape and dimensions of the column to which it is attached.

Dummy member

This is a member which is defined, but does not participate in the design process.

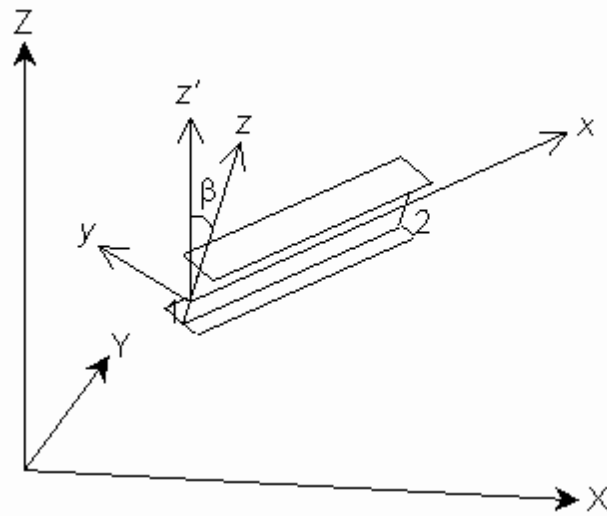
4.10.4 1D Member Axes

Member axes for linear members are defined in a similar manner to the axes of beam elements by two nodes locating the ends of the member. 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. For curved members the basic axis system of linear members holds.

The definition of the member **y** and **z** axes then depends on the member's orientation, verticality, orientation node and orientation angle. The member is considered vertical in GSA if the element is within the "[vertical element tolerance](#)".

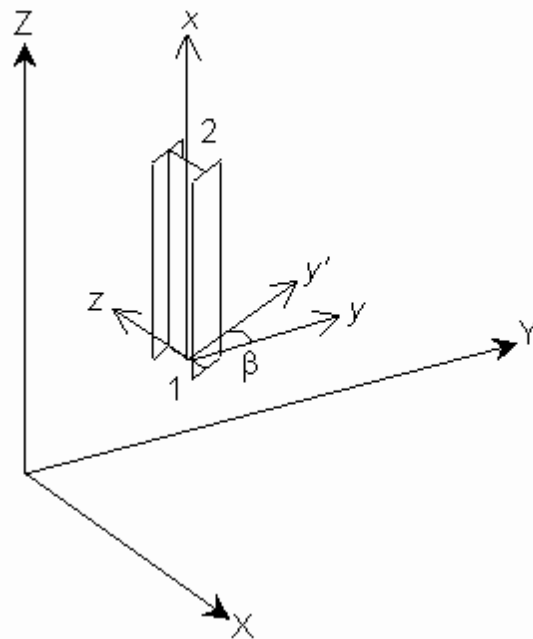
Non-vertical members

If an orientation node is not specified, the member **z** axis of a non-vertical member defaults to lying in the vertical plane through the member and is directed in the positive sense of the global **Z** direction. The member **y** axis is orthogonal to the member **z** and **x** axes. The member **y** and **z** axes may be rotated out of this default position by the orientation angle.



Vertical members

If an orientation node is not specified, the member y axis of a vertical member defaults to being parallel to and is directed in the positive sense of the global Y axis. The member z axis is orthogonal to the member x and y axes. The member y and z axes may be rotated out of this default position by the orientation angle.



Orientation Node and Angle

If an orientation node is specified, the member xy plane is defined by the member x axis and a vector from the first topology position to the orientation node, such that the node has a positive y coordinate. The member z axis is orthogonal to the member x and y axes. Specifying an orientation node overrides the "vertical member" and "non-vertical member" definitions described above. The member y and z axes may be rotated out of this default position by the orientation angle.

With an orientation angle the member y and z axes are rotated from their default positions about the member x axis by the orientation angle in the direction following the right hand screw rule. This occurs regardless of whether or not the member is vertical and of whether or not an orientation node is specified.

4.10.5 Restraints at Points

Members allow [restraint conditions](#) to be defined along their length. These restraints can either be at points or along spans. Points are marked by nodes that lie on the member whereas a span is made up of at least one constituent element along the member.

If the 'All Points' member restraint is set to advanced the spans restraint conditions can be specified for individual positions along the member.

Internal Points

Add, Delete, Modify, +, -

Buttons to add, delete and modify point restraints. Add and Modify invoke the Restraint Definition dialog for points, which then allows translational and rotational modes to be set. The 'Point' field in the Restraint Definition dialog relates the index of the node for which the restraint is being modified. For example if the internal nodes along the member are 3, 6, and 5, point 1 edits the restraint at node 3, point 2 edits the restraint at node 6 and so on.

The +/- allow the point numbers to be incremented or decremented.

4.10.6 Restraints at Spans

Members allow [restraint conditions](#) to be defined along their length. These restraints can either be at points or along spans. Points are marked by nodes that lie on the member whereas a span is made up of at least one constituent element along the member.

If the 'All Spans' member restraint is set to advanced the spans restraint conditions can be specified for individual spans.

Spans

Add, Delete, Modify, +, -

Buttons to add, delete, and modify span restraints. Add and Modify invoke the Restraint Definition dialog for spans. As with point restraints, the 'Span' field relates to the index of the span along the member.

The +/- allow the point numbers to be incremented or decremented.

4.11 Blocks

The block wizard can be activated from the Block table view using the wizard button. Blocks allow for rapid creation of models by allow replicas of groups of members to be defined.

Definition

Name

An optional identifier for the block.

Members

The members to be assigned to this block. These can be either 1D or 2D members.

Replicate

The replicate feature allows multiple copies of a member to exist in the model. The copies (replicas) remain linked to the geometry of the original member.

Offset spacings can be defined in the global x, y and z directions. This can be in the format "4 4 4" to define three replicas at 4m spacing or using the shorthand "3@4" to create the same result. The syntax types can be mixed e.g. "6 3@4 8" to expand to five replicas with spacings of "6 4 4 4 8".

4.12 Member End Conditions

In the definition of a member there is reference to condition at the ends of the member. Member ends define both end restraints for members as well as releases for the corresponding elements. The end [restraint conditions](#) are used to determine buckling effective lengths (for both Steel and Concrete members) whereas element releases form the basis for element end releases when creating or synchronizing elements from members.

Definition**Name**

The name is used only as a label for the connection.

Description

The description specified the degrees of freedom that are restrained by this connection. Individual rotational and translational terms can be restrained but the most common connections can be selected directly. These are

- **Pinned** – Top flange laterally restrained. Nominal torsional restraint. No restraint against rotation.
- **Fixed** – Encastré
- **Free**
- **Full rotational** – Top flange laterally restrained. Nominal torsional restraint. Both flanges fully restrained against rotation on plan.
- **Partial rotational** – Top flange laterally restrained. Nominal torsional restraint. Both flanges partially restrained against rotation on plan.
- **Top flange lateral**

Element release

The element release is used when mapping members to elements to provide appropriate element releases in the analysis model.

4.12.1 Steel Member Restraint Conditions

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.

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
Top Flange Warping	None		
	Partial		
	Full		
Bottom Flange Warping	None		
	Partial		
	Full		
Torsion	None	St. Venant torsion restraint	Full restraint is assumed if both flanges are laterally restrained.
	Friction/ Bearing on support only		
	Partial		
	Full		
Major Axis Rotation	None	Major (stiff) axis restraint	
	Partial		
	Full		
Minor Axis Rotation	None	Minor (weak) axis restraint	
	Partial		
	Full		
Top Flange Lateral	None		Full restraint is assumed if the bottom flange is laterally restrained and the full torsion restraint is also applied.
	Full		
Bottom Flange Lateral	None		Full restraint is assumed if the top flange is laterally restrained and the full torsion restraint is also applied.
	Full		
Major Axis Shear	None	Restraint of shear centre against movement in the direction associated with major axis bending.	
	Full		

Minor Axis Shear	None	Restraint of shear centre	Full restraint is assumed
	Full	against movement in the direction associated with minor axis bending.	if both flanges are laterally restrained.

4.13 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.

Elements can be created via the Element wizard or via the data tables.

The types of element offered depend on the [structure type specified](#). 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. 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.

Definition

Element Type

Sets the [element type](#). The element types are:

- 0D – Mass, Grounded Spring, Grounded Damper
- 1D – Beam Bar, Rod, Tie, Strut, Cable, Spring, Damper, Link, Spacer
- 2D – Quad 4, Quad 8, Triangle 3, Triangle 6
- 3D – Brick 8

Dummy Element

When checked, the element is specified as a dummy element and is ignored during analysis.

Beam Section or Element Property

This item is labelled according to the element type. It specifies 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.

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](#)
- **3D elements** reference [3D 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.

An element can have more than one property definition for form-finding analysis. 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.

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:

- mass and grounded spring elements have a single topology item;
- beams, bars, ties, struts and springs have two topology items;
- 2D elements, (depending on the element type), will have 3, 4, 6 or 8 topology items;
- 3D elements will have 8 topology items.

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".

For 3D elements the topology order is anti-clockwise when looking on the top surface of the element. The four lower corner nodes are identified first, followed by the upper corner nodes. The [local axes](#) of a 2D element will depend on the topology order unless overridden by the axis in the "2D Element Properties".

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. The angle is measured in degrees.

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.

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.

Translational and Rotational 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.

A release on an offset element is located at the offset position, not at the node.

On beams:

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 dropdowns give shortcuts 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

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.

Tapered 1D elements

An 1D element can be tapered from one end to the next if this element refers to a section that includes two different size sections. See [section definition](#) on how to define section for tapered elements.

See also:

[Element Types](#)

[Element Axes](#)

[Working with the Program – Element offsets](#)

4.13.1 Element Types

Beam Elements

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 rotational degrees of freedom are to be considered as pins it is preferable to use a bar or rod element. Offsets can be applied to beam elements where the flexible part of an element has to be offset from the node position.

Bar and Rod Elements

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.

A rod element is similar to a bar element but also includes torsional stiffness. The orientation of rods is the same as for a beam element with an orientation node and angle. Releases and offsets are not allowed.

Tie and Strut Elements

A tie element is a two-noded element with axial stiffness only. A tie will have zero stiffness under compressive forces.

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 tie and strut behaves exactly like a bar element.

Spring and Ground Spring Elements

A spring is a two-noded element in which stiffnesses are specified explicitly rather than derived from geometrical properties. The orientation of a spring is defined in the same way as a beam element with the x axis along the damper and the y and z directions defined by an orientation node and/or orientation angle.

For a zero length spring the orientation is defined by the axis specified in the spring properties.

In general a spring is defined by a stiffness, but there are variants on the springs, defined by the [spring properties](#), that allow more specialized behaviour.

A ground spring is a single-noded element that provides stiffness between the model and ground. 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.

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.

Quad and Triangle Elements

The quad and triangle elements are planar 2D elements. Quad elements can be either linear, with four nodes, or parabolic, with 8 nodes (four corner nodes and four 'mid-side' nodes). Similarly the triangle elements can be either linear, with three nodes, or parabolic, with 6 nodes (three corner nodes and three 'mid-side' nodes). For plane stress, plain strain or axisymmetric structure types these are limited to acting purely in the plane of the element, but in a space structure, these can act both in-plane and out of plane (in bending).

For the non-linear static solver, only the linear variants can be used. In this case the solver converts each quad element during the analysis into four triangular elements by creating an additional 'node' in the centre of the quadrilateral.

There are two methods for their out-of-plane formulation (Mindlin or MITC) and two methods for in-plane behaviour (Bilinear or Allman-Cook). See the [Advanced Solver Settings](#) for more details on the differences. The in-plane variants apply only to linear elements.

The parabolic elements have greater accuracy than the linear elements, but at the expense of a larger model. Quadrilateral elements provide greater accuracy than triangular elements.

Brick Elements

Brick elements are 3D elements defined by 8-nodes. They can be used for modelling any three dimensional item. They can be used in all analyses except non-linear static.

Infinite brick elements are defined by one face which then stretches in an infinite direction in an orthogonal axis.

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 applies as for a joint and equilibrium is not enforced.

See also [Link elements and rigid constraints](#).

Cable Elements

Cable elements are elements with only an axial stiffness. The way these elements work depends on the normal or sliding options.

A normal cable behaves like a tie element, only the properties are specified differently.

In a dynamic relaxation, non-linear analysis, with the sliding option, cable elements model a sliding cable.

chain. 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.

Damper and Ground Damper Elements

A damper is a two-noded element with no stiffness, so it does not contribute in static analysis. Instead it is defined by dashpot constants providing viscous damping. The orientation of a damper is defined in the same way as a beam element with the x axis along the damper and the y and z directions defined by an orientation node and/or orientation angle. For a zero length damper the orientation is defined by the axis specified in the damper properties. The damper characteristics are defined in the [damper properties](#).

A ground damper is a single-noded element that provides damping between the structure and the ground. It is specified in a similar way to a damper element, except that the orientation is always defined by the axis specified in the damper properties module.

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. 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.

4.13.2 Element Axes

For each element there is an element 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 element analysis results

The element axes are defined relative to the undeformed geometry of the structure. For linear analysis these remain unchanged during the analysis, but for non-linear analysis, element axes will be updated during the analysis according to the current deformed positions of the elements..

Beam Type Elements

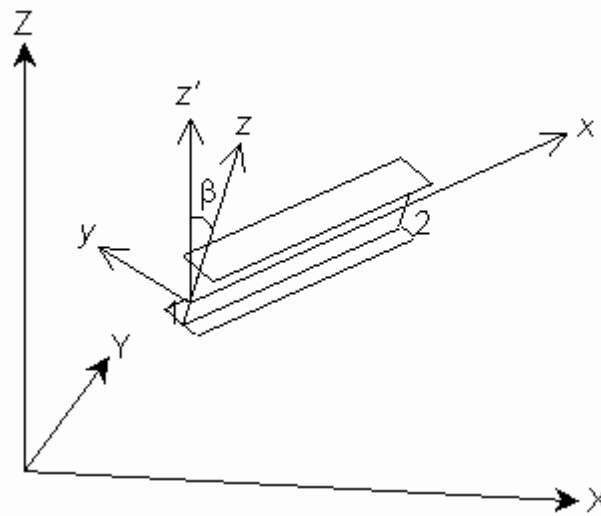
Beam elements including bars, rods, ties and struts are defined by two nodes locating the ends of the element. The **x** axis of the element is along the axis of the element (taking account of any offsets) from the first topology item to the second.

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

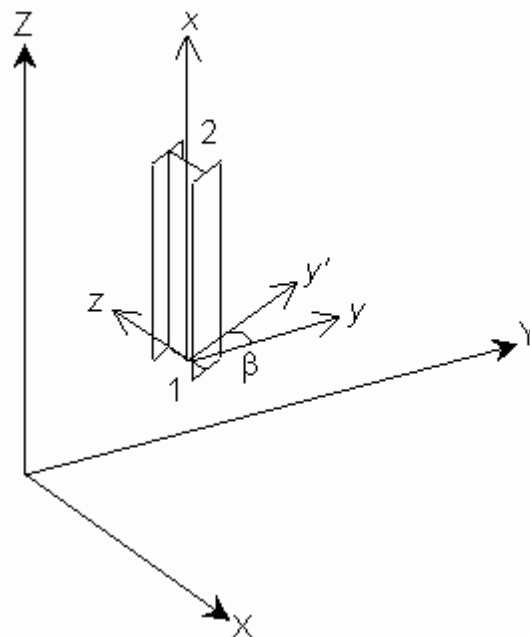
If an orientation node is not specified, the element z axis of a non-vertical element defaults to lying in the vertical plane through the element and is directed in the positive sense of the global Z direction. The

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 and Angle

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.

With an 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.

Spring and Damper Elements

For a two-noded spring or damper with finite length the element axis are defined in the same way as the [beam element axes](#). For a ground spring or a spring with zero length the axis defined in the spring or damper property is used to orientate the element. The axis can be Cartesian, cylindrical or spherical. For the cylindrical or spherical options a local Cartesian system is established at the spring or damper location.

Mass Elements

The definition of the element axes for a mass depends on the axis system set in the mass property. User defined axes can be Cartesian, cylindrical or spherical. For the cylindrical or spherical options a local Cartesian system is established at the spring or damper location.

Cable Elements

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](#).

Link Elements

The local axes of link elements are the same as the constraint axis of the master node.

Quad and Triangle Elements

2D element local axis is the axis referenced by the element stiffness and forces, moments and stresses, the xy plane of the 2D element local axis is always co-planar with the element plane. 2D element local axis can be defined by global axis or a user defined axis or by the nodes defining the 2D element (topological), the axis used by 2D element to determine its local axis is defined in 2D element property. User defined axes can be Cartesian, cylindrical or spherical.

Typically using a global axis or a user defined axis to define 2D element local axis can result in consistent local axis direction for all the elements in the mesh.

2D element local axis defined by element topology

If "topological" is used in 2D element property, 2D element local axis is defined as follows: The z axis direction of the 2D element local axis is always perpendicular to the 2D element plane and it is defined as

$$\mathbf{n} = (\mathbf{c}_3 - \mathbf{c}_1) \times (\mathbf{c}_4 - \mathbf{c}_2) \quad \text{for quad elements}$$

$$\mathbf{n} = (\mathbf{c}_2 - \mathbf{c}_1) \times (\mathbf{c}_3 - \mathbf{c}_1) \quad \text{for triangle elements}$$

Then the three axes of the 2D element local axis are defined as:

$$\mathbf{x} = \mathbf{c}_2 - \mathbf{c}_1 \quad \mathbf{y} = \mathbf{n} \times \mathbf{x} \quad \mathbf{z} = \mathbf{n}$$

where $\mathbf{c}_1, \mathbf{c}_2, \mathbf{c}_3$, & \mathbf{c}_4 are the coordinates of topology nodes 1, 2, 3 & 4 of the element, the coordinates of the node should include any offset.

2D element local axis defined by global or user defined axis

If global or user defined axis is used in 2D element property, the x axis of 2D element local axis is the

projection of the control vector "**X**" onto the 2D element plane and the other two axes of the 2D element local axis can be determined accordingly as shown below.

The control vector "**X**" depends on the type of the user defined axis,

If it is Cartesian, the control vector "**X**" is the same as the x axis of the Cartesian axis,

If it is Cylindrical, the control vector "**X**" is the projection of vector **R** onto the xy plane of the cylindrical axis, the vector **R** is the vector from the origin of the cylindrical axis to the element center.

If it is Spherical, the control vector "**X**" is equal to the vector from the origin of the spherical axis to the element center.

If the control vector "**X**" is perpendicular to the 2D element plane, then the **vector** normal to the plane defined by the control vector "**X**" and the z axis of the 2D element property axis will be used as the control vector "**X**". For Cartesian axis, the new **vector** will be the same as y axis of the 2D element property axis.

Once the control vector **X** is known, the 2D element local axis can be calculated as follows:

$$\mathbf{y} = \mathbf{n} \times \mathbf{X} \quad \mathbf{x} = \mathbf{y} \times \mathbf{n} \quad \mathbf{z} = \mathbf{n}$$

x, y & z are the three axes of the 2D element local axis.

If an orientation angle is defined, the calculated 2D element local axis is rotated by the orientation angle in a positive direction about the element z axis.

Brick Elements

3D 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 3D element property. If the axis system is 'global' or user defined then the specified axis set is used. If the axis system is 'topological' then the local axes are based on the element topology. User defined axes can be Cartesian, cylindrical or spherical.

Typically defining 3D element local axes by reference to an axis set results in more consistent local axes in the mesh.

4.14 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 an assembly does not have a clearly defined 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.

Entity

Assemblies can be defined in terms of members or elements. Only elements related to the member list or element list are considered. Spring and damper elements are ignored.

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.

Dimensions

y and z dimensions can be specified for the assembly. If these are specified the assembly is a rectangular prism centred on centreline defined by the topology. Only the elements, or parts of elements inside this prism are considered in the force and displacement calculations. Setting the dimensions to zero means that no clipping occurs.

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

This is how the assembly is defined internally (as used for assembly forces and moments). There are four 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 (e.g. 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.

Assemblies

Assemblies are a 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.

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 'Model | Investigate Model | Check Assemblies' tool checks all assemblies for missing topology items, and invalid geometry configurations.

Wall elements are excluded from the assembly calculation. Convert wall elements to 2D members to

avoid this problem.

4.15 Constraints

4.15.1 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. The generalised restraints and [nodal restraints](#) work together to apply restraint to the model. Care must be taken to avoid [conflicting constraints](#).

Definition

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).

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).

Restraint Hierarchy

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.

The total restraint on a node is the sum of all of the restraints acting on it.

4.15.2 Rigid Constraints

Rigid constraints define sets of nodes constrained to move as a rigid body. These are commonly used to model rigid diaphragms, where typically the nodes are rigid in the x-y plane but not in the z direction.

Rigid constraints are similar to link elements except that the number of nodes is not limited. The differences between [link elements and rigid constraints](#) is discussed elsewhere.

Unlike [joints](#) rigid constraints maintain equilibrium in the system.

Definition

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).

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. The master node may be included in the node list.

Type of Linkage

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
all (pin)	x, y, z
xy plane	x, y, zz
yz plane	y, z, xx

zx plane	x, z, yy
xy plane (pin)	x, y
xy plane (pin)	x, y
yz plane (pin)	y, z
zx plane (pin)	x, z
x plate	x, yy, zz
y plate	y, xx, zz
z plate	z, xx, yy
x plate (pin)	x
y plate (pin)	y
z plate (pin)	z
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.

4.15.3 Joints

Joints allow nodes to be tied in the specified translational or rotational directions. Unlike [rigid constraints](#), joints do not impose equilibrium on the model.

As with all constraints, care must be taken to avoid [conflicting constraints](#).

Definition

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(s).

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.

Joint Limitations

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.

4.15.4 Constraint Equations

Constraint equations allow a node, in a particular direction to be constrained relative to a set of other nodes. Constraint equations are the fundamental building block for all other constraint types.

Definition

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).

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.

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.

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:

$$u_{s,i} = \sum_m a_m \times u_{m,j}$$

So for a node whose displacement is the average of two other nodes this is of the form

$$u_{2,x} = 0.5u_{0,x} + 0.5u_{1,x}$$

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.

4.15.5 Tied Interfaces

Tied interfaces allow two sets of elements that do not have connectivity to be joined without the need for complex mesh refinement. Instead the elements on one side are taken as masters and the other side as slaves. The nodes on the boundary of the slave side are tied to the nodes on the boundary of the master side through a set of constraint equations.

As tied interfaces do not give proper continuity conditions across the boundary they should not be used in area where accurate determination of stresses is required.

Definition

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).

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 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 represents 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.

Tied Interface

A tied interface works by connecting the nodes associated with the slave elements to the elements on the master side to form a continuous entity.

A tied interface is a way of joining two meshes with dissimilar mesh densities by establishing a set of constraint equations to “stitch” the nodes on the slave side to those on the master side. The 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.

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). The master retains all its degrees of freedom.

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.

4.16 Loading

GSA provides a number of different ways to apply loads to a model. The simplest option is [nodal loading](#) where forces are applied directly to nodes. This is not recommended for 2D and 3D elements. The next level of loading applies loads to the elements, either [beam loading](#), [2D element loading](#) or [3D element loading](#). In the solver these use shape functions to give loading on the nodes compatible with the elements to which the load is applied. [Grid loading](#) is a different type of loading which is applied to a grid surface. An algorithm then distributes this loading from the grid surface to the surrounding beam elements. This is useful for models where floor slabs are not modelled explicitly.

[Gravity](#) is the final load type. This is different from the other load types as it is specified as an acceleration (in g). This is normally used to model the dead weight of the structure by specifying a gravity load of $-1 \times g$ in the z direction.

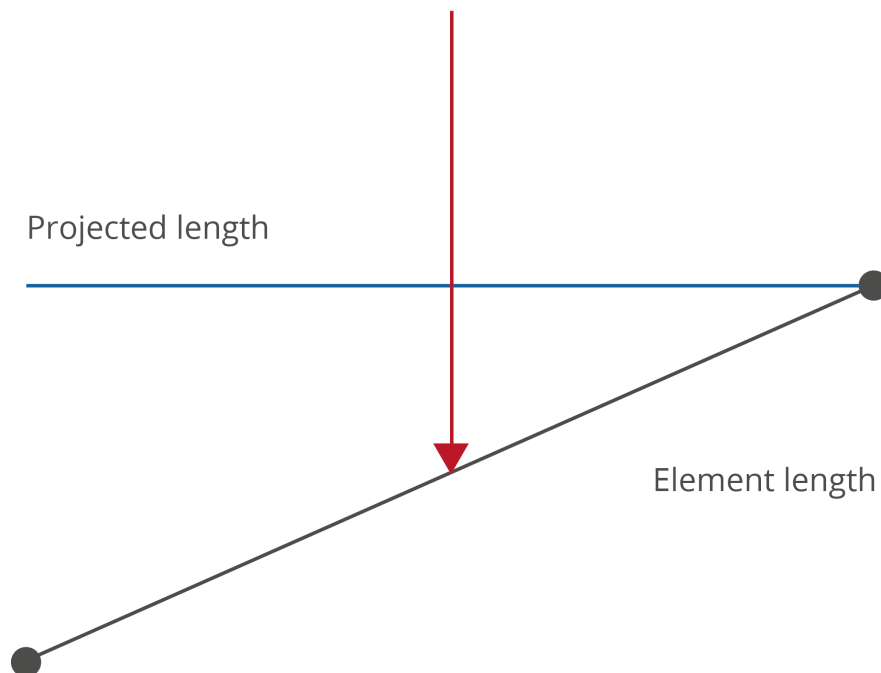
Selecting entities for loading

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](#).

Projected Loads

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.



Locking of Loads

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.

4.16.1 Load Case Specification

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.

Definition

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:

[Working with the Program – Load Cases](#)

4.16.2 Nodal Loading

Node Loading

Three type of nodal loading are possible in GSA. [Node loads](#) are the most direct type of loading and apply a force or moment directly to a node. [Applied displacements](#) provide a method of determining the state of the structure given a set of fixed displacements at selected nodes. [Settlements](#) work with constraints to also provide a way of specifying displacements at specified nodes, but in this cases these are unaffected by any other loads on the structure. Noded to which settlements are applied must be restrained in the settlement directions.

Load case and 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.

4.16.2.1 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.

Definition**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.

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.

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 – either a force or a moment.

4.16.2.2 Applied Displacements

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 are different to the other load types and are also distinct from [settlements](#).

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 other loading in the same analysis case.

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.

Definition**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.

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 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 or rotation.

4.16.2.3 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 as the settlement is in the node constraint axis.

Definition**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.

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 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 – displacement or rotation.

Settlements and Constraints

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.

4.16.3 Beam Loading

Beam Loading

Beam loading applies to beam, bar, tie and strut elements and to a more limited extent to cable elements. There are a number of ways in which loads can be applied but in all cases the loading is resolved during the solution 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%).

The most commonly used option is the [beam loads](#) which applies loads at specified positions along the elements.

[Prestress](#) allows prestress loads to be applied to elements. Beam [distortions](#) are a different and little used option which allows an element to have a displacement discontinuity along the element. This is only valid for linear elements. The final load type for beam element is [thermal](#) where temperature or temperature gradients can be applied to the elements.

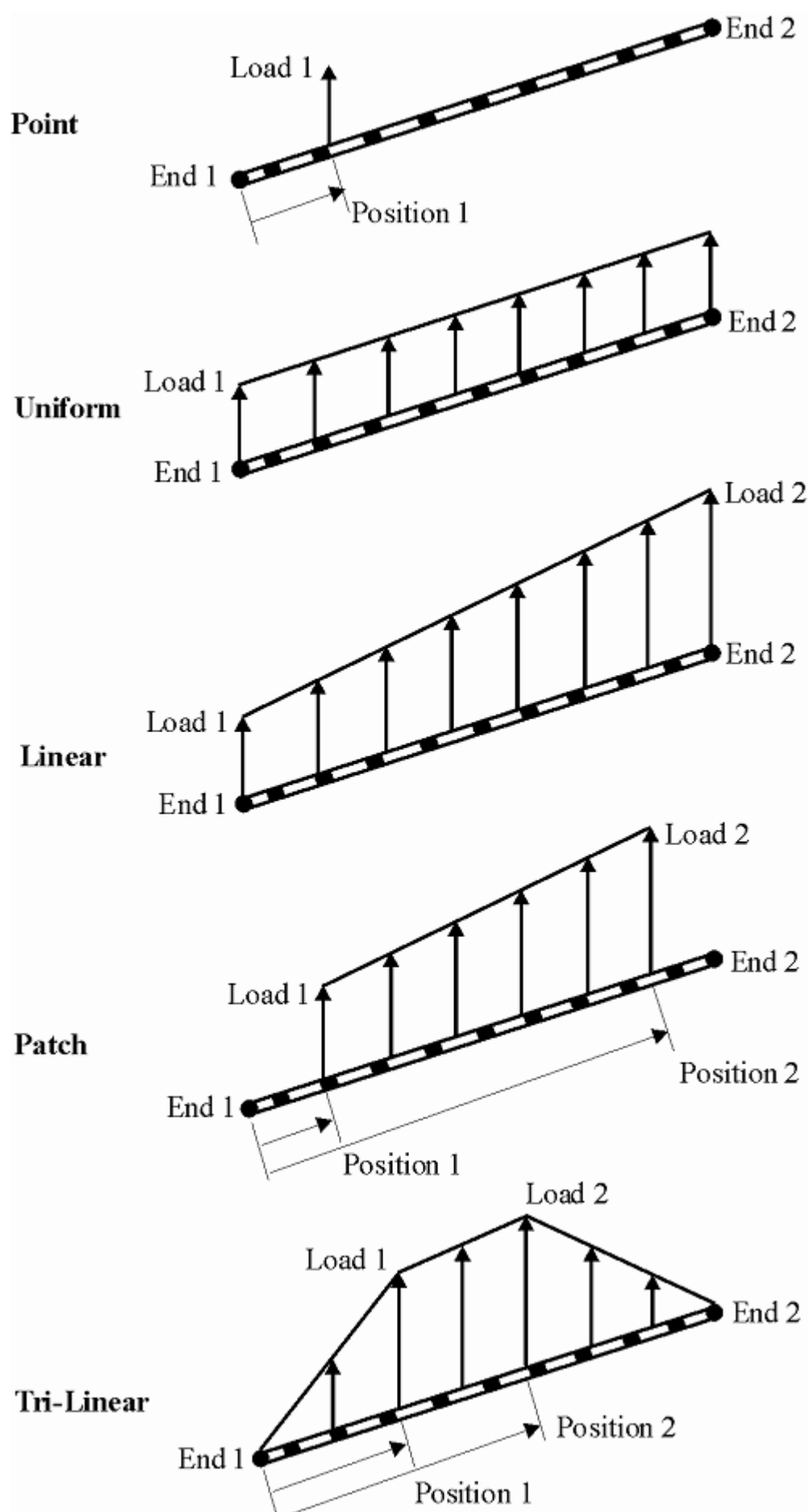
Load case and title

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.

4.16.3.1 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

$$W \times \cos$$

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.

Definition

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.

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

The [load case](#) in which the load applies. The load case gives a way of grouping load effects together.

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

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

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, with respect to the specified axis.

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%.

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.

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.

Beam loads in non-linear static 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.

4.16.3.2 Pre-stress

Prestress is a general description covering prestress loads, initial strains and initial lengths. In all cases the result is a prestress condition (set of forces and/or moments) in the element.

Definition

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.

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

The [load case](#) in which the load applies. The load case gives a way of grouping load effects together.

Type

The user has a choice of the type of loading applied to the elements.

- **Prestress 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 prestress types are related by the relevant element's stiffness and length alone. A positive prestress 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 Prestress** – Concrete elements are commonly prestressed 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 prestress 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 prestress applied would show a negative axial force despite having no load applied to the tip.

Prestress Force & Tendon Prestress

The magnitude of the prestressing load. This applies only to the prestress force and tendon prestress load types. A positive force implies a tensile prestress force or a tensile tendon prestress.

Prestress Offsets

The offsets in local (y, z) directions of the tendon prestress. This allows for the introduction of prestressing moments in addition to prestressing force. This applies only to the tendon prestress load type and can only be applied to beam elements. Note that the prestress offset cannot be used in the static non-linear solver.

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.

4.16.3.3 Distortion

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.

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.

Definition

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.

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

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, with respect to the specified axis.

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 applied to the element – either a displacement or rotation.

4.16.3.4 Thermal

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.

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:

$$\frac{dT}{dp} = \frac{T_{top} - T_{bot}}{p_{top} - p_{bot}}$$

This temperature gradient applies to the section as a whole.

Definition

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.

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

The [load case](#) in which the thermal load applies. The load case gives a way of grouping load effects together.

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

The temperature rise relative to ambient applied to the whole element.

Lower Surface, 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, Temperature

This defines the location and temperature rise that applies in the positive y or z directions relative to the neutral axis.

4.16.4 2D Element Loading

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.

[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. [Prestress](#) loads can be thought of either as prestress forces or tendon prestresses applied to the element or an initial strain. [Thermal](#) loads can be applied to consider either a uniform temperature rise or a thermal gradient on the element.

Load case and 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.

4.16.4.1 Face 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. Face loads can be applied in local, global or user defined axis directions. For face loads local is interpreted as element axis 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 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

$$W \times \cos$$

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.

Definition

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.

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

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 non-linear 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 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

- **Point** – force can be applied at a point on the face of the elements. The position of the point load on the face can be specified in (r, s) coordinates. (Based on two-dimensional shape function)

If the elements to which this relates are load panels then the load type must be uniform.

Direction

The direction in which the load applies, with respect to the specified axis.

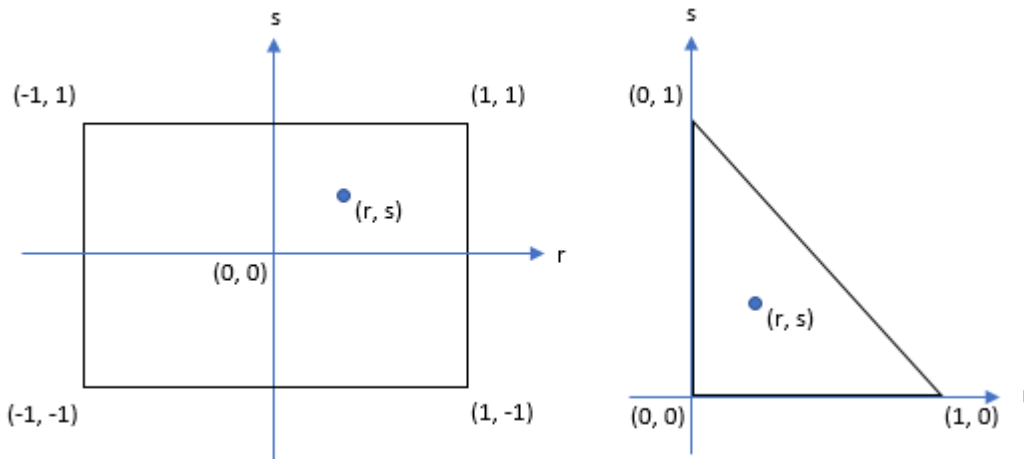
Pressure

The pressure to apply to the element. For point type load, force at the point to be specified (not pressure). 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.

Position of load

Position of load is applicable for point load type. The position of the point load to be specified in (r, s) coordinates based on two-dimensional shape function.

- Coordinates vary from -1 to 1 for Quad 4 and Quad 8.
- Coordinates vary from 0 to 1 for Triangle 3 and Triangle 6.



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.

4.16.4.1.1 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.

- **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



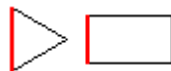
- **Two edges supported** - the load spans from the sides adjacent to the reference edge.



- **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. If the bounding beam/bar elements are dummy elements these elements cannot take load so an appropriate support condition should be selected to ensure that no load is attributed to the dummy elements.

For elements with rigid offsets the basic load distribution is modified to avoid loads on the rigid offsets.

4.16.4.2 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.

Edge loads can be applied in local, global or user defined axis directions. 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.

Definition

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.

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

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 non-linear analysis options.

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	4nono	1

Direction

The direction in which the load applies, with respect to the specified axis.

Stress at Ends

The stress to apply to the element. The stress along the edge is interpolated from the end values.

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.

4.16.4.3 Pre-stress

Prestress loads can be thought of either as prestress forces, tendon prestress applied to the element or an initial strain. The tendon prestress option allows for the prestress to be offset by a constant distance from the centre of the element in the thickness direction to induce bending.

The prestress or initial strain is applied in either the local x or y direction or in both directions.

An element with a positive prestress tends to contract, as in a prestressed slab.

Definition

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.

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

The [load case](#) in which the load applies. The load case gives a way of grouping load effects together.

Type

The user has a choice of the type of loading applied to the elements.

- **Prestress force** – an in-plane force applied to the element
- **Initial strain** – an initial in-plane strain is specified in the element
- **Tendon prestress** – an in-plane force is applied to the element at an offset from the neutral axis. 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 x and y directions where x and y are element local directions.

Prestress Force

The prestress force (load per unit length) to apply to the element.

Initial Strain

The initial strain in the element.

Tendon Prestress, Offset

The prestress 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.

4.16.4.4 Thermal

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.

Definition

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.

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

The [load case](#) in which the thermal load applies. The load case gives a way of grouping load effects together.

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

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.

4.16.5 3D Element Loading

There are two loading options for loading 3D (brick) elements. [Face loads](#) apply to specified faces of the brick element. [Thermal](#) loads allow temperatures to be applied to the elements.

Load case and 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.

4.16.5.1 Face Loads

Face loads on 3D elements are similar to face loads on 2D elements, with the additional requirement that the face be selected. Loads are specified as a pressure across the surface.

Definition

Name

The name is only used as a convenient way of identifying a load.

3D element list

The list of elements to which the load is to be applied.

In sculpt: The 3D element list is set to the 3D 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.

Face

The face to be loaded. The faces (1 to 6) are ordered: Bottom, Top, Front, Right, Back, Left

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.

4.16.5.2 Thermal

Thermal loads on 3D elements can either be uniform giving rise to a uniform thermal strain across the element, or variable with temperature specified at each node.

Definition

Name

The name is only used as a convenient way of identifying a load.

3D Element List

The list of elements to which the load is to be applied.

In sculpt: The 3D element list is set to the 3D elements in the current selection set. The 3D 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 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
- **General** – load which varies over the whole element

Temperature

The magnitude of the temperature change as required by the load type.

4.16.6 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 surface](#) 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 can be [grid point loads](#), [grid line loads](#) or [grid area loads](#). In all cases these loads are expanded to beam type elements.

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 rectangular and the simplified tributary areas option is set, otherwise two-way spanning panels are treated in the more general way described below. For simplified tributary areas the angle at each corner is bisected and the intersection of these lines is used to split the panel up in to three or four regions. For a rectangular panel this leads to the typical “back of an envelope” pattern. The load on each these regions is then assumed to be applied to the nearest edge.

Loads on two-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.

Load cases and titles

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.

4.16.6.1 Grid Point Loads

These are point loads located in space or, more precisely, on a grid surface which is located in space. The grid surface is also used to identify the elements that are considered during the grid load expansion. Refer to [Specifying Grid Loading](#) 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.

If the grid surface associated with the load is 2D element type, the grid point load will be transferred as a point load on the coincided 2D element. If the grid point load coincides with corners or edges of more than one 2D elements, the load will be transferred to the first element that program identifies and ignore the other elements.

Definition

Name

The name is only used as a convenient way of identifying a load. Optional.

Grid Surface

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).

In sculpt: The grid plane is set to the current grid. The grid plane cannot be edited in the dialog.

Grid Coordinates

The grid coordinates of the load point. Only the x and y coordinates (or r and if the grid plane is cylindrical) are defined, the z axis is taken as zero.

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.

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 implies panel local axes.

Direction

The direction of the loading, in the specified axis direction.

Value

The magnitude of the load.

4.16.6.2 Grid Line Loads

These are line loads located in space or, more precisely, on a grid surface which is located in space. The grid surface is also used to identify the elements that are considered during the grid load expansion. Refer to [Specifying Grid Loading](#) 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.

If the grid surface associated with the load is 2D element type, the grid line load will be transferred as point loads on each of the 2D elements which intersects with the grid line load. Position of the point load is at the centroid of the grid line segment that intersected with the 2D element. Intensity of the point load is also calculated from the intersected grid line segment. If the grid line load coincides with edges of more than one 2D elements, the load will be transferred to the first element that program identifies and ignores other element.

Definition**Name**

The name is only used as a convenient way of identifying a load. Optional.

Grid Surface

The surface in which the load is located and the surface that defines the elements that attract the load. Note that the load is not necessarily applied in the underlying grid plane axis directions (see [Axis](#), below).

In sculpt: The grid plane is set to the current grid. The grid plane cannot be edited in the dialog.

2D Polyline

Line loads are defined by a polyline. This can either be a reference to a polyline specified in the polylines module, or the polyline can be defined explicitly in the load record. 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 surface). Note that the vertices are defined by Cartesian x-y coordinates even when the grid plane is cylindrical.

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.

Axis

The axis in which the load is applied. Setting this to local implies panel local axes.

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.

4.16.6.3 Grid Area Loads

These are area loads located in space or, more precisely, on a grid surface which is located in space. The grid surface is also used to identify the elements that are considered during the grid load expansion. Refer to [Specifying Grid Loading](#) for details.

Grid area loads must be located so as to be bounded by a closed polygon of elements.

If the grid surface associated with the load is 2D element type, the grid area load will be transferred as point loads on each of the 2D elements which intersect with the grid area load. Position of the point load is at the centroid of the intersected polygon between the grid area load and the 2D element. Intensity of the point load is also calculated from the intersected grid area load polygon.

Definition**Name**

The name is only used as a convenient way of identifying a load. Optional.

Grid Surface

The surface in which the load is located and the surface that defines the elements that attract the load. Note that the load is not necessarily applied in the underlying grid plane axis directions (see [Axis](#), below).

In sculpt: The grid plane is set to the current grid. The grid plane cannot be edited in the dialog.

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 surface. 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.

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.

Axis

The axis in which the load is applied. Setting this to local implies panel local axes.

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.

4.16.7 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. Unlike the other load types, gravity loads are an acceleration applied to mass in the structure.

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 $1.4L1 + L2$.

Definition**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".

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

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).

Application

Gravity loads apply to the following element types:

- **Beams, bars, rods, 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
- **3D Elements** – through the volume and density

Spring, damper, grounded spring, grounded damper, link and spacer elements are considered massless, so attract no gravity load.

Load cases and titles

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.

4.17 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. For harmonic and time-history analysis a [damping table](#) can be used to define damping as a function of frequency or mode. Time-history analysis requires a spatial load distribution and a [load curve](#) defining the variation with time. [Dynamic load factors](#) are specified for periodic analysis. Footfall analysis requires a [frequency weighting curve](#).

4.17.1 Response Spectra

Basic Definition

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
- Ordinanza 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.

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

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 $\sqrt{10/(5+\eta)}$

4.17.1.1 ASCE 7 (05, 10), IBC (2006, 2009, 2015)

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.

4.17.1.2 Eurocode 8 (2004)

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)

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 (γ)

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 (γ)

A lower bound factor sets a minimum level below which the spectrum cannot fall. Design Spectra only.

4.17.1.3 Eurocode 8 (2003)

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)

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 (γ)

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.

4.17.1.4 Eurocode 8 (1994)

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 (γ)

The parameter γ 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 (γ)

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.

4.17.1.5 FEMA 356

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 (γ)

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)

$$T = C_t(h_n)$$

The building period coefficient, C_t , and the building period exponent, γ , 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 $\frac{2}{3}$ of that used for horizontal excitation. (FEMA356 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".

4.17.1.6 GB50011 (2001, 2010)

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 ()

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.

4.17.1.7 IBC (2000)

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 S_1

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 $\frac{2}{3}$ 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.

4.17.1.8 IS 1893 (Part 1) (2002)

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 ()

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 $\frac{2}{3}$ of that used for horizontal excitation. Checking this option applied the $\frac{2}{3}$ 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.

4.17.1.9 Ordinanza PCM 3274

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 ()

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.

4.17.1.10 UBC (1994)

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 $\frac{2}{3}$ 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.

4.17.1.11 UBC (1997)

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 $\frac{2}{3}$ 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.

4.17.1.12 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](#).

4.17.1.13 Damping

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

4.17.2 Frequency Damping

The frequency damping table defines the critical damping ratio as a function of frequency. It is used by harmonic analysis, time history analysis and footfall induced vibration analysis when the damping ratios for modes vary with frequency. Values are interpolated from the table. Frequencies outside the range use the nearest specified value.

Definition

Name

The name is used only as a label for this frequency damping table definition.

Description

Damping table consists of a series pair of data that gives the frequency and the corresponding total critical damping ratio. The description is a string in the form of '(f₁,d₁) (f₂,d₂) (f₃,d₃) ...' to describe each pair of the frequency and damping ratio. The description can be typed directly in the table or entered as a table in the wizard.

Frequency damping curves can be exported and imported as text (.txt or .csv) files.

4.17.3 Mode Damping

The damping table defines the critical damping ratio for the given modes of modal dynamic analysis. It is used by harmonic analysis, time history analysis and footfall induced vibration analysis when the damping ratios for some modes need to be defined individually.

Definition

Name

The name is used only as a label for this damping table definition.

Default Damping

If damping ratios of any modes have not been defined in this module, the default damping ratio is used for those modes in harmonic analysis, time-history analysis and footfall analysis.

Description

Damping table consists of a series pair of data that gives the mode number of modal dynamic analysis and its corresponding total critical damping ratio. The description is a string in the form $(m_1, d_1) (m_2, d_2) (m_3, d_3) \dots$ 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.

Mode damping curves can be exported and imported as text (.txt or .csv) files.

4.17.4 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.

Definition

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 $(t_1, v_1) (t_2, v_2) (t_3, v_3) \dots$ 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. When used in the analysis, if analysis time is beyond the defined range, the load factor will be taken as zero.

Load curves can be exported as text (.txt or .csv) and imported as text (.txt or .csv) or [PEER ground motion](#) files.

4.17.5 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.

Definition

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 $(f_1, d_{1,1}, d_{1,2}, d_{1,3}, d_{1,4}) (f_2, d_{2,1}, d_{2,2}, d_{2,3}, d_{2,4}) (f_3, d_{3,1}, d_{3,2}, d_{3,3}, d_{3,4}) \dots$ 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.

4.17.6 Frequency Weighting Curve

This defines a frequency weighting curve, 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 the curve, divided by 0.005 m/s^2 .

Definition

Name

The name is used only as a label for this frequency weighting curve.

Description

A 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 $(f_1, w_1) (f_2, w_2) (f_3, w_3) \dots$.

The description can be typed directly in the frequency weighting curve table or open the wizard to input the frequencies and weighting factor.

Frequency weighting curves can be exported and imported as text (.txt or .csv) files.

4.18 Raft

4.18.1 Soil

4.18.1.1 Soil Profiles

Soil profiles are vertical columns through the ground which are separated into levels of strata. A new table is created for each soil profile.

Layers must be entered in order of decreasing level.

The lateral extent of each soil profile is specified later as a soil zone.

Definition

Name

The name is a convenient way of identifying a particular soil profile.

Levels

The levels are defined in terms of the level at top of each stratum and the number of intermediate displacement levels.

Displacement levels sub-divide each layer of soil into a number of sub-layers. For example the specification of two intermediate displacement levels will divide the layer of soil into three sub-layers.

This also defines the number of stress calculation points, with depth, for the Boussinesq solution. The stresses are calculated at the centre of each sub-layer.

Soil Material Properties

The basic soil properties are the Young's modulus & Poisson's ratio. Where non-linear behaviour is required a non-linear soil curve can be assigned to individual layers of strata.

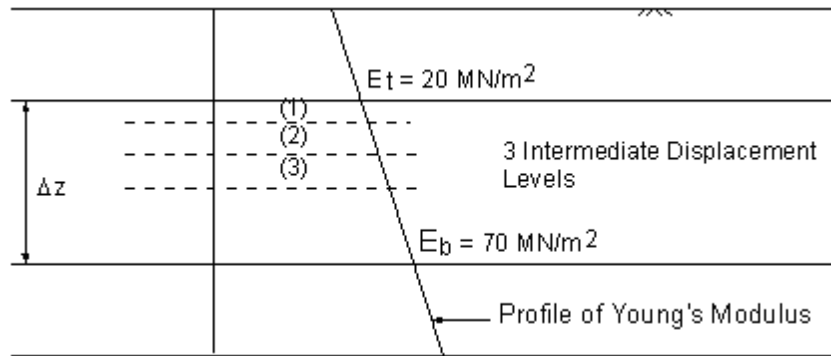
The pile-soil interaction property allows the selection of a pile-soil interaction property to be associated with each layer of a soil profile.

Notes

The number of displacement levels calculated is determined as the larger of:

a user-specified number (N) given for each soil layer

a generated number (n) taken from the Maximum Allowable Ratio (R_a) between values of Young's modulus (E).



The value of n is calculated by

$$n = \frac{\log(E_b/E_t)}{\log R_a}$$

So for the example illustrated above, if $R_a = 1.5$

$$n = \frac{\log(70/20)}{\log 1.5} = 3.089 \rightarrow 3$$

The value of n is always taken as the next lowest integer.

The value of $R_a = 1.5$ is normally sufficiently accurate for most problems.

4.18.1.2 Soil Zones

Soil zones along with soil profiles define soil blocks. The soil profile defines the variation with depth and the soil zones define the horizontal extents of the block.

Definition

Name

The name is a convenient way of identifying a particular soil zone.

Extents

This is defined in terms of rectangular areas specified by corner x and y co-ordinates. The soil zones are therefore aligned with the Cartesian coordinate system.

Soil zone areas may overlap. If this occurs then the profile relating to the soil zone with highest allocated number will be used in the calculation.

If no soil zone is specified then the first soil profile is assumed to be widespread. Similarly, if displacements are to be calculated for locations that are wide of any soil zones then the first soil profile will be used.

4.18.1.3 Soil Material Curves

The value of stiffness (Young's modulus or E) with depth is specified in the soil profiles. For linear materials each specified value of E will remain constant, at the given depth, throughout the calculation for stress, strain and thereby displacement.

For models of materials where the relationship between stress and strain is non-linear a non-linear curve must also be specified.

Definition

Name

This is only used as a way to easily identify the soil curve.

Stiffness strain curve

The nonlinear soil curve is specified as a factor on the stiffness (Young's modulus). This can be defined by the specification of a curve of strain versus stiffness factor. The secant stiffness at a particular strain is then the specified E value x factor.

4.18.1.4 Pile-Soil Interaction Properties

Pile-soil interaction properties allows for a connection between pile and soil. The soil interacts with piles (modelled by beam elements) through nodes along the piles in 3 dimensions (x, y, z). At the base of the pile, the z interaction is through normal bearing contact. Along the piles, z interaction is through shearing contact between piles and soil. The normal and shearing interaction stresses are given by

$$\frac{\sigma}{\sigma_{ult}} = C \left(\frac{\delta}{D} \right)$$

for vertical shear interaction along the piles and

$$\frac{\sigma}{\sigma_{ult}} = C \left(\frac{\delta}{B} \right)$$

for horizontal normal contact interaction along the piles

where: σ is the interaction stress (shear or normal as relevant),

σ_{ult} is the maximum value of the stress defined on this table,

$C()$ a coefficient obtained from Pile-soil interaction coefficients/curves explained in Pile-soil interaction coefficients table,

δ is the relative displacement explained below

D, B are geometry parameters explained below.

The values of δ represent the relative displacement between the pile and the elastic ground soil near it. That is, δ is the non-linear deformation of the soil local to the pile, which is usually behaving in a non-linear manner.

For circular piles $D = B_x = B_y$ = the diameter of the pile

For rectangular piles, B_x and B_y are the side width of the rectangular section, $p = 2B_x + 2B_y$ and $D = p/$

For H & I sections, B_x is the flange width, B_y the web height + 2 x flange thickness, p the complete perimeter (roughly 2 x web height + 4 x flange) and $D = (2B_x + 2B_y)/$.

Definition

Name

name of pile-soil interaction property.

Pile-Soil Interaction Curve (coefficients) in X, Y & Z or the curve used for pile base

These define the interaction curves used for x, y and z interaction and for the pile base interaction

The maximum soil bearing/shear stress-in X, Y, Z & pile base at both top and bottom of each layers of the soil.

These define maximum bearing/shear stresses in direction directions and locations

4.18.1.5 Pile-Soil Interaction Coefficients

This defines pile-soil interaction coefficients/curves, that are used by Pile-soil interaction properties.

Definition

Name

The name of the pile-soil interaction coefficients/curves.(PSIC)

Curve

For each PSIC, the coefficient $C(d)$ is a function of d , d is normalised relative displacement, i.e. $d = \delta/B$ or δ/D , B & D are explained in Pile-soil interaction property table. If d is outside the defined range on positive or negative side of PSIC curve, the last C value on positive or negative side of the curve will be used as the corresponding C value, i.e. the curve is extended horizontally on both positive and negative sides.

4.18.1.6 Soil Rectangular Loads

Normally in a raft analysis the load on the soil is from the structure, but additional soil loads can be specified. These act in addition to the raft loading.

Definition

Name

This is just used to make it easier to identify a soil load

Loading Plane Orientation

The orientation of the loading plane. A load in the vertical plane and no rotation angle is in the XZ plane.

Angle of local x from global X

Define the rotation angle of the load about the global z axis.

Centre of loaded area

The coordinates of the centre of the rectangular loading area (x, y, & z in global axis).

Width x

The width of the area in local x.

Length y

The length of the area in local y.

Load values in normal and tangent directions

- Normal load in local z direction.
- Tangent load in local x direction
- Tangent load in local y direction

Note that the soil loads follow the PDisp convention:

Loading plane orientation	Angle of local x from global x	Global direction of local z load (normal)	Global direction of local x load (tangential)	Global direction of local y load (tangential)
Horizontal plane	0	-Z	X	Y
Vertical plane	0	-Y	X	-Z
Vertical plane	90	X	Y	-Z

4.18.2 Raft Interaction

The Raft interaction module defines the interface between the nodes on raft and the soil. Each raft interaction node in the structure is assumed to interact with a rectangular area of soil.

Definition

Node List

A list of nodes that interact with soil.

Interaction Area

A rectangular area can be defined for the interaction nodes by x and y dimensions in global axis if "Automatic" option is not used. If 'Automatic' is selected, the interaction areas will be calculated by GSA automatically. If an interaction node is connected to one 1D element or parallel 1D elements, the interaction area cannot be generated. If an interaction node is connected to the middle nodes of parabolic 2D elements, the interaction area also cannot be generated, in these two cases, the interaction area has to be defined explicitly by the x and y dimension.

Interaction Elevation

This defines the elevation of the raft nodes that interact with soil, the elevation is relative to vertical level used in soil profile definition. If 'Automatic' is selected, the z coordinate of the interaction nodes will be used as the interaction elevation. .

Minimum & Maximum Pressure

The minimum pressure that can be exerted on the soil, with a positive value being compression.

If a negative minimum pressure is specified then this is the maximum tensile pressure the soil can sustain.

The maximum pressure that can be exerted on the soil. If the soil pressure is higher than the maximum pressure during raft analysis, the soil pressure will be constant and equal to this maximum pressure.

4.18.3 Pile Interaction

The Pile interaction module defines the interface between the nodes on pile and the soil.

Definition**Node List**

A list of nodes that interact with soil.

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.

4.19 Bridge

4.19.1 Alignments

An alignment is the basic setting out feature for bridges, defined in terms of chainage and straights or radii. The vehicle paths are then related back to alignments. A structure may have multiple alignments.

Definition**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

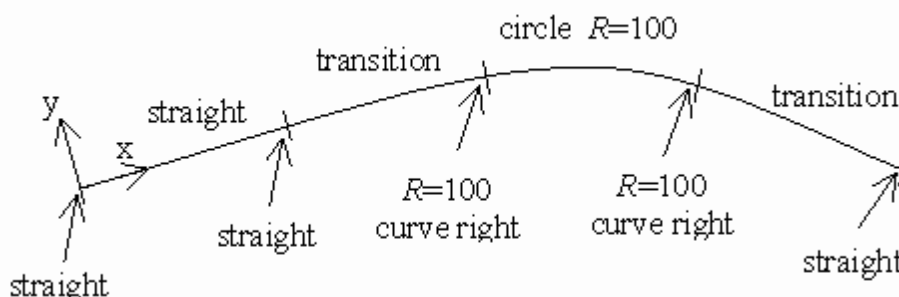
Distance along the Alignment from the origin. The origin of the alignment (chainage 0) is at the origin of the axis system of the Grid Plane, see below.

Curvature

Straight, Right Curve or Left Curve (looking in the 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.

4.19.2 Paths

Paths define traffic lines along a bridge related to the [alignments](#). These can be for trains, road vehicle or pedestrians. 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.

Definition

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.

4.19.3 Vehicles

For bridge analysis a vehicle is defined as a pattern of loading based on axle and wheel positions. User defined arrangements of wheel loads forming a vehicle are defined here for use in [bridge loading](#) or to [generate static vehicle](#) load. 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.

Definition

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 and Load

The wheel position is defined by the distance each wheel is offset from the centreline (i.e. $\pm y$ coordinate). A wheel load is then specified for the left and right respectively looking in the direction of increasing x.

Width

This specifies the width of the vehicle used when checking vehicle paths.

4.19.4 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.

Definition

Name

The name is used to refer to a particular VUDL.

Type

Different linear and non-linear variable UDL types can be selected.

Number of Segments

The number of segments used to define the complete VUDL (with a maximum of three segments).

Factor (A) and 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.

This provides a view of the load intensity as a function of load length. This is provided as visual confirmation of the VUDL definition.

4.19.5 Influence Effects

Influence effects give the effects on one part of the structure of a load in another part. GSA allows definition of both [node influence effects](#) and [beam influence effects](#).

4.19.5.1 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.

Definition**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.

Note: As the number of analysis cases in the bridge analysis task depends on the influence effects, the editing of influence effects will be disabled if there is a bridge analysis task.

4.19.5.2 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.

Definition

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.

Note: As the number of analysis cases in the bridge analysis task depends on the influence effects, the editing of influence effects will be disabled if there is a bridge analysis task.

4.19.6 Path Loading

Path Loading can be used with the results of an influence line analysis to derive optimized bridge loading load cases.

Definition

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).

Note: As the number of analysis cases in the bridge analysis task depends on path loading data, the editing of path loading will be disabled if there is a bridge analysis task.

4.19.7 Bridge Loading

Both user defined [static bridge loads](#) and [moving bridge loads](#) can be defined to define loading relative to defined paths. For code based analysis there are [standard bridge loads](#).

4.19.7.1 Static Bridge Load

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 “Model | Bridge Modelling | 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 “Model | Bridge Modelling | 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.

Definition

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.

If “Bridge design loading code” is selected in “Bridge Loading Specification”, the bridge analysis has to be done first, then the carriageways paths can be populated to the path dropdown list and the static bridge loads can be defined and expanded

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.

4.19.7.2 Moving Bridge Load

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 "Model | Bridge Modelling | 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.

Definition

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.

If "Bridge design loading code" is selected in "Bridge Loading Specification", the bridge analysis has to be done first, then the carriageways paths can be populated to the path dropdown list and the static bridge loads can be defined and expanded

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).

4.19.7.3 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)

4.19.7.3.1 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 axles (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 axles.

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 axles.
- **HB+HA(Hong Kong)** – Vehicle+VUDL – exclusion length for HB vehicles is 25.2m from the end axles.
- **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.

4.19.7.3.2 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 axles, 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.

4.19.7.3.3 US Highway Loading

Loading for two different US Loading codes has been included as detailed below.

The older AASHTO 16th 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 (i.e. 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.

4.19.7.3.4 Eurocode Loading

Load Model 1 from the Draft prEN 1991-2 dated 22/8/01 has been allowed for as follows:

Vehicles

- EU-TS300 – 2 axles 300kN each
- EU-TS200 – 2 axles 200kN each
- EU-TS100 – 2 axles 100kN each

Path Loads

- EU-LM1:Lane1 – UDL $9 \times (\text{path width})$ kN/m + Vehicle EU-TS300, zero exclusion length
- EU-LM1:Lane2 – UDL $2.5 \times (\text{path width})$ kN/m + Vehicle EU-TS200, zero exclusion length
- EU-LM1:Lane3 – UDL $2.5 \times (\text{path width})$ kN/m + Vehicle EU-TS100, zero exclusion length
- EU-LM1:Other – UDL $2.5 \times (\text{path width})$ kN/m

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.

4.19.7.3.5 Australian Highway Loading

Loading from the draft DR 00375 code dated September 2001 has been allowed for as follows:

Vehicles

- M1600-3.5 – 6 axle vehicle 120kN/axle 3.5m centre spacing
- M1600-5 – 6 axle vehicle 120kN/axle 5m centre spacing
- S1600-3.5 – 6 axle vehicle 80kN/axle 3.5m centre spacing
- S1600-5 – 6 axle vehicle 80kN/axle 5m centre spacing

Path Loading

- M1600 – UDL 6kN/m + Two Vehicles, M1600-3.5 and M1600-5, at least 6.5m clear between them, placed in worst positions, zero exclusion length
- S1600 – UDL 24kN/m + Two Vehicles, S1600-3.5 and S1600-5, at least 6.5m clear between them, placed in worst positions, zero exclusion length

4.19.7.3.6 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 axles 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.

4.20 Analysis Stages

There are situations where it is convenient to consider only part of the model or the model with different section or analysis properties. This can be accommodated in a model through the use of analysis stages. The basic [stage definition](#) defines the elements that participate in a stage. The properties in a particular stage can be mapped across from the base properties defined in the model in the [analysis stage properties](#) or the [analysis stage materials](#).

4.20.1 Stage Definition

The basic definition of a stage is a set of element which are associate with the stage. This can be 'all'.

Definition

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 "[Working with the Program](#)".

Creep Coefficient

This option allows creep to be considered for concrete materials by specifying a creep coefficient. The properties are modified as follows:

$$E \rightarrow \frac{E}{(1 + \phi)}, \quad G \rightarrow \frac{G}{(1 + \phi)}$$

Lock Releases on Elements

It may be useful to ignore releases that have been assigned to beam elements. Analysis stages allow releases to be excluded when this is required.

4.20.2 Stage Properties

Stage properties are used to specify element properties for analysis stages where these are to differ from the whole model element properties. The table maps from the element property number, to a property number to be used inn the analysis stage. Properties not referenced in the analysis stage properties retain the element property reference.

Definition

Stage

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

The property number as given in the Elements table. I.e. the whole model element property number.

Element Stage Property

The property to be mapped for this stage.

Example

<i>Stage No.</i>	<i>Type</i>	<i>Element Property No.</i>	<i>Property Record</i>
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.

4.20.3 Stage Materials

Stage materials are used to specify analysis materials for analysis stages where these are to differ from the whole model element materials. The table maps from the element materials (referenced by property), to a material number to be used in the analysis stage. Materials not referenced in the analysis stage materials retain the material defined in the element property reference.

Definition

Stage

The analysis stage for which the properties are to be adjusted.

Element Material

The material number defined in the element properties.

Stage Material

The material to be mapped for this stage.

Creep Coefficient

The creep coefficient used for concrete materials in stage. Click **Code** in the wizard to get a code-specific creep value.

4.21 Tasks and Cases

4.21.1 Analysis Case Definition

The case description definition dialog is made available when an analysis case definition is required, it is not invoked directly.

Definition

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.

4.21.2 Analysis Wizard

4.21.2.1 Solver Option

All analysis options are controlled from the analysis wizard and depend on the selected solver option.

Definition

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](#), [non-linear static](#) and non-linear static explicit 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 component buckling analysis.
- **Soil-structure** – [raft](#) analysis.
- **Bridge** – bridge optimisation analysis (pre-analysis and no analysis case should exist).
- **Miscellaneous** – mass and LS-DYNA analysis
- **Lightweight structures** – [form finding](#) analysis.
- **Post-processing** – [analysis envelopes](#).
- **Diagnostic** – [model stability](#) and model stability P-delta analysis

Solver, Task and Initial Case

The solver option depends on the analysis requested. The task number is determined by GSA, but the initial case can be specified by the user.

Imperfection

GSA can apply a range of imperfections in an analysis. The most commonly specified imperfection is an imperfection which is a function of height. This option allows for an imperfection in the x or y direction as a linear function of height. More complex imperfections are set in the Advanced options.

Analysis Stage

The analysis task will refer to the selected stage. If stages are not required the 'Whole mode' option includes everything in the model.

4.21.2.2 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.

Definition

The details of the loading associated with each analysis case are displayed in a list box

- **Case** – case identifier
- **Name** – case name
- **Description** – a description of the case

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 [Case Description Definition](#) 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.

4.21.2.3 Static P-delta

Definition

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
- Result 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 or combination case is specified and the geometric stiffness is constructed from the results in this analysis case. The combination cases must be a simple combination (only one permutation).

4.21.2.4 Modal Dynamic

For both a 'modal dynamic' and 'modal dynamic p-delta' analysis the user is brought to this page.

Definition

Number of Modes & Start Mode

Specify the number of modes required from the eigenanalysis. Occasionally it is only higher modes that are of interest. The start mode 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](#), the following three options are available:

- Lumped mass at nodes - in this option, the total element mass is assigned to the nodes of the element and form the element mass matrix, the proportion of the total mass at each node depends on the length (1D element), area (2D element) or volume (3D element) covered by the nodes for 1D, 2D and 3D elements respectively, e.g. the two nodes of a beam element will each have half of the total mass of the element, the 4 nodes of a rectangular Quad4 element will each have 1/4 of the total mass of the element. The element mass matrix from this option is a diagonal matrix, this option should be used in most cases unless local vibration modes of the elements are interested.
- Calculate mass from the element shape function - in this option, the element mass matrix, called consistent mass matrix, will be calculated using the same shape function for element stiffness matrix calculation. The principle of this method is that the work done by the inertia forces of the distributed mass within the element is the same as the work done by the inertia forces of the calculated nodal masses are equal. The element mass matrix from this option is not diagonal. This option is normally used if element local vibration modes are interested
- Ignore element mass except for mass elements - if this option is selected, the mass of all elements except explicitly defined mass elements will be ignored in the dynamic analysis. The masses of mass elements will be always considered in modal 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.

4.21.2.5 Explicit Time History Load

Explicit time history analysis - load definition

Analysis termination time

The analysis will start from time zero, this termination time defines the duration of the analysis, once the analysis time reaches or passes this time, the analysis will stop.

Load curve and load description

This table is used to define the dynamic loads and their variations based on the load curves. The load curve in the first column is used to define the variations of the load or load combination given in the second column. There is no limit for the number of load curves and loads to be used in the analysis.

Analysis option

Two options available and they are linear and geometrically nonlinear. If linear option is selected, the analysis will be geometrical linear analysis, i.e. geometrically nonlinear effects are ignored. If geometrically nonlinear is selected, the analysis will be geometrically nonlinear analysis, i.e. large deformation and element geometric stiffness are considered. If elastic-plastic material is used for some or all elements, the elastic-plastic material properties will be considered for the relevant elements.

Initial deformation and loads

This allows the analysis to be started from the end of another analysis, e.g. the time history analysis can start from the end of a static analysis with only self-weight loads. Once this option is defined by giving an analysis case number that has analysis results, the analysis will start after moving all the nodes to the deformed positions according to the given analysis case, the loads used in the given analysis case will be included and kept constant during the explicit time history analysis.

4.21.2.6 Explicit Time History Mass & Damping

Explicit time history analysis - mass and damping definition

Mass scale factor on element masses

Element masses will be lumped at the element nodes, this factor can be used to magnify or reduce the element masses used in the analysis.

Mass derived from loads

Case definition: define the load or load combination that is used to derive masses

Direction: specify which direction of the loads to be used in deriving masses

Scale factor: the factor is used to magnify or reduce the masses derived from loads

Damping option

Damper elements: check this box to include damping from damper elements. If there is no damper element in the model, checking this box will make no difference

Rayleigh damping: define Rayleigh damping by specifying two frequencies and corresponding two critical damping ratios. Wizard is available for defining the Rayleigh damping by giving alpha and beta parameters directly.

4.21.2.7 Explicit Time History Results

Explicit time history analysis - analysis results

Whole model results

Whole model results can be saved at constant time interval or at variable time intervals defined in the table. The whole model results are saved and associated with analysis cases as other analysis. Whole model results are normally saved as larger time intervals.

Selected node and element results

Node list: define the nodes that results are saved

Element list: define the elements that results are saved

Selected node and element results can also be saved at constant time interval or related to analysis time steps (multiple of analysis time step as the saving time interval)

Selected node and element results are saved as analysis task results, i.e. the results are associated with the analysis task rather than analysis case, the results can be presented with chart view.

4.21.2.8 Ritz Analysis

A Ritz analysis is similar to a modal analysis but is specified in terms of Ritz vectors.

Definition

Number of Ritz Vectors

The number of Ritz vectors in each of x, y and z, and thus 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. 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.

4.21.2.9 Buckling

Buckling is an eigenanalysis to find the load factors and modes of buckling.

Definition

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 $a.A_a + b.A_b + \dots$)

4.21.2.10 Mass Calculation

A mass analysis calculates the mass and mass distribution in the structure.

Definition

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, for example 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.

4.21.2.11 Nonlinear Static

Non-linear static analysis uses dynamic relaxation to allow for non-linear effects and large displacements.

Definition

Element Slenderness

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.

4.21.2.12 Nonlinear Static Control

This provides the control parameters for nonlinear static analysis

Definition

Load increment

In a nonlinear analysis the load can all be applied at the start - *single load increment* or in *automatic load increments*. If the latter then the proportion of the total loads applied in the first load increment should be specified.

Initial load condition

There are three initial load conditions

No initial load – no initial load is considered on the structure

Load case – this load is applied at the start of the analysis and remains constant throughout the analysis. This can be used, for example, for dead loads.

Analysis case – the analysis starts from the results of the specified analysis case - the initial analysis case can be a linear static analysis or nonlinear analysis.

4.21.2.13 Nonlinear Static Termination

When the automatic load increment option is selected the termination criteria must be set.

Definition

Nodal displacement/rotation limit

A node need be selected which will be monitored and the analysis will terminate when the displacement/rotation at the specified node and direction reaches the limiting displacement or rotation

Load factor limit

The analysis will terminate when the load factor has been reached or passed the specified load factor limit. The load factor is the multiplier on all the loads defined by the load combination of this analysis case

4.21.2.14 Form Finding

Definition

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.

Post Analysis

Spacer element are used to control form-finding and avoid poor meshes. Post analysis it is possible to save axial forces generated in spacer elements as prestress

Beam and similar element also generate load axial forces which can be stored as prestress.

4.21.2.15 Relaxation 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.

Definition

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.

4.21.2.16 Member Identification

This information is required when the "[Individual component 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.

Definition

Elements

Input a list of elements to be investigated for individual component buckling. These must be adjacent beam elements.

4.21.2.17 Relaxation Analysis Progress

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.

4.21.2.18 Response Spectrum

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".

Definition

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.

Separate or CQC3

Normally a response spectrum analysis considers each direction separately, and some rule is used to combine the effects from different directions. a CQC3 analysis considers all three directions concurrently.

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.

The combination of basic responses allows for an SRSS combination across the different directions.

Axis

In general the calculation is carried out in global axes, 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%.

Note

There are two combination in response spectrum analysis. The part of the calculation is for the response in each mode. These modal responses need to be combined to get the response to excitation in that direction. This can use any of the combination methods noted above but is generally CQC. Once the response in the individual direction has been calculated these need to be combined to get overall responses. The second combination is to combine the results for different directions. The simplest is a linear combination where there is a full contribution from one direction and reduce combinations from the others. This can be handled just like any other combination case. The other option is the SRSS option and this takes the responses calculated in the first step and then combines these using SRSS.

4.21.2.19 Response Spectrum Details

Unlike most other analysis options derived results from a response spectrum analysis cannot be derived from the basic response spectrum results. Instead the derived modal results must be calculated and then these combined.

Definition**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.

4.21.2.20 Response Spectrum Modes

Apseudo 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.

Definition

Pseudo Dynamic Details

The dynamic details must be supplied for each pseudo mode to be considered. These are the frequency, model mass and effective mass. Only the pseudo modes for which values are not specified will be included in the analysis

4.21.2.21 Linear Time History

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.

Definition**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

Excitation is either a force excitation which requires a 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. The other option is base excitation where 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.

A load curve defines the variation of the dynamic loads against time. The load curve will be normalized before use.

Output Options

Two output options are offered. The save results intervals creates an analysis case for each interval, and saves a full set of results. The time-history option provides for results at a much higher frequency which are normally displayed as time histories.

Time Step

Normally the automatic time step should be used but this can be overridden if required.

4.21.2.22 Harmonic Analysis

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. The load is assumed to vary as

$$f(t) = a \sin(\quad t)$$

Definition**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

The load description defines the spatial distribution of the harmonic load. The syntax of the description is the same as other analyses, e.g. 1.4L1+1.6L2.

Excitation Frequencies

The excitation is assumed to cover a range of 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.

4.21.2.23 Periodic Load Analysis

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.

Definition

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

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.

4.21.2.24 Footfall Analysis

This page defines the analysis parameters. If no modal dynamic analysis results are present, this option is disabled and this page cannot be opened.

Definition

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.

Number of Footfalls

The number of footfalls to be considered in the analysis.

Walker (mass or weight)

The mass/weight representing a sample walker

Direction of Responses

The direction of response in the GSA global axis direction. It can be in Z (vertical), X, Y or XY plane (horizontal). Note that it is the response direction in GSA global axis, not the Basicentric axes of the human body described in BS6472 and BS6841.

Frequency weighting curve

The Frequency Weighting Curve (FWC) is used in calculating the response factors. Standard and user defined curves can be used.

The following standard frequency weighting curves are available in GSA: Wg, Wb and Wd. These weightings are defined in BS 6841:1987, a document which also describes how they might be applied to comfort, perception, health and so on. These defined weightings have been adopted and applied in various ways in other subsequent BS documents, as summarised below in the case of the assessment of human exposure to vibration in buildings.

- Wg is the weighting adopted in BS6472:1992 for the assessment of human exposure to vertical vibrations in buildings. It is presented as a base curve with a multiplier corresponding to the response factor metric in the frequency range 1-80 Hz. The horizontal weighting is also presented as a base curve which is similar to the Wd weighting and is shown in the frequency range 1-100 Hz.
- Wb is the weighting specified in BS6472-1:2008 for the assessment of human exposure to vertical vibration in buildings; Wd is the corresponding weighting for horizontal vibration. The weighting functions are presented in the form of charts showing the weighting curves in the frequency range 0.1 to 100 Hz.

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 (AISC SDGS11) – AISC Steel Design Guide Series, Floor Vibration Due to Human Activity.
- Walking on floor (AISC SDGS11 2nd ed) – AISC Steel Design Guide Series, Floor Vibration Due to Human Activity.
- Walking on floor (CCIP-016) – Arup method.
- Walking on floor (SCI P354) – SCI P354 – Design of Floors for Vibration: A New Approach.
- Walking on stair (AISC SDGS11 2nd ed) – AISC Steel Design Guide Series, Floor Vibration Due to Human Activity.

- Walking on stairs (Arup) – Arup method.
- Walking on stairs (SCI P354) – SCI P354 – Design of Floors for Vibration: A New Approach. method is used.
- Running on floor (AISC SDGS11 2nd ed) – AISC Steel Design Guide Series, Floor Vibration Due to Human Activity.
- User defined Dynamic Load Factor – from the 'Dynamic Load Factor' table

Walking Frequency

Defines the range (minimum & maximum) of walking frequencies to be considered in the analysis.

4.21.2.25 Soil Structure Analysis

The soil-structure analysis is a static analysis of a single analysis case including soil structure interaction.

Definition

Maximum No. 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.

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 Residual

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 Residual

This is only used for piled-raft analysis and it is the tolerance of the pile-soil interaction force.

4.21.2.26 Soil-Structure Progress

Definition

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

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.

Soil stiffness from the analysis can be saved in the input data at the end of the analysis. Note that this option modifies the input data. Subsequent soil-structure analysis of any analysis case will use the saved support stiffnesses as initial soil stiffnesses, as usual.

This option is disabled if any results exist for other than soil-structure analyses.

4.21.2.27 Bridge Analysis

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.

Definition

In either case the subsequent steps can be carried out using the Tools: [Optimise Bridge Loading](#) and [Expand Bridge Loading](#), and a 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.

4.21.2.28 Envelopes

The use of analysis envelopes is described in the [Step-by-Step Guide](#).

Definition

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.

4.21.2.29 Model Stability

A model stability analysis is set up identify problems arising from modelling.

Definition

Low Stiffness Modes

In most cases a handful of modes (say 5) should suffice. This will identify area of the model that have zero (or low) stiffness. This may be due to lack of proper connectivity. 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.

High Stiffness Modes

The high stiffness modes are usually less of an issue, but these enlarge the condition number. In particular short elements with high section properties can give rise to ill conditioning.

See [stability analysis](#) for details about the Model stability analysis.

4.21.2.30 Cases Set Up

This page confirms that all the data required for the analysis is present.

Definition

Analyse Now

The task can be analysed immediately, or kept for later analysis.

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.

It is generally best to use default settings for analyses, adjusting these should be done carefully as it is possible to change parameters resulting in inaccurate solutions.

4.21.2.31 Damping

Modal analysis is carried out ignoring damping, however much of the dynamic post-processing relies on damping to get a solution.

Definition

Damping Option

The critical damping ratio options are

Constant – a constant damping ratio is specified which is used for all modes

Damping vs Frequency – the damping ratio is calculated for the required frequency by interpolation from the specified frequency/damping table. If the frequency is outside the range defined in the table the damping ratio of the nearest end value is used.

Damping vs Mode – the damping ratio for the mode is determined from the specified mode/damping table. For modes not defined in the table the standard damping ratio will be used.

Modal (material) Damping – the damping ratios calculated for the material in a modal dynamic analysis are used. If the damping ratio is not available from a modal dynamic analysis, the analysis cannot be carried out.

4.21.3 Advanced Solver Settings

4.21.3.1 Stiffness Solution

For many of the solver options GSA uses a matrix solver. There are a number of options which allow adjustments to be made to the assembly of the stiffness matrix.

Definition

Solution Details

There are three matrix solvers in GSA, each with different characteristics:

- **Active Column**
- **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 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. Generally the sparse parallel direct solver is significantly faster than the active column solver.

The preconditioner is not used at present.

Degree of Freedom Ordering

The way in which the degrees of freedom are ordered can have a significant impact of 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 work 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 can define the vector explicitly.

The sparse direct solver works only with the approximate minimum degree (AMD) algorithm.

Global Stiffness Factor

This allows a factor to be applied to the stiffness as a whole. Either stiffening the structure (>0) or introducing more flexibility (<0).

Lock Releases

For some analyses it may be desirable to ignore releases at the ends of beam elements and consider all connections fully fixed

Stiffness Matrix Reduction

- **Limiting minimum:maximum stiffness ratio** – this is the limit where Gss 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 and Condition Number

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 of 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.

4.21.3.2 Eigensolution

Both the modal dynamic analysis and buckling analysis make use of an eigensolver. These are iterative so it is useful to be able control the solution procedure.

Definition

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 values for the subspace iteration are the main (outer loop) in the eigensolver. The second (Jacobi) set are only use in the Subspace Iteration [Jacobi] option to control inner iteration loop.

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.

Ignore Offsets

Offsets in element mass can give rise to off diagonal terms in the mass matrix. This increases the memory footprint of the solution, and in general will have little influence on the overall solution, so these can be ignored.

4.21.3.3 Ritz Analysis

Definition

Randomize the Load Vector

A Ritz analysis sets up load vectors for each of the required directions which correspond to a unit gravity load. For symmetric structures this can sometimes lead to a problem in that torsional responses are not captured. This option applies a randomization to the gravity load vector to seed a torsional response. The same randomization is applied in each run.

4.21.3.4 2D Element Analysis

Definition

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-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.

By default both options are set to the recommended, later formulations. The default in-plane and out-of-plane formulations can be set in the preferences. This preference setting can be found in the miscellaneous tab of the GSA Preferences.

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.

When tied interfaces are used with shell elements with five degrees of freedom (no drilling degree of freedom) there can be issues in setting up the constraint equations used by the tied interface. Assigning a full set of degrees of freedom avoids this problem.

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

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.

4.21.3.5 Convergence Control

The relaxation solution is iterative, and these parameters control the iterative process. They can be adjusted if convergence problems are experienced, or to speed up analysis.

Definition

Number of Cycles to Check Residuals

The convergence of the solution depends on the magnitude of the residual vector, however checking the residual takes time so it is usual to not calculate this at every time step.

Dummy Mass Update

A dynamic relaxation solution assigns dummy mass to the nodes which is updated during the solution. In general it is best to leave the updating to the Automatic option, but the user can also take control of how frequently this happens. IN some cases it may be better not to update the dummy mass, and instead specify a dummy mass factor directly.

Damping

There is a choice of using viscous damping (based on the pseudo velocities) or artificial damping (based on displacements). This can be handled automatically. or with a user defined percentage of damping.

4.21.3.6 Reduced Integration

In an explicit solution the time taken to process elements is the most significant factor in the solution time. Reduced integration allows for a faster solution by accepting a less refined integration scheme. This mean that the elements are not stiff in all modes of deformation so are liable to 'hourglassing'. Consequently hourglass control is needed to stabilize the model.

Definition

Use reduced integration (this will override the 2D element formulation)

This mean that instead of full integration of the elements a simpler integration scheme with fewer integration points will be used leading to a faster solution.

2D elements / 3D elements

Select the appropriate hourglass control type and degree of control.

4.21.3.7 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.

Definition

Residual

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.

Termination

Set the maximum number of cycles or the length of time (clock time) for which the analysis should run before terminating.

4.21.3.8 Results

GSA can calculate a large range of results. However not all these results may be required so this allows for selection of particular results to be stored. For an explicit solver these can be selected separately for Full Model Results and for Selected Model Results

Definition

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.

Centre-only

When checked 2D element forces and stresses and 3D 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.

4.21.3.9 Pre-load

Definition

Element Pre-load

Allows the user to specify a pre-load to be applied to the elements. This pre-load must refer only to prestress beam or 2D loads. The internal loads are used to calculate a geometric stiffness (Kg 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.

4.21.3.10 Geometric Stiffness

Definition

Geometric Stiffness for Beam Elements

This allows certain terms to be excluded from the geometric stiffness matrix. The options are

- Include all terms in the K_g matrix - this is the default option
- Suppress torsional buckling terms in the K_g 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 K_g matrix - similar to previous option but removing more of the destabilizing terms.
- Treat the beam element as a bar for K_g terms - includes only the Euler buckling.

Elements Excluded

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- analysis to a linear analysis, so it should be used with care.

Element forces

When constructing the geometric stiffness element forces are required. When these are not available it is possible to synthesize these from displacements, but this is only an approximation when these relate to a non-linear analysis.

4.21.3.11 Imperfection

Real structures are not the perfect structure envisaged by the design but contain imperfections. The design codes try to ensure that imperfections are considered in the design process. This feature allows imperfections to be included in the analysis.

Definition

Geometric imperfection

A geometric imperfection is specified by a set of equations defining how the x, y and z coordinates are modified for the analysis. The simplest imperfection is typically where the x (or y) coordinate is a function of the height of the node on building

Scaled case imperfection

If for example a buckling analysis has been carried out then it may be appropriate to check an imperfection in which the structure assumes a scaled value of one of the buckling model shapes. Select and scale a result case as appropriate.

Random imperfection

This specifies a random imperfection

Clear imperfections

Removes any specified imperfections

4.21.3.12 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.

Definition

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

Raft analysis 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 an initial support stiffness is assigned to the all soil interaction nodes that do not have support stiffness explicitly defined.

4.21.3.13 Explicit Solver

Definition

Time Step

The explicit solver calculates a critical time step based on the natural frequency of the mesh. For stability the time step must be less than this. By default the time step is set as a percentage of the critical time step, but a user defined time step can be specified. To ensure stability nodal masses can be adjusted, but a maximum mass scaling factor should be specified to avoid too much divergence from the real inertia characteristics of the structure.

Progress Dialog

This displays a dialog with summary information for the analysis. Note this slows the analysis, so is more useful as a diagnostic tool.

4.21.3.14 LS-DYNA

This allows various key parameters to be included in the LS-DYNA export. Refer to the LS-DYNA manuals for a detailed description of these parameters.

Definition

Command line

This specifies the command line to run on exiting the analysis wizard

Memory / Number of processors

Determine the memory and number of processors for parallel execution available to LS-DYNA when running

Element formulation

This allows the user to specify the element formulations to be used for quad and triangle elements and for beams.

Integration factor k

LS-DYNA assumes the sections are a collection of point areas which are integrated to get the element forces. The parameter k determines how a section will be discretised.

Include file

This allows a pre-prepared data file to be include in the LS-DYNA keyword file.

4.21.4 Soil-Structure Analysis Progress

This dialog is opened when a soil-structure analysis is started and keeps the user informed of the solution progress during the 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 soil-structure analysis.

Stop

This will terminate the soil-structure analysis.

4.21.5 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. A more details description is given in [Working with the Program – Combination cases](#) and [Working with the Program – Enveloping in GSA](#)

Definition**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 "Working with the Program" 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.

Note

The note is not used by the combination case but allows a note to be stored against the combination case.

Notes

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.

4.21.6 Design Wizard

Once analysis has been carried out GSA can then proceed to member design. The Design task wizard allows you to automate section design. The design wizard can be activated by:

- Selecting New Design Task from the Design menu
- Opening the Design Tasks page in the Cases and Tasks folder in the data explorer, and selecting New Design Task from the right-click Context menu.

A Design Task is collection of specifications that guide the automated, iterative design or checking of members. A Design Task is analogous to an Analysis Task in that there can be multiple Design Tasks all of which are saved with the model. Design Tasks must be executed to carry out either a design or a check based on the parameters defined in the task.

The result of a Steel Member design is a new section reference.

Results of the checks are saved as design histories in members and member utilizations that are available for output and contouring.

4.21.6.1 Design Option

The Design task wizard allows you to automate section design.

Definition

Name

A name for the design task.

Option

A design task can be set up either for steel member or RC member design. The RC member design is not currently implemented.

Member list

Design is carried out for selected members. This specifies the members that are to be included in the

design task, although the list can be 'all'. Non-steel members are ignored in steel design

4.21.6.2 Steel Member Design

Steel design works on a member-by-member basis and checks each member for strength and/or serviceability using the combination cases specified. Sections are drawn from the Steel Section Pool that the member refers to and these serve as possible candidates for the design. If a member fails a check, it is assigned a fitter section from the pool. Sections are ranked based on the Design Objectives.

Definition

Strength Design

The combination case reference considered in strength design must be specified. Both an upper limit and lower limit can be set for the target utilization. The lower limit does not alter the design, it only produces an inefficiency warning.

Serviceability Design

The combination case reference considered in serviceability design must be specified. This combination case is distinct from that specified for strength. Limits are set on span deflection and absolute deflections.

Design Objectives

Select a primary and a secondary objective for design. The primary objective will control the choice of section but where there is ambiguity the secondary objective is used to determine the design.

Section Assignment

Members can be designed on an individual or a group basis. When on an individual basis each member can end up with a different section after the design. When grouped all members with the same pool reference will be assigned the same section. The grouped option will generate fewer variations, but provide a more conservative solution, as some members will be 'over-designed'. It is also possible, at least theoretically, that grouped design may fail to converge, especially when doing strength design.

4.21.6.3 Task Option

Definition

The task can be stored for design at a later point, or the design can be run immediately. A third option allows for a check to be carried out on the current section, to see if it has sufficient capacity. This option does not generate new sections for the member – only reporting if the member meets the design criteria.

4.22 General Data

4.22.1 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.

See also [List Definitions](#) and [Lists Toolbar](#)

Definition

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 “[Working with the Program](#)”.

4.22.2 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.

Definition

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”.

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.

Part V

5 Analysis in GSA

5.1 Linear Static Analysis

For most users the most common use of GSA is for linear static analysis. The [Getting Started](#) manual 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 [Running Analyses](#).)

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 [Results Display Options section](#).

5.1.1 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 of the structure as a whole due to the load.

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. The results of P-delta analysis are similar to those for a linear static analysis.

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 within the solution there are two passes. The first pass establishes the forces in the structure allowing the geometric stiffness to be established for the second pass. The GSA solver handles both passes in a single solution task.

A P-delta analysis is set up using the Analysis Wizard from the **Analysis / New Analysis Task** menu command. For a P-delta analysis there is then a choice

- P-delta analysis with each analysis case defining its own differential stiffness
- P-delta effects for all analysis cases defined by case...
- P-delta effects form an analysed 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 (or other) 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.

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 [Results Display 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.

5.1.2 Linear 2D Element Analysis

5.1.2.1 Modelling Implications

Some structures can only be adequately modelled using 2D elements. For examples 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 Feature Guide section on [Generating 2D element meshes](#).

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
- **Flat shell** – an element which has both in-plane and out-of-plane stiffness
- **Curved shell** – not available at present in the solver

So to model a shear wall the plane stress type may be appropriate, while for a slab the flat shell may be the best choice.

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. 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.

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**
- **Prestress**
- **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.

5.1.2.2 Analysis & Results

Analysis

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

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 | Settings | Contour Settings** or **Graphics | Display | Settings | Diagram Settings** 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 [Results Display 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** 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.

5.1.2.3 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, prestress 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

5.2 Nonlinear Static 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. Non-linear data types are made available in the data tables. The options given are the basic [Non-linear static analysis](#) and [Fabric 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](#)

5.2.1 Modelling Implications

Cases

The principle of superposition is not valid for non-linear behaviour so unlike linear static analysis, results from different analyses **must not** be added together. An analysis case can be defined as the summation of factored load cases. This total loading is then analysed. Combination Cases formed from previously analysed Gss, non-linear or GsRelax results will probably contain **invalid results**. Combination Cases should only be used for producing envelopes in these cases.

5.2.2 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.

Modelling implications of ties and struts

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.

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](#).

Results of non-linear static analysis

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 [Results Display Options section](#).

5.2.3 Dynamic Relaxation

5.2.3.1 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 or Fabric.

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. To mitigate this an analysis can be performed in steps with loads being applied incrementally during the analysis and the intermediate results stored. These incremental load options are offered by the Analysis Wizard.

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 relating to dynamic analysis included in the model will be ignored during GsRelax analysis.

5.2.3.2 Analysis & Results

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 the nonlinear solver. 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

steps and the particular non-linear analysis selected. The basic analysis option is *single increment*. 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. Only one analysis case is analysed at a time, so for multiple cases the solutions are sequential.

The dynamic relaxation solver uses an iterative process controlled by [damping](#). The results and progress of the analysis can be viewed and [adjusted](#) while it is in progress. The iterative process can be tuned in the analysis wizard, and the format and frequency of reporting analysis progress adjusted.

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 checks, and the intermediate results for the multi-increment analyses. This report window also displays the results summary for individual component buckling and automatic increment options.

5.2.4 Explicit Nonlinear Static Analysis

The explicit nonlinear static analysis option is also a dynamic relaxation approach. This is a new nonlinear solver that will in time replace the existing dynamic relaxation solver. This builds on the explicit dynamic (nonlinear time history) solver

5.2.5 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 offers features to deal with both of these issues.

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 geometry. 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 model.

Enter the [Fabric Material properties](#) in the fabric materials table. The [warp direction](#) will be aligned with the [element x axis](#) of each 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 Fabric Material property record number to assign the Material.

[2D element prestress loading](#) defines the prestress in warp and weft (element x and y) directions.

Alternatively the element prestress data (for membrane and 1D elements) will be generated during a Form-Finding analysis.

Spacer elements will probably have been used in a Form-Finding analysis to generate the model. These elements and associated prestress loading should be deleted prior to analysis as a real fabric structure.

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.

5.2.6 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 prestress that will result.

5.2.6.1 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.

5.2.6.2 Soap Film

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 prestress 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.

5.2.6.3 Force Density

Force density form-finding is a method of form-finding cable networks. The tension in each link of 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 links multiplied by the stiffnesses of the links.

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

5.2.6.4 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.

5.3 Dynamic Analysis

Dynamic analysis covers a range of analysis options from determining the dynamic characteristics of a structure, through looking at linear response, to full non-linear dynamic analysis.

The dynamic characteristics of a structure are best captured through a linear [modal analysis](#) or a [Ritz analysis](#). A modal analysis is used to determine the natural frequencies and the accompanying mode shapes, which represent free vibration of the structure without reference to any loads. A Ritz analysis is similar to a modal analysis, but can better target significant modes with approximations to the real modes.

There are a number of dynamic response analysis options which follow from a modal or Ritz analysis and which make use of these dynamic characteristics. These include seismic response using [response spectrum analysis](#), [linear time-history](#) and [harmonic](#) analysis and [footfall](#) analysis.

Where nonlinearity has to be considered the only option is a time history solution. This is available with the [explicit time history](#) analysis option.

5.3.1 Modal Dynamic Analysis

5.3.1.1 Modal Analysis

A modal dynamic analysis is set up using the Analysis Wizard from the **Analysis / New Analysis Task** command. 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.

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 global restraints should be avoided.

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.
- **Masses calculated from the element shape function** – the mass and inertia of the element (including the mass element) are accounted for.
- **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.

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 [Results Display Options section](#).

5.3.1.2 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.

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.

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.

5.3.2 Ritz Dynamic Analysis

5.3.2.1 Ritz Analysis

A Ritz analysis is a way of calculating approximations to the dynamic modes of a structure, so can appear similar to a modal dynamic analysis, even though the calculation is quite different. A Ritz analysis is based on a set of load dependent vectors derived from gravity loads applied in the three global axis directions. The advantage of a Ritz analysis is that it will recover modes of vibration that are significant in all three directions. This is the case even if the stiffness in one direction is significantly different from the other, which is a challenge for a normal dynamic modal analysis.

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.

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 [Results Display Options section](#).

5.3.2.2 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.

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.

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.

5.3.3 Explicit Dynamic Analysis

When the model is non-linear the only option for dynamic analysis is a time-history analysis which steps through the solution one time step at a time. There are two basic schemes for this time stepping: implicit and explicit. The time-history option implemented in GSA is explicit.

With an explicit time history solution the analysis time step depend on the mesh frequency, so small, stiff elements which have a high frequency will require a small time step that can control the overall solution computational effort.

Modelling Implications

An explicit time-history solution proceeds from a knowledge of the structure at a given time, being used to calculate an acceleration which is then used to predict the state at the next time step. With an explicit time history solution the analysis time step depends on the mesh frequency, so small, stiff elements (which have a high frequency) will require a small time step. This limiting time step controls the stability of the analysis, and so can control the overall solution computational effort.

An explicit solution can require a large number (1000s) of iterations, so results are only output at intervals requested by the user. Unlike other analysis options an explicit solution allows two types of output – results for the whole model and results for selected nodes and elements. The whole model results form a (relatively small) set of analysis cases where results such as displacements, forces, etc. are stored for all nodes and elements. The selected node and element results are normally output at a much small time step so that significant results can be tracked as a function of time. These are stored as task results, rather than analysis case results and are normally viewed as charts.

Analysis

An explicit time-history analysis is set up using the Analysis Wizard from the **Analysis / New Analysis Task** menu command. Select the **Explicit time history** option. The following page allow the user to specify the task details, including the spatial and temporal aspects of the load, and the distribution of the structure mass and specification of the damping.

Results

The results for an explicit time history analysis fall into two sets – whole model results and time history results. The whole model results are similar to those for a static analysis, but with the option of velocity and acceleration results for nodes. These display the state of the model at that snapshot in time.

The selected node and element results give the variation with time and they can be viewed charts.

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

5.3.4 Dynamic Response Analysis

5.3.4.1 Response Spectrum Analysis

See [Seismic Analysis](#).

5.3.4.2 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.

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 the modelling 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.

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 : [Linear Time History Analysis](#)

5.3.4.3 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 and 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.

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 Modelling 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.

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 [Results Display 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 : [Harmonic Analysis](#)

5.3.4.4 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 35, 29 and 1 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 frequencies etc. The following three design guides of footfall analysis can be considered:

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

In addition to the three design guides, user defined DLF (Dynamic Load Factor) curve can also be used in footfall analysis if you know the Fourier coefficients 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.

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 modelling 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. As the footfall analysis is only concerned with vertical excitation it may often be appropriate to constrain the modal analysis to exclude horizontal motion by applying [additional restraint](#).

For more details see [Analysis Wizard : Footfall Induced Vibration Analysis](#)

Results

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

Resonant analysis

- Maximum acceleration based response factors
- Peak acceleration
- Critical nodes
- Critical walking frequencies

Transient analysis

- Maximum velocity based response factors
- Peak velocity
- RMS (Root Mean Square) velocity
- RMQ (Root Mean Quad) velocity
- RMQ Critical nodes
- RMQ Critical walking frequencies

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

After a footfall induced vibration analysis, chart views that show the plot of response factor versus walking frequency, velocity versus time, etc. are also available for individual response nodes.

5.3.4.5 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.

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 the modelling 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.

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 : [Periodic Load Analysis](#)

5.4 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.

5.4.1 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 eigenvalue buckling analysis extends this concept from a simple column to the whole structure, to determine global modes of buckling.

Sufficient modes should be selected for the buckling analysis to ensure that all the modes of concern are identified.

5.4.1.1 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, it is inadvisable to define global restraints on the structure, as this can potentially provide an artificial degree of stiffness to the model.

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 GSA 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 Modal Buckling option. 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 GSA 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 and not the individual wall element buckling modes. This can potentially lead to an overestimate of the load factors.

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 **Model / Coordination Tools / 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.

5.4.1.2 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 typically 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 [Results Display 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.

5.4.2 Nonlinear Buckling Analysis

Non-linear analysis is performed by the dynamic relaxation (DR) 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 DR 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 DR analyses. In addition, the stiffness of individual beam elements is reduced when they are in compression.

There are a number of non-linear 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 modal buckling analysis first to get an estimate of the buckling load factor, then create a new model with imperfection introduced based on the first mode shape.

The automatic load increment option in non-linear static analysis should be used to search for the collapse / buckling load of a model, starting from a predefined first increment. Note that the loads are increased monotonically in this analysis.

Individual component buckling analysis can be used to estimate the maximum axial force capacity of an element or a chain of elements. This option should be used to estimate the effective restraint to the elements offered by the surrounding structure under a particular load combination.

Modelling implications for non-linear buckling analysis

Beam elements should be used if buckling within the element need to be considered, Bars and struts will not buckle no matter how large of the compressive forces.

Results for non-linear buckling analysis

The results for non-linear buckling analysis will include the real deflections and element forces from the applied loads. This differs from the [modal buckling analysis](#), which gives displacements that represent a scaled mode shape rather than the actual magnitude of the deflection.

The individual component buckling analysis will give axial force - deflection/rotation results for the elements under consideration. These are reported in the **Axial Force / Displacement Relationship** Output View option and the relationship can also be plotted using chart view.

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 and the relationship can be plotted using chart view

5.5 Dynamic Relaxation Analysis

A number of different analyses can be performed on the same data file using the dynamic relaxation solver. The basic analysis option is the single increment option. Using this option will generate forces and deflections in the model resulting from the imposed loads specified in a single analysis case.

This is similar to linear elastic static analysis except that material and geometrical non-linear effects will be considered. Also the post processing of results is different from linear static analysis as the principle of superposition is invalid in non-linear analysis. Therefore results from single increment analysis should not be combined and a different approach to load capacity factors and incremental loading is needed.

In order to match the loading from a linear analysis and combination cases the *loading* for a nonlinear analysis should be factored and combined. Thus the loading may be specified as 1.4L1+1.6L2. It is also possible to use a combination case such as 1.6C1. In this case the loading implied by that combination

will be used.

Note that each dynamic relaxation run relates to a single analysis case and that results **should not** be combined post analysis. 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, the solver generates analysis cases for each load stage.

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 component buckling analysis options are offered to investigate these issues. With these options, the imposed loading specified is used to define the loading pattern. Analysis cases are generated by the program using factored versions of the load pattern.

As the solver is iterative, 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 incremental loading options, the solver monitors the convergence rate for each analysis. If an initial run at low load converges the program stores the information about the convergence process. This initial run shows that the model is not inherently unstable. If convergence becomes significantly harder to achieve as the load increases it is likely that the model has reached 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 practice.

Another use of the nonlinear 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 and prestress. Selecting form finding analysis options tells the 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.

5.5.1 Nonlinear Analysis Options

5.5.1.1 Incremental Analysis

Single Increment

This is the default option. 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 structural model capacity has been exceeded and/or the model represents a mechanism. The [non-convergence](#) can be investigated by looking at the changes in the residual forces during analysis.

Multiple increment

In this option, the total applied loads will be divided into a number of equal or unequal increments. The first increment will be analysed, and then the load will increase in steps one increment at a time. An analysis case is generated with the total accumulated load for each load stage. Results are stored for each analysis case if convergence is achieved.

Automatic Increment

In automatic increment analysis, the program is searching for the structural capacity of the model, by successively increasing the load factor up to a maximum specified. The structural model capacity can be defined as the load at which analysis ceases to converge, or, the load at which predefined excessive

deflections occur. This option can also be used to investigate the load factor at which 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. Analysis cases are generated by the program with the loading increasing until specified limits are exceeded or the capacity of the model is found. These conditions can be limits on deflection or loading and are defined in the [Relaxation Analysis Task Control](#) dialog. The automatic load increment scheme is 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 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 cycles, the current increment will be $1.236 \times \text{Last_increment}$
 - c) If only the last increment analysis is converged within the limited cycles, the current increment will be $1.15 \times \text{Last_increment}$
 - d) If the analysis cannot converge within the limited cycles, the current increment will be reduced by a factor of 0.618, if it still cannot converge, the increment will be reduced by this factor again.
 - 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 nonlinear analysis under the current load level (previous loads plus the current load increment, previous nodal displacements are carried over from the previous loads (if converged) to speed up the process)
- 5) Check displacement and/or load factor limit, if satisfied, stop the analysis, otherwise Go to Step 3.

The program uses the convergence information from each run to pin-point the load at which convergence fails, or increases the load until the specified limits are exceeded. This is deemed to be the structural capacity of the model.

5.5.1.2 Individual Component 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 not be Bars or Struts. 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 ties and cables 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 moment to the member under investigation. (Buckling will not occur without a small disturbance to the member)

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.

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 data given in the report view. If a load deflection curve is not output, check the results carefully.

5.5.1.3 Form-finding

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 prestress 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](#) ([soap film](#) or [force density](#)) are used for elements where these have been defined. Otherwise uses [form-finding uses normal element properties](#).

5.5.1.3.1 Soap Film

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 prestress 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 prestress in the cable depends on the desired curvature of the cable. For a boundary cable to "soap film" Triangles and Quads

$$\text{prestress} \times \text{area} = (\text{soap film tension}) \times \frac{(\text{radius of curvature})}{2}$$

5.5.1.3.2 Force Density

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.

5.5.1.3.3 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.

5.5.1.4 Continuation Analysis

Because dynamic relaxation is iterative, it is sometimes desirable to start the iterations from a particular point in the analysis other than the initial position.

The initial position is the undeformed structure at rest with no forces other than prestress in the elements. As the iterations progress, the elements deflect and their internal forces increase until the deflected shape and internal forces are in equilibrium with the applied loads. If further loading is then to be applied, or damping or convergence parameters adjusted, it is quicker to start iterating again from this deflected, loaded position. This is termed continuation analysis.

A particular case of continuation analysis is form finding. After form-finding analysis, options will be given to update the nodal coordinates and save the current member forces. In this case the nodal coordinates will be overwritten with the coordinates of the deflected shape, the equilibrium element forces will be saved as prestressed forces equivalent distortion loads in an extra load case.

Form finding can be used to carry out continuation analysis. For example, any model, (not only zero stiffness form-finding models), can be analysed with the form-finding option, ignoring the form finding properties. This will perform the same analysis as single increment analysis. Opting to modify the node coordinates and save current member forces at the end of the form-finding analysis will guarantee that the current deformed state of the structure has been kept. If the resulting model is re-analysed with the load case combination used in previous form-finding analysis plus the extra load case just created, the structure will not deform any more since it is already in the equilibrium state.

The usefulness of this option is that it allows the structure to be modified between the first step form-finding analysis and the second stage normal analysis. For example, extra loading, elements or constraints can be added to the structure after form-finding analysis, then the structure can be analysed again with the extra load case being included in the load combination. This process could be used to model the construction stage of a structure.

5.5.2 Dynamic Relaxation Convergence & Damping

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.

Interrupting the 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.

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 details on [Optimising Convergence](#) are given below.

Termination

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.

5.5.2.1 Optimising Convergence

Mass and Inertia Factors

The user can modify the fictitious nodal masses and inertias used by the program by specifying Mass and Inertia Factors. It may be possible to speed up convergence by specifying factors of less than 1.0 to reduce the masses and inertias. However this method is less effective than adjusting the damping; the improvement in speed of convergence is limited to a factor of about 2. Factors of less than 0.5 should not be used and the user must be prepared for the solution process to "blow up" because the simulated damped vibration has become unstable.

The Inertia Factor only has an effect if the structure contains nodes to which moments are applied.

The mass button on the progress dialog box is used to adjust individual mass factors and it is normally not necessary. Setting individual mass factors is only necessary in the following circumstance. At a later stage of the analysis, the maximum residual stays at the same node in the same direction and the

residual decreases very slowly. This may indicate that the dummy mass in that direction is too large, so the convergence rate may be boosted by setting a small mass factor for the node in the relevant direction. However, be aware that if the mass factor is too small, the analysis might diverge.

If the structural flexibility cannot be judged before the analysis, use the default dummy mass and inertia factor (1.0). After the analysis starts, the dummy mass factor can be adjusted 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.

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.

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 GSA Theory manual. The structure is brought to rest and the process restarted. If artificial viscous damping is used, kinetic damping will be disabled.

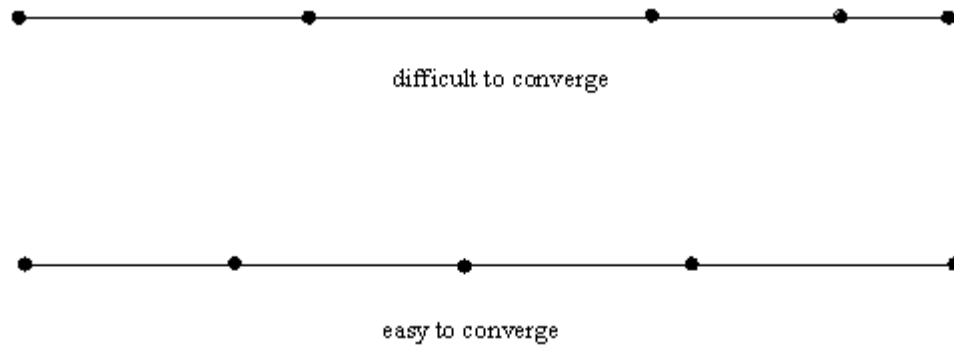
Modelling for Optimum Convergence

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.



5.6 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** in the Output Explorer.

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 modelling 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.

Running Model Stability Analysis

You can run a Model Stability Analysis by adding a new Analysis task and selecting Model stability (or Model stability P-delta).

5.7 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)

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 **Model | Create Storey Grid Planes** 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 **Model | Create Rigid Membranes from Storeys** 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 UBC 1994, UBC 1997, IBC 2000, FEMA 356 (rehabilitation code) Eurocode 8 (1994 and 2003) and Ordinanza PCM 3274 (Italian). 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.

5.7.1 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.

5.7.2 Response Spectrum Analysis

Response spectrum analysis is a method for estimating the seismic response of a structure from a set of modal dynamic or Ritz analysis results along with a response spectrum which capture the frequency content of an expected earthquake.

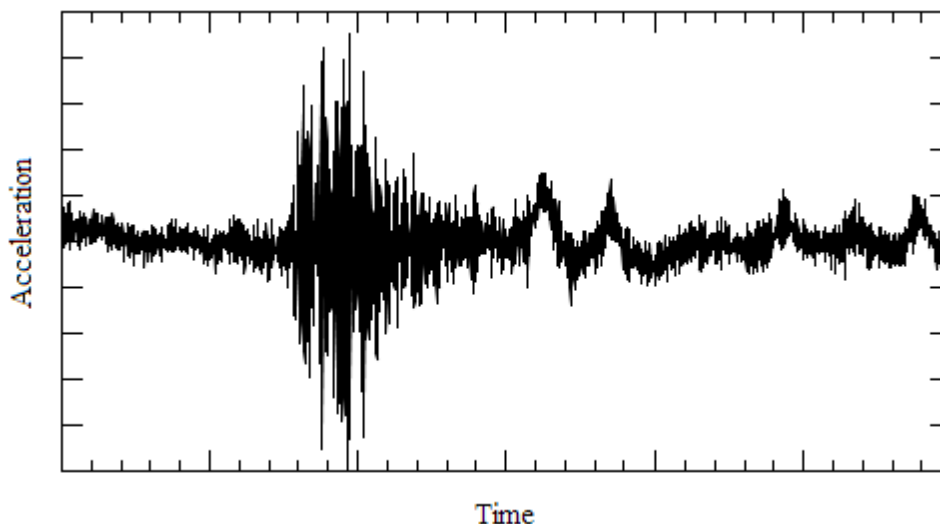
Response spectrum

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.

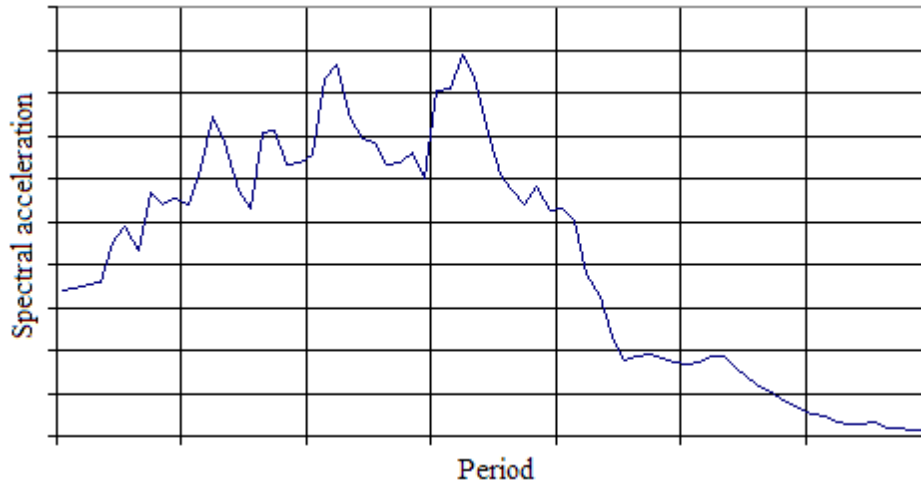
The modal analysis should contain sufficient modes to capture the dynamic characteristics of the structure. In general the sum of the effective masses (in the relevant directions) should be around 90% of the total mass of the structure. For tall structures it is typically easier to capture the effective 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

$$x_i = \Gamma_i \frac{a_{\text{spectral}}(f_i)}{(2\pi f_i)^2} \phi_i$$

In this equation:

$$\Gamma_i \frac{a_{\text{spectral}}(f_i)}{(2\pi f_i)^2}$$

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

$$\mathbf{F}_{jk} = a_{jk} \sum_i m_i \cdot \mathbf{u}_{ij}$$

$$\mathbf{M}_{jk} = a_{jk} \sum_i (m_i \cdot \mathbf{u}_{ij} \times (\mathbf{c}_i - \mathbf{o}) + \mathbf{I}_i \cdot \mathbf{u}_{ij})$$

Where m is the nodal mass, **I** the nodal inertia, **u** and **c** the modal displacement and rotation and **c** is node coordinates and

$$a_{jk} = p_{fac,jk} \cdot a_{spec,j}$$

Where PF_{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 $\mathbf{M}_{k,yy}$ and the torsional (twisting) moment is $\mathbf{M}_{k,zz}$.

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.

$$\text{mass ratio} = \frac{\sum_i m_{eff-i}}{M_{tot}}$$

$$\text{base shear coefficient} = \frac{V_d}{M_{tot}g} = \frac{V_d}{W_{tot}}$$

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).

Warning: any derived results 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.

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

5.8 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. Use the [Bridge Specification dialog](#) to select the design code, limit state(s) to be considered and loading type.
2. Define a User Axis – as method B
3. Define a Grid Plane – as method B
4. Define an Alignment – as method B
5. Define a series of [Paths](#) – carriageways/footways only (otherwise as method B)
6. Define influence effects – as method B
7. Carry out a Bridge Load Optimisation 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 Bridge 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. Use the [Bridge Specification dialog](#) to set the design code to "Undefined".
2. 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).
3. Define a [Grid Surface](#) on which the load is to be applied. Note that all bridge loading is applied in the Grid Surface, and that the elements selected in this plane need to either all be 2D elements or all form a simple grillage with 1D elements only that meet at nodes. Further details of these restrictions are given under [Grid Point Loads](#).
4. Define an [Alignment](#), defined by a series of curvatures at chainages and the Grid Plane.
5. 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 (or within the 2D element mesh), except at the ends of the structure.
6. Define a series of [Node Influence Effects](#) or [Beam Influence Effects](#) that are to be maximised/minimised by the optimiser. To optimise for stresses in 2D elements, add small beams joining adjacent nodes. For axial stress a single beam is sufficient, for bending stresses a straight line of beams is needed.
7. Define a series of [Path Loads](#) that are to be applied to Paths.
8. Carry out an [Influence Analysis](#) using the *Bridge* 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.
9. 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.
10. 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.
11. 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.
12. 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.

5.8.1 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 bridge analysis 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 Bridge Load Optimisation 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 (e.g. 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.

5.8.2 Delete Grid Loading Tool

Bridge Analysis generates Grid Point Loads, Grid Line Loads, Grid Area Loads, and 2D Polylines. These are not deleted automatically before carrying out a further bridge analysis, because there are circumstances where keeping these might be appropriate, but often it is appropriate for all grid loading to be deleted. The Delete Grid Loading Tool does this in a single operation, giving the option to delete 2D polylines at the same time.

5.8.3 Automatic Path Generation

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.

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.

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.

5.8.4 Analysis

[Bridge load optimisation](#) 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 [Model | Bridge Modelling | Expand Bridge Loading](#) 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".

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.

Lane by Lane Path 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.

5.9 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 and piled-raft analysis. The data modules in GSA relating to soil properties and analysis can be imported from or exported to the PDisp program.

An alternative modelling option is to create a solid finite element (FE) mesh using the **Model | Raft Modelling | Create Soil FE Mesh** command. With an FE mesh representing the soil the analysis can be run as a normal static analysis.

5.9.1 Data Requirements

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

Raft and piled-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 interaction nodes and kept free in all three directions for pile interaction 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 **Model / Data Modules / Raft / Soil** ... command or from the Data explorer using the **Raft / Soil** ... 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 PDisp rectangular load table. Only additional loads, i.e. extra to the loads from raft and piles should be defined in PDisp rectangular load table.

Soil-raft interaction and soil-pile interaction data

Soil-raft and soil-pile interactions are defined by the nodes on raft and piles. The Raft Interaction table defines the nodes on the raft to interact with soil. This table is opened from the **Model / Data Modules / Raft / Raft Interaction** command or from the Data explorer 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 elevation of the interaction can also be defined explicitly 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 **Model / Data Modules / Raft / Pile Interaction** menu command or from the Data explorer 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 explicitly or choose Automatic to take the nodal z coordinate as the interaction elevation.

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

5.9.2 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 Pile-Soil Interaction Coefficient (PSIC) curves. 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.

5.9.3 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 the Data explorer or from menu **Model | Data Modules | Specification | Raft Analysis Specification...** to define rigid boundary level etc soil settlement analysis parameters
4. Open **Soil Profiles** table from the Data explorer or from menu **Model | Data Modules | Raft | Soil | Soil Profile** to define soil properties. One soil profile defines the soil properties in a vertical line from the top surface of the soil to the rigid boundary level below. 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 the Data explorer or from menu **Model | Data Modules | Raft | Soil | 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 the Data explorer or from menu **Model | Data Modules | 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 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 always 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 needs 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 is considered. This is non-linear curve is only used when Boussinesq is selected for soil settlement analysis.

2. **Soil rectangular load** – The loads defined in the “Soil rectangular load” table are the loads acting directly on the soil in addition to the loads on soil come from the raft. Normally, these loads 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 on soil outside the building, they can be defined here to increase the flexibility of GSA raft analysis.

See also:

[Notes on raft analysis](#)

Iteration scheme in section [Raft Analysis](#)

5.9.4 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 the Data explorer or from menu **Model | Data Modules | Specification | Raft Analysis Specification...** to define rigid boundary level and other soil settlement analysis parameters.
4. Open **Soil Profiles** table from the Data explorer or from menu **Model | Data Modules | Raft | Soil | Soil Profile** to define soil properties. One soil profile defines the soil properties in a vertical line 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 the Data explorer or from menu **Model | Data Modules | Raft | Soil | 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 the Data explorer or from menu **Model | Data Modules | Raft | Soil | 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 each soil layer in X, Y & Z directions as well as at the pile base. The interaction in X, Y & base are normal contact strength and the interaction in Z is shear strength between pile surface and soil in vertical direction. A **Pile-soil interaction property** is referred to by one or more soil layers 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 a 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 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 the Data explorer or from menu **Model | Data Modules | Raft | Soil | 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 the Data explorer or from menu **Model | Data Modules | 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 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 further to allow soil local yield effect to be considered.
9. Open **Pile Interaction** table from the Data explorer or from menu **Model | Data Modules | 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. Normally the elevation of soil-pile interaction should be automatic as pile interaction nodes are not at the same level.
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.

See also the section on [raft analysis steps](#) and the iteration scheme in section [Raft Analysis](#)

5.9.5 Results

Raft & piled-raft analysis results

Results of raft and piles

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

Interaction results

The interaction results include the contact pressures between raft interaction nodes & soil as well as the contact pressure and shear pressure between pile interaction nodes and soil. These results can be contoured on graphic 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](#).

5.9.6 Notes

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 (PDD file)** 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 (PDD file)** 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 **Model | Raft Modelling | 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.

5.10 Analysis Stages

5.10.1 Modelling Implications

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).

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. 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.

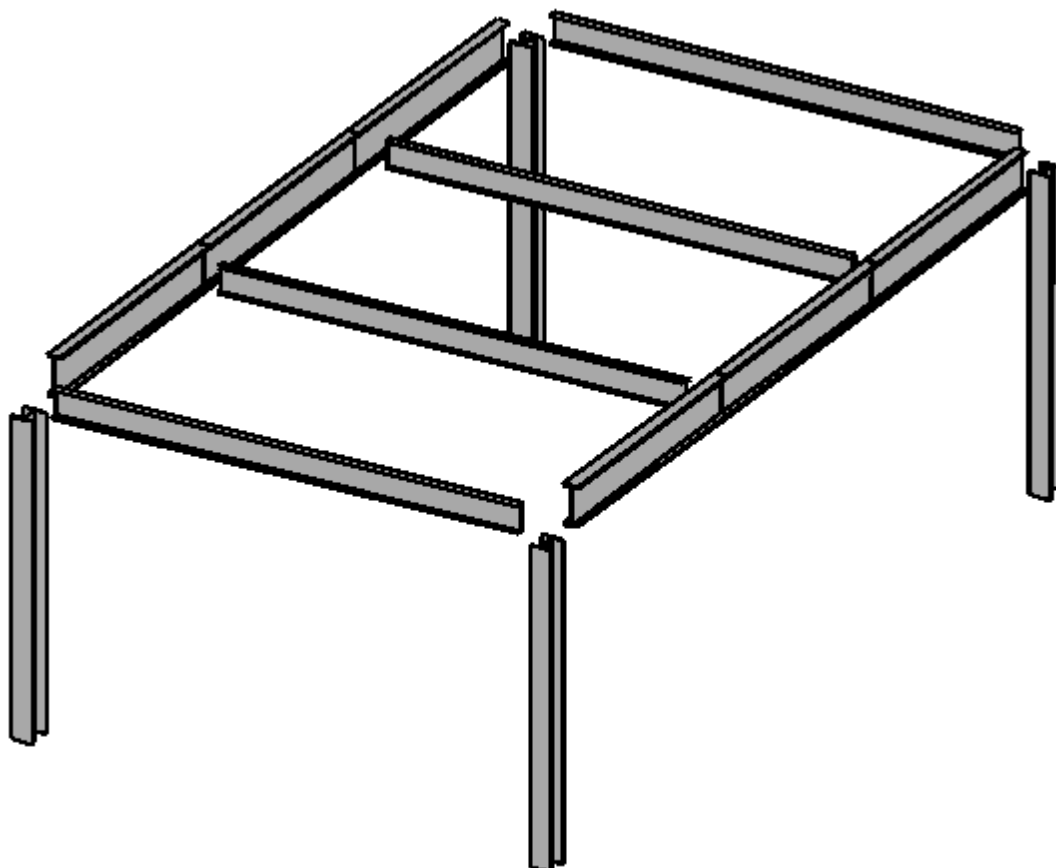
Analysis stages are not available for raft analysis.

5.10.2 Analysis and Design Layers

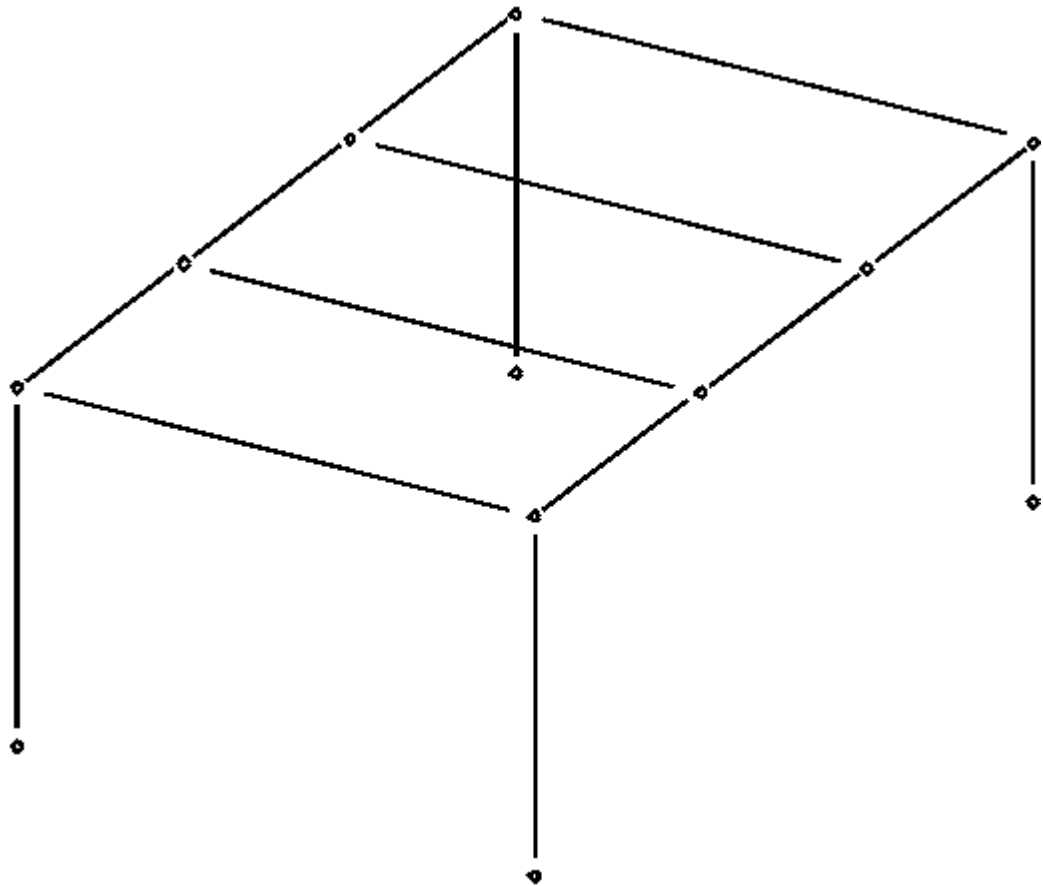
GSA can hold two versions of a model – these are best thought of as two “layers”, an analysis layer and a design layer. The analysis layer is composed of elements and provides the topological connectivity between the different parts of the structure. The design layer is composed of members and provides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

5.10.3 Analysis of Stages

The analysis of a stage is set up using the Analysis Wizard from the **Analysis / New Analysis Task** command. Select the option to create a new analysis and select the analysis stage. The remainder of the data in the wizard is entered in the usual way.

5.10.4 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.

5.10.5 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 [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.

5.11 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.

5.12 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 material grades via the Grade wizards. 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.

5.13 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 'Model | 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	Wave height	Wave period	Water depth	Wave direction
PHASE	Start phase	Finish phase	Phase increment	
STRUCT_Z_ORIGIN	Distance from seabed to GSA structure origin			
WAVE_THEORY	STOKES or AIRY			
CURRENT_PROFILE	Height	Current velocity		
DRAG_AND_INERTIA	Height	CD	CM	
MARINE_GROWTH	Height	Thickness		
SKIP_LEVEL	Height			
SKIPPED_ELEMENTS	Element list			
POLYLOAD_POINT	Number of points			

Unit: metre and kN

Note:

WAVE	Key word for wave profile (one entry only and compulsory)
Wave height	wave height, the vertical distance from valley to peak
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)
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, e.g. if sea-bed is at -60 on GSA coordinates, 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 to seabed
Current velocity	current velocity at the above given water level
DRAG_AND_INERTIA	Key word for drag and inertia coefficients (as many entries as needed and compulsory)
Height	distance to seabed
CD	Drag coefficient at the given water depth
CM	Inertia coefficient at the above given water level
MARINE_GROWTH	Key word for marine growth, radium growth of the tubular members (as many entries as needed and optional)
Height	distance to seabed
Thickness	growth thickness at the above given water level
SKIP_LEVEL	Key word for elements skip level (one entry only and optional)
Height	distance to seabed, 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      10
SKIPPED_ELEMENTS 1 to 5
POLYLOAD_POINT  15

```

5.14 Using Assemblies in Analysis

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

$$= \frac{1}{n} \sum_{i=1}^n \theta_i$$

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.

5.15 LS-DYNA Analysis

The GSA analysis options are focused on linear and mildly non-linear problems. In some cases it is necessary to use a more advanced analysis. This can be provided by LS-DYNA. This analysis option allows the GSA model to be used to create an LS-DYNA analysis

Modelling implications

The key idea is that the GSA model should form the reference model and that the extra data required for an LS-DYNA model should be included as part of the GSA data. Normally the different attributes required for a DYNA analysis are assigned by use of an analysis stage.

The key area where LS-DYNA requires more detail is in the material models. To achieve this a number of LS-DYNA specific material models are included. These are defined in the material wizard, by checking the DYNA material option

Once the task is created an LS-DYNA keyword file is written and opened in the selected LS-DYNA environment.

Results

The LS-DYNA results are not copied back to GSA for post-processing.

5.16 RC Slab Reinforcement Design

The RC slab reinforcement design process calculates reinforcement areas for RC slabs modelled using 2D elements of uniform thickness, subject to any combination of in-plane compression and/or shear force and out-of-plane bending moment and torsion. The process ignores through thickness shear.

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

To use these features, reinforcement grade needs to be defined which is referred to by concrete slab property, concrete slab property should be referred by 2D element property that include concrete material grade. Concrete design code need to be defined in design specification.

Theory

The input data for calculating reinforcement areas at a node of 2D elements include: forces and moments, reinforcement directions, section depth and concrete and reinforcement material properties.

RC slab design arrives at a solution by dividing the section into 3 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 the top and the bottom of the point in the two specified directions, if reinforcements cannot be calculated, empty results will be produced that indicates failure due to inappropriate input data..

Modelling Implications

RC slab reinforcement design is carried out directly on a 2D element model that represents a slab or a wall. The data required additional to the analysis model data are the Concrete Slab Properties, reinforcement grade, concrete grade and the selection of an RC Slab design code. Concrete slab property may be designed before or after analysis.

The *design property* number in the *2D elements properties* table identifies the Concrete Slab Design Property record associated with the element. Reinforcement is not calculated for elements that do not have Concrete Slab Design Properties. Refer to Concrete Slab Properties wizard for details of [Reinforcement](#) definition and [Calculation](#) definition.

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 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.

5.17 Results

5.17.1 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.

5.17.2 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 is 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 GSA Theory Manual section.

The natural frequencies (f), modal stiffness (k) and modal mass (m) 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 GSA Theory Manual section.

The load factor (λ), modal stiffness (k) and modal geometric stiffness (k_g) are related as follows:

$$\lambda = \frac{k}{k_g}$$

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.

5.17.3 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](#).

5.17.4 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](#). You can run a [model stability analysis](#) to check for disproportionately large stiffnesses.

To help differentiate the difference within a model, an entry 'Ignored loads' is reported that accounts for the first type of error above.

5.17.5 Soil Contact Bearing Pressure

The soil contact bearing pressure is available for output during or after a raft analysis. It is calculated as the vertical reaction at a node divided by the area of foundation as specified in the Soil Interaction table.

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 during raft analysis a temporary restraint may be applied 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.

5.17.6 Beam Element Results

5.17.6.1 Displacements and Rotations

Displacements reported are element displacements; end values may differ from nodal displacements when the element has end releases or is offset.

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.

$$\{u_r\} = [K]^{-1}(\{f_r\} - [K_{rf}]\{u_f\})$$

5.17.6.2 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 [Diagrams toolbar](#).

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 beam 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 of each end of every distributed load.

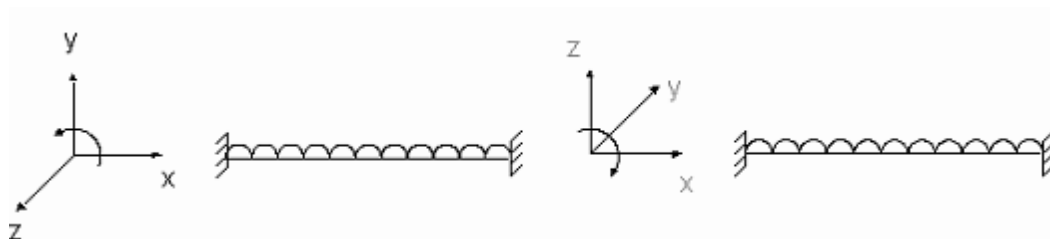
Sign Convention

The force in the beam at any point is the force required to maintain equilibrium if the beam is cut at that

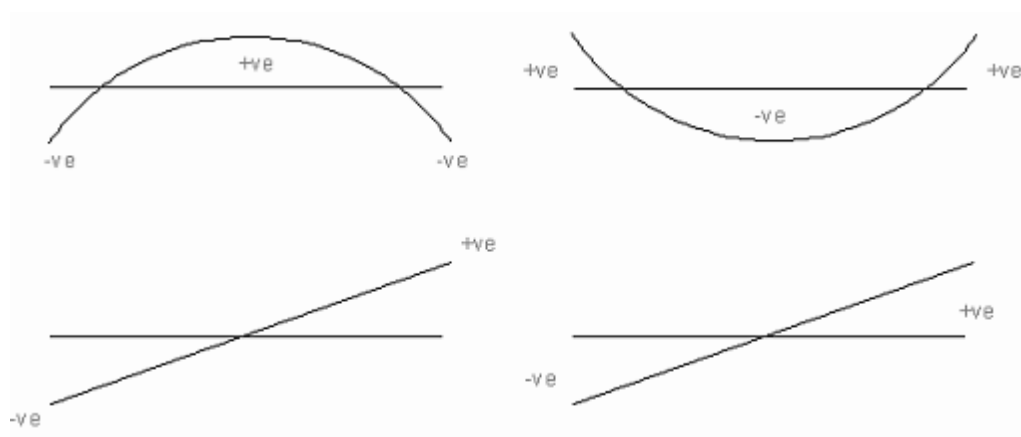
point and the end 2 part of the beam is discarded. Thus:

- 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.

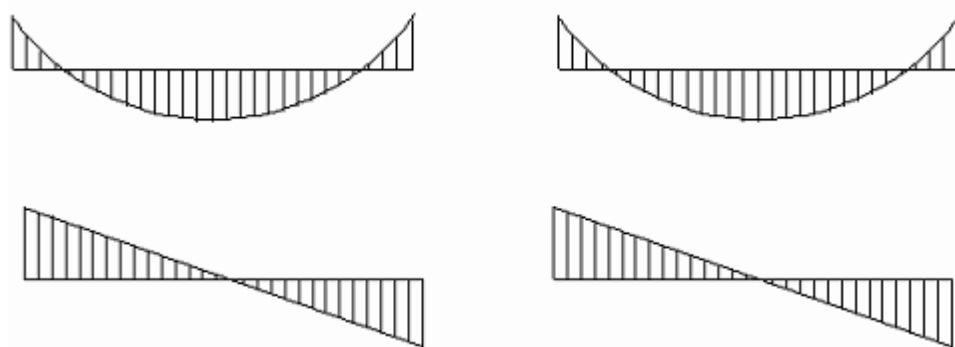
This is demonstrated for the following two beams:



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.

5.17.6.3 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:

$$A = \frac{F_x}{area}$$

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:

$$B_y = \frac{1}{I_{yy}I_{zz} - I_{yz}^2} (I_{zz}M_{yy} + I_{yz}M_{zz}) D_z,$$

$$B_z = \frac{-1}{I_{yy}I_{zz} - I_{yz}^2} (I_{yz}M_{yy} + I_{yy}M_{zz}) D_y$$

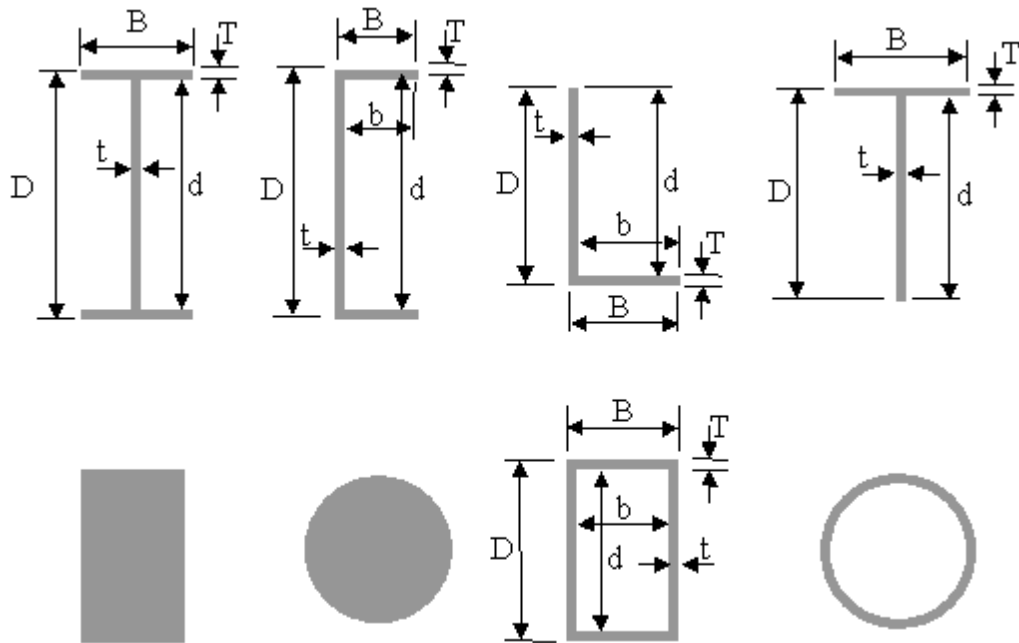
The combined stresses are:

$$C = A + B(y, z)$$

The shear stresses are:

$$S_y = \frac{F_y}{area_y}, \quad S_z = \frac{F_z}{area_z}$$

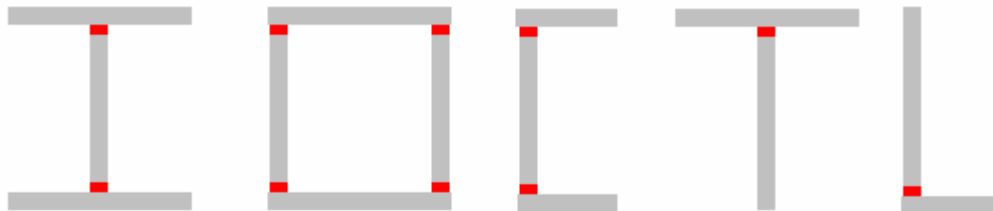
The shear stresses are of most use for steel structures so the shear stresses are calculated based on the shear area ($area_y$ and $area_z$) in BS 5950-1:2000 cl.4.2.3, and they depend on the section shape and if the section is rolled or welded. The section dimensions are shown below:



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	$0.8 \times D \times 1.6 \times B \times d \times t$		$1.8 \times B \times D \times t$			$1.8 \times B \times T$
	t	T		T		
Channel	$0.8 \times D \times 1.6 \times B \times d \times t$		$1.8 \times B \times D \times t$			$1.8 \times B \times T$
	t	T		T		
Rectangular hollow	$1.6 \times D \times 1.6 \times B \times 2 \times d \times t$		$1.8 \times B \times \frac{area \times D}{(D+B)}$			$\frac{area \times B}{(D+B)}$
	t	T		T		
Rectangular solid	$0.8 \times area$	$0.8 \times area$	$0.9 \times area$	$0.9 \times area$	$0.9 \times area$	$0.9 \times area$
Angle	$0.8 \times D \times 0.8 \times B \times 0.9 \times d \times t$		$0.9 \times B \times D \times t$			$0.9 \times B \times T$
	t	T		T		
Tee	$0.8 \times D \times 0.8 \times B \times d \times t$		$0.9 \times B \times D \times t$			$0.9 \times B \times T$
	t	T		T		
Circular	-	-	$0.6 \times area$	$0.6 \times area$	$0.6 \times area$	$0.6 \times area$

hollow			<i>area</i>	<i>area</i>		
Circular	$0.7 \times$	$0.7 \times$	$0.9 \times$	$0.9 \times$	$0.9 \times \text{area}$	$0.9 \times \text{area}$
solid	<i>area</i>	<i>area</i>	<i>area</i>	<i>area</i>		



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

$$\text{area}_y = \text{area} \times k_y, \quad \text{area}_z = \text{area} \times k_z$$

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 $2/3$ (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.

5.17.6.4 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.

5.17.6.5 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.

5.17.6.6 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 GSA theory manual.

5.17.7 2D Element Results

5.17.7.1 2D Element Displacements

The element displacements are the displacements of the flexible part of the element. In GSA displacements are calculated at degrees of freedom which for 2D elements are at nodes. However when elements are offset the nodal displacements need a transformation to the flexible part of the element.

Element rotations are not reported.

5.17.7.2 2D Element Stress Results

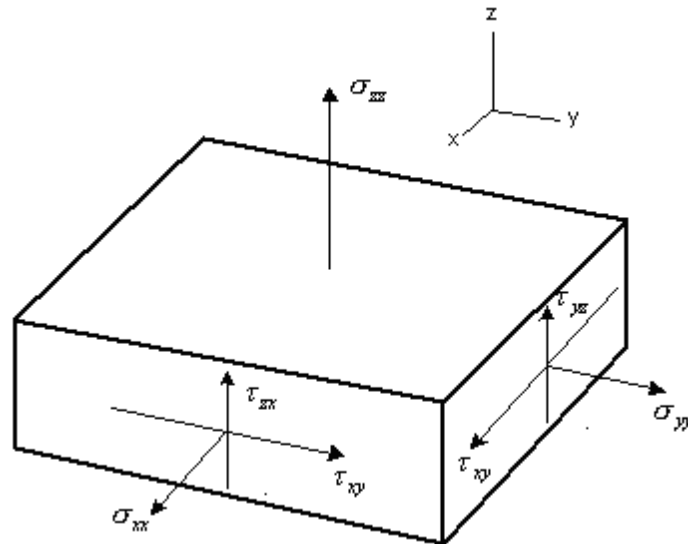
The method of calculating the stresses and forces in 2D elements depends to a certain extent on the solution method. The stresses are based on the strains, which in turn are based on the displacement gradients in the element. The stresses are calculated at the Gaussian integration points and then extrapolated to the nodes. Depending on the solution GSA either calculates the force and moment results from the stresses or by direct integration through the thickness of the element.

2D element results (forces and stresses) are most conveniently viewed as [contours](#) or [diagrams](#). Diagrams may be more useful in principal stresses with the direction information are required.

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

Projected and derived stresses

Stresses are a tensor value so there are six possible stress components that can be reported for any position on the element – three direct stresses and three shear stresses. The x and y stresses are in the plane of the element and z in the through thickness direction. When a user axis is selected the stresses are projected on to the plane of the element so that z remains normal to the element surface.

The derived stresses are useful ways of understanding how the element responds to the loading. The derived stress options are:

- **principal stresses** – stresses rotated so that there no shear stresses showing flow of stress in metals, concrete
- **maximum shear**
- **von Mises** – a measure of the distortional stress used in yield functions to see how close the material is to yield ($\sigma_{yield} - \sigma_{vm} = 0$)
- **average** – checking if material is in overall compression/tension

Details of these are given in the [GSA Theory](#) manual.

Limitations of the 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.

Checking 2D elements stresses

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

5.17.7.3 2D Element Force and Moment 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. Details of these are given in the [GSA Theory](#) manual.

The moment per unit length results are a tensor

$$M_x, M_y, M_{xy}$$

and the force results are a tensor (in-plane forces per unit length)

$$N_x, N_y, N_{xy}$$

and a vector (through-thickness shears per unit length)

$$Q_x, Q_y$$

The results

$$|M_x| + |M_{xy}| \text{ and } |M_y| + |M_{yx}|$$

carry the sign of M_x and M_y respectively.

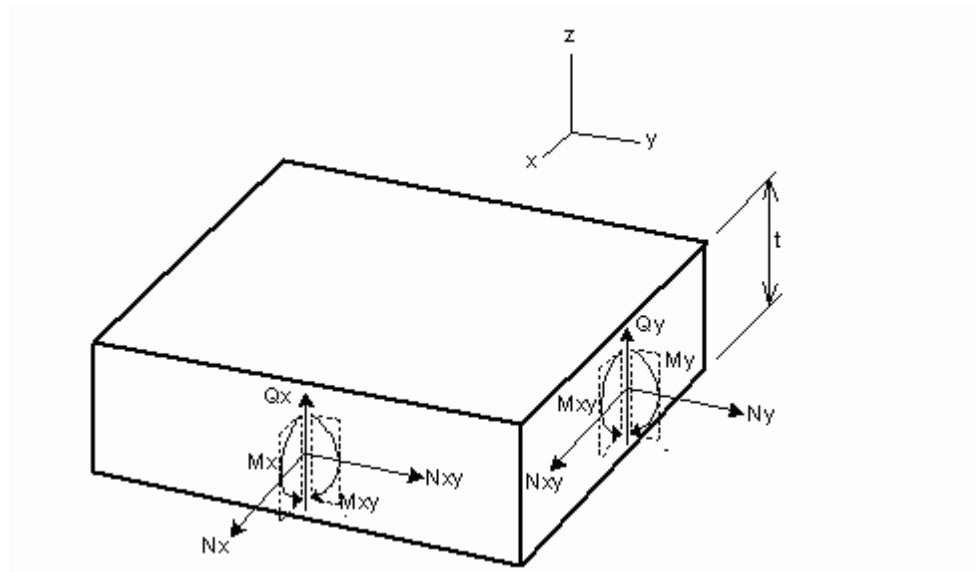
Note that the moment terms follow the Timoshenko convention in which the moment M_x is based on the stress in the x direction. With the Timoshenko convention if a slab is in compression on the top face in both the x and y directions the moments are both negative.

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).

Derived forces and moments

In-plane principal forces and principal moments can be calculated in a similar manner to [principal stresses](#):

- **principal forces** – no in-plane shear force
- **principal moments** – no in-plane twisting moment

5.17.7.4 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 [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 centre 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 centre located at the most negative side of the element Y axis to the opposite edge. The edge forces and moments are relative to the centre 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:

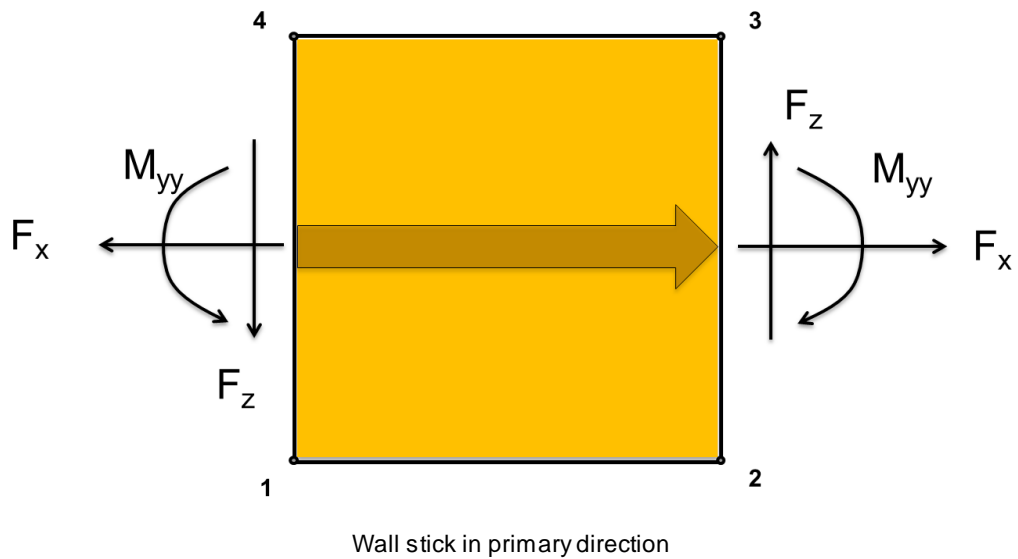
- x' – the same as wall element x axis direction

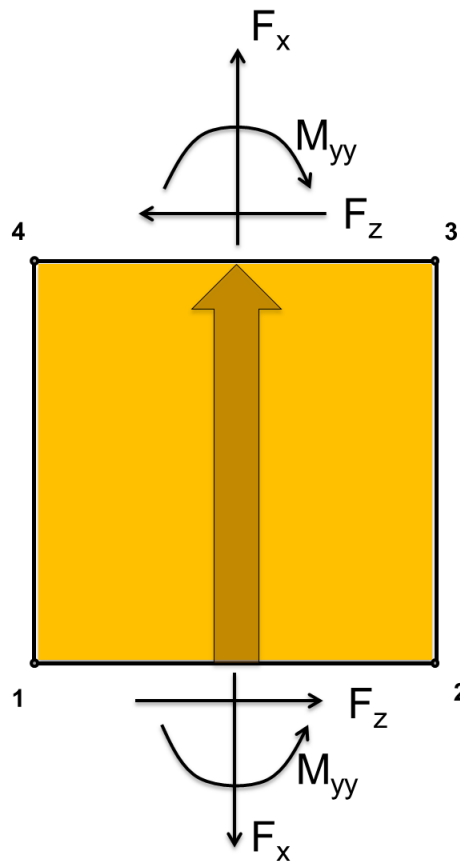
- y' – in the direction opposite wall element z axis direction
- z' – orthogonal to x' and y' (or the same as wall element local y direction)

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

5.17.7.5 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:

$$e = \sqrt{\frac{\sum (\sigma^2) - n(\sigma_{av}^2)}{n}}$$

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.

5.17.8 3D Element Results

Enter topic text here.

5.17.8.1 3D Element Displacements

The element displacements are the displacements at each node on the element. 3D elements have only translational degrees of freedom so there are no element rotations.

5.17.8.2 3D Element Stress Results

The stresses in 3D elements are based on the strains, which in turn are based on the displacement gradients in the element. The stresses are calculated at the Gaussian integration points and then extrapolated to the nodes.

Derived stresses

Stresses are a tensor value so there are six possible stress components that can be reported for any position on the element – three direct stresses and three shear stresses. The x and y stresses are in the plane of the element and z in the through thickness direction. When a user axis is selected the stresses are projected on to the plane of the element so that z remains normal to the element surface.

The derived stresses are useful ways of understanding how the element responds to the loading. The derived stress options are:

- **principal stresses** – maximum, minimum and intermediate stresses rotated so that there are no shear stresses showing flow of stress in metals, concrete
- **maximum shear**
- **von Mises** – a measure of the distortional stress used in yield functions to see how close the material is to yield ($\sigma_{yield} - \sigma_{vm} = 0$)
- **average** – checking if material is in overall compression/tension

Details of these are given in the [GSA Theory](#) manual.

Limitations of the 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.

Checking 3D elements stresses

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

5.18 Displaying Data and Results

Data and results are displayed in the following formats:

- In [Output Views](#), as tables of data.
- In [Graphic Views](#), as diagram, contours or labels.
- In [Chart Views](#), as graphs.

The topics within this section indicate how different data may be displayed.

5.18.1 Model Data Display Options

Data Type	Components ("all" unless noted)	Default axes (n/a unless noted)	Axes may be transformed	Output Views	Diagram	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				*
2D Element Shape Checks			*	
2D Element Projected Axes		*		
Nodal Stiffness (including Grounded Spring Stiffness)			*	
RC Beams	*			
Grid Planes	*			
Grid Plane Axes				*
Standard Materials	*		*	*
User Defined Materials	*		*	*
Fabric 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 Design Properties			*	
Spring Properties			*	
Non-linear Spring Properties			*	
Mass Properties			*	
	Mass		*	*
2D Element Properties			*	
	Thickness			*
	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			*	

5.18.2 Load Data Display Options

Data Type	Components	Default axes	Axes may be transformed	Output Views	Diagrams	Contours	Labels
	("all" unless noted)	(n/a unless noted)					

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	*		*		
Prestress				*			
Loads	Force	Global	*		*		
	Moment	Global	*		*		
Beam				*			
Thermal	Temperature				*		
Loads	Temperature Gradient				*		
2D Element				*	*	*	
Face Loads							
2D Element				*	*	*	
Edge Loads							
2D Element				*			
Prestress	Force	Global	*		*	*	
Loads							

	Moment	Global	*		*	*
2D Element				*		
Thermal						
Loads						
Temperature				*	*	
Gradient				*	*	
Gravity				*	*	
Loading						

5.18.3 Results Display Options

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	n/a		*	*	*

Energy Density					
Average Beam Strain	n/a		*	*	*
Energy Density					
Steel Utilisation			*	*	*
Torce Lines				*	
2D Element Displacements			*	*	*
2D Element Derived Forces and Moments	Principal 2D 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 Projected Moments	2D Moment	*		*	
	2D Moment, M_x	*	*		*
	2D Moment, M_y	*	*		*
	2D Moment, M_{xy}	*	*		*
	2D Moment, $ M_x + M_y $	*	*		*
	2D Moment, $ M_x + M_y + M_{xy} $	*	*		*

		$M_y + M_{yx} $			
2D Element	2D Force	*		*	
Projected Forces	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 Element Derived Stresses	2D Stress, Principal			*	
	2D Stress, Max		*		*
	2D Stress, Min		*		*
	2D Stress, Global Angle to Max	*	*		
	2D Stress, Max Shear		*		*
	2D Stress, Von Mises		*		*
	2D Stress, Average		*		*
2D Element Projected Stresses	2D Projected Global Stress, xx	*	*	*	*
	2D Projected Global Stress, yy	*	*	*	*
	2D Projected Global Stress, zz	*	*	*	*
	2D Projected Global Stress, xy	*	*	*	*
	2D Projected Global Stress, yz	*	*	*	*
	2D Projected Global Stress, zx	*	*	*	*

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 Reinforcement			*	*
Element Strain Energy			*	*
Assembly Displacements			*	*
Assembly			*	*

Forces and
Moments

5.18.4 Bridge Data Display Options

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				*			

5.18.5 Bridge Results Display Options

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	

5.18.6 Analysis Stage Data Display Options

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				*			

5.18.7 User Modules Display Options

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				✓		✓	

5.18.8 Analysis Diagnostics: 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 and solve for residual displacements

$$\{f_R\} = \{f\} - [K]\{u\}$$
$$\{u_R\} = [K]^{-1}\{f_R\}$$

and then the error 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:

$$e = \frac{\|[K]\{u\} - \lambda[M]\{u\}\|}{\|[K]\{u\}\|}$$

and for a buckling analysis:

$$e = \frac{\|[K]\{u\} + \lambda[K_g]\{u\}\|}{\|[K]\{u\}\|}$$

These are reported in the Analysis Details output.

5.18.9 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 , and ρ for a standard material in the three different formats.

		$E \text{ (N/m}^2\text{)}$		$\rho \text{ (t/m}^3\text{)}$
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](#).

Part VI

6 Design in GSA

6.1 Steel Design

For design to work, it is necessary for Member definitions to be complete and consistent. The recommended approach for creating a GSA model is to define members first, and then to create elements from them. This approach is also more efficient when coordinating the GSA model with a BIM model. The Create Elements from Members tool (under Model | Coordination menu) generates elements from Members.

Before starting to design members, the following must be ensured.

- [End Conditions](#) must be defined appropriately.
- Members and the corresponding element definitions must be in-sync with each other. The 'Highlight inconsistent elements' display under Graphic settings highlights Members and Elements that are out of sync.
- Steel Members must be appropriately restrained along their length.

Design Task

A Design Task must be set up and executed for designing Members.

Prior to setting to a Design Task, the following must be ensured.

1. The Design code must be specified from '[Specification | Design Specification](#)'.
2. Design grades available to the project must be populated under '[Material | Material Grades | Steel](#)'.
3. Populate the Section Library. The '[Add Catalogue Sections](#)' and '[Add Standard Sections](#)' tool under 'Model | Section Tools' can be used to add several section sizes to the library at once.
4. Group Members and assign each group to a [Pool](#). Add Sections into the relevant Pools from the Pool definition wizard.
5. Set up combination cases for strength and/or serviceability checks.
6. Set up a [Design Task](#).
7. From the Design Task view | Right click menu, choose to either design or check the task.

The Design Task uses a code checker as specified in the Design Specification. The 'Notes on Code Checks' section outlines the checks, design code references and assumptions for each code of practice.

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](#).

More:

[Tools](#)

[Results for steel design](#)

6.1.1 Generating Steel Member Restraints

In addition to [member ends](#), Steel Members must have either [internal restraints](#) or effective lengths defined to carry out buckling checks across member subspans. There are three ways of assigning Steel Member Internal restraints.

- Manually, by editing them from the [Steel Properties](#) page for a given Member. These can then be propagated to other Members using the [Modify Entities pane](#).
- Semi-automatically, by invoking the Generate Steel Member restraints tool on a list of Members.
- Automatically, while a Design Task is executed. When generating restraints automatically, GSA uses the parameters defined in the [Design Specifications](#).

6.1.2 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](#)

6.1.2.1 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	

6.1.2.2 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.

6.1.3 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.

6.1.4 Steel Checks to AISC 360-05

All cases provided to the checker are assumed to be factored loads including second order small p-delta effects. 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.

Definition

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.

6.1.5 Steel Checks to AS4100:1998

Input data

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.

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

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.

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.

6.1.6 Steel Checks to BS5950-1:2000

Input data

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.

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.

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.

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.

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.

6.1.7 Steel Checks to EN1993

EN1993 or Eurocode 3 provides a basic code supplemented by [National Annexes](#) to allow for variations from one country to another.

Input data

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.

Effective net area factor (β)

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.

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.

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.

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.

6.1.8 EN1993 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)

$$M_0 = 1.00, \quad M_1 = 1.00 \quad \text{and} \quad M_2 = 1.25$$

Clause 6.3.2.2.

Table 6.3 is used for imperfection factors.

Clause 6.3.2.3 (1)

$L_{T,0} = 0.4$, $\alpha = 0.75$ 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)

$f = 1 - 0.5(1 - k_c)[1 - 2.0(\lambda_{T0} - 0.8)^2]$ 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 value used for calculating interaction factors in any direction (say x or y) is the largest value of the following three

- 1) x or y depending on the direction.
- 2) T for torsion.
- 3) T_F 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)

$$M_0 = 1.00, M_1 = 1.00 \text{ and } M_2 = 1.10$$

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:

$$\lambda_{T,0} = 0.4 \text{ and } \lambda_{T,0} = 0.75$$

Welded sections:

$$\lambda_{T,0} = 0.2 \text{ and } \lambda_{T,0} = 1.00$$

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)

$$m_0 = 1.00, \quad m_1 = 1.00 \text{ and } m_2 = 1.25$$

Clause 6.3.2.2.

Table 6.3 is used for imperfection factors.

Clause 6.3.2.3 (1)

$\alpha_{LT,0} = 0.4$, $\alpha_{LT} = 0.75$ 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)

$f = 1 - 0.5(1 - k_c)[1 - 2.0(\alpha_{LT} - 0.8)^2]$ 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)

$$m_0 = 1.00, \quad m_1 = 1.00 \text{ and } m_2 = 1.25$$

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):

$$\alpha_{LT,0} = 0.2 + 0.1(b/h), \quad \alpha_{LT} = 0.4 - 0.2(b/h) \quad \alpha_{LT}^2 \geq 0 \text{ and } \alpha_{LT} \leq 1.0$$

Welded sections (doubly symmetric I sections):

$$L_{T,0} = 0.3(b/h), \quad L_T = 0.5 - 0.25(b/h) \quad L_T^2 \geq 0 \text{ and } L_T = 1.0$$

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 value used in interaction equations in any direction (say x or y) is the smallest value of the following three

- 1) x or y depending on the direction.
- 2) T for torsion.
- 3) T_F 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 C_1 and C_2 are taken from Table 1 and Table 2 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 1993-1-1:2007

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.

6.1.9 Steel Checks to HKSUOS:05**Input data**

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.

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.

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

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.

6.1.10 Steel Checks to IS800

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.

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 ()

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.

6.1.11 Steel Checks to SANS 10162-1:2011

Input data

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.

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.

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.

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

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 to [BS5950-1:2000](#) for other limitations.

6.1.12 Results

Running a Design Task produces the following results and output

- A report in the Report Pane as the task executes.
- Member Design History – accessed from Output Explorer | Member Results. This outline the design history for each member, recorded over multiple runs of Design Tasks.
- Member User Modules – accessed from Output Explorer | User Modules. These contain the utilizations of each Member in the Design Task post-design.

Apart from the above, GSA also presents force and displacements results for Members. These are accessed from **Output Explorer | Member 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 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 [Result Pane](#).

Select the type of results to be shown from the dropdown menu at the top of the window. 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 [Result Pane](#).

Steel design summaries, brief calculations, verbose calculations and effective lengths are also available in Output Views and the [Result Pane](#).

6.1.13 Tools

Various menu items are available to assist you in the construction and maintenance of design models. These are accessible from the Model | Coordination menu:

Menu Item	Description
Create Elements from Members	Automates the construction of an analysis model, given an existing design model.
Create Members from Elements	Automates the creation of members, given an existing analysis model.

Generate Steel Member Restraints	Automates the generation of member internal restraints
Synchronize Members and Elements	Updates member data from elements or elements from members.

6.2 Member Results

The following Results are available for Members. Note that Members are also associated with results from a Design Task run. These are described under [Steel Design | Results](#).

Result	Description
Member Displacements	Aggregate of Element displacements along the Member
Member Forces and Moments	Aggregate of Element forces and moments along the Member
Steel Utilisation	Worst utilisation from Member local strength and buckling checks
Steel Design Calcs (Verbose)	Detailed steel design calculation output for buckling checks and local strength along the member
Steel Design Calcs (Brief)	Detailed steel design calculation output for buckling checks and local strength at critical points along the member
Steel Member effective lengths	Member effective lengths calculated for each buckling sub-span along the member
Member deflection	Member displacement relative to support, as defined by restraint conditions at ends and internal points
Member span to max. deflection ratio	Ratio of deflection over the span that carries that deflection

Member Deflection and *Member Span to Max. Deflection Ratio* results require member end conditions to be set and are only applicable to members with no intermediate supports.

Part VII

7 GSA Tools

7.1 Welcome to GSA dialog

The Welcome to GSA dialog is displayed on entry to GSA and is designed to assist the user to get started quickly.

Start

New Model: Takes the user to the New Model Wizard to allow simple structures to be defined with the minimum effort.

Open Model: Browse for an existing file

Recent Model: Open a recent file

Help links (and Preferences)

Go to the Oasys website

Hide this dialog

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 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.

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 Analysis and Design Tasks

The option to undo analysis and design task results may be set in the [Input Data Preferences](#). When this is set, an analysis or design task 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.

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.

7.3 New Model Wizard

7.3.1 Titles

This allows the user to enter the job details. By default the job details of the previous job are used.

Definition

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, Subtitle & Calc Heading

The title of the job, the subtitle that this model relates to, and calculations 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.

7.3.2 Structure

This page sets up the basic details of the structure. Cancelling at any time will result in no new document.

Definition

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 stress** – 2D analysis option with plane stress conditions.
- **Plane strain** – 2D analysis option with plane strain conditions.
- **Axisymmetric** – 2D analysis option with axisymmetric conditions.

Previous versions of GSA offered 'Plane' and 'Grid' structure types. These have been removed as structure types but the same conditions can be achieved with a Space structure and setting the appropriate global restraints.

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.

Design Codes

Selects the steel and concrete design codes. These can be changed later, but code changes require mapping of some of the data, so it is useful to settle on the appropriate design code at an early stage.

Grid Data

[Grid lines](#) and [grid planes](#) can be assigned to give a framework for subsequent model generation.

7.3.3 Properties

Definition

Material Grades

In general steel and concrete grades will be needed when creating a model. This allows steel and concrete grade to be selected based on the selected design codes.

Sections

Sections are one of the key items in many GSA models. If there are a set of sections that relate to the current project it is convenient to be able to set these up as group of sections, rather than have to set up each individually. Options are provided for [catalogue sections](#) and [standard sections](#). Along with this a standard slab profile can be defined.

Generate Data

Selecting '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.

7.4 Data Generation Wizard

7.4.1 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](#)
- [Orthogonal frame](#)
- [Truss](#)
- [Vierendeel](#)
- [Pratt truss](#)
- [Pitched portal](#)
- [Roof truss](#)
- [2D element orthogonal mesh](#)
- [2D element grid](#)
- [2D element circular grid](#)
- [3D element orthogonal mesh](#)

Selecting one of these will lead directly to the page to define this structure.

7.4.2 Portal Frame

The information supplied on these pages provides a description of a 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.

Definition

Generation

The user can choose to create analysis and/or design models with

- **Generate members**
- **Generate elements**

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.

.

7.4.3 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.

Definition**Generation**

The user can choose to create analysis and/or design models with

- **Generate members**
- **Generate elements**

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.

Beams

Opens the [Section Wizard](#) to allow the section to be selected for the beams spanning in the x and y directions. 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.

7.4.4 Orthogonal Frame

The information supplied on these pages provides a description of an 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.

Definition**Generation**

The user can choose to create analysis and/or design models with

- **Generate members**
- **Generate slabs for floors**
- **Generate elements**

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.

Slab

Open the 2D Property Wizard to allow definition of the slab properties to apply to floor members.

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

If this box is ticked, generate a storey [grid plane](#) at each level.

7.4.5 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.

Definition

Generation

The user can choose to create analysis and/or design models with

- **Generate members**
- **Generate elements**

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

The end style of the truss can be selected from

- **Cornice** – overhanging.
- **Facia** – vertical.
- **Mansard** – sloping.

Use Bars for Bracing

Bar elements should be used for bracing. By default beam elements will be used.

Top & Bottom Chords, Diagonals & Verticals

Opens the [Section Wizard](#) to allow the top and bottom chord, bracing and vertical sections to be selected. This is marked as unspecified until a section is assigned when the actual section will be displayed.

The verticals are only relevant if the facia option is selected, otherwise this item is disabled.

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.

7.4.6 Vierendeel

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.

Definition

Generation

The user can choose to create analysis and/or design models with

- **Generate members**

- **Generate elements**

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.

Top & Bottom Chord & Verticals

Opens the [Section Wizard](#) to allow the top and chord and vertical sections to be selected. 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.

7.4.7 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.

Definition**Generation**

The user can choose to create analysis and/or design models with

- **Generate members**
- **Generate elements**

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.

Use Bars for Bracing

Bar elements should be used for bracing. By default beam elements will be used.

Top & Chord, Diagonals & Verticals

Opens the [Section Wizard](#) to allow the top and bottom chord, diagonal and vertical sections to be selected. 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.

7.4.8 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.

Definition

Generation

The user can choose to create analysis and/or design models with

- **Generate members**
- **Generate elements**

Bay Width, Number of Bays

The width of the structure is defined by the number of bays of equal bay width.

Eaves & Eaves-ridge Height

Specify the height from ground to the eaves and the height from the eaves to the ridge.

Frame Spacing, Number of Frames

The length of the structure is defined by the number of frames of equal frame spacing.

Column, Rafter & Beams

Opens the [Section Wizard](#) to allow the column, rafter and beam section to be selected. This is marked as unspecified until a section is assigned when the actual section will be displayed.

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. By default no loading is applied to the model.

7.4.9 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.

Definition

Generation

The user can choose to create analysis and/or design models with

- **Generate members**
- **Generate elements**

Width & Depth

Specify the width of the truss between supports and 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, Ties & Bracing

Opens the [Section Wizard](#) to allow the rafter, tie and bracing section to be selected. 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.

7.4.10 2D Element Orthogonal Mesh

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.

Definition**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

This generates spacer elements along the 2D element edges for use in form finding.

Internal & Edge

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) and around the edge.

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.

7.4.11 2D Element Skew Mesh

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.

Definition**Coordinates**

Specify the coordinates of the four corners of the grid. The program will generate a mesh within 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

This generates spacer elements along the 2D element edges for use in form finding.

Internal & Edge

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) and around the edge.

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.

7.4.12 2D Element Elliptical Mesh

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.

Definition

Bottom, Top and Height

Specify the major and minor axis dimensions at the bottom and top of the mesh. If height is non-zero the mesh will be 3D, and if zero it will form a planar 2D mesh.

Blend to Square

If selected, the bottom or top 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

This generates spacer elements along the 2D element edges for use in form finding.

Internal & Edge

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) and around the edge.

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.

7.4.13 3D Element Orthogonal Mesh

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.

Definition**Geometry**

Specify the coordinates for left-right, front-back and bottom-top surfaces

Number

Specify the number of elements to generate in the three primary directions.

7.4.14 Generate

This page confirms that you have entered the data to generate a model if you select Finish, the model will be generated.

7.5 Preferences & Settings

7.5.1 Preferences

7.5.1.1 Basic

Basic preferences are those that apply across the program.

Definition**Company Information, Language, File Locations, Page Setup**

The company logo, specified as a bitmap and name can be adjusted as required.

GSA has limited support for languages other than English. Select the language as appropriate.

The file locations specifies the [Preferred Files and Folders](#) where GSA will look for various data and temporary files

Page setup controls the appearance of printed output - borders, logos, etc.

Tolerances, Units, Section, Design

GSA models hold a set of [tolerances](#). Preferred values for these can be stored for creation of new models.

The units set up a preferred set of [units](#) for use when creating a new model.

GSA has a number of section catalogues available. This allows a preferred catalogue to be selected. The section wizard will then default to this catalogue.

The design option allows preferred [design codes](#) to be selected.

Data IDs

When there are references between modules it is possible to reference by number, name or name and number. This sets the preferred option.

Labels

Set preferences for labelling restraints, releases and rigid constraints. For restraints and releases the choice is between “Text” labelling or “Symbol” to indicate the type of restraint or release. For rigid constraints the option is to include or exclude dots at the slave and master nodes.

Refer to [Labels](#) for more details.

Property Tips

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 [Message Pane](#) 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](#).

Geometric Entities

The geometric entities (lines, areas and regions) have largely been replaced by 2D members, so are hidden by default.

7.5.1.2 Graphics

This page handles preferences that relate to Graphic Views.

Definition

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.

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.

7.5.1.3 Sculpting

This page handles preferences that relate to sculpting and to entity selection in Graphic Views.

Definition**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, 0D 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 with click

When checked, previous selections are not emptied by a plain click or drag. Otherwise the shift key must be used while selecting, to append to the selection.

Highlight selected elements by line

When checked, selected entities 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 list expanded

Lists are normally copied collapsing ranges to condense the string. This option permits list to be copied with each item included explicitly. The separator can be select to suit.

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 °

.

7.5.1.4 Input

This page handles preferences that relate to the input data.

Definition**Precision**

Defined the precision to which new nodes are located.

Show Drop-down Lists

When tables require a selection from a set of options this can either be displayed as a conventional drop-down, or can be an edit file which matches the options as the user types.

Number of analysis case factors in combination tables

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 and Redo](#) for more details.

Undo analysis and design task results

Enables the option to undo analysis and design. Refer to [Working with the Program – Undo and Redo](#) for more details.

Material models

GSA has a limited number of material models which can be used in analysis, however it has links to other programs which include many more material models. This provides an option to offer material models specific to [LS-DYNA](#) and [OpenSees](#).

7.5.1.5 Result

This page affects the way output is presented.

Definition**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.

Units

The unit separator when outputting dimensional units

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”.

Entity Output

Selecting text output for a large model with many cases can be slow. To speed up the output it can be limited by default to a set number of entities when the total number of entities exceed the user defined limit. This can be adjusted/overridden in the output specification.

2D/3D element analysis results

Full results for 2D and 3D elements (i.e. results at all topology items) can use lots of memory and this level of detail is often not required, so the results at centre only allows the user to only keep one set of results per element - the values at the element centre.

Selecting text output for a large model with many cases can be slow. To speed up the output it can be limited by default to a set

7.5.1.6 Miscellaneous

This page handles miscellaneous preferences that do not fit clearly into any other category.

Definition**Messages**

It is often useful to be able to limit the number of error and warning messages that are displayed. Limiting the number of messages is achieved by setting the number of messages. Selecting 'omit lists in

notes' reduces the amount of output by giving the details but not a list of entities to which that applies.

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. Other options allow tailoring of what is included in the backup.

Startup

The 'Welcome' dialog when opening GSA is optional and is controlled by the [Show Welcome](#) dialog option. When opening a file initial graphic and output views can be activated.

Batch Output

Batch 'save to file' options for *Saved Graphic Views* and *Saved Output Views* will be saved in the format specified here.

COM Objects

When using the GSA COM interface it can be beneficial to create a new GSA instance for each COM object.

Solver options

GSS – The default solution method for GSS should be set up in the '[Preferred Solution](#)' dialog box. The LS-DYNA environment to launch when setting up an LS-DYNA task can be specified. Also it is possible to set the D3PLOT version to be used when exporting from GSA. 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. 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.

LS-DYNA – GSA can be used to create analysis tasks for LS-DYNA, but has no direct link with LS-DYNA. The user can set up a link to the LS-DYNA shell environment to provide a direct connection from GSA.

Advanced – The advanced preferences allow threading to be controlled and diagnostic information to be requested.

7.5.2 Miscellaneous Preferences

Enter topic text here.

7.5.2.1 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 [Basic](#) page of the 'Preferences'.

Definition

Resolution of png and jpeg

The resolution of images can be either set to *screen resolution* or selected *print resolution*. The latter is preferred for higher quality output.

Image size

Adjust the image size based on the window size or specify a size directly.

7.5.2.2 Preferred Section

The Preferred Section dialog is available from the [Basic page](#) of the Preferences.

Definition

The default steel section catalogue is selected. The other catalogues remain available for selection in the [Section Wizard](#).

7.5.2.3 Preferred Folders

The preferred folders dialog is available from the [Basic](#) page of the Preferences. This is where the user can select locations for files.

Definition

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.

Mappings Database

Pathname for the file holding the section mappings for the GSA to Revit link.

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.

7.5.2.4 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.

Enforce old OpenGL in Graphic Views – <TO-DO>.

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.

Antialiased fonts in Graphic Views – <TO-DO>.

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 model data – when checked, undo is enabled for edits carried out on the model. Refer to [Working with the Program – Undo and Redo](#) for more details.

Use whole model undo logic – changes the way that data is saved to support undoing sculpting and bulk tool edits.

7.6 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.

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 pane](#).

7.6.1 Sculpt Dialogs

7.6.1.1 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:

[Creating user axes graphically](#)

7.6.1.2 Connect 1D Elements

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. Also, a 1D element aligned with the edge of a 2D element will be split at each 2D element node.

The elements that are to be connected must be [selected](#) prior to opening this dialog box. Two or more elements must be selected.

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.

7.6.1.3 Split 1D Elements

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” or “Sculpt | Geometric Entity Operations | Split Lines” 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.

7.6.1.4 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

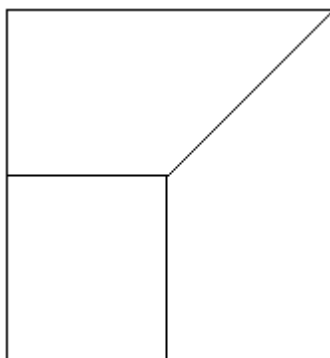
Display the consequences of this operation in the current Graphic View without committing to applying the operation. Exiting this dialog box by “Cancel” will “undo” the preview.

See also:

[Splitting elements graphically](#)

7.6.1.5 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 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:

[Splitting elements graphically](#)

7.6.1.6 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.

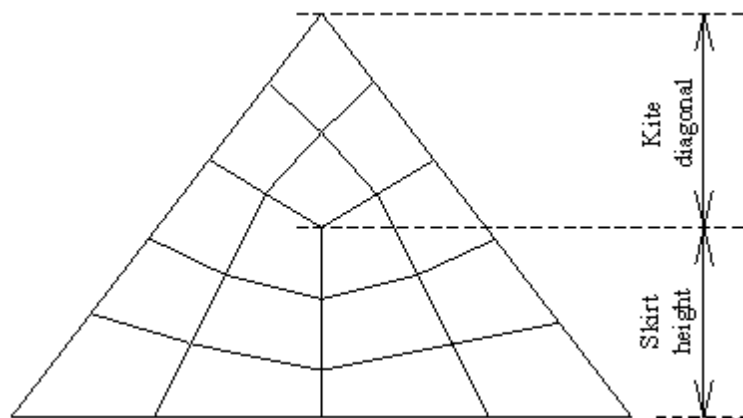
The split may be specified in one of the following ways:

- **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:

[Splitting elements graphically](#)

7.6.1.7 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:

[Disconnecting elements graphically](#)

7.6.1.8 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.

Axes

Opens the [Axes dialog box](#). The move or copy performs coordinate transformations with respect to the current 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 definition 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 definition 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 definition axes.

Number of copies

Specifies the number of copies to be made. This is disabled when moving entities or when copying by reflecting.

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.

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.

Further options for moving and copying can be found in [Moving and copying entities graphically](#) and [Definition Axes](#)

7.6.1.9 Extrude

A set of nodes, elements or members must be [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 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.

Extruding a set of members

If the extrusion is based on a set of members then copies of the selected members are included at each increment. If the **Include 2D members along extrusion** option is selected a 2D member is created between each increment for each 1D member 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:

[Extruding nodes and elements graphically](#)

7.6.1.10 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 Line of Nodes" menu command when the current view is a Graphic View. "Flex Line of Nodes" 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 Line of Nodes" 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:

[Definition Axes](#)

7.6.2 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.

7.6.2.1 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.

7.6.2.2 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.

7.6.2.3 Add Elements Sculpt Tool

The type of element created by this tool is as set in the [Data Defaults pane](#). 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.

7.6.2.4 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.

7.6.2.5 Modify Entities Sculpt Tool

Click on an existing node, element, member, line, area or region. The attributes of the selected entity are immediately changed to the data defaults. This may be useful for applying restraints to nodes. Another way of achieving the same is to use the [Modify Elements sculpt command](#).

7.6.2.6 Smart Create Areas/Region Tool

7.6.2.6.1 Create Areas from Lines

The command is used to find and create areas from a set of lines selected by the user:

1. The command finds one prevalent plane (the one which contains the most of the selected lines).
2. The selected lines are then searched for closed loops on that plane. The closed loops are used to create new areas.
3. If area already exists, it is not created again.
4. The area attributes are set according to default GSA settings for areas.
5. If an area consists of smaller areas then the outer area is not created.
6. Created areas cannot overlap. If two areas would overlap the one made of longer lines is given priority and the other is ignored.

Tolerance

The GSA preference "Tolerance for coincidence" affects the command behaviour:

1. When the prevalent plane is being determined.
2. When evaluating if a particular line is on the plane.

Coincident nodes

The user is advised if a selection contains coincident nodes. Note that a loop is not closed if a vertex is formed of two nodes although they have identical coordinates. The GSA preference "Tolerance for coincidence" affects the search for these nodes.

7.6.2.6.2 Create Areas on Grid Plane(s)

This command is a variation of the [Create areas from lines](#) command. There are two distinctions:

1. The plane to create areas on is given by the chosen grid planes.
2. No line selection is required. All lines on the specified plane are used.

7.6.2.6.3 Create Region from Objects

This command is a variation of the [Create areas from lines](#) command but is more complex. Nodes, lines and areas can be selected. The command creates areas and includes them in a region in a single run.

1. The command finds one prevalent plane from the input (as in the Create areas from lines command) and creates areas.
2. These areas and selected objects are included in the created region.

Lines intersection

The user is advised if the selection contains intersecting lines. If the lines do not form a closed loop (i.e. an area) then it is assumed they are to be part of a mesh. For that reason the user is offered a choice to split such lines in their intersections.

7.6.2.6.4 Create Region(s) on Grid Plane(s)

This command is a variation of the [Create areas on grid plane\(s\)](#) command. There are two distinctions:

1. The plane to create regions on is given by the chosen grid planes.
2. No object selection is required to run the command. All nodes, lines and areas on the specified plane (s) are found and worked with.

7.6.3 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.

7.6.4 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 is the new axis, if required. The option is given to set this new grid plane as the current grid.

7.6.5 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 pane](#).
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.

7.6.6 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 [Modify Entities pane](#) when the cursor mode is set to Select Nodes. “Modify” 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 [Nodes](#) dialog.

7.6.7 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.)

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.

7.6.8 Rounding Nodal Coordinates Graphically

Nodal coordinates can be rounded to the nearest specified length using the “Sculpt | Node Operations | 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.

7.6.9 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.

Create Lines from 1D Elements

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.

7.6.10 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](#). The [Adding a string of 1D elements](#) and [Create 1D Elements from Lines command](#) also allow for simple sculpting of elements.

Other non-graphical tools are also available for constructing a model.

7.6.10.1 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 pane](#).
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 pane.
4. Select grid points or nodes. 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.
5. Give the “Add String of 1D Elements” command.

7.6.10.2 Create 1D Elements from Lines

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 pane](#). 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.

7.6.11 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.

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.

A 1D element aligned with the edge of a 2D element will be split at each 2D element node.

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 or 2D will be ignored by this operation and no action will be taken if only 2D elements are selected.
3. Give the "Connect 1D Elements" command.
4. Specify how the elements are to be split in the [Connect 1D Elements](#) dialog box.

7.6.12 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.

Splitting 1D Elements

1D elements can be split using the "Sculpt | 1D Element Operations | 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](#).

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 | 2D Element Operations | 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 | 2D Element Operations | 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 | 2D Element Operations | 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 | 2D Element Operations | 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.

7.6.12.1 Splitting 1D Elements

1D elements can be split using the “Sculpt | 1D Element Operations | 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.

- Specify how the elements are to be split in the [Split 1D Elements dialog box](#).

7.6.12.2 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 | 2D Element Operations | Split Quad Elements" menu command. The procedure is as follows.

- Set the [cursor mode](#) to Select Elements.
- Select the elements to be split. Elements that are not Quad 4 or Quad 8 will be ignored by this operation.
- Give the "Split Quad Elements" command.
- 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.

- Set the [cursor mode](#) to Select Nodes.
- Select the node at which attached 2D elements are to be refined. Non-2D elements will be ignored by this operation.
- 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 | 2D Element Operations | 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.

- Set the [cursor mode](#) to Select Elements.
- Select the elements to be refined. Elements that are not Quad 4 or Quad 8 will be ignored by this operation.
- Give the "Refine Quad Elements" command.
- 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 | 2D Element Operations | Split Quad to Tri Elements" menu command. Each quad element is split

into two tri elements such that the split occurs 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 | 2D Element Operations | 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.

7.6.13 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.

7.6.14 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.

7.6.15 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.

7.6.16 Modifying 2D Elements from Parabolic to Linear

Parabolic 2D elements (i.e. Tri6 and Quad8) can be converted to linear (i.e. Tri3 and Quad4, respectively) using the “Sculpt | 2D Element operations | Modify Parabolic 2D to Linear” 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 Parabolic 2D to Linear” command.

7.6.17 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.

7.6.18 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.

7.6.19 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 + 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 | Node Operations | 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.

7.6.20 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 Line of Nodes” menu command. “General commands | Flex Line of Nodes” 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 Line of Nodes” 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.
 - o The nodes are ordered along a vector pointing from the most extreme node to the centre of the set of nodes.
 - o The end nodes are identified.
 - o 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:
 - o Any of the set of nodes may be selected as the shift node.
 - o The nodes to be flexed are dispersed along the line(s) formed between the end nodes and the shifted node.
 - o “Even” spacing results in separate, even spacing on either side of the shifted node.
 - o “Proportional” spacing results in spacing that is proportional to the position of the nodes **on the flex line**.
4. When flexing in circular shape:
 - o Only internal nodes (not end nodes) of the set of nodes may be selected as the shift node.
 - o 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.
 - o 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.
 - o Spacing is then based upon angles around the centre.
 - o “Even” spacing results in separate, even angles on either side of the shift node.
 - o “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.

7.6.21 Straightening Lines of Nodes Graphically

Ragged lines of nodes can be straightened using the “Sculpt | Node Operations | Straighten Spacing Proportionally” and “Sculpt | Node Operations | 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.

7.6.22 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. The procedure is as follows.

1. Set the [cursor mode](#) to Select Elements.
2. Select the elements to be flipped.
3. Give the “Flip 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 flipped with the elements.

7.6.23 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 | 2D Element Operations | Spin 2D Elements” menu command. The procedure 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.

7.6.24 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 [Modify Entities pane](#) when the cursor mode is set to Select Elements. “Modify” 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 Entities dockable pane.

7.6.25 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](#).

7.6.26 Deleting Nodes and Elements Graphically

Nodes and elements can be deleted using the “Edit | Delete” (Del) menu command.

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 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” command.

Note that the option to delete references to the entity is offered.

7.6.27 Creating Rigid Constraints Graphically

[Rigid constraints](#) can be created using the “Sculpt | Constraint Operations | Create Rigid Constraint” 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 Constraint” 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.

7.6.28 Creating Joints Graphically

[Joints](#) can be created using the “Sculpt | Constraint Operations | 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.

7.6.29 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.

7.6.30 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)
 - Prestress
 - Distortions
 - Thermal
- 2D Element Loading (only offered when 2D element analysis is enabled)
 - Face Loads
 - Edge Loads

- Prestress
- 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 Prestress Definition](#), [Beam Distortion Definition](#), [Beam Thermal Load Definition](#), [2D Element Face Load Definition](#), [2D Element Edge Load Definition](#), [2D Element Prestress 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.

7.6.31 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](#).

7.6.32 Deleting Loading Graphically

Any loading applied to nodes or elements 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. Give the “Delete Displayed Loading” command.

The consequence of this operation is that the lists in the affected loading records have “not (<deleted entities>)” appended unless the resulting list is empty, in which case the loading record is deleted.

7.7 File Tools

7.7.1 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.

7.7.2 Footfall Response Data

This dialog can be accessed from the “File | Export | Footfall Response (CSV file)” menu command. This allows control of the data written for analysis in the Footfall Response spreadsheet.

Definition

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)

Check one of the mode shape directions to be used in the export.

7.8 Output View Dialogs

7.8.1 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.

Definition**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 [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 Output Views](#)

[Working with Saved Views and Preferred Views](#)

[Working with Sets and Lists](#)

7.8.2 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](#).

Definition**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. Otherwise 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.

Output in spreadsheet format

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.

See also:

[Enveloping in GSA](#)

7.9 Graphic View Dialogs

7.9.1 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.

Definition**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 [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 [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 [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 [Printing from Graphic Views](#). This also affect the saving of the image 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 [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 [Analysis and Design Layers](#) for details.

Stage

Select the analysis stage to be displayed.

Refer to [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 [Labels 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](#). Commonly used contour options are available on the [Contours toolbar](#).

[Diagram](#) – “Graphics | Display | Settings | Diagram Settings” menu command or via the [Display toolbar](#). Commonly used diagram options are available on the [Diagrams 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.

Further details can also be found in [Working with Graphic Views](#), [Working with Saved Views and Preferred Views](#) and [Working with Sets and Lists](#).

7.9.2 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 [Labels 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](#)

7.9.2.1 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 grid plane IDs**
- **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](#)

7.9.2.2 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 IDs** – also displays names for grid nodes.
- **Constraint axis IDs** 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.
- **Tied interfaces** – Each node that is part of an interface has a brown dot.
- **Raft interaction numbers** are labelled in the format “R” followed by the interaction number.
- **Raft interaction area** is drawn in orange over the raft.
- **Pile interaction numbers** are labelled in the format “P” followed by the interaction number.
- **Pile interaction area** is drawn in brown to the depth of the pile..

Data references, such as constraint axis references, may be labelled 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.

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](#)

7.9.2.3 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 IDs**
- **Property IDs** – labelled in the format “pb” (etc. depending on element type) followed by the number.
- **Member end condition IDs** – labelled in the format “c” followed by the number.
- **Member pool IDs** – labelled in the format “p” followed by the number.
- **Group numbers** – labelled in the format “g” followed by the number.
- **Analysis material IDs** – labelled with the material name. Analysis material names are preceded by “m” followed by the number.
- **Material grade IDs** – labelled with the grade name. Grade names are preceded by “st”, “c”, “a”, “g”, “frp”, “r” or “t” 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** – 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.
- **Element edges** – an arrow aligned with edge one is drawn from the first node to the second node of each 2D element.
- **Initial stage IDs** – labelled in the format “s” followed by the number of the first analysis stage in which

the element appears.

- **Element lengths** – 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.
- **Member end conditions** – drawn with a symbol which is a stylized representation of the restraint.
- **Member restraints**
- **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. 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](#)

7.9.2.4 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**
- **Line IDs** – displays both numbers and names.
- **Constraint axis IDs**
- **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 IDs** – displays both numbers and names.
- **Area types**
- **Property IDs** 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 IDs** – displays both numbers and names.
- **Region types**

Assembly label options are:

- **Assembly IDs** – displays both numbers and names.
- **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. 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](#)

7.9.2.5 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**
- **Outline Edges**
- **Outline Edges & Filled**
- **Outline Edges, 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.

3D Elements may be drawn as follows.

- **Outline**
- **Outline & Filled**
- **Outline Edges**
- **Outline Edges & Filled**
- **Filled**
- **Off** – 3D elements are not drawn.

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 pool**
- **By analysis material**
- **By material type**
- **By material grade**
- **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.

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 a line from the node to the offset position for element topologies that

have offsets defined.

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 polylines** option draws all polylines.

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](#)

7.9.3 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.

For additional information see [Deformed image](#) and [Scaling of diagrams, contours and deformations](#)

7.9.4 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](#).

Definition

Data (tree control)

The selected data will be contoured for the first case only in the Cases list on the [Lists toolbar](#).

RC Slab Reinforcement

This contour setting will be offered in the tree control if RC Slab design is enabled and if the RC Slab design code has been set to a valid code, or to “undefined”.

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/3D 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/3D 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/3D Element Results, Average stresses and forces at nodes

When checked, results are averaged across elements at common nodes. (See [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.

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 [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, ...

Apply

Applies the current settings to the image without exiting the dialog box.

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](#).

For more information see [Contours](#), [Scaling of diagrams, contours and deformations](#) and [Displaying Data and Results](#)

7.9.5 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 [Diagrams 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](#).

Definition

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 [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](#). 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:

[Diagrams](#)

[Scaling of diagrams, contours and deformations](#)

7.9.6 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.

Definition**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.

7.9.7 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.

Definition

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.

7.9.8 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 | Animate” menu command or from the [Orientation toolbar](#).)

Definition

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 ($0.1\text{s} < \text{modal period} < 60\text{s}$).

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:

[Animation](#)

7.9.9 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](#).

Definition

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:

[Changing the orientation of the image](#)

7.9.10 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.

Definition

Colour over-rides

Over-ride the default colours to produce **Grey in-fill**, **Black lines**, **Black text**, **Black symbols** or **White background** on **Print** or **Screen**.

Negative colours

Inverts the default colours on screen to produce, for example, the image on a black background.

Scaling factors

Factor the default **Font height** or **Symbol size** on **Print** or **Screen**.

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.

7.10 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](#)

[Harmonic analysis](#)

[Periodic load analysis](#)

[Linear time history analysis](#)

[Footfall analysis](#)

[Storey displacements and forces](#)

[Forces on 2D element cut](#)

7.10.1 Non-linear Analysis

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. Component 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.

7.10.2 Modal Analysis Details

This dialog is accessible from the “View | New Chart View | Mode 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.

7.10.3 Harmonic Analysis

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 Harmonic 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.

7.10.4 Periodic Load Analysis

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.

7.10.5 Linear Time-history Analysis

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 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.

7.10.6 Footfall Analysis

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 curves can be used to tell how far/close of the calculated responses from the criteria. 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 to be used 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.

7.10.7 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.

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.

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.

7.10.8 Forces on 2D Element Cut

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.

7.11 Model Tools

7.11.1 Manage Data

This dialog is accessible from the 'Model | 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.

Nodes

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.

Elements**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 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.

Properties and materials

Blank unused properties and materials

Deletes unused properties and materials without renumbering anything. If 'Exclude sections' is ticked no beam sections are deleted.

Delete blank properties and materials

Deletes all blank properties and materials, renumbering subsequent records and the references to these in other modules. References to blank records are also deleted.

Loading

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.

7.11.2 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.

Definition

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.

7.11.3 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.

Definition

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 not set, the minimum and maximum x and y extents of the region of grid points.

7.11.4 History

This dialog is accessed from the 'Model | History' menu command.

Definition

A history of automatically recorded events is displayed in chronological order. An event is an occasion when the model has been opened in GSA and saved. The date, time, duration of each event is reported together with the name of the user.

Notes

A note can be added to existing items and new items to record significant stages in the model history.

New

Creates a new history record.

Delete, Delete All

These delete individual or all records. This might be of use is an old model is to become the basis of a new model.

7.11.5 Coordination

7.11.5.1 Create Elements from Members

If a model is defined in the design layer then it is relatively straightforward to create an analysis model from this. For the selected members the properties of the members allow the generation of appropriate 1D elements and 2D elements or rigid diaphragms as appropriate.

A check allows existing elements to be retained or overwritten with new elements based on the member definition.

7.11.5.2 Delete Elements from Members

The idea is that models are generated and edited in the design layer. To allow for rapid adjustment of the model this allows for rapid removal of the elements associated with a member, to allow the member to be updated prior to regeneration of the analysis model.

7.11.5.3 Synchronise Members and Elements

The process of design can leave members being assigned new sections, leading to incompatibility between the sections assigned to members and those assigned to the corresponding elements.

In addition it is possible for other attributes such as orientation and offsets to get out of sync.

This option allows either elements to be updated from members or vice versa.

7.11.5.4 Create New Model from Deformed Geometry

Using a deformed shape from analysis and new model is created reflecting the deformed geometry.

Definition

Case

The deformed geometry is specified as an analysis case.

Scaling

The deformed geometry can be scaled by a factor or to a predefined maximum displacement.

Orientation

The orientation angle can be adjusted to take account of the deformation.

7.11.5.5 Extract 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.

Definition

Titles

The titles and job details to include in the new model

Sub-model & Tasks

This selects the part of the model to be extracted, and the analysis tasks whose results are required. For dynamic tasks there are option for global versus local treatments See below.

Options

There are a number of options to guide the creation of the sub-model. The most significant is the option to trim rigid constraints that cross the boundary introduced by 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 modal 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

$$K_i - \lambda_i M_i = 0$$

This can be rearranged in the form

$$K_i = \lambda_i M_i$$

Which we can consider as a static (pseudo modal) analysis of

$$K_i = p_i$$

where

$$p_i = \lambda_i M_i$$

When extracting the sub-model the frequency (eigenvalue) and mode shape (eigenvector) can be used to create a set of node loads:

$$^n p_i = \lambda_i ^n M_i$$

$$^n p_i = (2 \pi f_i)^2 \cdot ^n M_i$$

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.

7.11.5.6 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.

Definition

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.

7.11.5.7 Rationalize Sections for BIM Coordination

GSA does not in general make any significant distinction between beams, columns and braces, and this can result in difficulty when coordinating with BIM models. The option to rationalize section for BIM allows for sections to be separated into distinct sections for vertical and non-vertical elements or members.

7.11.5.8 Create Members from Regions

The standard approach in the past to model generation for 2D elements has been to define lines, areas and regions. This is superseded by 2D members. This option allows a translation of the geometric entities in an old model into members.

7.11.5.9 Create Members from Elements

This dialog can be accessed from the “Model | Coordination Tools | Create Members from Elements” menu command.

The Create Members from Elements command generates members from 1D elements.

Definition**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.

Creation Options

The options are:

- **One element = one member**
- **Many elements = one member**

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.

7.11.6 Section Tools

7.11.6.1 Add Catalogue Sections

This allows selection of a group of sections from a catalogue which are added to the section library.

Definition

Member Type & Material

Depending on the context the member type can be preset and a material grade is then chosen for the section. An analysis material can be assigned or the option to derive properties from the grade can be selected.

Catalogue & Type

A catalogue and type are selected and then the corresponding sections are listed and can be selected.

Name

Normally each section should be assigned a unique name, so the name specified is a template that can be expanded for the individual sections.

<g> – material grade

<t> – member type

<d> – section description

– section number

So for example for a steel section with a grade S450 and a profile UB457x191x89 and a name 'Steel <g> <d>' would result in a name Steel S450 CAT UB UB457x191x89'.

Steel Design Attributes

Details for steel design, such as net : gross area ratio can be specified.

7.11.6.2 Add Standard Sections

This allows creation of a group of sections from standard shapes which are added to the section library.

Definition

Member Type & Material

Depending on the context the member type can be preset and a material grade is then chosen for the section. An analysis material can be assigned or the option to derive properties from the grade can be selected.

Section & Dimensions

This only provide a limited number of section types - those most commonly used: rectangular, circular, tee and I sections.

The dimensions required change depending on the chosen profile. The dimensions are specified as lists, so the depth can be for example '400 500 600' for discrete sizes or '400 to 600 step 50' for ranges.

Name

Normally each section should be assigned a unique name, so the name specified is a template that can be expanded for the individual sections.

<g> – material grade

<t> – member type

<d> – section description

– section number

So for example for a concrete section with a grade C40/50 and a rectangular profile 400 × 400 and a name 'Concrete <g> <d>' would result in a name 'Concrete C40/50 STD R 400 400'.

7.11.6.3 Modify Section Properties

In many models, particularly when targeting steel design, there can be a large number of sections. It is convenient to be able to make bulk changes to groups of sections, e.g. the material grades and analysis materials, or adding sections to pools.

7.11.7 Checking Tools

7.11.7.1 Check Entity Lists for Completeness

This gives a check across multiple lists to see if they give a complete description of the model.

Definition

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.

7.11.7.2 Comparing Models

There are two ways to compare models. The option 'Model | Checking Tools | Compare with Reference Model...' prompts for a reference model and then lists some aspects that are different to the current model:

- Titles
- Specification
- Grid lines
- Planes
- Polylines
- Nodes
- Elements
- Members
- Sections
- 2D Properties
- 3D Properties
- Spring Properties
- Link Properties

It does not list what the differences are, only that there are additions, modifications, or deletions.

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.)

7.11.8 Manipulate Model

7.11.8.1 Create Mass from Loads

This dialog can be accessed from the "Model | Manipulate | Create Mass Elements | 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.

Definition

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.

7.11.8.2 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.

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.

Definition**Case Definition & Scale**

The case definition allow the loading to be specified – either a single analysis case (e.g. A2) or a combined case (e.g. 1.4A1 + A2). The overall case description can also be scaled by the scaling factor.

Expansion

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.

Elements

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.

7.11.8.3 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.

7.11.8.4 Update Superseded Sections

With time sections can be superseded - this allow old sections to be replaced with the new/updated sections. This should make minimal impact on the model

7.11.8.5 Collapse Joint Nodes

This allows for the nodes defining joints to be collapsed - effectively removing the joint.

7.11.8.6 Flip Upside-Down 2D Elements

Swaps the top and bottom faces of 2D elements.

Definition

An element list specifies the 2D elements to be flipped.

7.11.8.7 Apply Offsets

Typically models are built with beam elements connecting at nodes, but this implies an overlap as it ignores the dimensions of the elements modelled as lines.

Definition

Element List

Specifies the elements to be considered.

Offset

There are a number of offset options available. Beams can be offset to element faces (offset in xy plane), beams can be offset to be level with column nodes (offset in z), or alternatively the [reference point](#) of the section can be modified. The most general option allows offsets to be assigned to elements by individual section.

7.11.8.8 Trim Overlapping Members

The allows members to be trimmed where there is a small overlap due to tolerance issues.

7.11.8.9 Convert Spring Support to Ground Spring

A spring support is like a ground spring element attached to a node that is otherwise restrained. Spring supports are not modifiable for analysis stages so it can be useful to convert these to ground spring elements

7.11.8.10 Split Warped 2D Elements

This dialog can be accessed from the “Model | Manipulate | 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.

7.11.8.11 Rationalize Lines

This dialog can be accessed from the “Model | Manipulate | 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.

7.11.8.12 Create Grid Loads from Element Weight

This creates a set of grid loads from the element weight.

Definition

Load Case

Specifies a load case to which these loads are assigned.

Elements & Properties

The elements that will provide grid loading from their weight. This can be based on the specified properties of the elements or the analysis properties.

7.11.8.13 Clean Combination Cases

As models evolve it is possible to get combination cases which refer to non-existent analysis cases. This option allows combinations cases to be cleaned by purging references to non-existent cases.

7.11.8.14 Generate Steel Member Restraints

This is tool which generates internal restraints on members by check member connections.

Definition

Members

This defines the members that will get generated constraints.

Axial buckling

Specifies that the secondary member can provide restraint against major and/or minor axes buckling of the primary. An angle tolerance provide a limit on deciding which secondary members can provide restraint.

Lateral torsional buckling

Specifies that the secondary member can provide restraint to the top and/or bottom flange of the primary against lateral torsional buckling. An angle tolerance provide a limit on deciding which secondary members can provide restraint.

Span restraint

Specifies that the spans can provide lateral torsional restrain to the top and/or bottom flange.

7.11.9 Mesh Generation

7.11.9.1 2D Mesh from 1D Grillage

In many cases a model is constructed as a frame using beam elements and for many situations the floors can be modelled as rigid diaphragms. When a more detailed floor model is needed it is then necessary to create a finite element mesh for the floor slabs. This tool provide a quick way to model these situations.

Definition

Grid planes and tolerance

The grid planes are used to define the floors, but to ensure that the correct beam elements are considered there are three options

- **all elements** – picks up any beam elements that lie within the given tolerance of the grid planes

- **element on grid surface** – the grid surfaces on the grid planes are used and the elements considered are those that exist on the grid surface
- **element list** – this is similar to the first option with the exception that only the elements in the selected list are considered

2D Property

Choose either an existing property to assign to the mesh or create a new 2D property.

Target Element Size

GSA requires a typical element size to target for meshing. This should be small enough to ensure that the mesh is refined enough across the floor panels without creating an excessive number of elements.

7.11.9.2 2D Mesh from Regions

There are two ways to generate 2D element meshed for irregular shapes. The preferred option is to define members, but the other option is to use areas and regions. The definition of the region lines nodes, lines and areas as the base information needed to generate a mesh.

7.11.9.3 Delete 2D Mesh for Regions

IN order to keep a model up to date there is a linkage between the 2D element mesh and the region from which they were generated. It is good practice to delete the 2D mesh in order to update the region definition.

7.11.9.4 Disassociate 2D Mesh from Regions

Once a 2D mesh has been generated from regions there is a link between the two. This option allows the link between the two to be broken so that the mesh can be adjusted manually or the region changed

7.11.9.5 Check All Regions for Mesh Generation

Prior to generating a mesh it is good practice to check for any issue that might result in a failure to generate a mesh or would result in a poor mesh.

7.11.9.6 Generate Mesh from Soil Zones/Profiles

GSA provide two methods for soil structure analysis. The legacy version defines the soil with profiles and zones. For a finite element approach to modelling the soil the needs to be meshed with brick elements.

Definition

Element Size

This defines the target size for the elements in the soil mesh.

Boundary Conditions

The soil boundary in the horizontal direction can be modelled in different ways – either restraining the

boundary nodes or be creating infinite elements around the boundary.

Tolerance

The limits the distortion in the mesh that is acceptable when moving from one soil zone to another.

7.11.9.7 Delete 3D Soil Mesh

GA can generate a soil 3D mesh from the definition of soil zones and soil profiles. This options allows the mesh to be deleted prior to modifying ths zone or profile definitions.

7.11.10 Contact Constraints**7.11.10.1 Create 2D Face to Ground Contact**

It can be convenient to model a foundation stiffness under a mesh of 2D elements. This tool creates a set of springs and constraint equations to model this interaction

Definition**Elements**

For each 2D element in the element list, a node will be created at the element centre and that node will be connected to the 2D element by a constraint equation.

Stage

The stages to which this constraint should apply.

Springs

The centre node will optionally have a soil spring specified either by a nodal stiffness or an attached ground spring element.

Raft Interaction

If this is to be used for a soil-structure interaction the relevant raft interaction areas can be determined.

7.11.10.2 Create 2D Edge to Ground Contact

It can be convenient to model a foundation stiffness under a mesh of 2D elements. This tool creates a set of springs and constraint equations to model this interaction

Definition**Nodes**

For each 2D element edge represented in the node list, a node will be created at the element edge centre and that node will be connected to the 2D element by a constraint equation.

Stage

The stages to which this constraint should apply.

Springs

The centre node will optionally have a soil spring specified either by a nodal stiffness or an attached ground spring element.

7.11.10.3 Create a Joint in Disparate Axes

Joints work in the constraint axis of the nodes, but at times there may be a need to create 'joints' where the nodes have disparate axes. This can be achieved through a set of constraint equations.

Definition

Stage List

The stages to which this constraint should apply.

Master & Slave

The two nodes to be joined.

Coupled directions

The directions to be coupled referred to the local directions of the slave node.

7.11.10.4 Create a Node/Beam to 2D Element Constraint

It can be convenient to connect a node or beam element to a 2D element mesh where there is no topological connectivity. This can be achieved through a set of constraint equations.

Definition

Node/Beam

The node or beam element to be connected to the face of the 2D element.

2D Element

The 2D element which forms the other half of the connection

Link Type

The link can be either a tied or rigid constraint. The rigid option constrains the 2D element to which the node is tied.

Stages

The stages to which this constraint should apply.

Tolerance

If there is a node on the 2D element within this tolerance, the node will be linked directly to this node rather than a set of constraint equation connecting the whole face.

7.11.11 Create Grid Lines

Normally a set of grid lines will be defined prior to building a GSA model. This option tries to generate a series of grid lines based on the geometry defined in the model.

This has a reasonable chance of success for a simple model but may give unexpected results for a more complex model.

7.11.12 Building Modelling

7.11.12.1 Create Storey Grid Planes

This is available from the “Model | 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).

7.11.12.2 Create Rigid Diaphragms from Storeys

This is available from the “Model | 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.

7.11.12.3 Create Load Panels on Grid Planes

If a floor is modelled as a rigid diaphragm it is useful to be able to apply loads in a convenient way. This can be achieved in different ways but one method is to create load panels

Definition

Grid planes and tolerance

The grid planes are used to define the floors, but to ensure that the correct beam elements are considered there are three options

- **all elements** – picks up any beam elements that lie within the given tolerance of the grid planes
- **element on grid surface** – the grid surfaces on the grid planes are used and the elements considered are those that exist on the grid surface
- **element list** – this is similar to the first option with the exception that only the elements in the selected list are considered

2D Property

Choose either an existing property to assign to the mesh or create a new 2D property.

7.11.12.4 Create Load Panels on Surface

This tool gives a quick route to creating load panels on a surface.

The [Create Facade List](#) tool can be useful in identifying the nodes on facades.

Definition

Element List

An element list is used to define the surface - either the current selection, a saved list or a user defined list of beam elements.

Group

Sets a group for the load panel elements which will be created.

Properties

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.

Panel List

A new list can be created to make it easier to identify the load panels.

Tolerance

Specifies the tolerance used to determine if edges are straight.

7.11.12.5 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.

7.11.12.6 Create Base Dummy Elements

Models are typically generated with columns attached to ground. In order to use load panels it is necessary to close loops of beam (or 1D) elements. Create Base Dummy Elements creates a set of dummy beam elements to form enclosed areas. The base level is considered to be z elevation value specified in the specification.

7.11.12.6.1 Create Load Panels for Facade

This provides a shortcut way to apply load panel elements to the facade of a structure. The operation consists of

- [Create Dummy Base Elements](#) – ensure that ground level can accept load panels -
- [Create Facade List](#) – creating a list of elements on the exterior of the structure
- [Create Load Panels for Element List](#) – based on the list of elements on the exterior of the structure create a set of load panels (but with default properties)

7.11.12.7 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](#).

Definition**Elements**

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 (G_{sup} and G_{inf}) for dead and the adverse and beneficial factors (Q_{sup} and 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](#).

7.11.12.8 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](#).

Definition

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 (f_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 f_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	$A3 + (C1_{abs} + C2_{abs} + C3_{abs} + C4_{abs} + C5_{abs})$
C7	$A3 - (C1_{abs} + C2_{abs} + C3_{abs} + C4_{abs} + C5_{abs})$

See also [Create Patterned Load](#).

7.11.12.9 Create Member Timeline

When considering the construction stages of a model, particularly when creep and shrinkage are a concern it is useful to be able to assign a timeline to a member or set of members. This is not used directly by GSA at present.

Definition

Members

Select the list of members to be updated. Any existing timeline for these members will be overwritten.

Timeline

The timeline specifies when the selected members are created, and for concrete members when they start drying and age at which they are loaded.

7.11.13 Raft Modelling

7.11.13.1 Create a New Raft Model

This dialog can be accessed from the “Model | Raft Modelling | 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 raft analysis case.

Raft analysis case

The raft analysis case upon which the new support stiffnesses are to be based.

7.11.14 Bridge Modelling

7.11.14.1 Generate Static Vehicle Load

This dialog is accessible from the “Model | Bridge Modelling | 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.

Definition

Vehicle

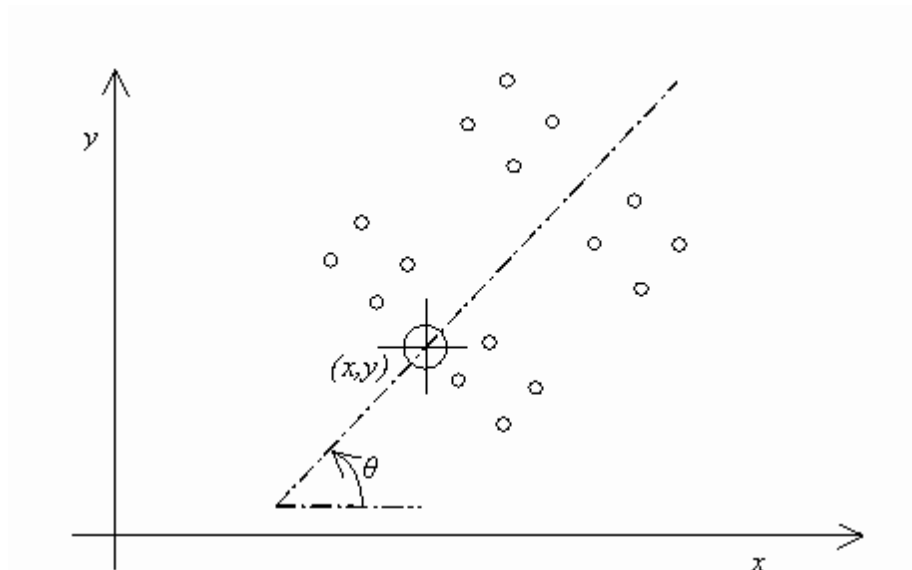
The vehicle which contributes the load. This can be either a standard vehicle or a [user defined vehicle](#).

Load Case

The load case in which the vehicle load will be generated.

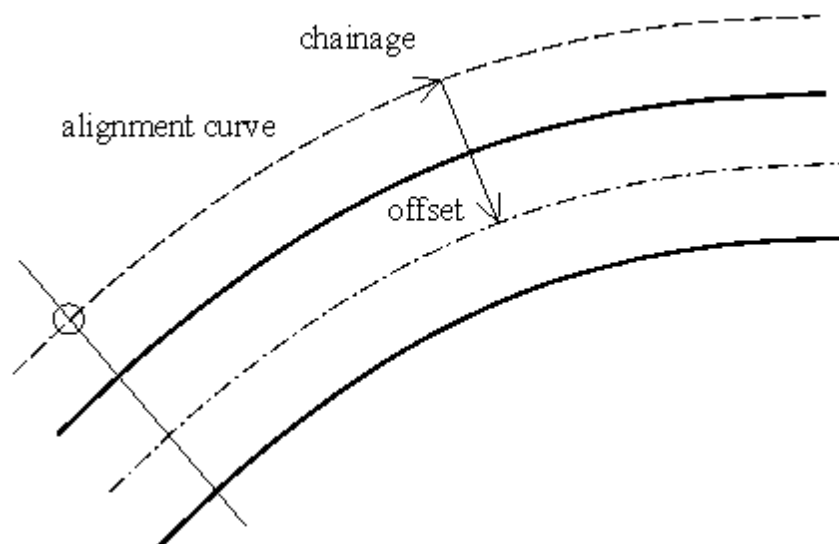
In Space

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.

7.11.14.2 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.

Definition**Axle Number**

The user selects the axle number to define.

Contact Area

The length of the contact area along the length of the vehicle and 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.

7.11.14.3 Optimise Path Loading

This dialog is available from the “Model | Bridge Modelling | 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.

Definition**Optimisation**

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.

7.11.14.4 Expand Bridge Loading

This dialog is accessible from the “Model | Bridge Modelling | 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.

Definition**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.

7.11.14.5 Delete Grid Loads

The process of bridge analysis creates grid loading. This allows the model to be cleaned by deleting the auto generated bridge grid loads.

7.11.14.6 Extrude Along Alignment

A bridge model is typically built to follow an alignment. It is convenient therefore to be able to generate a model by specifying an extrusion along an alignment.

Definition**Alignment and Chainage**

Choose the alignment along which to extrude a model and the start and end chainages for the extrusion. This defines the basic geometry of the model.

Target Element and Section

The extrusion places a set of elements along the alignment. This controls both the element size and the section to apply to the bridge deck.

7.11.15 Dynamics

7.11.15.1 Modal Forces

In a modal analysis the structure can be considered as a series of single degree of freedom (spring / mass) oscillators. To carry out post-processing for effect of loading using the modal results requires construction of the corresponding modal forces. These are defined as the dot product of the load vector and the mode shape.

The output is reported in the message pane.

Definition

Analysis Task

Specify the analysis task that provides the mode shapes

Load Cases

Specify the load cases that are to provide modal forces.

7.11.15.2 Create CEAP Modal Damping File

This dialog can be accessed from the “Model | Dynamics | Create CEAP Modal Damping File” menu command. This command creates a modal damping file for use in Oasys CEAP (Civil Engineering Application Package).

Definition

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.

7.11.15.3 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

$$\tilde{m}_{ij,part} = \frac{\left(\sum_k (\phi_{ik}^T [m_k] u_{jk}) \right)^2}{\hat{m}}$$

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

$$\tilde{m}_{j,part-sum} = \sum_i \tilde{m}_{ij,part}$$

Definition

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.

7.11.15.4 Create Model Principal Axes

This determines from an analysis task the centre of mass principal axes of the particular analysis model. These results are used to generate a new axis set.

7.11.16 Seismic

7.11.16.1 Equivalent Static / Accidental Torsion Load

This dialog can be accessed from the "Model | Seismic | Equivalent Static / Accidental Torsion Load" menu command. This command creates a load case corresponding to either an equivalent static load or an accidental torsion load to a selected seismic code.

Definition

Load Type

The same dialog is used for both Lateral load (equivalent static load) and Accidental torsion.

Dynamic Response Analysis and Base Shear

A list of the dynamic response cases allows the particular case to be selected. The base shear calculated in this analysis will then be used.

Seismic Code Procedure

To calculate either equivalent static or accidental torsion loads requires a seismic code to be selected and a set of code related parameters (e.g. peak accelerations, importance factors, etc.). The required seismic

parameters are defined in the response spectrum, so either an existing can be used or a new spectrum defined. The appropriate procedure from the code is followed. Both the fundamental period and base shear are code values by default, but can be overridden.

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 e.g. 1.4L1 +0.25L2. Leave blank if none.

Details

The direction of loading will be generated for either the X or Y direction. Z is assumed to be vertical. The mass offset is required for accidental torsion calculations. 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. Both the base level and number of storeys can be controlled by the user.

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.

7.11.16.2 Response Spectrum Generator

In most cases a code based response spectrum is what is required, but on occasions it is useful to be able to generate a spectrum from a given time history. When selecting this option a time history must first be selected. This is a text file either in tab or comma separated form, or a [PEER](#) file.

Definition

Time History Curve

If the file contains more than one time-history record select the appropriate record.

Spectrum Details

Specify the details such as title, number of spectral values and acceleration option to be used in the new spectrum.

Damping

Set the level of damping for use in the spectrum calculation. This must be greater than zero.

Peak Acceleration

By default the peak acceleration is derived from the input time history but the user can override this value and set a desired maximum acceleration

7.11.16.3 Spectrum Broadening

When a response spectrum is generated from a time history it can have sharp peaks. A response spectrum analysis may have modes which are just off these peaks and so the response may be underestimated.

Definition

Reference spectrum

The spectrum to be broadened

Broadening factor (f)

Each peak is smeared across by the broadening factor to avoid unrealistic dips.

Overwrite reference / create new spectrum

The existing spectrum can be updated with the broadened spectrum or a new spectrum can be generated from the reference spectrum.

7.11.16.4 Time-History Load Curves

In general the load curves used in seismic analysis are from reference curves or a database of ground motions. When selecting this option a time history must first be selected. This is a text file either in tab or comma separated form, or a [PEER](#) file.

Definition**Time History Curve & Title**

Select the appropriate time history curve from those in the file, and set a title as appropriate.

7.11.16.5 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 results are output to a csv file which can be opened in Excel.

This dialog can be accessed from the “Model | Seismic | 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.

Definition**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.

7.11.16.6 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 $f_i Am$.

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.

7.11.16.7 Create Positive Result Set

A set of results can be converted to a positive only set of results. This can be useful for combining results with a response spectrum analysis.

7.11.16.8 Rationalize Spring Properties for LS-DYNA

The way in which GSA defines spring properties is different to the way that they are defined in LS-DYNA. If the model is to be transferred to LS_DYNA this can be run to ensure that the spring will transfer properly.

7.11.16.9 Split Column to Column Beams

This will split beams where they intersect columns.

7.11.17 Offshore

7.11.17.1 Wave loading

See [wave loading](#) in the Step By Step Guide.

7.11.18 Manage User Modules

These dialogs can be accessed from the “Model | Manage User Modules” menu command and from the right-click menu displayed by right-clicking on ‘Nodal User Modules’, or ‘Element User Modules’ or ‘Member User Modules’ on the [Output Explorer pane](#).

Definition

User modules can be:

- **deleted**
- **renamed**
- **exported**
- **generated**

These operations may be carried out for individual user modules by first selecting the required user module.

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.

7.11.19 Miscellaneous Tools

7.11.19.1 Expand Grid Loading

This dialog is accessible from the “Tools | Grid/Face Loading | Expand Grid/Face 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.

Definition**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 save 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](#)

7.11.19.2 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.

7.11.19.3 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.

Definition

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.

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.

7.12 Miscellaneous Dialogs

7.12.1 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.

7.12.2 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.

7.12.3 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.

7.12.4 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.

Go to what

In multipage tables, choose to go to a record or a page.

Record number

Specifies the record or page to go to in the table.

7.12.5 Modify Curve

This dialog is made available when a response 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.

7.12.6 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).

7.12.7 GWA Import Options

This dialog is accessed from the "File | Import | Text (GWA File)" menu command or from the Data explorer 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

The import mode setting only affects the handling of node, element, load case and analysis task and case numbering. Other imported data always overwrites existing data.

Import mode, Overwrite – When overwrite is selected, imported nodes and elements overwrite original nodes and elements of the same number and imported loads are imported into their respective loading modules, with no adjustment to the load case numbers.

Import mode, Append – When append is selected, imported nodes, elements and load case numbers are renumbered so as to follow on from the existing data. Items in the imported model are incremented by the highest number in the original model; 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 6 8. References to the renumbered nodes and elements in the imported data are renumbered.

Note that the following data are overwritten, even when the import mode is set to Append:

- All properties
 - Materials
 - Analysis cases
 - Combination cases
- existing data will be overwritten.

7.12.8 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.

7.12.9 CAD Import Options

[CAD Import: Select Layers to Import](#)

[CAD Import: Map layers to properties](#)

[CAD Import: Import Options](#)

[CAD Import: Post import options](#)

7.12.9.1 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.

7.12.9.2 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.

7.12.9.3 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.

7.12.9.4 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](#).

7.12.10 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.

7.12.11 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 PolyFace 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.

7.12.12 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.

7.12.13 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.

7.12.14 ADC AdBeam Export

This dialog is accessed from the "File | Export | ADC (AdBeam file)" menu command. The data exported

Design code

Selected concrete design code in project specification.

RC beam list

The list of RC beams (members defined as concrete beams) to export. The defaults is all.

Loading and results

Select load combination for which result will be exported.

Save File

Browse path to save exported file.

Open AdBeam on completion

If this option is checked GSA will try to open AdBeam with the first file exported.

7.12.15 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 cannot open the file generated by GSA.

Section List

This gives a choice of exporting a list of section "all" or specific section.

Members List

This gives a choice of exporting a list of members. If the member option is chosen, one section will be generated for each of the positions.

Cases

Specific analysis or combination results can be exported as AdSec loads.

Save File

Browse path to save exported file.

Open AdSec on completion

If this option is checked GSA will try to open AdSec with the first file exported.

7.12.16 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.

7.12.17 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.

7.12.18 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:

[Numeric Formats](#)

7.12.19 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:

[Axis Sets](#)

Part VIII

8 GSA Text Files

8.1 Text Data File Format

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 GWA 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 [Text File Format](#) document.

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 [string ids](#) for information on sID formatting.

Format

The format of the specific records is as follows:

8.2 Reinforcement Template File

In order to simplify the creation of reinforced concrete sections, it is possible to create a reinforcement template file. This allows several templates to be defined which can be selected when template reinforcement is selected in the section wizard.

Rebar Template

This file is delimited by a <rebar> </rebar> pair of tags. Inside these tags the following tags define the different types of reinforcement

```
<beam> </beam> – beam template
```

```
<col-rect> </col-rect> – rectangular column template
```

```
<col-circ> </col-circ> – circular column template
```

```
<col-perim> </col-perim> – perimeter template
```

Comments can be included by starting the line with an exclamation mark.

Beam Template

The beam template is of the form

```
<beam>
```

```
Name, Cover, Grade
```

```
<<link>>
```

```
<<layer>>
```

```
<<layer>>
```

```
<<layer>>
```

```
<<layer>>
```

```
</beam>
```

where

Name – is a name used to identify this template

Cover – is the cover in mm

Grade – is the name of the reinforcement grade

<<link>> – is the link definition

<<layer>> – is the layer definition (one for top & bottom, and middle & ends)

Rectangular Column Template

The rectangular column template is of the form

```
<col-rect>
```

Name,Cover,Number,Arrange,Size,Grade,Bundle

```
<<link>>
```

```
</col-rect>
```

where

Name – is a name used to identify this template

Cover – is the cover in mm

Number – is the number of bars

Arrange – is the arrangement number

Size – is the bar size

Grade – is the name of the reinforcement grade

Bundle – is the number of bars per bundle (default 1)

<<link>> – is the link definition

<<layer>> – is the layer definition (one for end-top, end-bottom, mid-top, mid-bottom)

Circular Column Template

The circular column template is of the form

```
<col-circ>
```

Name,Cover,Grade,Bundle

```
<<link>>
```

```
<<layer>>
```

```
<<layer>>
```

```
</col-circ>
```

where

Name – is a name used to identify this template

Cover – is the cover in mm

Grade – is the name of the reinforcement grade

Bundle – is the number of bars per bundle (default 1)

<<link>> – is the link definition

<<layer>> – is the layer definition (one for outer, inner)

Perimeter Column Template

The perimeter column template is of the form

```
<col-perim>
```

Name,Cover,Grade,Spacing,Size,Bundle

```
<<link>>
```

```
</col-perim>
```

where

Name – is a name used to identify this template

Cover – is the cover in mm

Grade – is the name of the reinforcement grade

Spacing – is the bar centre to centre spacing

Size – is the bar size in mm

Bundle – is the number of bars per bundle (default 1)

<<link>> – is the link definition

<<layer>> – is the layer definition (one for outer, inner)

Link Template

The link template is of the form

```
<link>
```

Size,Legs

```
</link>
```

where

Size – is the bar size in mm

Legs – is the number of link legs

Layer Template

The perimeter column template is of the form

```
<layer>
```

Pos,Number,Size,Rows,Bundle

where

Pos – is the position of the layer

Number – is the number of bars in the layer

Size – is the bar size in mm

Rows – is the number of rows

Bundle – is the number of bars per bundle (default 1)

The position options for beams are

end-top

end-bottom

mid-top

mid-bottom

and for circular columns

inner

outer

Example

```
!  
! rebar template  
!  
<rebar>  
  
  <beam>  
    Beam C-1,0.025,500B  
    <link>  
      0.02,2  
    </link>  
    <layer>  
      end-top,4,0.02,1,1  
    </layer>  
    <layer>  
      end-bottom,2,0.012,1,1  
    </layer>  
    <layer>  
      mid-top,2,0.02,1,1  
    </layer>  
    <layer>  
      mid-bottom,4,0.020,2,1  
    </layer>  
  </beam>  
  
  <col-rect>  
    Rect A-1,0.025,8,2,0.032,500B,2  
    <link>  
      0.02,2  
    </link>  
  </col-rect>  
  <col-rect>  
    Rect B-1,0.020,4,1,0.025,500B,2  
    <link>  
      0.02,2  
    </link>  
  </col-rect>  
  
  <col-circ>  
    Circ D-1,0.025,500B  
    <link>  
      0.02,2
```

```

        </link>
        <layer>
            outer,4,0.02,1,3
        </layer>
    </col-circ>

    <col-perim>
        Perim E-1,0.027,0.15,0.016,500A,3
        <link>
            0.02,,2
        </link>
    </col-perim>
</rebar>

```

8.3 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.

Curve File

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.

Unit Types

The current options for TYPE are:

- UNDEFINED
- FORCE
- LENGTH
- MASS
- TIME
- TEMPERATURE
- 1ST_MOMENT_AREA
- 2ND_MOMENT_AREA
- VOLUME
- PER_LENGTH
- PER_AREA
- PER_VOLUME
- ANGULAR_ACCELERATION
- FLOW_RATE
- FREQUENCY
- ANGULAR_FREQUENCY
- PERIOD
- THERMAL_EXPANSION

- | | | |
|-----------------------|----------------------|------------------------|
| • DISP | • CS_AREA | • TEMPERATURE_GRADIENT |
| • PRESSURE | • CS_AREA_PER_LENGTH | • IMPULSE |
| • ACCEL | • INERTIA | • MOMENTUM |
| • WORK | • DENSITY | • ANGULAR_MOMENTUM |
| • MOMENT | • AREA_DENSITY | • DAMPING |
| • STRESS | • WEIGHT_DENSITY | • ENERGY |
| • FORCE_PER_LENGTH | • MASS_PER_LENGTH | • ENERGY_DENSITY |
| • MOMENT_PER_LENGTH | • PER_MASS | • DEGREE |
| • MOMENT_PER_ROTATION | • VELOCITY | • PERCENT |
| • STRAIN | • ROTATION | |
| • AREA | • ANGULAR_VELOCITY | |

The FACTOR is the value by which the data values should be multiplied to convert to SI units (e.g. 0.01 for cm).

Example

```

Test curve
Time <TIME>
Relative acceleration <ACCELERATION/0.001>
Reference point 1
0, 0
1, 1
2, 4
3, 9
4, 16

```

8.4 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 IX

9 API and Command Line Interface

There are two methods by which GSA can be driven as a non-interactive application. These are through the [GSA API](#) or by using a [command file](#).

The API uses COM automation to allow 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
```

9.1 GSA API

The GSA API uses COM automation to allow 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 below.

Note that function names are case sensitive.

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' in the [GSA API Reference](#)).

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.

For late binding code

```
Dim gsaObj As Object
Set gsaObj = CreateObject("Gsa_8_4.ComAuto.1")
```

... 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 GSA.exe in the program files folder of the particular version.)

Further details are available in the [GSA API Reference](#) Manual.

9.2 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")

Command File Commands

The GWC file commands are listed as either core commands or 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.

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).

SaveAs (filename)

Save the data to GWB, GWA or CSV file.

filename – the name of the file to be saved, including path and extension.

Close ()

Close the current file.

Exit ()

Close the file and exit GSA.

Analyse ()

If task exists, analyse the task.

If no task is specified (or task is 0), analyse all unanalysed tasks if analysis tasks exist; otherwise do linear static analysis of each specified load case.

Delete (option)

Delete results.

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.

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.

SaveViewToFile (option, filetype)

Save saved or preferred Graphic Views or Output Views to file.

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.

Part X

10 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. The options cover [export](#) of data and/or results from GSA, [import](#) of data into GSA and links with [Revit](#) and through [IFC](#) files.

10.1 Analysis in LS-DYNA

GSA provides most of the analysis capabilities for structural models, however for advanced analysis there is an option to create an LS-DYNA analysis task.

This feature is provided to make it simpler to maintain a single reference model and analyse in the appropriate tool.

Mapping between GSA and LS-DYNA

Some features available in GSA are not available in LS-DYNA and other features have been added to GSA, to facilitate LS-DYNA analysis, but which are not available for analysis in GSA. Analysis stages provide a way of coordinating between the GSA and LS-DYNA models. Using the analysis stages non-linear (LS-DYNA) materials can be assigned to the model while retaining the linear materials used by GSA.

Analysis Parameters

The Analysis Wizard allows an LS-DYNA keyword file to be created, generally for a selected analysis stage. The analysis parameters for specification of timings, excitation and elements formulations. Termination and output times (whole model and time history) can be set. Excitation can be selected as a force excitation or base excitation. Assemblies can be specified – these are exported as cross sections.

In addition to the mass of the structure additional mass can be added for the LS-DYNA analysis. This is selected as a loading which is converted to a mass elements when creating the LS-DYNA model.

The final stage in the analysis task selects a keyword file for output and the environment to launch. A default environment can be set in the Preferences.

Materials

GSA is used primarily for materially linear problems, while LS-DYNA generally uses non-linear materials. To accommodate this a number of LS-DYNA material models can be defined in GSA. The following models are available for LS-DYNA:

- Concrete EC2
- General Nonlinear Beam
- Hysteretic Beam
- Hysteretic Reinforcement
- Park Ang Beam
- Plastic Kinematic
- Seismic Isolator

Some of these material models refer to Load Curves in LS-DYNA. These are defined as Material Curves in GSA, and transferred as load curves.

Spring/damper elements

LS-DYNA considers some spring elements as variants of beam elements and in general regards springs

as single degree of freedom entities. These simple springs are written as *ELEMENT_DISCRETE. For simple springs the orientation option (VID) is set to 0 so the spring acts along the axis between the nodes. For more general springs the GSA element is written as a DYNA element for each stiff direction. In this case the orientation option is set to the number of an *DEFINE_SD_ORIENTATION record with option (IOP) set to 1 and a vector defining the direction of action (XT,YT,ZT). Springs and dampers are differentiated by using different material definitions. For a spring the element relates to a *MAT_SPRING definition and a damper refers to a *MAT_DAMPER_VISCOUS definition.

The properties for a linear elastic spring are written as *MAT_SPRING_ELASTIC defining the stiffness (K). A non-linear spring is written as *MT_SPRING_NONLINEAR_ELASTIC with the spring curve written as a load curve referred to by LCD. Tension and compression only springs are written as *MAT_SPRING_INELASTIC and a load curve is generated for the stiffness (LCFD) and the compression tension flag is set (CTF) as required

GSA springs

Linear spring in x - This is written as a *ELEMENT_DISCRETE with a *MAT_SPRING_ELASTIC for the stiffness

Linear spring in all 6 dof - This is written as a *ELEMENT_BEAM with ELFORM set to 6 with a *MAT_LINEAR_ELASTIC_DISCRETE_BEAM defining the stiffness in all 6 dof

Non-linear spring in all 6 dof - This is written as a *ELEMENT_BEAM with ELFORM set to 6 with a *MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM defining the material curves of stiffness in all 6 dof. If some of the dof are linear these must be converted to (non-linear) material curves

Tension-only and compression only springs - These must be converted to a nonlinear spring and can then be written as *ELEMENT_DISCRETE with material *MAT_SPRING_INELASTIC

Lockup elements - These must be converted to a nonlinear spring and can then be written as *ELEMENT_DISCRETE with material *MAT_SPRING_INELASTIC

Connector elements - These are exported as *ELEMENT_BEAM with ELFORM set to 6 with a *MAT_LINEAR_ELASTIC_DISCRETE_BEAM defining a stiffness in all 6 dof of 1e10.

Gap elements - These are not exported at present

Friction elements - These are not exported at present

GSA Damper

Linear damper in x - This is written as a *ELEMENT_DISCRETE with a *MAT_DAMPER_VISCOUS for the damping

Linear damper in all 6 dof - This is written as a *ELEMENT_BEAM with ELFORM set to 6 with a *MAT_LINEAR_ELASTIC_DISCRETE_BEAM defining the stiffness in all 6 dof.

Rayleigh damping

In addition to damper elements , Rayleigh damping can be specified. Rayleigh damping is written as and in the *DAMPER_GLOBAL record and *DAMPING_PART_STIFFNESS respectively.

Composite shells

GSA does not provide options to analyse composite shells. To allow for this in LS-DYNA there is an option in the 2D element properties wizard to define a composite shell. This allow a series of layers to be specified in terms of a material, a thickness and an angle. If these are specified the element thickness is the sum of the thicknesses of the layers.

10.2 Export options

The export options are available from the “File | Export” menu command.

[Input data](#)

[SAP2000](#)

[Nastran](#)

[OpenSees](#)

[AutoCAD](#)

[PDisp](#)

[ADC](#)

[AdSec](#)

[Steel Member\(s\)](#)

[Footfall Response](#)

[Text output](#)

10.2.1 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.

10.2.2 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.

10.2.3 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.
- Prestress 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 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.

10.2.4 OpenSees

The model data can be exported to an OpenSees TCL file.

Export

The user can select:

- Which analysis stage should be exported.
- Which case loads are exported in.
- To export bars as OpenSees trusses or corotational trusses.
- To export steel and concrete beams (as separate choices) as OpenSees elastic, nonlinear or displacement beam columns.
- 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.
- Prestress and in-plane loads are converted to equivalent thermal loads.

10.2.5 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.

10.2.6 PDisp

This exports the [PDisp](#) data (compatible with PDisp 17.7.2) used in a raft model in the PDisp format. 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.

10.2.7 ADC

This exports RC beam data for design in [ADC/AdBeam](#).

The GSA model must contain RC beams (members defined as concrete beams) and analysis results for this option to work.

Having set up RC beams corresponding properties the "[ADC AdBeam Export](#)" dialog allows the user to select a list of beams for selected load combinations. The export creates an AdBeam 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 result is exported for selected load combination only.
- Minor axis bending and shear, axial loads and torsion in the beams are ignored.
- The beam member properties in the design layer may differ from the corresponding element properties in the analysis layer.

10.2.8 AdSec

This exports section data for design in [AdSec](#).

The GSA model must contain either concrete beam or column members 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.

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.

10.2.9 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.

10.2.10 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.

10.2.11 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.

10.3 Import options

The import options are available from the “File | Import” menu command.

[Text](#)

[AutoCAD](#)

[PDisp](#)

10.3.1 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.

10.3.2 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.

10.3.3 PDisp

PDisp data (PDD) 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.

10.4 Links With 3D Modelling Packages

10.4.1 Revit

The link between Revit Structure 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 synchronise an existing GSA model with changes from a Revit model.
- [Update from Oasys GSA](#) – synchronise an existing Revit model with changes from a GSA model.
- [Import from Oasys GSA](#) – create a new Revit model from a GSA model.

Following the [Recommended Practice](#) will improve coordination between the GSA and Revit models.

10.4.1.1 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.

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 filename 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. The user may require manual editing of the said areas. Nonetheless, lines that form the opening get exported as GSA Lines.
- When exporting to the analysis layer, the Analytical Projection Plane is handled as follows:
 - For beams,
 1. if set to 'Center of Beam' the GSA element is offset downwards by half the beam depth.
 2. if set to 'Top of Beam' then no offset is applied.
 3. Other settings are not handled
 - For columns
 1. Only 'Top of Column' at the top and 'Bottom of Column' at the bottom are handled. No offset is applied.
 2. 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 columns, beams and braces 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 symbol 'Type Comments' field.

2. 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 catalogue 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, hence the 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 written.
- Other data in the GSA model is not affected by the update.
- Please note that this does not apply to Floors and Walls and their types.

Known Issues

- Restraints are not handled.
- Translational releases are not handled.
- GSA objects that relate to deleted Revit objects are not deleted.

10.4.1.2 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:

- 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.
 - Columns can be vertical or slanted.
 - Column lengths can be altered.
 - Orientation 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.
 - Orientation can be altered.
 - Revit offsets are changed as necessary to accommodate changes in the elevation of reference levels.
 - Changes in beam orientations can be translated – although this may not necessarily yield useful results.
 - 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 a. 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 b. If referencing new section property, the section in question must be imported into Revit as new Family.
- Sections 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 is **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 behaviour such as the Revit elements becoming disjointed.

10.4.1.3 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.

Map Sections

Click 'Map Sections' to begin mapping GSA section properties to the corresponding Revit Family types. This may take a little long depending on the number of sections in the GSA model. During this, GsRevit

tries to load the new family types into the Revit model. If prompted for overwriting a pre-loaded Family Type, please choose 'Yes'.

GSA - Revit section mapping

This table displays the Revit Family types that will be used for the corresponding GSA sections. Please note that a given GSA section can either be 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 ignored.

Note: If a GSA section maps to 'N/A' for a Revit Family type, it signifies that GsRevit has not been able to find a suitable match for the section among the Family Types loaded in the project. If a family type refers to N/A, elements referring to the corresponding section will not be imported.

The section mapping utility currently works only for catalogue steel sections. For standard sections and for concrete sections, load the appropriate Family Type into the model before starting the import process. See ['Recommended Practice'](#) for more information on this.

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.

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.

Coordination

- 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.
- GSA area and region can be imported as a wall or slab in Revit as per the section
- 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.
- Option to delete Revit entities where there is no matching of GSA entities.
- Option to map Revit entities using GSA entity id shared parameter where there is no mapping by 'unique ID' : 'SID'

Filter

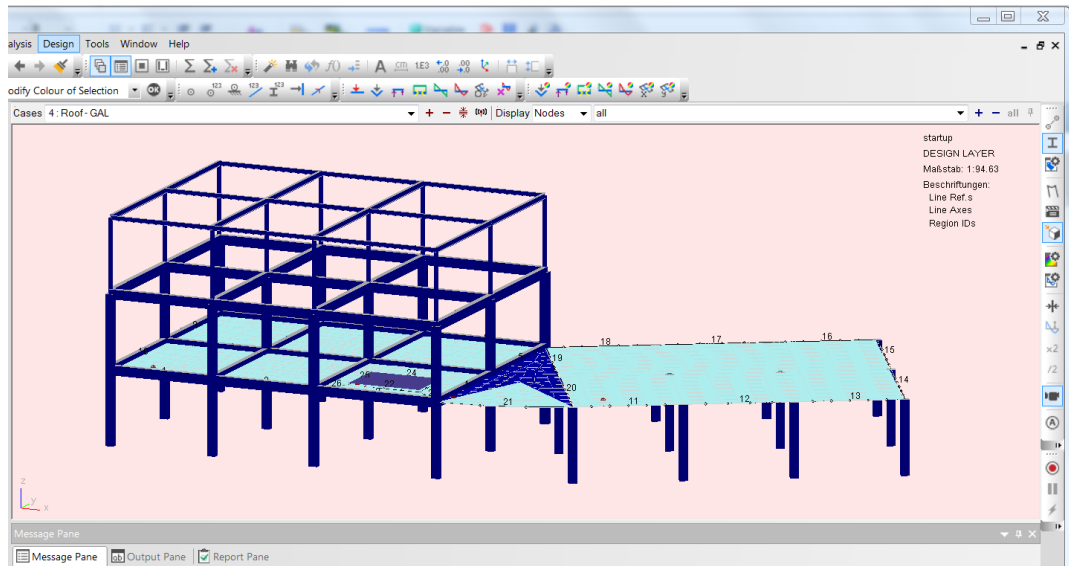
- The filter allows selective import of entities

10.4.1.4 Import from Oasys GSA-An Example

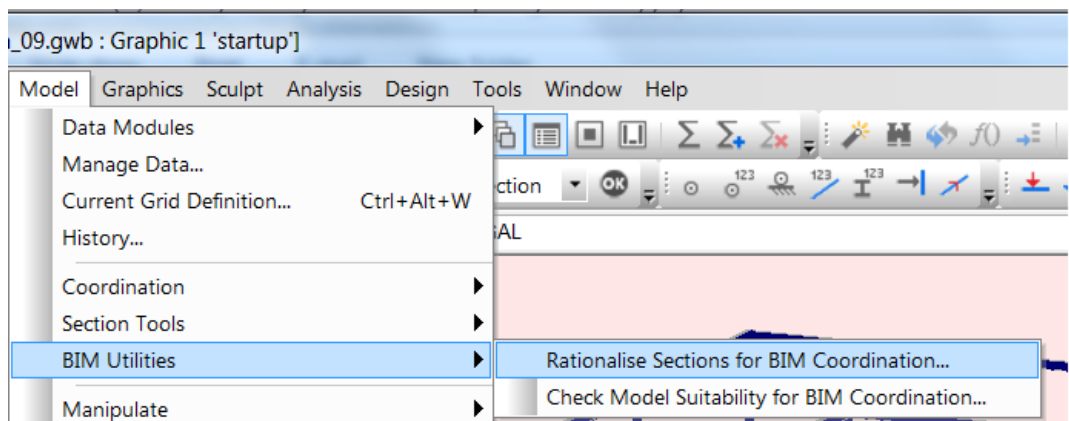
Importing a GSA model into Revit

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.

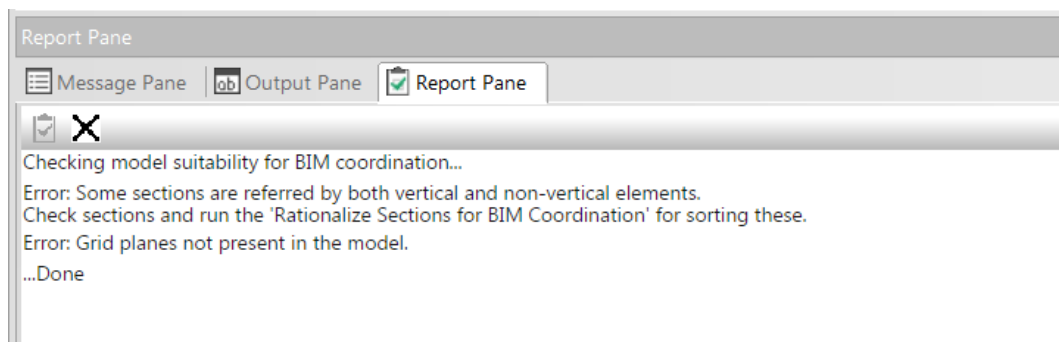
The model used in this tutorial, "TrGen_10.gwb" can be found under the Samples folder in the GSA installation folder, typically <C:\Program Files\Oasys\GSA 10.0\Samples\Training_General>.



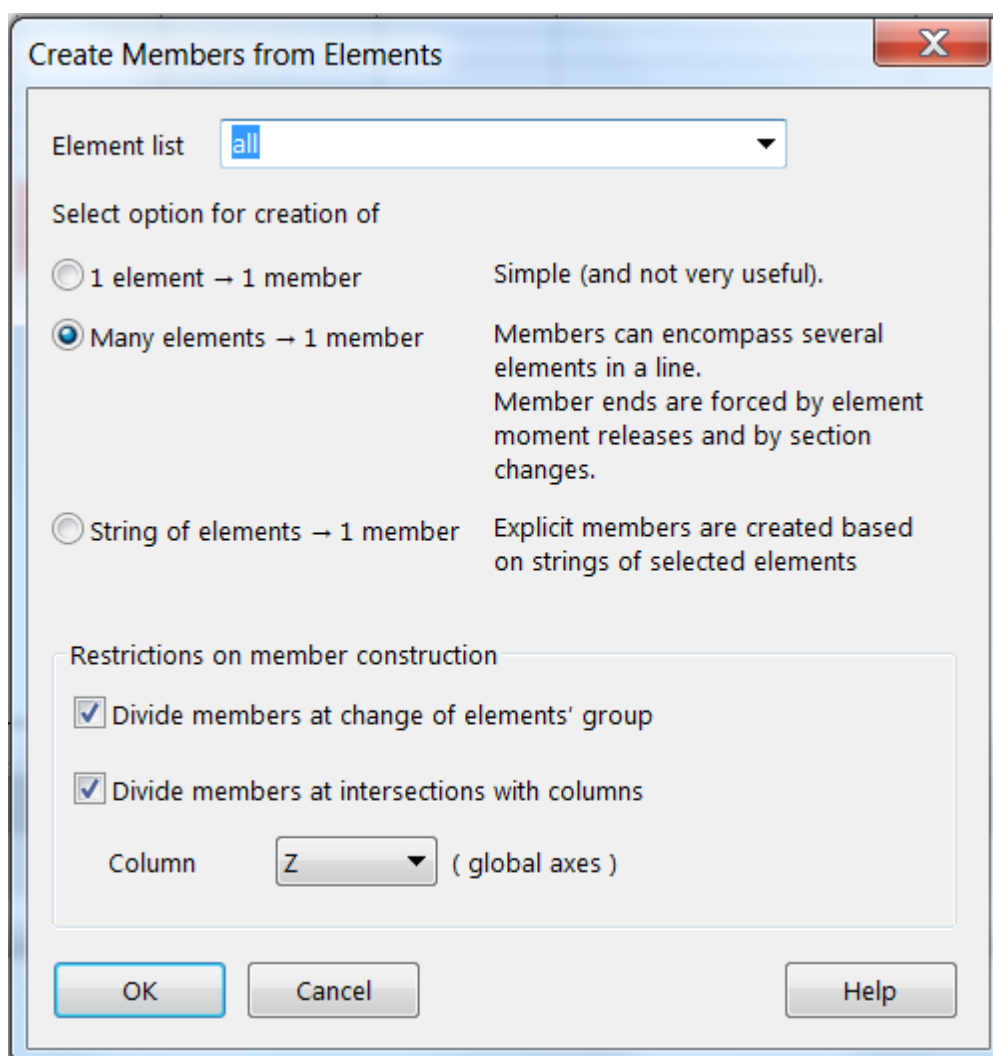
1. Open the file in GSA 8.7, or later.
2. From Model | Checking Tools, run 'Check Model Suitability for BIM Coordination' Command



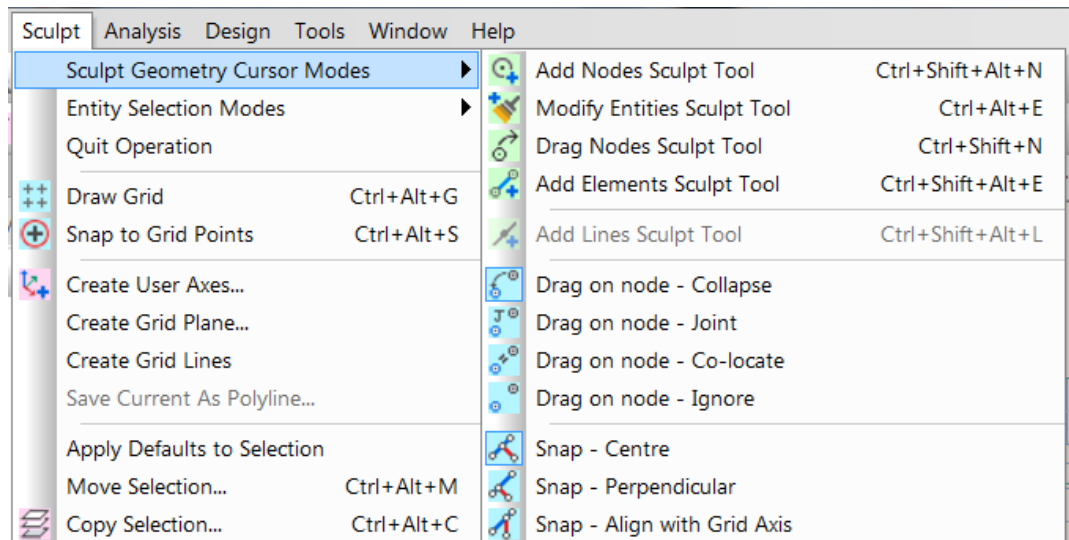
3. GSA produces the following output in the Report Pane after checking model suitability for BIM coordination:




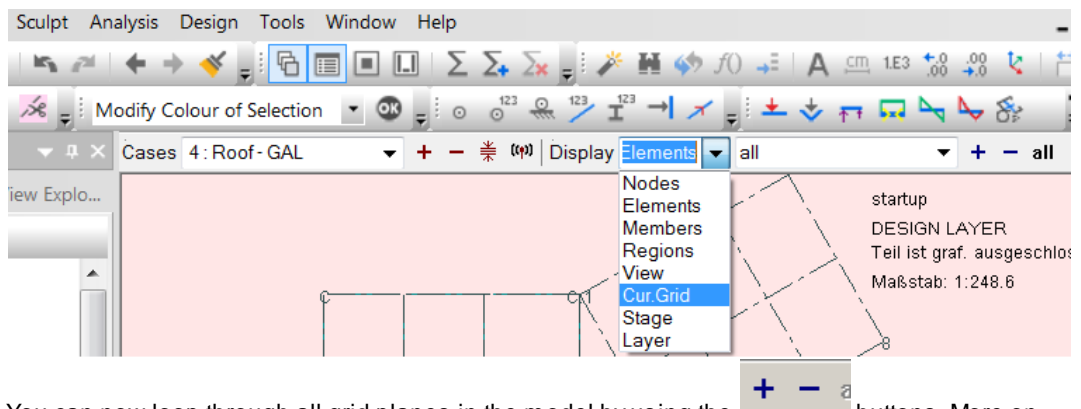
4. 'Warning: Members are not present in this model': It's advisable to coordinate with the GSA Design layer. Use 'Model | Coordination Tools | Create Members From Elements...' to automatically generate members.




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 'GSA concepts | 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.

Property	Name	Colour	Material			Member Type	Section Profile
			Type	Grade	Analysis		
Defaults	Section #		Steel	1	from Grade	Column	STD I 1000 500 15 25
1	Column	...	Concrete	1	1 : Concrete long	Column	STD R(m) 0.4 0.3
2	Beam - x	...	Concrete	1	1 : Concrete long	Generic	STD R(m) 0.45 0.25
3	Beam - y	...	Concrete	1	1 : Concrete long	Generic	STD R(m) 0.45 0.25
4							
5							
6							

Open 'Mappings.db3' from <C:\Program Files\Oasys\GSA 10.0\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:

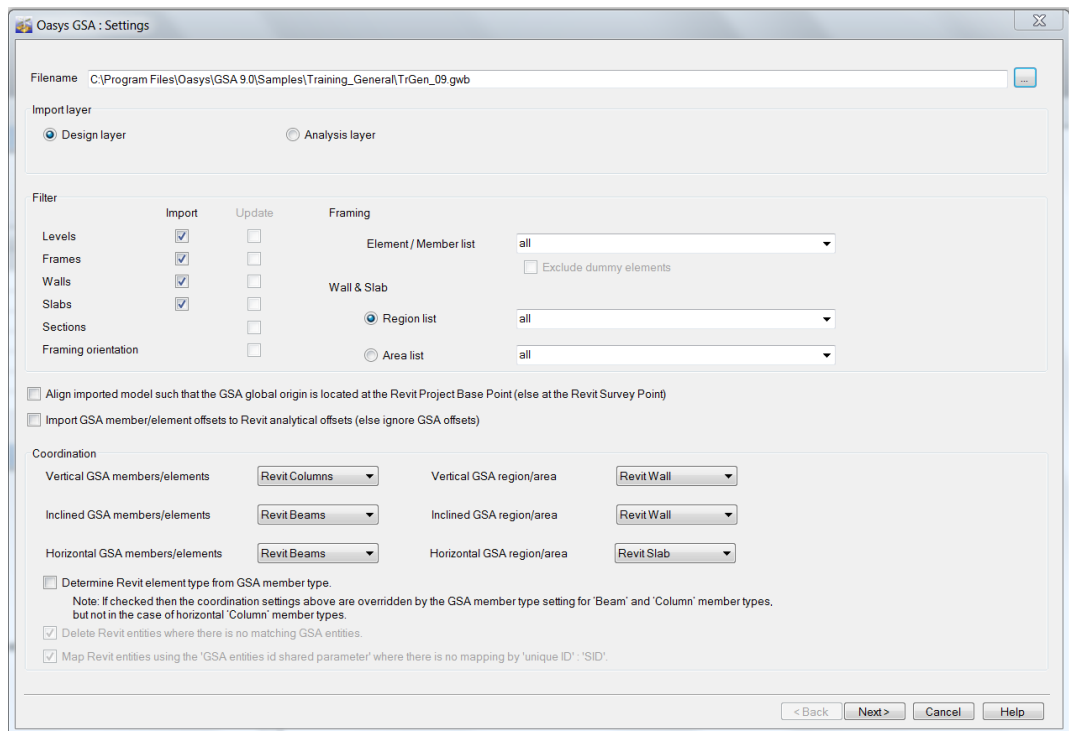
Database: mappings Table: GsRvt_Columns File: C:\Program Files\Oasys\GSA 8.6\mappings.db3

RecNo	GSA_Section_Description	Revit_Family_Type	Revit_Family
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

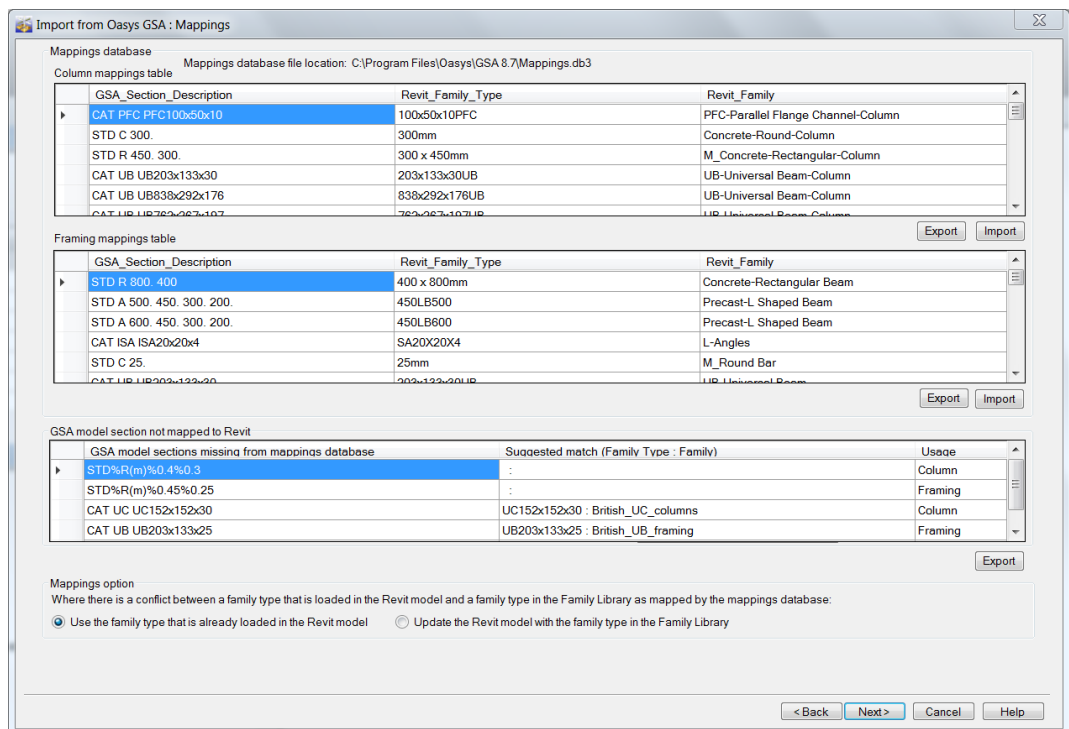
Caution: If you modify the mappings.db3 file, make sure to save it in a different location – since reinstalling GSA would overwrite this file. In this case, the path to your version of mappings.db3 can be set in GSA by 'Tools | Preferences | Basic | File Locations | Mappings Database file'.

Save and close the mappings.db3 file.

- 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 specify mapping here and on clicking Next, changes will be saved in mapping database at path displayed on top.



11. When mapping sections, make sure no sections are <Unmapped>. Sections mapped INVALID will not be imported.

Import from Oasys GSA : Mapped Sections

Map GSA Sections to Revit Family Types

GSA Section | 2D Element

Section No.	GSA Section Name	GSA Section Description	Revit Family Type	Structural Type/Usage
1	Column	STD%R(m)%0.4%0.3	Concrete-Rectangular-Column>...	COLUMNS
2	Beam - x	STD%R(m)%0.45%0.25	Concrete-Rectangular Beam>40...	FRAMING
3	Beam - y	STD%R(m)%0.45%0.25	Concrete-Rectangular Beam>40...	FRAMING
11	Column - top floor	CAT%UC%UC152x152x30%19990407	Concrete-Rectangular-Column>...	COLUMNS
12	Beam - x - top floor	CAT%UB%UB203x133x25%19990407	Concrete-Rectangular Beam>40...	FRAMING
13	Beam - y - top floor	CAT%UB%UB254x146x37%19990407	Concrete-Rectangular Beam>40...	FRAMING

Legend:
 1) <Unmapped>: Match not available, choose suitable section from the dropdown
 2) NOT USED: Section is not used as a column/beam in the GSA model
 3) INVALID: Section is referred to by both Vertical and Non Vertical GSA Members

< Back Finish Cancel Help

Import from Oasys GSA : Mapped Sections

Map GSA Sections to Revit Family Types

GSA Section | 2D Element

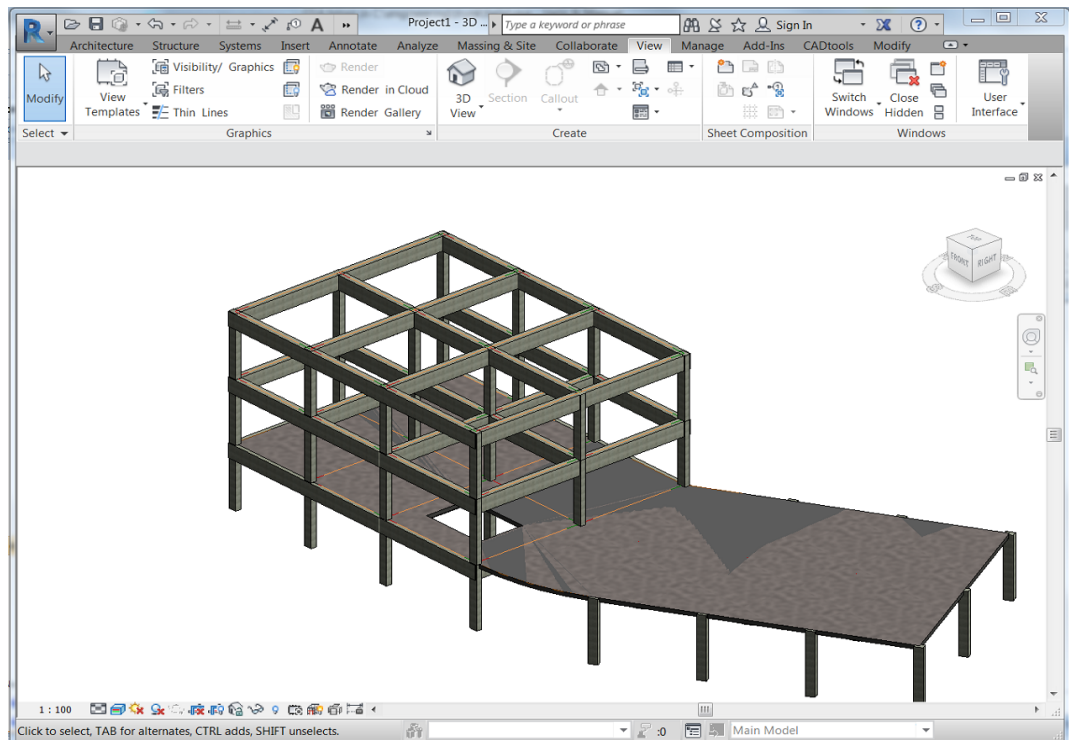
Wall Family Basic Wall>Exterior - 300mm Concrete

Slab Family Floor>160mm Concrete With 50mm Metal Deck

Legend:
 1) <Unmapped>: Match not available, choose suitable section from the dropdown
 2) NOT USED: Section is not used as a column/beam in the GSA model
 3) INVALID: Section is referred to by both Vertical and Non Vertical GSA Members

< Back Finish Cancel Help

Select 'Unjoin Elements' if Revit throws up warnings.



10.4.1.5 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 can try to find suitable matches for most steel catalogue sections, but for the rest, the best option is to manually load the desired (equivalent) family type before starting the import. During the import process, one can then select the appropriate family type in the drop-downs.

Section Property References

For each section property in GSA, it must be ensured the section is not referred to by elements/members that are both vertical and non-vertical. Sections referred to by both will not be imported.

GSA has a tool that facilitates splitting section properties into Beam(non-vertical) sections and Column(vertical) sections. The command 'Rationalize sections for BIM coordination' in the Tools menu examines all sections in the section property table. If any of them is referred to by both vertical and non-vertical entities, the tool creates a new section property and gets the vertical entities to refer to the new section.

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 'Model | Manipulate | Create Elements from Members' and 'Model | Manipulate |

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's 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.
2. Open a new Revit model from one of the templates for Metric or for Imperial.
3. If you plan to use grid planes for creating levels, for best results, 'Level 1' in the Revit model may be deleted. If not, levels may be defined at appropriate elevations manually.
4. 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, make sure to load them separately in the Revit model, under Structural Column Family types and Structural Framing Family types.
5. It's best to start the import from an 'Elevation' view rather than the default 3D view.

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.

10.4.1.6 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 installed at the time GSA is installed then the GSA installation will automatically register GsRevit with Revit. Successful registration is indicated by the presence of the GSA options in the Revit 'Tools | External Tools' menu (and that they work!).

GsRevit can be registered interactively by running GsaRegister.exe, which may be found in the GSA program folder.

Possible reasons for failure in the registration process are:

- Revit Structure is not installed in the default location. The automated install-time registration assumes that Revit is installed in its default location, <C:\Program Files\Autodesk Revit Structure x>. When running GsaRegister.exe interactively there is an opportunity to specify the location of Revit.
- The version of Revit Structure that you have installed is not supported.

Refer to file GsRevit_ReadMe.pdf (installed in the Docs folder beneath the GSA program folder) for the latest information on GsRevit, including supported versions.

10.4.1.7 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 and 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 [Section Profile](#).

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'.

10.4.2 IFC

The IFC file format is a widely-used format supported by variety of software for civil engineers. It describes building and construction data. It is not meant to replace proprietary formats of any particular software but to provide a universally recognized general format, even if the functionality to interchange data is limited.

Oasys GSA includes both IFC Import and Export with some limitations.

[Export from Oasys GSA to IFC](#): exports the whole GSA model into a new IFC file.

[Import from IFC to Oasys GSA](#): imports IFC objects into the current GSA project. Not all objects are transferred into Oasys GSA. A log of the import process is available to users.

Update from IFC to Oasys GSA: applying changes onto current GSA objects from a modified IFC file is not currently implemented.

Credit: the implementation uses IfcOpenShell library which depends on Open Cascade library.

10.4.2.1 Export from Oasys GSA to IFC

The export is intended for:

- viewing a GSA model in IFC graphical viewers
- transferring GSA models into Tekla Structures..

There may be other applications that have not been tested. Note that exporting to Autodesk Revit is recommended to be done primarily by [GSA Revit plugin](#) (GsRevit), not via IFC.

All entities will be exported to IFC, i.e. the user has no option to reduce the export to part of a model. The user can choose whether to export the design layer, analysis layer or both layers.

The following options are available:

Generate assemblies for Tekla

If ON, IFC objects are augmented with additional properties to facilitate import into Tekla Structures. It has no compatibility impact on importing to another software package. This switch can always be turned ON unless you aim to reduce the file size. The default is ON.

Layer(s) to export

1. Design layer: regions and members are exported. Elements are skipped during export (default).
2. Analysis layer: elements are exported. Regions and members are skipped during export. The IFC file will be typically larger than if the design layer is exported since each GSA element has to be represented as one IFC object.
3. Both: both design and analysis layer are exported to IFC file. This is not recommended.

Colors

GSA lets you decide how the IFC objects are colored.

1. None: in case you want to keep the IFC file small.
2. By type: different entity types within GSA are assigned different colors. (default)
3. By property: according to assigned property number within GSA.
4. By group: according to assigned group number within GSA.

Shell elements

There are several ways how to represent shell surfaces in IFC.

1. As shells with no thickness (a plane shell). The thickness information is lost.
2. As faces (e.g. a box consists of six faces so the thickness is determined). Not an efficient setting.
3. As solids (an area is extruded by thickness, i.e. usually by thickness of a slab or a wall). This is the default setting and the format expected by Tekla Structures.

Beams

The cross-sections of beams can be represented in two ways.

1. By perimeter. Every section behaves as a user section. (default)
2. As standard sections. IFC format supports basic standard sections as L, U, C, I shapes.

Which option to select depends on the target software.

10.4.2.2 Import from IFC to Oasys GSA

IFC is a loose format: a physical object (e.g. a beam or a slab) can be described in many ways (the same shape can use different representations). Not all possible IFC representations are supported when an IFC file is opened in GSA. Therefore it is likely that some entities will be lost or not transferred neatly. This behaviour can be improved in the future.

IFC files are imported into the current GSA document (typically a blank document). The smaller the IFC file is, the faster the process.

The recommended settings when exporting from Tekla Structures are:

- (optional) select the objects to export,
- select File format: IFC,
- select Export type: Coordination view,
- for Tekla Structures rev. ≤ 20 , unselect Pour objects in "Advanced tab" (other settings in the Advanced tab are not known to have a functional impact on import into GSA).

Part XI

11 Bibliography

1. AISC Steel Design Guide Series, Floor Vibration Due to Human Activity.
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.

Part XII

12 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.

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.

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.

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.

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 tables 39

- / -

/= command
in tables 39

- + -

+= command
in tables 39

- = -

= command 39

- - -

-- command
in tables 39

- = -

== command
in tables 39

- 1 -

1D Element Results Dialog 533

- 2 -

2D circular mesh
data generation 427

2D derived force results
output options 378

2D element
axis 217

2D element displacements

output options 376

2D element edge loads
create dialog 241
definition dialog 241

2D element face loads
create dialog 239
definition dialog 239

2D element facet angle
change default 432

2D element loads
load panel fundamentals 240

2D element pre-stress loads
create dialog 243
definition dialog 243

2D element stresses
output options 376

2D element thermal loads
create dialog 244
definition dialog 244

2D elements
Allman-Cook 302
centre-only results 305
convert Quad4 to Quad8 457
convert Quad8to Quad4 457
convert Tri6 to Tri3 457
geometry checks 302
linear analysis introduction 315
linear to parabolic 457
load panels 189, 240
loading data 238
mesh generation 111
MITC 302
modelling and loading introduction 314
parabolic to linear 457
properties 189
settings 302
split warped 505
wall results 379

2D elliptical mesh
data generation 427

2D mesh
delete 507
disassociate 507

2D orthogonal mesh
data generation 426

2D polylines
check for validity 150
definition dialog 150

- 2D properties
 - composite slab 192
 - hollow slab 192
- 2D skew mesh
 - data generation 426

- 3 -

- 3D element
 - axis 217
- 3D element displacements
 - output options 382
- 3D element face loads
 - create dialog 245
 - definition dialog 245
- 3D element stresses
 - output options 382
- 3D element thermal loads
 - create dialog 245
 - definition dialog 245
- 3D elements
 - properties 193
- 3D orthogonal mesh
 - data generation 428

- A -

- abs combination case operator 104
- accelerators 129
- Accidental Torsion Load 518
- ACI 18, 20
- Acknowledgements 22
- active column solver 300
- ADC
 - export options 554
- add elements sculpt tool 446
- add lines sculpt tool 446
- Add Nodes command 449
- add nodes sculpt tool 446
- Add String of 1D Elements command 451
- adornments
 - apply to selection 65
- advanced preferences 434
- AISC LRFD checker
 - documentation 399
- AISC360 checker
 - documentation 400

- align
 - beams and columns 505
- alignment
 - extrude 516
- alignments
 - data 268
 - extrude geometry along 443
- Allman-Cook 302
- aluminium
 - material grade 157
- Analyse All command 85
- Analyse All Marked Tasks command 97
- analysis
 - bridge step by step 345
 - bridge structures 349
 - buckling step by step 328
 - damping 300
 - dynamic step by step 321
 - Elements 78
 - explicit time history 324
 - explicit time-history 365
 - footfall analysis 326
 - harmonic analysis 325
 - imperfection 306
 - introduction 84
 - linear static step by step 313
 - LS-DYNA 307
 - non-linear step by step 316
 - P-delta 313
 - periodic load analysis 327
 - piled-raft analysis 350
 - raft 143
 - raft analysis 350
 - response spectrum 341
 - seismic step by step 339
 - step by step summary 85
 - Tasks & Cases 77
 - threading 434
 - Time history analysis 325
 - Types 83
- analysis cases
 - definition dialog 283
 - fundamentals 101
- analysis envelopes
 - Analysis Wizard 298
 - step by step 360
- analysis features 15
- analysis layer basics 356

- Analysis Layer in graphic views 70
- analysis materials
 - data 161
- analysis options
 - Analysis Wizard 283
- Analysis Progress Dialog
 - GsRaft 308
- analysis specification
 - dialog 141
- analysis stages
 - colour elements by 476
 - creep 282
 - data 281
 - definition data 282
 - in graphic views 359
 - introduction to analysis 358
 - introduction to analysis results 358
 - long term analysis 282
 - modelling implications 355
 - property data 282
 - results introduction 358
- analysis tasks
 - fundamentals 102
- analysis tasks and cases
 - Analysis Wizard 99
 - copy and paste 100
- Analysis Wizard
 - analysis envelopes 298
 - analysis options 283
 - buckling parameters 289
 - cases set up 299
 - footfall analysis parameters 295
 - form finding options 291
 - generic dynamic analysis parameters 295
 - GsBridge control 298
 - GsRaft control 297
 - GsRaft progress 297
 - GsRelax analysis progress information 292
 - GsRelax analysis task control 291
 - GsRelax member identification 291
 - harmonic analysis parameters 294
 - linear time history analysis parameters 294
 - mass calculation 289
 - modal dynamic parameters 285
 - non-linear static options 289
 - pseudo response spectrum analysis 293
 - response spectrum analysis 292
 - Ritz analysis parameters 288
 - static analysis cases 284
 - static p-delta 285
 - working with 99
- angle between nodes 33
- Animate 71
- Animate command 71
- animation
 - Animation Settings Dialog 487
 - in graphics 71
 - settings for graphics 487
- Animation Settings command 71
- Annotate Selection command 65
- annotating graphic diagrams etc 65
- annotation of diagrams options 486
- Append GWA data 465
 - from Data Explorer pane 31
- applied displacements
 - create dialog 230
 - data 230
 - definition dialog 230
- Apply Adornments to Selection command 65
- approximate minimum degree 300
- areas
 - copy / move graphically 457
 - Copy Areas Dialog 441
 - definition 152
 - labels in graphic views 475
 - Move Areas Dialog 441
 - working with 111
- AS 18, 20
- AS4100 checker 400
 - Buckling checks 400
 - documentation 400
 - Local checks 400
- ASCE 7 -05
 - ASCE7-05 spectrum 253
- ASCII (GWA) file
 - syntax 536
- assemblies 364
 - data 220
 - definition dialog 220
 - labels in graphic views 475
- auto views
 - details 96
- AutoCAD
 - export options 553
 - import options 556

- automatic constraints
 - fundamentals 117
 - automatic increment
 - dynamic relaxation analysis 331
 - axes 89
 - coordinates, non-global definition fundamentals 92
 - Create User Axes Dialog 436
 - creating user graphically 448
 - cylindrical and spherical fundamentals 91
 - data 145
 - definition dialog 145
 - global cylindrical 89
 - grid fundamentals 92
 - grid planes 148
 - grid surfaces 149
 - in table views 43
 - joint in disparate axes 509
 - labels in graphic views 472
 - Member 207
 - modify axis 145
 - output 44
 - projected fundamentals 92
 - sets fundamentals 89
 - use of fundamentals 91
 - vertical 89
 - x elevation 89
 - y elevation 89
 - Axes command
 - in graphic views 66
 - Axes Dialog 534
 - axis
 - elements 217
 - axisymmetric
 - new model wizard 419
- B -**
- backup
 - set time interval 432
 - backups
 - working with 117
 - bar
 - axis 217
 - base
 - create dummy elements 511
 - basic concepts in GSA 29
 - Batch Analysis command 98
 - batch output file formats 432
 - batch printing views and view lists 97
 - beam
 - axis 217
 - offset 505
 - split at columns 521
 - beam cross strain energy density
 - output options 375
 - beam derived stresses
 - output options 371
 - beam displacements
 - output options 369
 - beam distortion loads
 - create dialog 236
 - definition dialog 236
 - beam forces
 - output options 369
 - beam loads
 - create dialog 232
 - definition dialog 232
 - beam pre-stress loads
 - create dialog 235
 - definition dialog 235
 - beam sections
 - modifiers 185
 - simple beams 185
 - beam strain energy density
 - output options 374
 - beam stresses
 - output options 371
 - beam thermal loads
 - create dialog 237
 - definition dialog 237
 - Beams
 - Influence effects 273
 - load data 232
 - loading data 232
 - bearings
 - data 224
 - beta angle
 - beam axis 217
 - BIM 568
 - constructing a model 27
 - blank in tables 41
 - block 209
 - definition 209

- block 209
 - replica 209
- bricks
 - properties 193
- bridge
 - analysis 349
 - Expand Bridge Loading 516
 - Generate Static Vehicle Load 513
 - influence analysis 349
 - lane-by-lane path optimisation 349
 - Optimise Path Loading 515
- bridge analysis
 - modelling implications 347
 - step by step 345
- bridge analysis specification 144
- bridge beam
 - tendon 185
- bridge beams 175
- bridge modelling
 - delet grid loads 516
- Bridge Options Dialog 485
- bridges
 - bridge loading data 275
- broadening
 - response spectrum 519
- BS 18, 20
- BS5950 checker 402
 - buckling checks 402
 - buckling interaction check 402
 - combined local check 402
 - documentation 402
 - limitations 402
 - local axial check 402
 - local bending check 402
 - local checks 402
 - local shear check 402
 - local torsion check 402
- buckling analysis
 - step by step 328
- buckling analysis results
 - output options 367
- Buckling checks
 - AS4100 400
- buckling parameters
 - Analysis Wizard 289
- building
 - create storeys 510

- C -

- cables
 - properties 194
- case description
 - definition dialog 311
- case descriptions 311
- cases 308
 - Analysis Case Definition Dialog 283
 - analysis case fundamentals 101
 - Case Description 311
 - combination case data 308
 - combination case fundamentals 102
 - Combination Cases dialog 308
 - data 309
 - fundamentals 101
 - load case data 229
 - load case fundamentals 101
 - Load Case Titles dialog 229
 - output by 47
 - selected for output 46
 - selecting for graphic views 64
 - syntax of combination descriptions 104
- cases set up
 - Analysis Wizard 299
- catalogue
 - section 169
- CEAP Modal Damping 517
- cell operators
 - in tables 39
- centre-only results 305
- chart view
 - chart view settings 489
- chart views
 - Footfall analysis 491
 - Forces on 2D Element Cut 493
 - Generic periodic load analysis 491
 - Harmonic analysis 490
 - Linear time history analysis 491
 - Modal analysis details 490
 - non-linear analysis 490
- check
 - regions 507
- Check Data command 98
- check for completeness
 - entity lists 502

- Chinese Standards
 - GB 50011-20012 spectrum 256
- clean
 - combination cases 506
- Clear Colour Modifications command 67
- clipboard
 - copy graphic image to 74
 - copy output to 466
- Codes
 - Design 18, 20
- coexistent values in envelopes
 - specifying 468
- coexistent values in output view envelopes 47
- coincidence tolerance
 - changing 430
- coincident elements
 - deleting graphically 58
 - highlighting in graphics 70
 - selecting 58
- coincident nodes
 - collapsing graphically 449
 - deleting graphically 58
 - highlighting in graphics 70
 - selecting 58
- Collapse Coincident Nodes command 449
- colour
 - in graphics 67
 - in tables 43
- colour elements by property etc. 476
- colours
 - assigning to entities 56
- colours in graphic views 489
- column
 - reinforcement 182
- COM interface 544
- COM numeric precision
 - change default 432
- COM units 536
- COM version 544
- combination cases 308
 - clean 506
 - data 308
 - definition dialog 308
 - fundamentals 102
 - syntax of descriptions 104
- Combine model 465
- command file (GWC)
 - run by 544
- command line interface 544
- Command Toolbar 125
- company information 428
- company name
 - setting 432
- Comparing models 503
- composite slab
 - 2D properties 192
- concrete
 - Design Codes 18, 20
 - material grade 155
 - section 180
- concrete beam
 - dialog 532
 - export 532
- concrete beam design 532
- concrete slab
 - calculation properties 199
 - reinforcement properties 199
 - strength reduction properties 200
- condition number 300
- conflicting constraints
 - fundamentals 117
- Connect 1D Elements command 451
- Connect 1D Elements Dialog 436
- connecting 1D elements graphically 451
- Connection
 - Member 210
- constraint equations
 - data 225
 - definition dialog 225
- constraints
 - automatic fundamentals 117
 - axes fundamentals 93
 - conflicting fundamentals 117
 - constraint equation data 225
 - creating rigid graphically 462
 - fundamentals 115
 - joint data 224
 - rigid constraint data 223
 - rigid fundamentals 116
 - tied interface data 226
- constructing a model
 - introduction 27
- construction (analysis) stages
 - in graphic views 359

- construction (analysis) stages
 - introduction to analysis 358
 - introduction to analysis results 358
 - modelling implications 355
 - results introduction 358
- construction grid
 - in graphic views 56
- contact
 - edge to ground 508
 - face to ground 508
 - node to 2D element 509
- contour barrel widths in graphic views 489
- Contour Settings command 63
- Contour Settings Dialog 480
- Contour Values Dialog 480
- contours
 - apply to selection 65
- contours on graphic views 63
- Contours Toolbar 126
- control
 - nonlinear analysis 290
- Convert Nodes to Grid command 201
- Convert Nodes to Standard command 201
- convert orientation node to angle 456
- coordinates
 - axes, non-global definition fundamentals 92
 - grid fundamentals 92
 - rounding graphically 450
- copy 41
 - Copy Areas Dialog 441
 - Copy Elements Dialog 441
 - Copy Lines Dialog 441
 - Copy Members Dialog 441
 - Copy Nodes Dialog 441
 - Copy Regions Dialog 441
 - graphic image to clipboard 74
 - nodes, elements, members, lines, areas, regions graphically 457
 - output to clipboard 466
- copy and paste analysis tasks and cases 100
- Copy command 48, 59, 74
- copy GWA data
 - from Data Explorer pane 31
- Copy Selection command 457
- copying
 - cutting and pasting in tables 41
- Copying between tables and spreadsheets 43
- cover
 - section 184
- create
 - base dummy elements 511
 - grid lines 509
 - rigid membranes from storeys 510
- Create 1D Elements from Lines command 451
- Create 2D Edge Load command 463
- Create 2D Face Load command 463
- Create 2D Pre-stress command 463
- Create 2D Thermal Load command 463
- Create a New Raft Model command 513
- Create a New Raft Model Dialog 513
- Create Applied Displacement command 463
- Create Beam Distortion command 463
- Create Beam Load command 463
- Create Beam Thermal Load command 463
- Create Facade List 511
- Create Footfall Response Data command 465
- Create Gravity Load command 463
- Create Grid Area Load command 464
- Create Grid Line Load command 464
- Create Grid Plane command 448
- Create Grid Point Load command 464
- Create Joints command 462
- Create lines from 1D elements 450
- create load panels on element list 510
- create load panels on facade 511
- create mass elements 504
 - from element mass 504
 - from member mass 504
- create mass from grid loads 504
- create mass from loads 503
- Create Members from Elements 500
- Create Node Load command 463
- Create Patterned Load Combinations 512
- Create Patterned Loads 511
- Create Pre-stress command 463
- Create Rigid Constraints command 462
- Create Settlement command 463
- create storeys 510
- Create User Axes command 448
- Create User Axes Dialog 436
- creep
 - analysis stages 282
- cross strain energy density

- cross strain energy density
 - output options 375
- Cross Strain Energy Density Results command 375
- CSA 18, 20
- CSV
 - batch saving saved views to 97
- CSV (GWA) file
 - syntax 536
- CSV file
 - export options 551
 - import options 555
- current grid
 - definition dialog 495
 - fundamentals 92
 - in graphic views 56
- current polyline in Graphic Views 60
- Cursor Mode Toolbar 127
- cursor modes
 - in Graphic Views fundamentals 72
- Curve Data Selection Dialog 526
- curves
 - alignment data 268
 - extrude geometry along alignment 443
 - Modify Curve Dialog 525
- custom links
 - fundamentals 116
 - properties 194
- cut 41
 - Forces on 2D Element Cut 493
- Cut section forces
 - Assemblies 220
- cutting
 - copying and pasting in tables 41
- cutting 1D elements graphically 451
- cylindrical axes fundamentals 91
- D -**
- dampers
 - properties 195
- damping 287
 - analysis wizard 300
 - CEAP Modal Damping 517
 - frequency damping table 261
 - mode damping table 261
 - settings 304
- soil-structure analysis 307
- data
 - delete from Data Explorer pane 31
 - extents in output views 47
 - Manage Data Dialog 493
 - selecting what to output 46
- Data Explorer pane 31
- data generation
 - 2D elliptical mesh 427
 - 2D orthogonal mesh 426
 - 2D skew mesh 426
 - 3D orthogonal mesh 428
 - constructing a model 27
 - grid 421
 - orthogonal frame 422
 - pitched portal frame 425
 - portal frame 420
 - Pratt truss 424
 - roof truss 425
 - structure types 420
 - truss 423
 - Vierendeel 423
- Data Management
 - comparing models 503
 - import GWA data 465
- Data Options Toolbar 124
- data preferences 431
- data scaling unreliable message 50
- Decrease Precision command
 - in graphic views 66
- default view settings
 - details 94
 - introduction 94
- Define Current Grid command 57
- defining current grid
 - in graphic views 57
- definition
 - 2D properties 189
 - areas 152
 - block 209
 - environmental impact 197
 - grid layouts 496
 - lines 151
 - links 194
 - masses 189
 - material curve 164
 - material wizard 161
 - member 204

- definition
 - non-linear spring curve 164
 - regions 152
 - section 165
 - springs 186
- definition axes
 - fundamentals 92
- definition dialogs
 - 2D element edge loads 241
 - 2D element face loads 239
 - 2D element pre-stress loads 243
 - 2D element thermal loads 244
 - 2D polylines 150
 - 3D element face loads 245
 - 3D element thermal loads 245
 - Analysis Case 283
 - applied displacements 230
 - assemblies 220
 - axes 145
 - beam distortion loads 236
 - beam loads 232
 - beam pre-stress loads 235
 - beam thermal loads 237
 - constraint equations 225
 - Current Grid 495
 - Element Wizard 212
 - generalised restraints 222
 - gravity loads 250
 - grid area loads 249
 - grid line loads 248
 - grid lines 147
 - grid planes 148
 - grid point loads 247
 - grid surfaces 149
 - joints 224
 - node loads 229
 - polylines 150
 - Response Spectrum Wizard 251
 - rigid constraints 223
 - settlements 231
 - tied interfaces 226
 - Variable UDL Wizard 271
- Deformation Settings command 64
- Deformation Settings Dialog 479
- deformed geometry
 - new model 497
- Deformed Image command 64
- deformed image on graphic views 64
- delete
 - 2D mesh 507
 - data from Data Explorer pane 31
 - Delete All Polyline command 60
 - Delete All Results command 85
 - Delete command 41, 462
 - Delete Displayed Loading command 465
 - Delete Element command 70
 - Delete Element Loading command 70
 - delete in tables 41
 - Delete Member command 70
 - Delete Nodal Loading command 70
 - Delete Node command 70
 - Delete Polyline Point command 60
 - delete results 84
 - Delete Results From Files command 118
 - deleting nodes and elements
 - data management 493
 - derived stress output
 - beams 371
 - descriptions
 - cases 311
 - design
 - section 110
 - steel design results step by step 414
 - steel design step by step 397
 - steel design tools step by step 414
 - design codes 18, 20
 - preferred 428
 - design features 16
 - design layer basics 356
 - Design Layer in graphic views 70
 - design pools
 - steel 186
 - design specification 143
 - Design Task 397
 - members 309
 - option 309
 - run 310
 - steel 310
 - destabilizing load
 - member 206
 - Diagram Settings command 62
 - diagrams
 - annotation options 486
 - apply to selection 65
 - change scale on change of case switch 429

diagrams

- Diagram Settings Dialog 483
- simple/detailed & draw in plane of picture switches 483

diagrams on graphic views 62

Diagrams Toolbar 125

dialogs

- 1D Element Results 533
- 2D Element Edge Load Definition 241
- 2D Element Face Load Definition 239
- 2D Element Pre-stress Load Definition 243
- 2D Element Thermal Load Definition 244
- 2D polyline definition 150
- 3D Element Face Load Definition 245
- 3D Element Thermal Load Definition 245
- Advanced Preferences 434
- Analysis Case Definition 283
- analysis specification 141
- Animation Settings 487
- Applied Displacement Definition 230
- Axes 534
- axis definition 145
- Beam Distortion Definition 236
- Beam Load Definition 232
- Beam Pre-stress Definition 235
- Beam Thermal Load Definition 237
- bridge analysis specification 144
- Bridge Options 485
- CEAP Modal Damping 517
- Combination Cases 308
- concrete beam 532
- Connect 1D Elements 436
- Constraint Equation Definition 225
- Contour Settings 480
- Contour Values 480
- Copy Areas 441
- Copy Elements 441
- Copy Lines 441
- Copy Members 441
- Copy Nodes 441
- Copy Regions 441
- create 2D element edge load 241
- create 2D element face load 239
- create 2D element pre-stress load 243
- create 2D element thermal load 244
- create 3D element face load 245
- create 3D element thermal load 245
- Create a New Raft Model 513

- create applied displacement 230
- create beam distortion load 236
- create beam load 232
- create beam pre-stress load 235
- create beam thermal load 237
- create gravity load 250
- create grid loading 247, 248, 249
- create node load 229
- create settlement 231
- Create User Axes 436
- Current Grid Definition 495
- Curve Data Selection 526
- Deformation Settings 479
- design specification 143
- Diagram Settings 483
- Disconnect Elements 441
- Display Methods 476
- DXF Export Options 530
- DXF Import Options 530
- Element Wizard 212
- environmental impact definition 145
- Export Member Input Data to CSV 533
- Extrude 443
- Find 524
- Flex 444
- Footfall Response Data 465
- Further Options 486
- Further Options (output) 468
- Generalised Restraint Definition 222
- Go To 525
- Graphic Fonts and Styles 489
- Graphic Settings Wizard 469
- Gravity Load Definition 250
- Grid Area Load Definition 249
- grid line definition 147
- Grid Line Load Definition 248
- grid plane definition 148
- Grid Point Load Definition 247
- grid surface definition 149
- GsRaft Analysis Progress 308
- GWA Import Options 526
- Joint Definition 224
- Labels and Display Methods 472
- Lighting Settings 67
- Load Case Titles 229
- material curve 164
- Material Wizard 161
- Modify 525

- dialogs
 - modify axis 145
 - Modify Curve 525
 - Move Areas 441
 - Move Elements 441
 - Move Lines 441
 - Move Members 441
 - Move Nodes 441
 - Move Regions 441
 - NASTRAN Export Options 531
 - Node Load Definition 229
 - non-linear spring curve 164
 - Numeric Format 534
 - OpenSees Export Options 531
 - Orientation Settings 488
 - Output Settings Wizard 466
 - Path Loading 274
 - polyline definition 150
 - preferred folders 434
 - RC beam 532
 - Refine Quad Elements 439
 - Replace 524
 - Response Spectrum Wizard 251
 - Rigid Constraint Definition 223
 - Settlement Definition 231
 - Split 1D Elements 437
 - Split Quad Elements 438
 - Split Tri Elements 440
 - Storey Displacements and Forces 492, 523
 - Tied Interface Definition 226
 - titles 138
 - Translucency Settings 68
 - Unwrap Settings 68
 - Variable UDL Wizard 271
 - View List 119
 - Welcome to GSA 417
- disassociate
 - 2D mesh 507
- Disconnect Elements command 461
- Disconnect Elements Dialog 441
- disconnecting elements graphically 461
- displacements
 - applied displacement data 230
 - intermediate results 533
 - output options 368
 - Storey Displacements and Forces Dialog 492, 523
- Display Methods Dialog 476
- display methods in graphic views 62
- display options 383
 - analysis stage data 394
 - bridge data 393
 - bridge results 393
 - error norm 394
 - load data 386
 - model data 383
 - numeric formats 395
 - results 388
 - user modules 394
- distance between nodes 33
- distortion loads
 - data 236
- documentation 18
- Double size of diagrams command 65
- drag nodes sculpt tool 445
- Drag on node - Collapse command 445
- Drag on node - Co-locate command 445
- Drag on node - Ignore command 445
- Drag on node - Joint command 445
- Draw Selection command 54
- drawing part of the model 53
- drilling degrees of freedom
 - Allman-Cook 302
- dummy elements
 - draw 476
- dummy mass
 - settings 304
- DXF Export Options Dialog 530
- DXF Import Options Dialog 530
- Dynamic analysis
 - scaling of modes 520
 - step by step 321
- dynamic analysis results
 - output options 367
- dynamic load
 - dynamic load factor 262
 - load curve 262
- dynamic load factor
 - data 262
- dynamic relaxation
 - incremental analysis 331
- dynamic response
 - data 251

- E -

- EC3 checker
 - National Annexes 406
- edge
 - contact to ground 508
- edge loads
 - data 241
- edges of elements
 - highlighting in graphics 69
- Edit Design Property command 70
- Edit Element command 70
- Edit Material command 70
- Edit Member command 70
- Edit Node command 70
- Edit Property command 70
- Edit Restraint Property command 70
- Edit Section Property command 70
- editing models in graphics 435
- effective lengths
 - member 206
 - steel members 414
- Effective mass
 - by part 517
- eigensolver 301
 - convergence 301
 - settings 301
 - shifting 301
 - Sturm sequence check 301
 - subspace iteration 301
- eigenvalue
 - introduction to buckling analysis 329
 - model stability 338
- eigenvalue buckling analysis 328
 - modelling implications 329
- eigenvalue buckling results
 - introduction 329
- element
 - axis 217
- element formulation
 - LS-DYNA 307
- element properties
 - Object Viewer 33
- Element Properties command 70
- element user modules 118
- elements
 - 2D element release 212
 - 2D mesh generation 111
 - add/modify one by one sculpting 445
 - adding graphically introduction 451
 - adding string of 1D graphically 451
 - Beam release 212
 - colour in graphics 67
 - Connect 1D Elements Dialog 436
 - connecting 1D graphically 451
 - convert Quad4 to Quad8 457
 - convert Quad8 to Quad4 457
 - convert Tri3 to Tri6 457
 - convert Tri6 to Tri3 457
 - copy / move graphically 457
 - Copy Elements Dialog 441
 - create members 500
 - creating from lines graphically 451
 - data 212
 - delete from members 497
 - deleting 493
 - deleting graphically 462
 - Disconnect Elements Dialog 441
 - disconnecting graphically 461
 - dummy 212
 - End stiffness 212
 - Extrude Dialog 443
 - extrude graphically 458
 - flip 505
 - flipping graphically 460
 - from members 497
 - highlighting coincident in graphics 70
 - highlighting edges in graphics 69
 - joining 1D graphically 456
 - labels in graphic views 474
 - length of selection 33
 - link fundamentals 116
 - manage user modules 521
 - mass of selection 33
 - modifying graphically 461
 - Move Elements Dialog 441
 - offset fundamentals 116
 - orientation node, convert to angle 456
 - parabolic to linear 457
 - Refine Quad Elements Dialog 439
 - renumbering 493
 - selecting for output 46
 - shrinking in graphics 66
 - spinning 2D graphically 461

- elements
 - Split 1D Elements Dialog 437
 - Split Quad Elements Dialog 438
 - Split Tri Elements Dialog 440
 - splitting 1D and 2D graphically 452
 - surface area of selection 33
 - synchronise from members 497
 - upside-down 505
 - wizard 212
- EN1993
 - Eurocode 3 405
- EN1993 checker 405
 - limitations 405
- EN19933 checker
 - buckling checks 405
 - buckling interaction check 405
 - combined local check 405
 - documentation 405
 - local axial check 405
 - local bending check 405
 - local checks 405
 - local shear check 405
 - local torsion check 405
- End condition
 - members 207
- End stiffness
 - Beam element 212
- Engineering Scale command 50
- engineering scale for structure
 - default switch 469
- entities
 - check for completeness 502
- Entity Panes
 - working with 36
- Entity Selection Mode 58
- envelope subject component
 - specifying 468
- enveloping
 - fundamentals 103
 - operator fundamentals 104
 - optimising fundamentals 104
 - permutations 104
 - stored step by step 360
 - syntax of combination descriptions 104
- enveloping in output views 47
- environmental impact
 - specification dialog 145
- step by step 360
- wizard 197
- environmental impact specification
 - dialog 145
- Equivalent Static Load 518
 - seismic 340
- error messages
 - limit numbers 432
- error norm 300
- Eurocode 18, 20
 - Eurocode 8 - 1994 spectrum 254
 - Eurocode 8 - 2003 spectrum 253, 254
- Eurocode 3
 - EN1993 405
- Evaluating expressions 120
- Expand Bridge Loading 516
- Expand Gravity Loading 523
- Expand Grid Loading 522
- Explicit 287, 288
- explicit solver
 - settings 307
 - time step 307
- explicit time history
 - analysis 324
 - modelling implications 324
 - results 324
- explicit time-history
 - analysis 365
- Explorer Panes
 - working with 31
- export
 - ADC 554
 - AutoCAD 553
 - DXF Export Options Dialog 530
 - footfall response 555
 - GWA 551
 - input data 551
 - NASTRAN 552
 - NASTRAN Export Options Dialog 531
 - OpenSees 553
 - OpenSees Export Options Dialog 531
 - output views 555
 - Pdisp 554
 - Revit 557
 - SAP2000 551
 - steel members 555
- Export command 41, 43

Export Member Input Data to CSV Dialog 533
 export options 551
 Expressions 120
 Map beam loads on 1D elements 463
 Map node loads on nodes 463
 Map pressures on 2D elements 463
 extents of output
 specifying 468
 external matrix 196
 extract
 sub-model 497
 extrude
 alignment 516
 nodes and elements graphically 458
 Extrude Dialog 443
 Extrude Selection command 458
 Eye Distance In command 53
 Eye Distance Out command 53

- F -

fabric
 material grade 159
 fabric structures
 step by step 319
 fabricated
 section 177
 facade
 create load panels 511
 facade list
 create 511
 face
 contact to ground 508
 face loads
 data 239
 facet angle of 2D elements
 change default 432
 FEMA
 FEMA 356 spectrum 255
 field types
 in tables 39
 file backups
 working with 117
 file locations
 change default 432
 find
 in tables 42
 Find command 42, 59
 Find Dialog 524
 find in Graphic Views 59
 Flex Dialog 444
 Flex Selection command 458
 flexing lines of nodes graphically 458
 flip
 elements 505
 Flip Elements command 460
 folders
 preferred folders dialog 434
 font
 in graphic views 71
 in table views 43
 fonts
 Graphic Fonts and Styles Dialog 489
 footfall analysis
 chart view 491
 modelling 326
 results 326
 step by step 326
 footfall analysis parameters
 Analysis Wizard 295
 footfall response
 export options 555
 Footfall Response Data Dialog 465
 force density
 1D properties 197
 2D properties 198
 force density form-finding 321
 forces
 intermediate results 533
 Storey Displacements and Forces Dialog 492, 523
 Forces on 2D Element Cut command 493
 form finding options
 Analysis Wizard 291
 format
 Numeric Format Dialog 534
 format of numeric output
 change default 432
 formats
 numeric fundamentals 395
 form-finding
 by force density method 321
 by soap film method 320
 modelling implications of analysis 320

form-finding
 properties 197
 structures step by step 320
 frequency damping table
 data 261
 front ordering 300
 FRP
 material grade 157
 Further Options Dialog 486
 Further Options Dialog (output) 468

- G -

GB 50011-2001
 GB 50011-2001 spectrum 256
 general reinforcement
 section 183
 generalised restraints
 data 222
 definition dialog 222
 generate
 new model wizard 428
 orthogonal frame 499
 generate member restraints
 points 209
 spans 209
 Generate Static Vehicle Load 513
 generate steel member restraints 506
 Generate User Modules command 118
 generating an imperfect geometry 329
 generating models
 2D element meshes 111
 generic dynamic analysis parameters
 Analysis Wizard 295
 Generic periodic load analysis
 chart view 491
 geometric entities
 adding graphically introduction 450
 working with 111
 geometric section
 properties 178
 geometric stiffness
 options 306
 pre-load 305
 geometry
 generating imperfect 329
 geometry checks
 settings 302
 Getting started with GSA 27
 glass
 material grade 159
 global cylindrical
 axes 89
 go to
 in tables 42
 Go To command 40, 42
 Go to Design Property Data command 70
 Go To Dialog 525
 Go to Element Data command 70
 Go to Material Data command 70
 Go to Member Data command 70
 Go to Node Data command 70
 Go to Property Data command 70
 Go to Restraint Property Data command 70
 Go to Section Property Data command 70
 graft GWA data
 from Data Explorer pane 31
 Graft model 465
 Graphic Display Toolbar 128
 Graphic Fonts and Styles Dialog 489
 Graphic Settings command 49
 graphic view
 save to file preferred format 432
 Graphic view settings 49
 Graphic view wizard 49
 graphic views
 animation 71
 annotating diagrams etc 65
 axes 66
 axes label settings 472
 case list 64
 colour in 67
 colours 489
 contour barrel widths 489
 contour settings 480
 contour values 480
 contours 63
 current grid 56
 cursor mode fundamentals 72
 defining current grid 57
 deformation settings 479
 deformed image 64
 diagram settings 483
 diagrams 62

- graphic views
 - display method settings 476
 - display methods 62
 - double-clicking 49
 - drawing part of the model 53
 - element label settings 474
 - formatting of numbers 66
 - highlighting coincident elements 70
 - highlighting coincident nodes 70
 - highlighting edges of elements 69
 - labels 61
 - layer 70
 - lighting settings 67
 - node label settings 473
 - Object Viewer 33
 - opacity 68
 - polylines in 60
 - printing from 72
 - properties 33
 - rendering quality 434
 - resetting adornments 70
 - resetting entities 53
 - right click menus 70
 - scaling diagrams etc 65
 - scaling for printing 52
 - sculpting (editing) models in 435
 - Settings Wizard 469
 - shrinking elements in 66
 - storey label settings 472
 - surface shading 67
 - symbol sizes 489
 - threading 434
 - translucency settings 68
 - units 66
 - unreliable data scaling 50
 - unwrap settings 68
 - working with 48
- graphics
 - graphic view preferences 429
 - sculpting preferences 430
- Graphics Palettes
 - working with 34
- gravity
 - data 250
- gravity loads
 - create dialog 250
 - definition dialog 250
- grid
 - in graphic views 56
 - in table views 43
- grid area loads
 - data 249
 - definition dialog 249
- grid axes
 - fundamentals 92
- Grid command 45
- grid coordinates
 - fundamentals 92
- grid frame
 - data generation 421
- grid layouts
 - definition 496
 - disabling 495
- grid line loads
 - data 248
 - definition dialog 248
- grid lines 93
 - create 509
 - definition dialog 147
- grid loading
 - create dialog 247, 248, 249
 - create graphically 464
 - data 246
 - graphical display 108
- grid loads
 - delete bridge loads 516
 - from weight 506
 - tolerance data 142
- grid nodes
 - definition 201
 - working with 111
- Grid output view
 - opening 45
- grid planes 93
 - creating graphically 448
 - data 148
 - definition 148
 - graphical display 108
 - mesh 506
- grid point loads
 - data 247
 - definition dialog 247
- grid points
 - disabling 495
- grid surfaces 93

grid surfaces 93
 data 149
 definition dialog 149
 Grid Window command 45
 grids
 Expand Grid Loading 522
 grillage
 mesh 506
 Ground View command 49
 group
 output by 47
 GSA
 analysis features 15
 basic concepts 29
 design features 16
 documentation 18
 overview 15
 program features 17
 where do I start? 27
 working with the program 29
 GSA Dialog
 Welcome to 417
 GSA Toolbar 123
 GsBridge control
 Analysis Wizard 298
 GsRaft Analysis Progress Dialog 308
 GsRaft control
 Analysis Wizard 297
 GsRaft progress
 Analysis Wizard 297
 GsRelax analysis progress information
 Analysis Wizard 292
 GsRelax analysis task control
 Analysis Wizard 291
 GsRelax member identification
 Analysis Wizard 291
 GsRevit 557
 GWA file
 export options 551
 import options 555
 string IDs (sID) 536
 syntax 536
 units 536
 GWA Import Options Dialog 526
 GWA numeric precision
 change default 432
 GWA, TXT, CSV 551, 555

GwaCommand units 536
 GWC command file
 run by 544

- H -

Halve size of diagrams command 65
 Harmonic analysis
 chart view 490
 step by step 325
 harmonic analysis parameters
 Analysis Wizard 294
 Hide Areas command 53
 Hide Contours command 63
 Hide Diagrams command 62
 Hide Elements command 53
 Hide Labels command 61
 Hide Lines command 53
 Hide Members command 53
 Hide Regions command 53
 Highlight Coincident Elements command 70
 Highlight Coincident Nodes command 70
 Highlight Edges command 69
 history 496
 HKSUOS checker 409
 Buckling checks 409
 documentation 409
 Local checks 409
 hollow slab
 2D properties 192
 Hong Kong 18, 20
 bridge analysis 348
 hourglass control 304
 How do I get started? 27
 HTML
 batch saving saved views to 97

- I -

IBC
 IBC 2000 spectrum 256
 IBC 2006 (ASCE 7-05) spectrum 253
 IBC 2009 (ASCE 7-05) spectrum 253
 ill conditioning
 model stability 338
 imperfect geometry
 generating 329

imperfection 306
 import
 AutoCAD 556
 CSV 555
 DXF Import Options Dialog 530
 GWA 555
 GWA Import Options Dialog 526
 Pdisp 557
 Revit 557
 TXT 555
 Import GWA data 465
 import options 555
 Increase Precision command
 in graphic views 66
 Indian Standards
 IS 1893 - 2002 spectrum 257
 influence analysis
 bridge 349
 Influence effects
 Beams 273
 Nodes 272
 initial pre-load 305
 input data
 export options 551
 input preferences 431
 Insert command 41
 insert in tables 41
 intermediate beam forces
 output options 369
 intermediate displacements and forces 533
 introduction
 constructing a model 27
 Introduction to GSA 27
 Invert Selection command 58, 65
 IRS 18, 20
 IS 18, 20
 IS 1893
 IS 1893 - 2002 spectrum 257
 IS800 checker
 documentation 411
 Isometric command 49
 Italian
 Ordinanza PCM 3274 spectrum 258

- J -

Join String of 1D Elements command 456

Join String of Lines command 456
 joining strings of 1D elements graphically 456
 joining strings of lines graphically 456
 joint
 as constraint equation 509
 joints
 adding by disconnecting elements graphically 461
 collapse node 504
 creating graphically 462
 data 224
 definition dialog 224
 JPEG
 batch saving saved views to 97

- K -

keyboard accelerators 129
 keyboard shortcuts 129
 keystrokes in graphics
 changing sensitivity 429
 Kg
 settings 306

- L -

labels
 apply to selection 65
 Labels and Display Methods command 61, 62
 Labels and Display Methods Dialog 472
 Labels in graphic views 61
 Labels Toolbar 125
 lane-by-lane path optimisation
 bridge 349
 layer basics 356
 legend panel in graphics
 changing 469
 length of element selection 33
 lighting settings
 in graphics 67
 Lighting Settings command 67
 line segment
 section 177
 line segment section
 properties 178
 linear static analysis
 step by step 313

- Linear time history analysis
 - chart view 491
- linear time history analysis parameters
 - Analysis Wizard 294
- lines
 - copy / move graphically 457
 - Copy Lines Dialog 441
 - creating from 1D elements graphically 450
 - definition 151
 - Grid 93
 - joining graphically 456
 - labels in graphic views 475
 - Move Lines Dialog 441
 - rationalize lines 505
 - working with 111
- link
 - node to 2D element 509
- links
 - fundamentals 116
 - properties 194
- links / stirrups
 - section 184
- Lists
 - check for completeness 502
 - data 310
 - effective mass 517
 - embedded 86
 - working with 86
- Lists Toolbar 128
- load cases
 - data 229
 - deleting 493
 - fundamentals 101
 - renumbering 493
 - titles dialog 229
- load curve 287
 - data 262
 - time history 520
- load panels
 - 2D element fundamentals 240
 - 2D properties 189
 - create on facade 511
 - create on surface 510
- loading
 - 2D Element Edge Load Definition dialog 241
 - 2D Element Face Load Definition dialog 239
 - 2D Element Pre-stress Load Definition dialog 243
 - 2D Element Thermal Load Definition dialog 244
 - 3D Element Face Load Definition dialog 245
 - 3D Element Thermal Load Definition dialog 245
 - Applied Displacement Definition dialog 230
 - Beam Distortion Definition dialog 236
 - Beam Load Definition dialog 232
 - Beam Pre-stress Definition dialog 235
 - Beam Thermal Load Definition dialog 237
 - create 2D element edge load dialog 241
 - create 2D element face load dialog 239
 - create 2D element pre-stress load dialog 243
 - create 2D element thermal load dialog 244
 - create 3D element face load dialog 245
 - create 3D element thermal load dialog 245
 - create applied displacement dialog 230
 - create beam distortion load dialog 236
 - create beam load dialog 232
 - create beam pre-stress load dialog 235
 - create beam thermal load dialog 237
 - create element load graphically 463
 - create gravity load dialog 250
 - create grid load graphically 464
 - create grid loading dialog 247, 248, 249
 - create nodal load graphically 463
 - create node load dialog 229
 - create settlement dialog 231
 - delete from Data Explorer pane 31
 - Gravity Load Definition dialog 250
 - grid Definition dialog 247, 248, 249
 - map loading graphically 463
 - Node Load Definition dialog 229
 - Settlement Definition dialog 231
 - Types 227
- loads
 - 2D element loading data 238
 - applied displacement data 230
 - beam load data 232
 - beam loading data 232
 - beam pre-stress load data 235
 - create mass from grid loads 504
 - create mass from loads 503
 - deleting graphically 465
 - distortion load data 236
 - edge load data 241
 - Expand Bridge Loading 516
 - Expand Gravity Loading 523
 - Expand Grid Loading 522
 - face load data 239

loads

- Generate Static Vehicle Load 513
- gravity data 250
- grid area loads data 249
- grid line loads data 248
- grid loading data 246
- grid point loads data 247
- nodal loading data 229
- node load data 229
- Optimise Path Loading 515
- pre-stress 2D load data 243
- settlement data 231
- thermal 2D load data 244
- thermal beam load data 237

local axes

- constraint axes fundamentals 93

Local checks

- AS4100 400

LS-DYNA 365

- composite shell 193
- control parameters 307
- spring properties 521

- M -

Manage Tasks command 98

manage user modules 521

manual backup

- working with 117

Manual Backup command 117

Map Beam Loads on 1D Elements command 463

Map beam loads on beam elements

- Expressions 120

Map Face Loads on 2D Elements command 463

Map node loads on nodes

- Expressions 120

Map Node Loads on Nodes command 463

Map pressures on 2D elements

- Expressions 120

mapping file 569

margin around image

- changing 429

Mark Task For Analysis command 97

marking views and view lists for printing and saving to file 97

mass 133

- axis 217

create mass from grid loads 504

create mass from loads 503

mass calculation

- Analysis Wizard 289

mass matrix

- external 196

mass of element selection 33

masses

- properties 189
- storeys 520

Material | Material Grades | Steel 397

material grade

- aluminium 157
- concrete 155
- fabric 159
- FRP 157
- glass 159
- reinforcement 160
- steel 155
- timber 158

material properties

- data 154, 161

materials

- analysis data 161
- wizard 161

matrix

- external 196
- sping 165

matrix singularity

- model stability 338

matrix solver

- settings 300

max combination case operator 104

measure distance between nodes 33

member

- Axes 207
- Connection 210
- definition 204
- destabilizing load 206
- steel 206
- timeline 513

member properties

- Object Viewer 33

Member Properties command 70

member restraints

- points 209
- spans 209

- member restraints
 - steel restraint description 210
- member user modules 118
- members
 - copy / move graphically 457
 - Copy Members Dialog 441
 - create elements 497
 - create from elements 500
 - delete elements 497
 - design task 309
 - end conditions 207
 - labels in graphic views 474
 - length of selection 33
 - manage user modules 521
 - mass of selection 33
 - Move Members Dialog 441
 - orientation node, convert to angle 456
 - selecting for output 46
 - steel design step by step 397
 - straightness tolerance 142
 - surface area of selection 33
 - synchronise from elements 497
 - trim 505
- mesh
 - delete for soil zones/profiles 508
 - floor 506
 - regions 507
 - soil zones/profiles 507
- mesh generation
 - 2D elements 111
 - modelling regions 114
 - tips on modelling 115
- Message Pane
 - working with 33
- Message Pane command 33
- messages in report
 - limit numbers 432
- methods
 - Display Methods Dialog 476
- min combination case operator 104
- Miscellaneous
 - deleting results from files 118
 - file backups 117
 - unlocking files 117
- MITC 302
- Modal analysis details
 - chart view 490
- modal analysis results
 - output options 367
- modal damping
 - CEAP Modal Damping 517
- modal dynamic parameters
 - Analysis Wizard 285
- modal dynamics
 - introduction to analysis 321
- modal force 517
- modal P-delta analysis
 - step by step 323
- mode damping table
 - data 261
- model
 - Create a New Raft Model Dialog 513
 - generate orthogonal frame 499
 - principal axes 518
 - units fundamentals 140
- Model comparison 503
- model construction
 - introduction 27
- model stability 338
- modelling
 - footfall analysis 326
 - for analysis (construction) stages 355
 - for bridge analysis 347
 - for eigenvalue buckling analysis 329
 - for form-finding analysis 320
 - for non-linear static analysis 318
 - for seismic analysis 339
 - RC slab reinforcement design 365
 - using 2D elements 314
- modelling implications
 - explicit time history 324
- models
 - generating 2D element meshes 111
- modify
 - in tables 42
- Modify Colour command 43
- Modify Colour of Selection command 67
- Modify Colour of This command 67
- Modify command 42
- Modify Curve Dialog 525
- Modify Dialog 525
- Modify Elements command 461
- modify elements sculpt tool 447
- modify linear 2D to parabolic 457

Modify Nodes command 449
 modify nodes sculpt tool 447
 modify properties
 section 502
 modify/parabolic 2D to linear 457
 mouse sensitivity preferences
 changing 429
 move
 Move Areas Dialog 441
 Move Elements Dialog 441
 Move Lines Dialog 441
 Move Members Dialog 441
 Move Nodes Dialog 441
 Move Regions Dialog 441
 nodes by expression graphically 458
 nodes, elements, members, lines, areas, regions
 graphically 457
 Move Selection command 457
 multi-page tables 38
 multiple increment
 dynamic relaxation analysis 331

- N -

NASTRAN
 export options 552
 NASTRAN Export Options Dialog 531
 New Analysis Task command 98
 New Graphic View (main thread) command 48
 New Graphic View command 48
 new model
 deformed geometry 497
 new model wizard
 generate 428
 properties 420
 structure type 419
 titles 419
 New Output View command 44, 48
 New users introduction 27
 nodal user modules 118
 Nodal User Modules command 521
 node loads
 create dialog 229
 definition dialog 229
 node properties
 Object Viewer 33
 Node Properties command 70

node stiffness
 ground spring 505
 node to 2D element
 contact 509
 nodes
 add more than one graphically 449
 add/modify/drag one by one sculpting 445
 adding graphically 449
 collapsing coincident graphically 449
 coordinates, non-global definition fundamentals
 92
 copy / move graphically 457
 Copy Nodes Dialog 441
 definition 201
 deleting 493
 deleting graphically 462
 draw shrunk 476
 Extrude Dialog 443
 extrude graphically 458
 Flex Dialog 444
 flexing lines graphically 458
 highlighting coincident in graphics 70
 Influence effects 272
 labels in graphic views 473
 load data 229
 loading data 229
 manage user modules 521
 modifying attributes graphically 449
 Move Nodes Dialog 441
 Node Grid Settings 201
 Node Grid Settings Definition dialog 201
 Node Mesh Attribute 201
 Node Mesh Attribute Definition dialog 201
 Node Stiffness 201
 Node Stiffness Definition dialog 201
 renumbering 493
 rounding coordinates graphically 450
 selecting for output 46
 shrink 476
 straightening lines graphically 460
 transform graphically 458
 non-linear
 combination fundamentals 102
 termination time 305
 nonlinear analysis 290
 control 290
 damping 304
 load factor 290

- nonlinear analysis 290
 - termination 290
 - non-linear analysis
 - chart view 490
 - step by step 316
 - non-linear buckling analysis 330
 - non-linear spring
 - material curve 164
 - non-linear static
 - introduction to analysis 318
 - non-linear static analysis 317
 - modelling implications 318
 - non-linear static options
 - Analysis Wizard 289
 - non-stiff degrees of freedom 302
 - numeric format
 - in preferred and saved views 96
 - output 44
 - Numeric Format command
 - in graphic views 66
 - Numeric Format Dialog 534
 - numeric formats
 - fundamentals 395
 - numeric output format
 - change default 432
- O -**
- offset
 - section 180
 - offsets
 - apply 505
 - draw 476
 - draw marks 476
 - element fundamentals 116
 - members 207
 - opacity
 - in graphics 68
 - open command file (GWC) 544
 - OpenSees
 - export options 553
 - OpenSees Export Options Dialog 531
 - operations
 - in tables 40
 - operators
 - abs 104
 - combination case descriptions 104
 - in tables 39
 - max 104
 - min 104
 - signabs 104
 - Optimise Path Loading 515
 - optimising enveloping
 - fundamentals 104
 - options
 - Bridge Options Dialog 485
 - DXF Export Options Dialog 530
 - DXF Import Options Dialog 530
 - Further Options Dialog 486
 - GWA Import Options Dialog 526
 - NASTRAN Export Options Dialog 531
 - OpenSees Export Options Dialog 531
 - Orientation
 - Member 207
 - orientation angle
 - beam axis 217
 - orientation node
 - beam axis 217
 - Orientation Settings Dialog 488
 - Orientation Toolbar 126
 - orthogonal frame
 - data generation 422
 - generate 499
 - output
 - case and entity lists 46
 - change default numeric format 432
 - data extents 47
 - enveloping 47
 - grid view 45
 - of data on a selection set 46
 - selecting what to 46
 - summary 47
 - tabbed tables 45
 - to spreadsheets 48
 - totals 47
 - working with views 44
 - Output command 48
 - Output Explorer pane 31
 - output options 383
 - 2D element derived force results 378
 - 2D element displacements 376
 - 2D element stresses 376
 - 3D element displacements 382
 - 3D element stresses 382

- output options 383
 - analysis stage data 394
 - beam cross strain energy density 375
 - beam derived stresses 371
 - beam displacements 369
 - beam force sign convention 369
 - beam forces 369
 - beam strain energy density 374
 - beam stresses 371
 - bridge data 393
 - bridge results 393
 - buckling analysis results 367
 - cross strain energy density 375
 - displacements 368
 - dynamic analysis results 367
 - error norm 394
 - intermediate beam forces 369
 - load data 386
 - modal analysis results 367
 - model data 383
 - numeric formats 395
 - reactions 368
 - results 388
 - SED 374
 - static analysis results 366
 - strain energy density 374
 - stress averaging 381
 - torque lines 376
 - user modules 394
- Output Panes
 - working with 33
- output preferences 432
- output view
 - change spacing 45
 - save to file preferred format 432
- output view settings
 - axes 44
 - numeric format 44
 - units 44
- output views
 - by case/property/group 47
 - export options 555
 - printing from views 48
 - Settings Wizard 466
 - threading 434
- overview 15

- P -

- page setup
 - preferred 428
- Page Setup command 72
 - output 48
- Pan Down command 51
- Pan Left command 51
- Pan Right command 51
- Pan Up command 51
- panels
 - draw 476
 - graphical display 108
- Paste 41
- Paste command 41, 59
- paste GWA data
 - from Data Explorer pane 31
- paste special Dialog 526
- pasting in tables 41
- path generation
 - Hong Kong 348
 - UK 348
- Path Loading dialog 274
- paths
 - data 269
 - Optimise Path Loading 515
- patterned load
 - Create Patterned Load Combinations 512
 - Create Patterned Loads 511
 - fundamentals 133
- P-delta analysis
 - step by step 313
- P-delta modal dynamic analysis
 - step by step 323
- P-delta results
 - introduction 313
- P-delta Ritz analysis
 - step by step 323
- Pdisp
 - export options 554
 - import options 557
- perimeter
 - section 175
- perimeter reinforcement
 - template 183
- periodic load analysis

- periodic load analysis
 - step by step 327
- permutations
 - fundamentals 104
- Pick Colour of This command 67
- pick tolerance
 - changing 430
- piled-raft
 - introduction to analysis 353
- piled-raft analysis
 - step by step 350
- pitched portal frame
 - data generation 425
- Plan command 49
- plane strain
 - new model wizard 419
- plane stress
 - new model wizard 419
- Planes
 - Grid 93
- PNG
 - batch saving saved views to 97
- point void
 - tendon 185
- point voids
 - section 180
- points
 - member restraints 209
- Polyline Properties command 60
- polylines
 - check for validity 150
 - data 150
 - definition dialog 150
 - draw 476
 - in Graphic Views 60
 - length 60
 - selecting within 58
- pools
 - section 186
- portal frame
 - data generation 420
 - pitched 425
- positive result 521
- post-analysis 84
- Prat truss
 - data generation 424
- precision 338
- preferences
 - basic 428
 - steel catalogue 434
- preferred folders 434
- preferred units
 - fundamentals 140
- preferred views
 - batch printing 97
 - batch saving to file 97
 - details 94
 - introduction 94
 - opening 32
 - units and numeric format 96
- pre-load 305
- pre-stress 2D loads
 - data 243
- pre-stress beam loads
 - data 235
- principal axes
 - model 518
- print
 - page setup 432
- print by case
 - changing 469
 - printing 72
- Print command 72
 - output 48
- Print Preview command 72
 - output 48
- printing
 - view lists 96
 - views and view lists remotely 97
- printing from graphic views 72
- printing from output views 48
- profile
 - component 167
 - section 167, 169
 - sections 110
- program features 17
- programming interface 544
- Progress Dialog
 - GsRaft Analysis 308
- projected axes
 - fundamentals 92
- properties
 - 2D elements 189
 - 3D elements 193

properties

- analysis stage property data 282
- cables 194
- concrete slab calculation 199
- concrete slab reinforcement 199
- concrete slab strength reduction 200
- dampers 195
- force density 1D 197
- force density 2D 198
- form-finding properties 197
- links 194
- masses 189
- material data 154, 161
- new model wizard 420
- non-linear spring curve 164
- Object Viewer 33
- output by 47
- RC slab calculation 199
- RC slab reinforcement 199
- RC slab strength reduction 200
- section modifiers 185
- section profile 178
- soap film 1D 198
- soap film 2D 198
- spacers 195
- spring curves data 164
- spring matrix 165
- springs 186

property tables

- changing number of points 431

Pseudo response spectrum analysis

- dynamic details 293

Pseudo response spectrum parameters

- Analysis Wizard 293

- Q -

quads

- properties 189
- Refine Quad Elements Dialog 439
- Split Quad Elements Dialog 438

- R -

raft

- Boussinesq 143
- Create a New Raft Model Dialog 513
- introduction to analysis 352

Mindlin 143

raft analysis

- specification 143
- step by step 350

raft analysis data requirements 350

raft analysis notes 354

raft results

- introduction 354

raft solution method 351

randomized load vector

- Ritz analysis 302

rationalise sections 568

rationalize

- section for BIM 500

rationalize lines 505

RC beam

- dialog 532
- export 532

RC beam design 532

RC slab

- calculation properties 199
- reinforcement properties 199
- strength reduction properties 200

RC slab reinforcement design

- introduction 365
- modelling implications 365
- negative areas 365
- step by step 365

reactions

- error norm 368
- output options 368

redo

- summary 417

reduced integration 304

reference point

- offset 505
- section 180

Refine Mesh at Nodes command 452, 455

Refine Quad Elements command 452, 455

Refine Quad Elements Dialog 439

Regenerate command 50

region

- delete mesh 507
- disassociatemesh 507

regions

- check 507
- copy / move graphically 457

- regions
 - Copy Regions Dialog 441
 - definition 152
 - labels in graphic views 475
 - mesh 507
 - modelling for mesh generation 114
 - Move Regions Dialog 441
 - tips on modelling for mesh generation 115
 - working with 111
- reinforcement
 - general 183
 - links / stirrups 184
 - material grade 160
 - over 184
 - section 180
 - smeared 184
 - template 181, 182
- Reinforcement Template 537
- Release
 - 2D element 212
 - Beam element 212
- renumbering
 - elements 493
 - nodes 493
- repeated freedoms
 - data 224
- replace
 - in tables 42
- Replace command 42
- Replace Dialog 524
- replica
 - block 209
- Report Pane
 - working with 34
- Report Pane command 34
- Rescale Data command 65
- Reset Display Adornments command 70
- Reset Pan command 51
- Reset Preferences command 32
- Reset to All Entities command 46, 70
- resetting graphics display adornments 70
- resetting graphics entities 53
- residual
 - settings 305
- response
 - dynamic response data 251
 - Footfall Response Data Dialog 465
- response spectrum
 - ASCE 7-05 253
 - broadening 519
 - Eurocode 8 - 1994 254
 - Eurocode 8 - 2003 253, 254
 - FEMA 356 255
 - GB 50011-2001 256
 - generate 519
 - IBC 2000 256
 - IBC 2006 253
 - IBC 2009 253
 - IS 1893 - 2002 257
 - Ordinanza PCM 3274 258
 - UBC 1994 258
 - UBC 1997 259
 - user defined 260
- response spectrum analysis
 - combination method 292
 - dynamic details 293
 - rigorous or fast 292
 - seismic 341
 - spectrum 292
- Response spectrum parameters
 - Analysis Wizard 292
- response spectrums
 - data 251
 - wizard 251
- Restore Manual Backup command 117
- Restore Timed Backup command 117
- restraints
 - generalised restraint data 222
 - steel member restraint description 210
- result
 - steel design results step by step 414
- Result Pane
 - working with 34
- Result Pane command 34
- results
 - change output view type 45
 - explicit time history 324
 - footfall analysis 326
 - selection 305
 - steel design 398
- Revit link
 - export from Revit to GSA 557
 - import to Revit from GSA 560
 - introduction 557

- Revit link
 - Model suitability for BIM Coordination 557, 559, 560, 568
 - recommended practice 568
 - registration 569
 - update Revit from GSA 559
 - right click menu 70
 - right click menus in graphic views 70
 - rigid constraints
 - data 223
 - definition dialog 223
 - fundamentals 116
 - Rigorous or fast
 - response spectrum analysis 292
 - Ritz analysis
 - randomized load vector 302
 - settings 302
 - Ritz analysis parameters
 - Analysis Wizard 288
 - Ritz P-delta analysis
 - step by step 323
 - rods
 - axis 217
 - roof truss
 - data generation 425
 - Round Coordinates command 450
 - rounding coordinates graphically 450
 - run by command file (GWC) 544
- S -**
- SANS10162 checker
 - buckling checks 412
 - buckling interaction check 412
 - documentation 412
 - limitations 412
 - local checks 412
 - SAP2000
 - export options 551
 - Save Default View Settings 94
 - Save Graphic View 95
 - Save Output View 95
 - Save Selection As 2D Polyline command 61
 - Save Selection As List command 59
 - saved image settings 433
 - saved view
 - save to file preferred format 432
 - saved views
 - batch printing 97
 - batch saving to file 97
 - details 95
 - introduction 94
 - opening 32
 - units and numeric format 96
 - saving to file
 - view lists 96
 - views and view lists remotely 97
 - scale
 - diagrams change scale on change of case switch 429
 - structure to engineering scale default switch 469
 - Scale to fit command 50
 - scaling
 - graphic diagrams etc 65
 - printed graphic image 52
 - Scaling of modal results 520
 - sculpt 450
 - adding elements introduction 451
 - adding geometric entities introduction 450
 - adding string of 1D elements 451
 - collapsing coincident nodes 449
 - connecting 1D elements 451
 - copy / move nodes, elements, members, lines, areas, regions 457
 - create element loading 463
 - create grid loading 464
 - create joints 462
 - create nodal loading 463
 - create rigid constraints 462
 - creating 1D elements from lines 451
 - creating lines from 1D elements 450
 - delete elements 462
 - delete nodes 462
 - deleting loading 465
 - disconnecting elements 461
 - extrude nodes and elements 458
 - flex lines of nodes 458
 - flipping elements 460
 - geometry cursor modes 445
 - grid planes 448
 - highlighting 430
 - joining lines 456
 - joining strings of 1D elements 456
 - map loading on elements 463

- sculpt 450
 - map loading on nodes 463
 - models in graphics 435
 - modify node attributes 449
 - modifying elements 461
 - nodes adding more than one 449
 - nodes/elements one by one 445
 - references 430
 - rounding nodal coordinates 450
 - spinning 2D elements 461
 - splitting 1D and 2D elements 452
 - straightening lines of nodes 460
 - transform nodes 458
 - user axes 448
- sculpt tool
 - add elements 446
 - add lines 446
 - add nodes 446
 - drag nodes 445
 - modify elements 447
 - modify nodes 447
- Sculpt Toolbar 124
- sculpting
 - coincidence tolerance 430
 - constructing a model 27
 - pick tolerance 430
 - reference point for selection 430
 - straightness tolerance 430
- section
 - catalogue 169
 - concrete 180
 - cover 184
 - definition 165, 169
 - design parameters 110
 - fabricated 177
 - general reinforcement 183
 - line segment 177
 - links / stirrups 184
 - modify properties 502
 - offset 180
 - perimeter 175
 - point voids 180
 - pools 186
 - preferred catalogue 428
 - profile 167, 169
 - property calculation 178
 - reference point 180
 - reinforcement 180
 - smeared reinforcement 184
 - standard shspe 170
 - steel design 180
 - template reinforcement 181
 - tendon 185
- section analysis properties
 - modifiers 185
- section design pools
 - steel 186
- Section Display command 62
- section mapping, Revit
 - mappings 569
- section profile
 - properties 178
- sections
 - design parameters 109
 - material types 109
 - member types 109
 - modifiers 185
 - preferred catalogue 434
 - profile 110
 - rationalize for BIM 500
 - sorting into vertical and non-vertical 568
 - superseded 504
 - working with 109
- SED
 - output options 374
- seismic
 - Equivalent Static / Accidental Torsion Load 518
 - equivalent static load 340
 - Response Spectrum Generator 519
 - Storey Masses 520
 - time history 520
- Seismic analysis 339
 - modelling implications 339
 - scaling of modes 520
 - step by step 339
- Select 2D Polyline command 61
- Select All command 58, 65
- Select Closest To Vertical 1D Elements command 58
- Select Highest Coincident command 58
- Select List command 59
- Select None command 58, 65
- Select Out-Of-Range command 58
- Select Polyline command 60
- Select String command
 - elements and members 58

- Select String command
 - for polyline points 60
- selecting entities in sculpt
 - Entity Selection Mode 58
 - forming selection sets 58
 - selecting topologies 58
 - selecting within polylines 58
- selection
 - Curve Data Selection Dialog 526
- selection properties
 - Object Viewer 33
- Selection Properties command 33
- serviceability
 - steel design 310
- Set Current Grid to This
 - in graphic views 57
- Set Current Grid to This command 70
- sets
 - working with 86
- settings
 - 2D element analysis 302
 - Animation Settings Dialog 487
 - Contour Settings Dialog 480
 - Contour Values Dialog 480
 - damping 304
 - Deformation Settings Dialog 479
 - Diagram Settings Dialog 483
 - dummy mass 304
 - eigensolver 301
 - explicit solver 307
 - geometric stiffness 306
 - Graphic view 49
 - lighting settings 67
 - matrix solver 300
 - Orientation Settings Dialog 488
 - Output Settings Wizard 466
 - output view 44
 - pre-load 305
 - results 305
 - Ritz analysis 302
 - Saved Image 433
 - soil-structure analysis 307
 - termination time 305
 - translucency settings 68
 - unwrap settings 68
- settlements
 - create dialog 231
 - data 231
 - definition dialog 231
- shade surfaces
 - changing 469
- Shade Surfaces command 67
- shading surfaces
 - in graphics 67
- shell 193
 - composite 193
 - LS-DYNA 193
- Shine Light From Here command 67
- shortcuts 129
- show in Graphic Views 59
- Shrink command 66
- shrinking elements
 - in graphics 66
- sIDs
 - convention 134
 - format 134
 - fundamentals 134
 - GWA 536
- signabs combination case operator 104
- simple beams
 - section modifiers 185
- simple static analysis
 - introduction 84
- single increment
 - dynamic relaxation analysis 331
- single tables 38
- size columns to fit
 - in table views 43
- size rows to fit
 - in table views 43
- Skew View command 49
- slab reinforcement
 - layers 182
 - template 182
- slab reinforcement design
 - modelling implications 365
 - step by step 365
- sliding joints
 - data 224
- smeared reinforcement
 - section 184
- Snap - Align with Grid Axis command, adding elements 446
- Snap - Align with Grid Axis command, adding lines 446

- Snap - Centre command, adding elements 446
- Snap - Centre command, adding lines 446
- Snap - Perpendicular command, adding elements 446
- Snap - Perpendicular command, adding lines 446
- Snap to Grid Points command 57
- soap film
 - 1D properties 198
 - 2D properties 198
- soap film form-finding 320
- soil
 - interaction data 267, 268
 - rigid boundary 143
 - zones and profiles display 476
- soil interaction
 - data 267, 268
- soil profiles
 - 3D mesh 507
 - delete 3D mesh 508
- soil zones
 - 3D mesh 507
 - delete 3D mesh 508
- soil-structure analysis 307
 - damping 307
 - settings 307
- solid sections in graphic views 62
- solver options
 - Analysis Wizard 283
- solver preferences 431
- space
 - new model wizard 419
 - structure type 419
- spacers
 - 2D elliptical mesh 427
 - 2D orthogonal mesh 426
 - 2D skew mesh 426
 - properties 195
- spans
 - member restraints 209
- sparse direct solver 300
- specification
 - analysis specification definition 141
 - bridge analysis specification 144
 - design specification definition 143
 - environmental impact definition 145
 - Output Settings Wizard 466
 - unit data 138
- spectrums
 - response spectrum data 251
- spherical axes fundamentals 91
- Spin 2D Elements command 461
- Split 1D Elements command 454
- Split 1D Elements Dialog 437
- split beams
 - at columns 521
- Split Quad Elements command 452, 455
- Split Quad Elements Dialog 438
- Split Quad to Tri elements command 452, 455
- Split Tri Elements command 452, 455
- Split Tri Elements Dialog 440
- split warped 2D elements 505
- splitting elements 452
- Spreadsheet Format command 48
- spreadsheets
 - copying to and from tables 43
 - data from output views 48
- spreadsheets & CAD
 - constructing a model 27
- spring
 - axis 217
 - non-linear spring curve 164
- spring properties
 - LS-DYNA 521
- spring support 201
 - ground spring 505
- springs
 - curves data 164
 - matrix 165
 - non-linear spring curve definition 164
 - properties 186
- stages
 - analysis stage data 281
 - definition data 282
 - property data 282
- standard shape
 - section 170
- Standard Toolbar 122
- starting to construct a model 27
- Starting to use GSA 27
- startup
 - show welcome dialog 432
- static analysis
 - introduction 84
 - non-linear 317

- static analysis
 - step by step 313
- static analysis cases
 - Analysis Wizard 284
- static analysis results
 - output options 366
- static p-delta
 - Analysis Wizard 285
- steel
 - Design Codes 18, 20
 - design task 310
 - material grade 155
 - member restraint description 210
 - section pools 186
- steel design
 - results step by step 414
 - section 180
 - step by step 397
 - tools step by step 414
- steel design results
 - effective lengths 414
 - introduction 414
- steel member
 - design results step by step 414
- steel member restraints
 - description 210
 - generate 506
- steel members
 - export options 555
 - restraint description 210
- step by step
 - analysis envelopes 360
 - analysis summary 85
 - bridge analysis 345
 - buckling analysis 328
 - dynamic analysis 321
 - environmental impact 360
 - footfall analysis 326
 - harmonic analysis 325
 - linear static analysis 313
 - modal P-delta analysis 323
 - non-linear analysis 316
 - P-delta analysis 313
 - periodic load analysis 327
 - piled-raft analysis 350
 - raft analysis 350
 - RC slab reinforcement design 365
 - response spectrum analysis 341
 - Ritz P-delta analysis 323
 - seismic analysis 339
 - steel design 397
 - Time history analysis 325
 - Wave loading 361
- Step by step guide
 - introduction 27
- stiffness matrix
 - external 196
 - model stability 338
- stiffness options
 - matrix solver 300
- Stop 71
- Stop command 71
- Storey Displacements and Forces command 492, 523
- storey masses 520
- storey views
 - details 96
- storeys
 - Displacements and Forces command 492, 523
 - Displacements and Forces Dialog 492, 523
 - Storey Displacements and Forces Dialog 492, 523
- Straighten Spacing Evenly command 460
- Straighten Spacing Proportionally command 460
- straightening lines of nodes graphically 460
- straightness tolerance
 - preferred 430
- strain energy density
 - output options 374
- strength
 - steel design 310
- stress averaging
 - output options 381
- stress output
 - 2D elements 376
 - 3D elements 382
 - beam derived stresses 371
 - beams 371
- string IDs
 - convention 134
 - format 134
 - fundamentals 134
 - GWA 536
- structure
 - default to engineering scale switch 469

- Structure comparison 503
 - structure types
 - data generation 420
 - styles
 - Graphic Fonts and Styles Dialog 489
 - in graphic views 71
 - subject component in output view envelopes 47
 - sub-model
 - extraction 497
 - subspace iteration
 - convergence 301
 - shifting 301
 - Sturm sequence check 301
 - summary output 47
 - superseded
 - sections 504
 - surface
 - create load panels 510
 - surface area of element selection 33
 - surface shading
 - changing 469
 - in graphics 67
 - Surfaces
 - Grid 93
 - sustainability 360
 - Switch Layer in graphic views 70
 - symbol sizes in graphic views 489
 - synchronise
 - elements from members 497
 - members from elements 497
 - syntax
 - combination case descriptions 104
- T -**
- Tab Separate Tables command 45
 - table views
 - colour in 43
 - data display in 43
 - Object Viewer 33
 - properties 33
 - working with views 38
 - tables
 - basic operations 40
 - cell operators 39
 - copying cutting and pasting in 41
 - copying to and from spreadsheets 43
 - delete blank and insert in 41
 - field types 39
 - find replace go to and modify in 42
 - from Data Explorer pane 31
 - navigation and editing 40
 - operators 39
 - single and multi-page 38
 - task view
 - working with 97
 - task view right-click menu
 - working with 98
 - tasks
 - Analysis Wizard 99
 - copy and paste 100
 - fundamentals 102
 - tasks (analysis) and cases
 - fundamentals 101
 - Template
 - Reinforcement 537
 - template reinforcement
 - definition 181
 - main 181
 - perimeter 183
 - rectangular column 182
 - side 182
 - slab 182
 - slab / wall 182
 - transverse allowance 183
 - tendon
 - bridge beam 185
 - position 180
 - section 185
 - tendon positions 175
 - termination
 - nonlinear analysis 290
 - termination time
 - settings 305
 - Text (GWA) file
 - syntax 536
 - thermal 2D loads
 - data 244
 - thermal beam loads
 - data 237
 - tied interfaces
 - data 226
 - definition dialog 226
 - timber

- timber
 - material grade 158
 - time history 287, 288
 - select 520
 - Time history analysis
 - step by step 325
 - time step
 - explicit solver 307
 - timed backup
 - set interval 432
 - working with 117
 - timeline
 - member 513
 - titles
 - definition 138
 - new model wizard 419
 - tolerances 142
 - preferred 428
 - preferred model 431
 - toolbars 121, 122
 - Command 125
 - Contours 126
 - Cursor Mode 127
 - Data Options 124
 - Diagrams 125
 - Graphic Display 128
 - GSA 123
 - Labels 125
 - Lists 128
 - Orientation 126
 - Sculpt 124
 - Standard 122
 - tools
 - create rigid membranes from storeys 510
 - steel design step by step 414
 - torse lines
 - output options 376
 - totals in output 47
 - touch gestures
 - generally 121
 - in Graphic Views 121
 - touch screen 121
 - Transform geometry
 - Expressions 120
 - Transform Geometry command 458
 - translucency
 - in graphics 68
 - Translucency Settings command 68
 - transverse allowance
 - template reinforcement 183
 - triangles
 - properties 189
 - Split Tri Elements Dialog 440
 - trim
 - overlapping members 505
 - trimming 1D elements graphically 451
 - truss
 - data generation 423
 - Pratt 424
 - Vierendeel 423
 - TXT
 - batch saving saved views to 97
 - TXT file
 - export options 551
 - import options 555
- U -**
- UBC
 - UBC 1994 spectrum 258
 - UBC 1997 spectrum 259
 - UK
 - bridge analysis 348
 - undo
 - enable/disable undo, preference 434
 - summary 417
 - undo steps
 - change default 431
 - units
 - change default 431
 - GWA 536
 - in model fundamentals 140
 - in preferred and saved views 96
 - in table views 43
 - output 44
 - preferred 428
 - preferred fundamentals 140
 - specification data 138
 - Units command
 - in graphic views 66
 - Unlock File command 117
 - unreliable data scaling 50
 - unwrap mode
 - in graphics 68

Unwrap Settings command 68
 update
 sections 504
 user axes
 Create User Axes Dialog 436
 creating graphically 448
 user modules 118
 manage 521
 user-defined materials
 data 161
 USNRC
 spectrum broadening 519
 utilization
 steel design results step by step 414

- V -

variable UDLs
 wizard 271
 VBA interface 544
 vehicles
 data 270
 Generate Static Vehicle Load 513
 vertical
 axes 89
 Vierendeel
 data generation 423
 View Explorer pane 32
 View List Dialog 119
 view lists
 details 96
 introduction 94
 view settings 94
 View settings painter
 Graphic view 49
 Output view 44
 views
 auto view details 96
 batch printing 97
 batch saving to file 97
 default view settings details 94
 saved and preferred 94
 saved view details 95
 storey view details 96
 Visual Basic interface 544
 volume clipping 54

- W -

wall elements
 limitations 316
 results 379
 warped 2D elements
 split 505
 Wave loading
 step by step 361
 Weight 133
 Welcome to GSA Dialog 417
 wheel
 patch definition 515
 Wheel Patch Definition 515
 Where do I start? 27
 wizard
 Graphic view 49
 Output view 44
 Wizard command 49
 wizards
 element 212
 environmental impact 197
 Graphic Settings 469
 material 161
 Output Settings 466
 Response Spectrum 251
 tables 38
 Variable UDL 271
 WMF
 batch saving saved views to 97
 working with
 section 109
 working with GSA 29

- X -

X Elevation command 49

- Y -

Y Elevation command 49

- Z -

Zoom In command 51
 Zoom Out command 51

Endnotes 2... (after index)

Back Cover