



Oasys AdSec

Help Manual

Oasys

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 AdSec

© Oasys Ltd. 2019

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: May 2019

Table of Contents

Part I About AdSec	8
1 Overview	8
2 Design Codes	9
Part II Step by Step Guide	14
1 Creating a Simple Section	14
2 Creating a Compound Section	15
3 Load Cases and Analysis Cases	16
Part III Working with AdSec	19
1 Using AdSec	19
2 Sign Convention	21
3 Working with the Gateway	21
4 Working with the Section View	22
5 Working with Table Views	23
Editing Tables	24
Find, replace, go to and modify in tables	25
6 Ultimate Limit State Analysis Options	26
7 Serviceability Limit State Analysis Options	29
8 Strain Discontinuities	30
9 Staged Analysis	31
Part IV Program Data	33
1 Titles	33
2 Specification	34
General Specification	34
Units	34
3 Material Properties	34
Concrete Properties	35
Rebar Material Properties	35
Steel Material Properties	36
FRP Material Properties	37
4 Sections	37
Definition	38
Reinforcement	38
5 Loads	39
Load Titles	39
Loading	39
Reference Point	40
6 Analysis Cases	40

ULS Analysis Cases	41
SLS Analysis Cases	41
Part V Dialogs and Wizards	44
1 General Section Wizard	44
General Section Wizard : Titles	44
General Section Wizard : Design Option	45
General Section Wizard : Wrap/Cast Section	45
General Section Wizard : Definition	46
General Section Wizard : Reinforcement	46
2 Section Wizard	47
Section Wizard : Section Type	47
Section Wizard : Catalogue Section	48
Section Wizard : Standard Shapes	48
Section Wizard : Perimeter Section Definition	50
Section Wizard : Section Definition	50
3 Section Material	51
4 Compound Section Definition	51
5 Section Builder	51
6 Reinforcement Wizard	51
Reinforcement Wizard : Template Definition	52
Reinforcement Wizard : Links and Cover	52
Reinforcement Wizard : Main Reinforcement	52
Reinforcement Wizard : Side Reinforcement	53
Reinforcement Wizard : Rectangular Column Reinforcement	53
Reinforcement Wizard : Circular/Elliptical Reinforcement	53
General Reinforcement Definition	53
7 Analysis	54
8 Advanced Analysis Control	55
9 Chart Analysis Dialogs	55
N/M Chart	56
Myy/Mzz Chart	56
M/Curvature and M/EI Chart	57
10 Miscellaneous Dialogs	57
Stress-strain Chart	58
Expand Section	58
Label and Display Options	58
Chart Style	59
Additional Points on N/M and Myy/Mzz Charts	59
Output Specification	60
11 Preferences	60
Part VI Output Options	63
1 Graphical Output	63
2 Tabular Output	63
Part VII Command File	66
1 Command File Commands	66

Part VIII AdSec Text File

Part I

1 About AdSec

Oasys AdSec is a program for non-linear analysis of sections with a particular emphasis on concrete sections. Analysis options are available for ultimate and serviceability limit states in accordance with various design codes.

Permission to reproduce extracts from the British Standards is granted by BSI. British Standards can be obtained from:

BSI Customer Services,
389 Chiswick High Road,
London W4 4AL.
Teleph +44 (0)20 8996 9001
one
Email cservices@bsi-global.com

1.1 Overview

Oasys AdSec is a program for non-linear analysis of sections with a particular emphasis on concrete sections. Analysis options are available for ultimate and serviceability limit states in accordance with selected [concrete design codes](#)

Analysis – ULS

For the Ultimate Limit State (ULS) the section analysis options are:

- the ultimate moment capacity of the section
- stresses from the ultimate applied load
- ultimate axial force/moment (N/M or P/M) interaction charts
- ultimate moment (M_{yy}/M_{zz}) interaction chart (for biaxial bending only)

Analysis – SLS

For the Serviceability Limit State (SLS) the program calculates:

- cracking moment
- stresses, strains, stiffness and crack widths for each applied loading and strain
- moment-curvature and moment stiffness charts

Applications

Concrete, steel and fibre-reinforced polymer (FRP) sections of either standard or user defined shape.

Steel sections can be selected from steel catalogues used in various countries. Reinforcement can be defined for concrete sections: this can be steel, prestress tendons or FRP bars.

These basic sections can be combined into compound sections, by joining or wrapping sections.

Reinforcement can be prestressed with a force or initial strain.

The following load types cannot be input explicitly, but can be simulated:

- nominal eccentricity moments
- prestress using unbonded tendons

1.2 Design Codes

Concrete

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

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	2009	2001
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>	Germany	2004	
DIN EN 1992-2	<i>Eurocode 2-2</i>	Germany	2010	
DS/EN 1992-1-1	<i>Eurocode 2-1-1</i>	Denmark	2004	
DS/EN 1992-2	<i>Eurocode 2-2</i>	Denmark	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	Eurocode 2-2	Italy	2006

Part II

2 Step by Step Guide

The Step by Step Guide is intended to give detailed instructions of how to carry out a number of typical AdSec operations. The details of the working with the program and the program data are give in separate chapters.

Welcome to AdSec

When launching AdSec the *Welcome to AdSec* dialog is displayed. There are options to

- Create a new section – will open the General Section Wizard guiding the user through the definition of the titles, specification and section.
- Work on your own – allows access directly to the AdSec menus.
- Open an existing file – a file dialog is opened so that the user can select an existing AdSec file to work on
- Select recent file – the recently used files are displayed and the user is able to select the file with which to continue working. The file can also be opened by double-clicking on the file in the list.

The "About AdSec" option give access to license and version details.

See also

[Creating a Simple Section](#)

[Creating a Compound Section](#)

[Load Cases and Analysis Cases](#)

2.1 Creating a Simple Section

When opening AdSec a *Welcome to AdSec* dialog offers a selection of options. Select the option to *Create a new section*. This opens the General Section Wizard — this is used in various ways in AdSec for creating and editing sections.

Titles

When creating a new section job titles and details are entered here and then the design code is chosen. As well as the design code the units can be changed here and the bending axis can be selected between bi-axial and uni-axial bending. Uniaxial ending has the additional option of specifying the section as a slab or wall in which case cracking checks are omitted on the "side" faces.

Uni-axial bending constrains the neutral axis to remain horizontal despite asymmetric geometry or loading. Moments generated about the vertical axis are ignored. Use this option for constrained sections (e.g. angles restrained by walls or slab) and for standard rectangular beams to Eurocode to allow crack widths to be calculated. For other cases a biaxial bending analysis should be used, allowing the neutral axis to rotate from horizontal as a result of any asymmetry of reinforcement or section, even for applied bending about the horizontal axis.

Section

The next stage is to define the basic section. The section is given a name as a convenient way of referring to and identifying the section. The material type allows for concrete, steel or fibre reinforced polymer (FRP) sections. Depending on the material selected the material grades available will change. The section button gives access to the normal section wizard to select the section shape and size or alternatively if the section description syntax is known it can be entered directly. The section shape is drawn to provide visual feedback on the section shape.

The next stage only applies for concrete sections as this is where the reinforcement is defined. Reinforcement can be either "general" or "template" (although the template option is not available for all section shapes).

General Reinforcement

In the General Reinforcement Definition the section extents are displayed as a guide for placing the reinforcement. General reinforcement defines single bars, lines of bars or arcs/circles of bars and the bars are specified by type and diameter. The actual coordinates required to define the location of the bars will depend on the definition type selected.

Prestress in bars is specified here as part of the bar definition. Prestress force is specified as a force in each bar, or an initial strain applied to the bars.

The modify allows the reinforcement definition to be edited while the shift option allows the reinforcement to be repositioned.

Template Reinforcement

Template reinforcement starts by considering the section as a beam or column and then offers reinforcement patterns appropriate to that section shape. The template option allows large numbers of bars to be defined in a quick and convenient manner.

The section is now complete.

Further sections can be created however in this case the process starts at the section definition stage.

2.2 Creating a Compound Section

A compound section is one built from a number of component sections. This can be used to analyse situations such as:

- where a slab may be cast after a beam is in place
- where the slab and beam may be of different strengths
- for sections of different materials such as concrete cased steel sections or
- for sections where FRP is added to strengthen the section.

A number of approaches are possible to build a compound section assuming the reference or base section has already been defined. The most suitable approach depends on the final section to be assembled.

Compound Section Definition

The [compound section definition](#) is the lowest level approach to assembling a compound section. The different sections are selected and the offsets from the reference or base section are specified.

Section Builder

The [section builder](#) option allows a compound section to be assembled from two existing sections by specifying the position of one section relative to the other. AdSec will calculate the offsets based on the section dimensions.

Wrap/Cast Section

The "wrap/cast section" option allows a compound section to be created by introducing a new section which is wrapped or cast around or inside the base section. The options are:

- **Apply to face** – this is used to apply a thin section to the faces of the base section. This can either warp the entire section or apply a strip of material to the bottom or top face of the section. This option is typically used to "apply" FRP to an existing section.
- **Cast rectangle or circle** – these options allow a new section to be cast around the existing section. These options are typically used to create concrete cased steel sections.
- **Fill void** – this option which is only available for hollow sections allows the void to be filled. This option is typically used for concrete filled tubes.

Once the secondary section option is defined the definition of the section is as before but the wizard takes care of the section shape creating a secondary section that giving the correct intersection with the reference section.

2.3 Load Cases and Analysis Cases

It is important to understand the difference between load cases and analysis cases. The load cases group together loads that apply to the section, however the analysis case refers to a load case or cases and allows combinations of load to be considered.

Use the Loads table or the Load Definition dialog to define load records. Any load records that belongs to a particular load case will be grouped together.

Analysis cases are set up for ULS or SLS cases separately and both ULS and SLS analysis cases are numbered from 1.

For a [ULS Analysis](#) use the ULS Analysis Cases table or ULS Analysis Case Definition dialog to define the analysis cases.

For an [SLS Analysis](#) use the SLS Analysis Cases table or SLS Analysis Case Definition dialog to define the analysis cases. Analysis cases for an SLS analysis are more complicated than for ULS analysis as the load duration and creep effects need to be taken into account. The load duration options will depend on the design code selected. The basic options are long term where creep effects are considered and short term where no creep takes place. The creep factor is specified for each component of the section and effect of creep is modelled by modifying the Young's modulus

and the slope of the stress strain relationship.

$$E_{long} = \frac{E}{1 + \phi}$$

If the loading is dead load the long term option (which includes creep) should be used. If the load is a short term imposed load then the short term option is selected which excludes the effect of creep. Most real loading will have a component of long term load plus short term imposed loads. AdSec offers two options for modelling this type of situation.

When the design code is BS8110 the effect of the combined load can be modelled using the "long + short" term option. With this the long term load is allowed to creep but the additional short term load is only applied after the creep effects of the long term load have been taken into account.

When the design code is BS5400, or a related code, the effect of combined load is modelled using the intermediate term analysis option. In this the creep effect is varied depending on the ratio of live load to dead load. In this cases the analysis case must define the live to dead load ration M_q/M_g .

When selecting an analysis (either ULS or SLS) you are give the opportunity to select the analysis cases that you wish to analyse.

Note for a ULS analysis if no analysis cases have been defined then a single analysis case is created for each load case.

Part III

3 Working with AdSec

AdSec is a general non-linear section analysis program.

Output from AdSec includes:

- Section properties
- Section strength
- Force moment (N/M or P/M) and moment (M_{yy}/M_{zz}) interaction charts
- Section stress/stain at serviceability state
- Crack widths and cracking moment
- Flexural stiffness (EI) of section
- Moment curvature and moment stiffness charts

AdSec provides different material models for concrete and reinforcement. These are detailed in the AdSec Theory Manual

Sections can be loaded with force/moment, component strains/curvature, concrete-only strain/curvature and prestress.

The features of the non-linear solution are:

- Solution method is iterative
- Plane sections remain plane
- Program searches through possible strain planes
- Three variables – one strain, ϵ_x and two curvatures, κ_y and κ_z .
- Strain at point (y,z) = $\epsilon_x + \kappa_y \cdot z + \kappa_z \cdot y$
- Loops until a plane is found that satisfies three conditions

3.1 Using AdSec

When using AdSec it is important to understand how the model is constructed and loaded. Also important is how the general properties are applied to the particular section or component of a section.

Creating and Editing Sections

The General Section wizard will guide the user through the process of creating a section when a new file is created. The file can contain more than one section. To create subsequent sections the user should select the "Data | New Section" menu option or where the new section is closely related to the shape of the existing section the "Data | Wrap Current Section" menu option. Note that the main purpose of multiple sections is to build up Compound Sections (see below), and that generally a file should only contain one section.

Perimeter sections can be entered clockwise or anti-clockwise. The perimeter is closed automatically. The same applies to voids in perimeter sections.

Once a section is created the *Gateway* gives access to all aspects of the section. Select *Section | Dimensions* if you want to change its dimensions. Only select *Section | Definition* if you wish to change the section shape and re-create the reinforcement.

Reinforcement

Reinforcement can be defined as either:

- General reinforcement (i.e. single bars, lines, arcs and circles of bars) which can be placed for any section.
- Template reinforcement (i.e. beam and column arrangements) is available for selected (common) section shapes. The Reinforcement wizard will guide the user through the process of creating Template reinforcement.

A section may contain both template and general reinforcement.

Compound Sections

A compound section is made up of existing sections (component sections). There are several benefits to this:

Component sections can be analysed separately from the compound section without having different files.

Sections with Template beam and column reinforcement arrangements can be used to make up the Compound section.

The positioning of component sections is easy and flexible (see below)

To create a compound section, select the "Data | New Compound Section" option. In a compound section the original section is referred to as section "A". The origin of section A will be the origin of the Compound section. Position additional section(s) by specifying the offset of their origins relative to the origin of section A. You should ensure that component sections are in contact but do not overlap.

The "Data | Wrap Current Section" menu option also creates a compound section.

Current Section

AdSec can contain multiple section so it has the concept of a current section. The current section is:

- the section displayed in the Section View
- the section on which an analysis will be performed
- the section whose data will be changed on selecting any of the items in the Data menu

The current section is displayed on the toolbar. It can be changed here, or by clicking on any item in the Sections tab of the Gateway.

Analysis Cases

Analysis cases contain data to be used in a ULS Capacity (Strength), ULS Loads, or SLS Loads (Serviceability) analysis. Each analysis is performed for a specified case or cases. AdSec stores ULS and SLS analysis cases. The load descriptions specify factored load cases to be included in the analysis. For example an analysis case load description is of the form 1.4L1+1.6L2.

3.2 Sign Convention

Horizontal axis is labelled y , and is positive to the right

Vertical axis is labelled z , positive upwards

Applied Forces

N is axial load is positive for compression (referred to as P in some codes)

M_{yy} is the moment about the y axis, positive for compression on the top of the section.

M_{zz} is the moment about the z axis, positive for compression on the right of the section.

Strain planes

ε_{ax} is axial strain, positive for compression.

κ_y is the curvature about the y axis, positive for compression on the top of the section.

κ_z is the curvature about the z axis, positive for compression on the right of the section.

$$\varepsilon = \varepsilon_{ax} + \kappa_y \cdot z' + \kappa_z \cdot y'$$

Note: Moment angles and neutral axis (NA) angles are measured positive anticlockwise from the positive y axis. A negative concrete-only strain models shrinkage in the concrete. A negative prestress gives tension in the steel.

3.3 Working with the Gateway

When an AdSec file is read the Gateway view is opened. This is a view giving access to all the modules that go to make up an AdSec model. If this view has been closed it can be re-opened using the *View / Gateway* menu command.

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

Clicking the right mouse button when the cursor is pointing at an item in the Gateway displays a floating menu that relates to that item. Double clicking on an item will open the appropriate view.

The Gateway organizes the data into several categories. Against each item is reported the number of records currently specified for the data module. These are updated as the data is created or edited.

3.4 Working with the Section View

When an AdSec file is read the Gateway view is opened and normally also a graphic view. The graphic view displays the basic section including the reinforcement. The section view always displays the current section and changing the current section results in the view being updated to reflect this change.

If results are present some basic results are display in the graphic view – details of these are given elsewhere.

The appearance of the section can be modified using the *Label and Display Options*.

The view is adjustable dynamically and via the Graphic menu. The redraw option just redraws the image without altering the view. Dragging a rectangle in the window causes the image to be zoomed and panned to result in that rectangle filling the Window.

Operation	Short-cut
Scale to fit	Ctrl+Home
Zoom in	Ctrl+Up
Zoom out	Ctrl+Dn
Reset pan	Shift+Home
Pan right	Shift+Rt
Pan left	Shift+Left
Pan up	Shift+Up
Pan down	Shift+Dn

The section view can also be saved to file in one of the following formats:

- WMF – Windows Metafile
- JPEG
- PNG – Portable Network Graphics
- DXF – AutoCAD DXF
- SVG – Scalable Vector Graphics

See also:

[Graphical Output](#)

[Label and Display Options](#)

3.5 Working with Table Views

Tabular input

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

Tables in AdSec are similar to spreadsheets, but there are a number of significant differences.

The current cell is 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.

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.

The '=' and '==' commands may be used to copy data from other records in a table. Instead of typing a value in a cell, the following instructions may be entered:

- = to copy the value from the cell above.
- =n to copy the value from the same cell in record n.
- == to copy the remainder of the record from the record above.
- ==n to copy the remainder of the record from record n.

Selecting

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 several 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.

See also

[Editing tables](#)

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

3.5.1 Editing Tables

Data Entry

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

Changes made in a table are immediately reflected in other tables and the graphic view.

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

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.

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.

Information can be cut/copied from tables in AdSec and pasted into spreadsheets and vice versa. Also results can be copied from Output Views and pasted into spreadsheets. Data copied from AdSec 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 AdSec and copied to form a 'template' in the spreadsheet.

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 toolbar.

The font that is used in the table can be selected. This gives a standard Font selection dialog.

3.5.2 Find, replace, go to and modify in tables

Find

Use the 'Edit | Find' (Ctrl+F) menu command or the 'Find' button on the Data 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**, use the 'Edit | Go To' (Ctrl+G) menu command or the 'Go To' button on the Data toolbar.

Replace

Use the 'Edit | Replace' (Ctrl+H) menu command or the 'Go To' button on the Data toolbar to perform a search, as for Find, and also replace the specified text or numbers. The 'Replace' dialog 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 toolbar to modify numerical cell entries. Cells containing text remain unmodified. The Table View 'Modify' dialog 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).

3.6 Ultimate Limit State Analysis Options

Strength Analysis

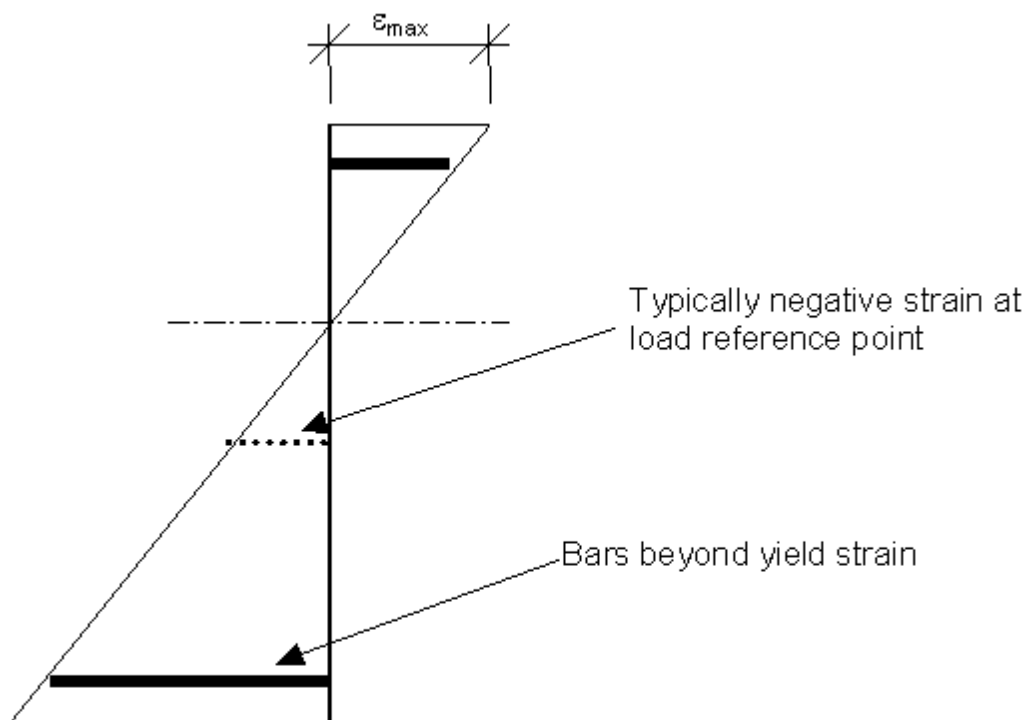
An ultimate capacity moment is found for each selected analysis case, solving for the input values of axial force, moment orientation and limiting strain. The resulting ultimate moment capacity and neutral axis position are output. The resulting stresses and strains can also be viewed.

Loads Analysis

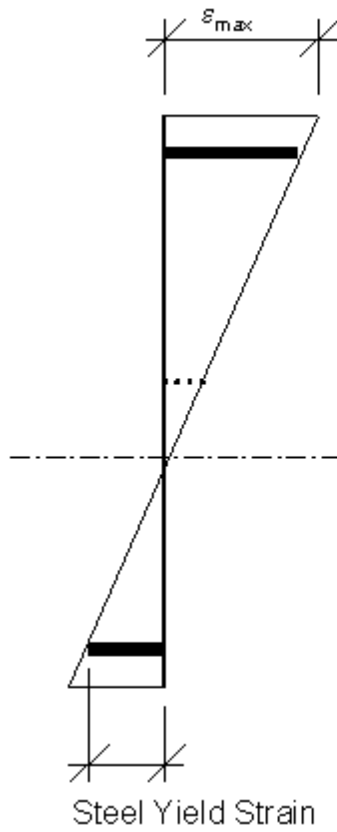
The program finds the state of strain corresponding to the input (factored) force and moments. Factored material curves are used. The resulting stresses and strains can be viewed.

Force Moment Interaction Charts

The Force/Moment Interaction (N/M) chart gives a capacity envelope for a given moment orientation. The 'no tension' and 'balanced yield' values are found for the input value of moment orientation. An ultimate capacity analysis is performed to generate ultimate moments for a range of axial force values between maximum tension and compression capacity of the section. If the reference point is offset from the plastic centroid, the peak of the graph will be offset from the Y-axis ('moment=0' position). This is due to the moment of the maximum possible axial force about the reference point. In addition, the peak force on the graph may not be equal to the maximum ultimate resistance for constant strain for some moment angles for biaxial problems. This is because the maximum ultimate capacity may cause secondary bending at 90 degrees to the specified angle about the reference point. As secondary bending is ignored in uniaxial problems, the graph should reach the maximum ultimate resistance for these problems.



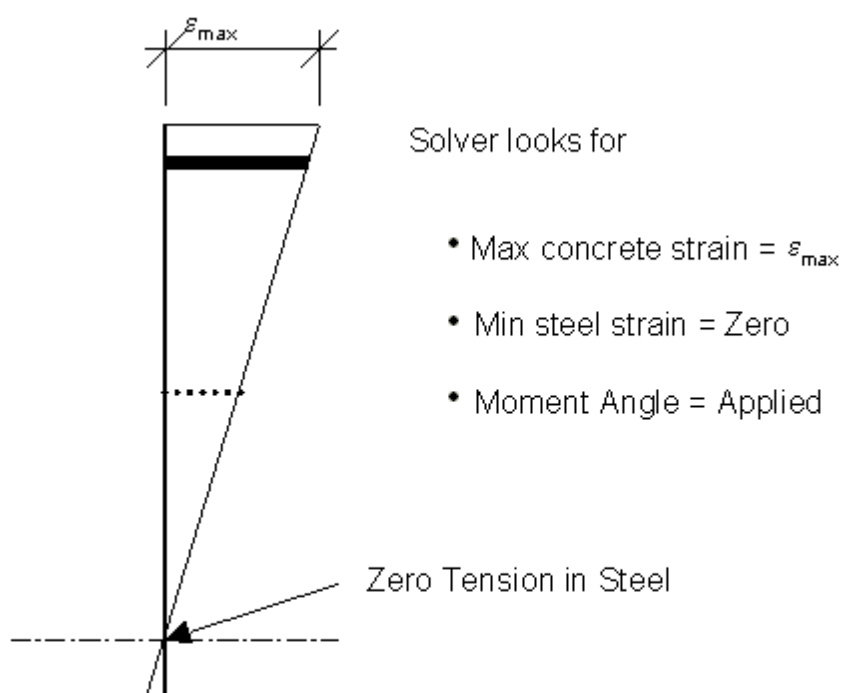
N/M chart – Typical strain plane for point on Lower Chart



Solver looks for

- Max concrete strain = ϵ_{max}
- Min steel strain = Yield strain
- Moment Angle = Applied

N/M chart – Balanced Yield Point



N/M chart – Typical strain plane for No Tension point

The user can superimpose force-moment coordinates on the chart "Graphic | Additional Point" menu option a shortcut is available on the Graphic toolbar. This is useful for demonstrating that applied loads are within the capacity envelope.

Moment interaction charts

The Moment Interaction (M_y/M_{zz}) charts give a capacity envelope for a given axial force. The moment orientation is varied from zero to 360 degrees. An ultimate moment capacity analysis is performed for each orientation and the input value of axial force. The results are plotted and can be output in a table. The program assumes the prestress factor to be equal to one for the chart. A number of plots can be output on one chart for different values of axial force.

User input M_y, M_{zz} coordinates can be plotted on the chart. This is useful for demonstrating that applied loads are within capacity.

3.7 Serviceability Limit State Analysis Options

Loads Analysis

The Serviceability Limit State (SLS) Loads analysis can be used to investigate a number of serviceability issues.

- Stiffness
- Cracking
- Stress / Strain in section
- Staged loading
- Strain discontinuities

AdSec offers a choice of material properties for serviceability, allowing accurate modelling of material non linear behaviour. The choice of material models will depend on the design code selected.

The various material options are discussed in the AdSec Theory Manual section.

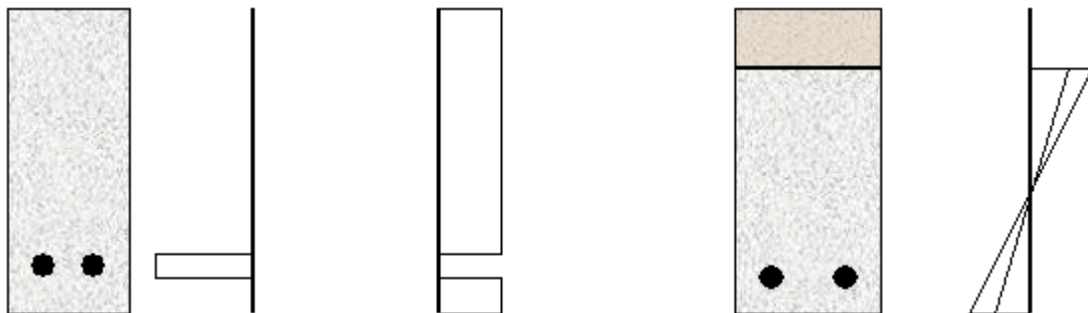
Moment Curvature and Moment Stiffness Charts

The Moment/Curvature (M/κ) and Moment/Stiffness (M/EI) charts give the variation of the stiffness or curvature of the section with increasing moment. This gives an way to find an appropriate equivalent bending stiffness for a given level of moment. This is the value that should be used in analyses to predict displacements of concrete structures.

A number of curves can be produced for different levels of axial force.

3.8 Strain Discontinuities

AdSec can be used for non-linear analysis of sections where there are locked in strain planes. This can occur when there is prestress, creep and shrinkage or locked in strain planes in a compound section.



Example of strain discontinuities

Prestress

Single bars and lines of bars can be prestressed with a force or a strain. A negative value of prestress gives tension in the bar. Specify prestress forces or strain in the reinforcement table. Template reinforcement cannot be prestressed. Prestress can be factored in the SLS analysis cases. Factor 0.0 gives behaviour without prestress. Use different prestress factors to compare prestress levels within the same data file for design and include relaxation.

Note that an AdSec analysis will automatically generate prestress losses due to shortening of the concrete and creep. So the prestress value, or prestress factor should not allow for these effects.

Creep and Shrinkage

Both these effects can be modelled by 'concrete only' strain planes, however for a single stage of loading, creep is accounted for automatically by the program using the creep factor and need not be modelled by concrete only strain planes.

3.9 Staged Analysis

With AdSec it is possible to analyse the effects of adding more structure onto existing sections that are already loaded. In conducting a staged analysis you must ensure to transfer any load and strain conditions from one stage to the next.

For example, consider a situation where you are modelling a reinforced concrete beam that is unpropped before the slab is cast on top.

To do this you would create three sections within AdSec:

- i. A beam section that stops at the slab soffit,
- ii. A slab section that represents the effective width, and
- iii. A [compound section](#) that has the two appropriately adjacent.

Then to conduct the staged analysis:

- 1) Create a load for the Beam, say a Section Force applying the bending moment
- 2) Create a SLS load case that applies that load to the Beam
- 3) Run the SLS analysis for the Beam section
- 4) Go to **Tools -> Extract Strain Plane...** and chose the appropriate strain plane. This will create a new load that is a Component Strain on the Beam section
- 5) Set the Compound section as active
- 6) Create a new Section Force, etc. that represents the additional load
- 7) Create a SLS or ULS load case that analyses the compound section for the initial loading (from step 1) plus the extracted strain plane (from step 4) and the new load (from step 6).

This method also works, for example, for strengthening existing in situ beams, columns, and slabs with steel or carbon fibre plates.

Part IV

4 Program Data

This chapter describes the different types of data that can be used to describe the model. The data is organized in modules and displayed for input and editing in a number of dialogs and tables. All the tables can be accessed from the Data pull down menu, or from the General or Sections tab in the Gateway.

The same input data is used for both Ultimate and Serviceability Limit States as the program generates factored and long-term data as required. Analysis to BS8110:1997 (UK) and BS8110:1985 (Hong Kong), BS5400 and EN 1992-1-1 (Eurocode) is available. Code-specific defaults are set by the program on selection of a code of practice.

Ultimate capacity analysis will give limiting moments based on material factors, load factors and limiting concrete and steel strains.

Serviceability analysis will give the stiffness, stresses, strains, crack widths and cracking moments generated using loading and user selected material properties. The material properties include a choice of compression and tension stiffness options.

Stresses and Strains can be calculated to BS5400 Appendix A A2.2 by selecting Linear Concrete compression and BS8110-2 tension stiffness. However due to inconsistencies within BS5400 this curvature analysis will not give a crack width which complies to BS5400.

4.1 Titles

The titles view contains the job details. This data is for information only and has no effect on the rest of the model. The data in this section can be displayed in the Oasys Columbus document management system.

The title entries, excluding the Notes and Bitmap, are printed at the top of each page of output. For a new file, certain entries default to those in the last file that was saved, but can be changed here.

Job Number – This is a number used to identify a particular job.

Initials – The initials of the engineer responsible for the model.

Edit Date – This is supplied automatically and records when the model was last edited.

Job Title, Subtitle, Calc. Heading – These fields give a brief description of the job and the calculation particulars.

Notes – This gives the user a place to record any notes that should be kept with the model.

4.2 Specification

The data described below is required to define a new problem for analysis.

See also

[General Specification](#)

[Units](#)

4.2.1 General Specification

This defines the basic parameters that will govern the analysis

Design code – In the general specification the user chooses the [design code](#). Where the code has been updated there may be several variants of the code identified by date. An option allows inclusion of superseded codes.

Bending axes – The problem is defined as biaxial (bending about the y and z axes) or uniaxial (bending about the y axis only). The uni-axial option is provided for cases where some external action on the section constrains it to bend about a single axis.

Slab/wall – the model can be defined as slab or wall. If a wall the section is assumed to represent a strip and where the sides are part of a continuous material. This will set the bending axes to biaxial

Show parameters – opens a dialog to display Eurocode nationally determined parameters

Advanced – opens a dialog for code specific items

Surface tolerance – this is used when generating circular sections to determine the number of facets required to represent the circle as a polygon.

4.2.2 Units

The user can select any system of units that is convenient and change units at anytime. Changing the units does not change the values stored in AdSec, only the values presented to the user. It is therefore possible to define the problem using SI units and examine the output using *kip* and *in* units.

A set of base units (force, length, section dimensions and stress) are defined and other units (e.g. moment) are derived from these. A number of preset units selections is available but the user may select any set of units to suit.

All data is stored internally in SI units.

4.3 Material Properties

AdSec works with concrete, steel and fibre reinforced polymer (FRP) materials and these are defined separately. There are also separate materials for rebar.

In all of these material definition dialogs there is an option to display the stress-strain curves.

The material properties are defined as below.

See also

[Concrete Properties](#)

[Rebar Material Properties](#)

[Steel Material Properties](#)

[FRP Material Properties](#)

4.3.1 Concrete Properties

Concrete properties defined are short-term (ignoring creep) and unfactored. The basic material property data defined during input is used to generate stress-strain relationships used during analysis. These stress-strain curves are generated using material factors, and creep coefficients.

Name – used to identify the concrete material.

Set properties to – allows a material to be initialized from a standard grade. The standard values can be overridden as desired.

Strength, density & stiffness – The concrete strength is characterized by the compressive strength (cube or cylinder strength) and density. The tensile strength is assumed to be a function of the compressive strength and density but can be overridden. The stiffness is characterized by the elastic (or Young's) modulus.

Extra data – gives access to extra data to better define the material, but not required for AdSec.

Material models & strain – these are the different material models for compression and tension, and for serviceability and strength calculations. These are characterized by strain limits.

Partial factors / resistance factors – depending on the design code either partial factors or resistance factors may be defined. These are normally set to the code specified value for strength calculations, but can be varied for other cases.

Confined properties – For a Mander confined material or Eurocode confined material extra data is required to generate the confined material curve from the base curve. For Eurocode this will be the confining stress from which a confined strength is derived. For the Mander curve the confined strength is defined directly. In both cases the strain limits are modified.

4.3.2 Rebar Material Properties

Reinforcement can be define as standard steel reinforcement, prestress tendons or GFRP/CFRP reinforcement. The basic material property data defined during input is used to generate stress-strain relationships used during analysis. These relationships are generated using material factors.

Name – used to identify the rebar material.

Set properties to – allows a material to be initialized from a standard grade. The standard values can be overridden as desired.

Bar type – used to select from steel rebar, prestress tendons or FRP bars

Material model – defines the material model and stress-strain curves are specified for the rebar. The same curve is used for both ULS and SLS. The stress-strain curves are described in detail in the AdSec Theory Manual.

Strength and Stiffness – the rebar strength is characterized by the tensile strength. The stiffness is characterized by the elastic (or Young's) modulus. A hardening modulus is also use for material models involving hardening.

Limiting strains – the strains at which characterize the rebar material model is deemed to have failed.

Ductility – is used in some codes to differentiate between different rebar properties for the same strength.

Extra data – gives access to extra data to better define the material, but not required for AdSec.

Partial factors / resistance factors – depending on the design code either partial factors or resistance factors may be defined. These are normally set to the code specified value for strength calculations, but can be varied for other cases. For some codes there is a factor on both strength and stiffness.

4.3.3 Steel Material Properties

The basic material property data defined during input is used to generate stress-strain relationships used during analysis. These relationships are generated using material factors.

Name – used to identify the steel material.

Set properties to – allows a material to be initialized from a standard grade. The standard values can be overridden as desired.

Material model – defines the material model and stress-strain curves are specified for the steel. The same curve is used for both ULS and SLS. The stress-strain curves are described in detail in the AdSec Theory Manual.

Strength and Stiffness – the steel strength is characterized by the tensile strength. The stiffness is characterized by the elastic (or Young's) modulus. A hardening modulus is also use for material models involving hardening.

Limiting strains – the strains at which characterize the steel material model is deemed to have failed.

Extra data – gives access to extra data to better define the material, but not required for AdSec.

Partial factors / resistance factors – depending on the design code either partial factors or resistance factors may be defined. These are normally set to the code specified value for strength calculations, but can be varied for other cases.

4.3.4 FRP Material Properties

The basic material property data defined during input is used to generate stress-strain relationships used during analysis. These relationships are generated using material factors.

Name – used to identify the FRP material.

Set properties to – allows a material to be initialized from a standard grade. The standard values can be overridden as desired.

Strength and Stiffness – the FRP strength is characterized by the tensile strength. The stiffness is characterized by the elastic (or Young's) modulus. A hardening modulus is also used for material models involving hardening.

Bond strain limit – the strains at which the bond between FRP and the base material will fail.

Extra data – gives access to extra data to better define the material, but not required for AdSec.

Partial factors / resistance factors – depending on the design code either partial factors or resistance factors may be defined. These are normally set to the code specified value for strength calculations, but can be varied for other cases. FRP uses a factor on both strength and stiffness.

4.4 Sections

General Sections

AdSec works with concrete, steel and FRP sections. The concrete sections are the most complex with a definition consisting of two parts – the basic concrete section and the reinforcement.

More than one section can be included in a model to facilitate the analysis of compound (or composite) sections, sections belonging to a family with similar geometry or different design options subjected to similar load.

Compound Sections

In many cases sections are built up from component parts into a final section. This process can be modelled in AdSec using compound sections. A compound section is composed of a number of simple sections that are offset relative to one another.

This is accessed from the "Data | New Compound Section" menu command or the AdSec toolbar shortcut.

See also

[Definition](#)

[Reinforcement](#)

4.4.1 Definition

Sections can be defined in two different ways – either as a standard section shape with dimensions or as a perimeter (with voids).

Name – name used to identify that section.

Material type and grade – the type of section (e.g. concrete) and the material grade (e.g. C40/50). Depending on the code the options are chosen from concrete, steel and FRP.

Section – Standard shapes such as rectangles, circles, etc are defined. The section is then specified by dimensions such as depth and breadth, diameter, etc. The section can be specified directly if the syntax is known, so for example a rectangular section 500 mm deep and 300 mm wide would be STD R 500 300. Alternatively the Section button can be used to open the Section Wizard to help define the section.

Perimeter sections are defined by a series of coordinates that define the outline of the section. The perimeter definition can include voids and may be defined in a clockwise or anti-clockwise direction.

Note: changing the *definition* will mean that all template reinforcement is lost. Changing the *dimensions* will result in the template reinforcement being adjusted to fit the new section dimensions.

Aggregate size – used when placing reinforcement in a concrete section

4.4.2 Reinforcement

Reinforcement can be defined in two ways – either as template reinforcement for selected standard section shapes or as individual bars or groups of bars. Template reinforcement cannot be prestressed.

General reinforcement

Definition type – The type is where either individual bars (single), lines, arcs, circles or perimeters of bars is specified.

Diameter and Bundle – the bar diameter must always be specified. When the reinforcement is a line either single bars or bundles of bars can be specified.

Number of bars/bundles – when the reinforcement is a line, arc or circle of bars the number of bars or bundles must be specified.

Rebar – the material to be used for the reinforcement.

First Bar, Last Bar, Point on Arc, Centre – the coordinates of a single bar, or the coordinates at the start and end of a line or arc. When an arc of bars is to be defined an intermediate point has to be defined through which the arc passes but this need not be the position of any bar. When a circle is to be defined the centre of the circle should be specified.

Prestress – can be applied to bars either as a force or as a strain. If force is selected the force per bar is specified. In all cases the prestress must not exceed the elastic limit of the material. A negative prestress value will give tension in the steel. When applied loads *Exclude* prestress anchorage and curvature forces, AdSec will calculate the additional axial force and bending moments induced by the prestress in the reinforcement. If the applied loads are set to *Include* then no additional loads will be created.

Add Perimeter Bars

This allows bars to be defined around the perimeter of the section, based on bar size and spacing.

Template

Template reinforcement is defined for either a column or beam section. Template reinforcement must always be defined using the wizard.

4.5 Loads

Loads are defined in two parts. Firstly there is the loading which can be either forces and moments or applied strains and curvatures and secondly there is the reference point – the position at which the loads act.

Unfactored loading can be defined in the Loading Table and combined and factored to ULS or SLS when defining the Analysis Cases.

See also

[Load Titles](#)

[Loading](#)

[Reference Point](#)

4.5.1 Load Titles

The Load Titles is a way to associate a name with a particular load case. It plays no part in the AdSec analysis.

4.5.2 Loading

Loads can be specified as either a force and moment or as a strain plane

Load case – used to group together different load actions. This is primarily of use for composite sections where different forces and/or strains can be applied to the individual component sections.

Load Type – loading can be in the form of:

- Section force – forces and moments
- Component strain – strain and curvature applied to a single component
- Concrete-only strain – strain and curvature applied only to the concrete
- Whole section strain – strain applied to the whole section

Note: prestress is defined in the Reinforcement Table.

Force and Moment – the loading applied to the section is a combination of axial force (N) and two moments (M_{yy} and M_{zz}).

Axial Strain and Curvature – for applied strains the strain is input in the form of a strain plane:

$$\varepsilon = \varepsilon_x + \kappa_y z' + \kappa_z y'$$

An applied strain plane is allowed for each load case. For each line the section strain and curvature about axes parallel to the user y- and z-axis are input. The strain axes' origin is at the reference point. The loading is generated along with applied loads at the start of the analysis.

For a *whole section* strain plane the program translates the section origin to the reference point then calculates the force and moment from the applied strain plane on the unstrained section using the chosen material properties. y' and z' are the translated coordinates.

The forces and moments calculated will be affected by load factors, material ultimate factors and serviceability creep factors. The data is treated the same as an applied load thereafter.

A *component strain* plane or concrete only strain plane is stored and added to the strain generated during analysis. This ensures that the difference in strain between zones of a composite section is modelled correctly. For this reason the program does not calculate an equivalent load from the applied strain plane for 'component section' strain planes.

When the section is compound the component and concrete only strains allow pre-loading or beams strains to be applied to a particular component of the compound section.

4.5.3 Reference Point

The reference point is the location in the section where the force and moment are assumed to act. It is also the axis origin for the strain plane definition (y,z).

Geometric centroid – is defined as the centre of the concrete outline alone. This is the default location.

User specified point – The reference point can be directly specified using (y,z) coordinates.

4.6 Analysis Cases

To perform a ULS or SLS analysis it is necessary to define an analysis case. This describes how the loading that has been specified is to be interpreted for analysis along with any analysis specific details.

To perform an SLS analysis it is necessary to define one or more SLS analysis cases.

See also

[ULS Analysis Cases](#)

SLS Analysis Cases

4.6.1 ULS Analysis Cases

To perform a ULS or SLS analysis it is necessary to define an analysis case

Name – used to identify the analysis case.

Description – a description of the loading. The description syntax is of the form

$$a_1Lc_1 + a_2Lc_2 + \dots$$

where a_i is the factor that applies to the load defined in load case c_i . E.g. $1.2L_1 + 1.4L_2$

Prestress factor – the prestress is applied to the reinforcement, defined for the section. For analysis this prestress may be factored as required.

Note: if no ULS analysis cases exist, the program will generate an unfactored analysis case equivalent to each load record.

4.6.2 SLS Analysis Cases

To perform an SLS analysis it is necessary to define one or more SLS analysis cases.

Name – used to identify the analysis case

Section – the section to which this case applies

Analysis type – types available depend on the design code selected and are one of:

- Long term – the Young's modulus and material curves are adjusted depending on the creep factor to model the behaviour of the section under sustained loading.
- Short term – the user specified Young's modulus and material curves are used to model the behaviour of the section under short term loads
- Intermediate term – applied to BS5400 related codes only. This uses a Young's modulus and material curve which is interpolated between the long and short term values depending on the ration of live load to dead load.
- Long+short term – uses the long term properties for a first analysis. The concrete creep is then calculated and stored as a creep strain plane. A second analysis using the short term properties looks at the section under the combined long term and additional short term loading.

Load description – a description of the loading. The description syntax is of the form

$$a_1Lc_1 + a_2Lc_2 + \dots$$

where a_i is the factor that applies to the load defined in load case c_i . For long + short load there is an additional short term load.

Prestress factor – the prestress is applied to the reinforcement, defined for the section. For analysis

this prestress may be factored as required.

Creep factors – This specifies the creep factors that modify the concrete properties so that for linear stress-strain curves

$$E_{long} = \frac{E_{short}}{(1 + \phi)}$$

where ϕ is the creep factor. The theory section describes how creep is applied to other concrete curves.

Part V

5 Dialogs and Wizards

Most of the data in AdSec can be edited in dialogs or wizards. If the item is simple a single page dialog is usually adequate but for more complex data where there are inter-dependencies a wizard is provided to lead the user through the various steps.

See also

[Section Wizard](#)

[Compound Section Definition](#)

[Reinforcement Wizard](#)

[Chart Analysis Dialogs](#)

[Miscellaneous Dialogs](#)

[Preferences](#)

5.1 General Section Wizard

The General Section Wizard provides a single means of entering new simple sections, editing existing sections and creating compound sections. The use of the wizard depends on the particular context in which it is invoked.

See also

[General Section Wizard : Titles](#)

[General Section Wizard : Design Option](#)

[General Section Wizard : Wrap/Cast Section](#)

[General Section Wizard : Definition](#)

[General Section Wizard : Reinforcement](#)

5.1.1 General Section Wizard : Titles

When setting up a new model AdSec expects job details – this is entered in this dialog. This data is for information only and has no affect on the rest of the model.

Job Number

This is a number used to identify a particular job.

Initials

The initials of the engineer responsible for the model.

Edit Date

This is supplied automatically and records when the model was last edited.

Job Title, Subtitle, Calc. Heading

These fields give a brief description of the job and the calculation particulars.

Notes

This gives the user a place to record any notes that should be kept with the model.

5.1.2 General Section Wizard : Design Option

The design option allows the specification information to be set up prior to defining the section geometry etc.

Design Code and Country

In the general specification the user chooses the [design code](#).

Bending Axes

The problem is defined as bi-axial (bending about the y and z axes) or uni-axial (bending about the y axis only). The uni-axial option is provided for cases where some external action on the section constrains it to bend about a single axis.

Slab/wall

A problem can be defined as slab or wall in which case the section is assumed to represent a strip and where the sides are part of a continuous material.

Units

The units can be modified at any time however it is convenient to be able to define a set of units to use when creating a new section.

5.1.3 General Section Wizard : Wrap/Cast Section

In many situations a compound section is constructed by wrapping or casting around (or inside) another section. The wrap/cast option is used to define a compound section in these circumstances.

Apply

The apply option is used to apply material to the face (top or bottom) or wrap a section. This option

is typically used to apply FRP to an existing section. The material thickness is specified and in the case of a material applied to a face this can either be over the full width or of a specified width. The section created is typically a rectangle (if applied to a face) or perimeter (if wrapping).

Create rectangle/circle

These options "cast" either a rectangle or circle around a section. While the outer shape is simple the section must contain a void shaped according to the inner section. No checks are made that the created section is larger enough to enclose the inner section.

Fill

This option is used to fill the void in a section.

5.1.4 General Section Wizard : Definition

A section is defined by a material and a shape and these can be interdependent. The graphic shows the shape of the section as it is defined give visual checking of the geometry.

Name

The name is simply a convenient label by which to refer to the section.

Material Type and Grade

The material is defined by a type which is one of:

- concrete
- steel
- FRP

and a material grade, which defined the strength, stress-strain relationship etc. The material grades depend on the type of material selected

Section

The section defines the geometrical properties of the general section. The sections can be entered directly or can be selected from the section wizard. The valid section shapes will depend on the material type chosen.

5.1.5 General Section Wizard : Reinforcement

If the section is defined with a concrete material types then it is possible to add reinforcement to the section. The graphic shows the bars added to the section and the percentage reinforcement is reported.

General Reinforcement

General reinforcement allows complete flexibility in position of reinforcement. The Add, Modify, Shift and Delete options all allow the reinforcement to be specified and adjusted.

Template Reinforcement

Template reinforcement is placed in the section based on rules for typical placement of bars for different section shapes and for beam or column arrangements. The template option is not available for all section shapes.

5.2 Section Wizard

The section wizard is where sections can be defined. The wizard takes the user through the following pages. Note that not all section options are available for all materials

See also

[Section Wizard : Section type](#)

[Section Wizard : Catalogue Section](#)

[Section Wizard : Standard shapes](#)

[Section Wizard : Perimeter section definition](#)

[Section Wizard : Section definition](#)

[General Section Wizard](#)

5.2.1 Section Wizard : Section Type

The section type is where the basic method of selection is defined.

Name

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

Material

The material will list all the material defined, and fill in with the material number where there are gaps in the numbering. If the required material has not yet been defined the number of that material can be entered directly.

Definition method

The definition method specifies how the material is to be defined. This breaks down into three main options.

- catalogue sections — selected from online section catalogues
- standard sections — defined by a shape and its dimensions
- geometric sections — defined by a perimeter

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

5.2.2 Section Wizard : Catalogue Section

The catalogue sections are stored in an database, installed with AdSec.

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.

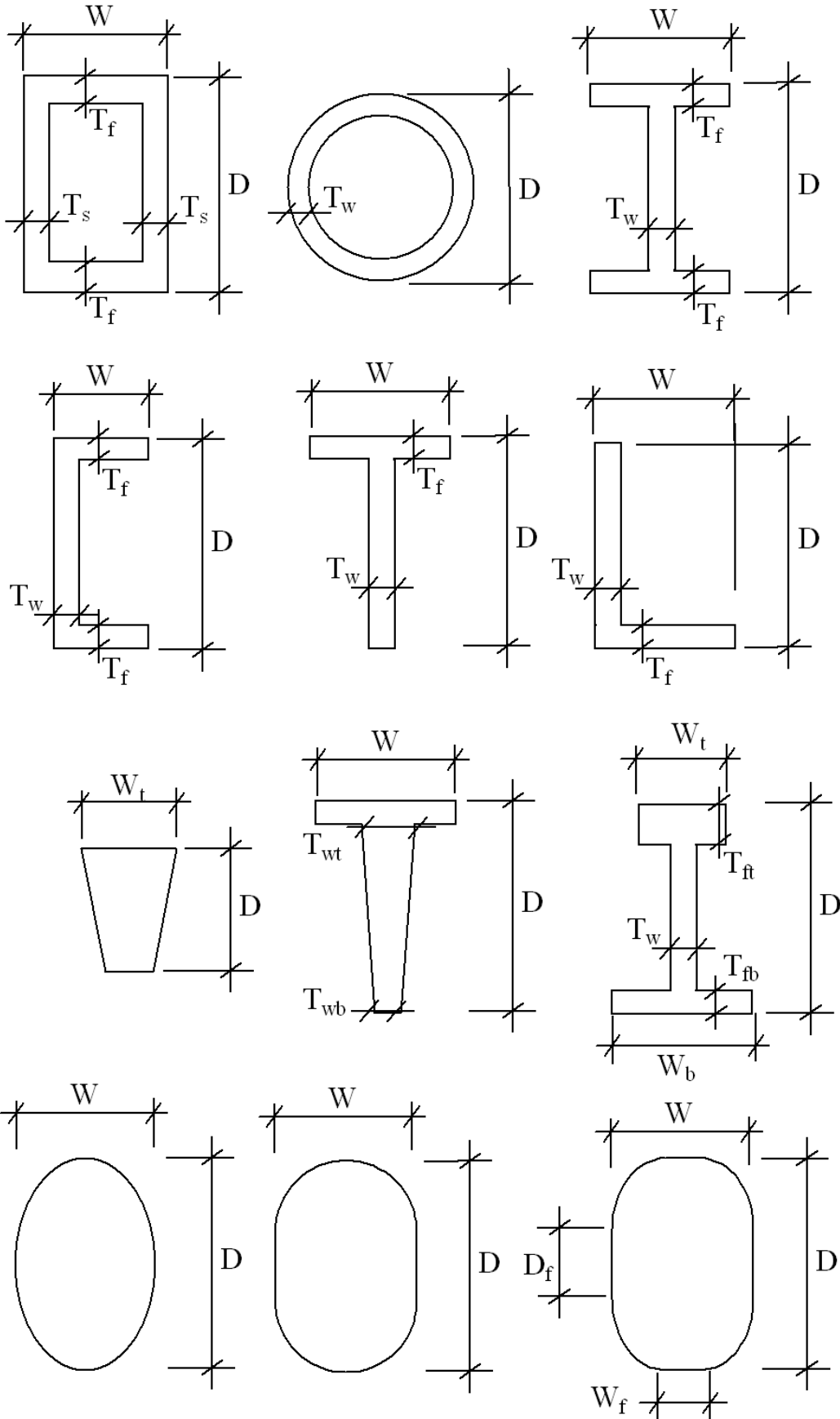
Include superseded sections

Superseded sections are to be displayed. If the model contains superseded sections this will be checked on entry.

5.2.3 Section Wizard : Standard Shapes

Section dimensions

Standard sections can be rectangular, circular, I, channel, T or angles. Rectangular and circular sections can be either solid or hollow. Specify the overall external dimensions of the section and the thickness of component parts as shown below. Note that top and bottom flanges are always the same thickness, as are left and right walls of a rectangular hollow section.



Ellipse index (n)

This field is only available for the ellipse section type. It allows superellipses to be entered. A value of 1 corresponds to a diamond, 2 to an ellipse, and ∞ to a rectangle.

5.2.4 Section Wizard : Perimeter Section Definition

The outline of a perimeter section is defined by a series of coordinates describing a polyline. The polyline is automatically closed to form a polygon so an end point coincident with the start point need not be entered. Polyline segments may not intersect.

In addition, any number of voids may be defined in the section, again by a series of coordinates describing an unclosed polyline of non-intersecting segments. Voids may not intersect with each other or with the outline.

The section displayed in the wizard is as viewed from end 1 of the element towards end 2.

The centroid is calculated for the section and the section is assumed to lie centred at its centroid, — not at the datum coordinates.

The section can be imported or exported from a DXF file. For the import to work the DXF file should contain only LWPOLYLINE or POLYLINE entities that described the perimeter and void in the section. The export option allows the section to be exported as a series of LWPOLYLINE entities.

The bridge beam option is only enabled if the bridge beam database is available. This gives access to standard bridge beam sections.

5.2.5 Section Wizard : Section Definition

This page summarizes the section definition.

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.

Properties

This shows the values of the elastic section properties for the concrete outline for reference.

More – displays a more comprehensive list of (unmodified) section property values.

5.3 Section Material

This allows for quick modification of the material grade.

For EC2 sections this also allows the minimum value of ζ to be modified. ζ is used in EC2 equation 7.18 to simulate cracking from a previous load event where a value of zero mean no previous cracking.

5.4 Compound Section Definition

The Compound Definition dialog is where the component sections are assembled into a compound section.

Compound Section Name

The name is used to identify the section.

Component Section Table

This defines the sections that compose the compound section. Up to four component sections can be defined referenced as A, B, C, ...

Component sections B, C, ... are offset relative to A.

The reference point is defined with reference to the current section. In the case of a compound section the "geometric centroid" used is the geometric centroid of the primary section (A).

5.5 Section Builder

The section builder is a way of quickly placing one section relative to another. A reference section and section to be added are selected. The user then selected the positioning of the additional section relative to the original section from the positions offered.

A compound section with the appropriate offsets is then created.

5.6 Reinforcement Wizard

The Template Reinforcement Wizard is where the reinforcement can be specified for selected standard shapes. The patterns of reinforcement depend on the particular shape but break down into either beam or column reinforcement patterns. General Reinforcement allows bars to be located by coordinates inside the section. General and template reinforcement can both be used in the same section.

See also

[Reinforcement Wizard : Template Definition](#)

[Reinforcement Wizard : Links and Cover](#)

[Reinforcement Wizard : Main Reinforcement](#)

[Reinforcement Wizard : Side Reinforcement](#)

[Reinforcement Wizard : Rectangular Column Reinforcement](#)

[Reinforcement Wizard : Circular/Elliptical Reinforcement](#)

[General Reinforcement Definition](#)

5.6.1 Reinforcement Wizard : Template Definition

The basic choice when defining reinforcement using a template is between beam and column arrangements.

The beam option allows reinforcement to be specified in terms of top, bottom and side bars.

For the column option the specification of the reinforcement depend on the shape of the section.

For rectangular sections a standard bar pattern is chosen and bar sizes and covers specified. For circular sections a pattern of rings is assumed and the user gives details of the bars in the rings.

5.6.2 Reinforcement Wizard : Links and Cover

Links and covers are specified so that the main steel can be positioned. They are not used in the AdSec calculations.

Cover to Links

Cover to the links can be uniform – the same on all sides or variable in which case covers are specified for top/bottom and left/right sides of the section.

Links

The diameter of the bars and the type of steel is specified. The steel can refer to a standard or user defined steel material.

5.6.3 Reinforcement Wizard : Main Reinforcement

Top

This specifies the reinforcement at the top of the section.

This specifies the number of bars in each row, the number of rows of bars and the distance between the rows. Either single bars or pairs of bars can be placed. Where pairs of bars are used the spacing of the bars must also be specified. When the section can be classified as having a rib and flange the option is offered to place the bars within the rib width or across the flange width. The diameter of the bars and the type of steel is also specified. The steel can refer to a standard or user defined steel material.

Bottom

This defines the reinforcement at the bottom of the section. The data to define this is the same as for the top reinforcement.

5.6.4 Reinforcement Wizard : Side Reinforcement

Side Bar Arrangement

This specifies the number or pitch of the side bars. The bars can then be located relative to the top or bottom main steel. Where a section is tapered there is the option of sloping rather than vertical placing of side bars.

Side Bars

The diameter of the bars and the type of steel is specified. The steel can refer to a standard or user defined steel material.

5.6.5 Reinforcement Wizard : Rectangular Column Reinforcement

Column Bar Arrangement

The reinforcement is defined in terms of a number of bars and a layout inside the section. Where the arrangement has more than one layer of bars the distance between the bars must also be specified.

Column Bars

The bars can all be assigned the same diameter or diameters can be specified for each of the reinforcement regions. If the variable diameter option is chosen the corner, top, bottom left and right bar diameters are specified individually.

The diameter of the bars and the type of steel is specified. The steel can refer to a standard or user defined steel material.

5.6.6 Reinforcement Wizard : Circular/Elliptical Reinforcement

Circular/Elliptical Arrangement

The reinforcement is defined in terms of a number of bars per ring of reinforcement and the number of rings. Where the arrangement has more than one ring of bars the distance between the rings must also be specified.

Bars

The diameter of the bars and the type of steel is specified. The steel can refer to a standard or user defined steel material.

5.6.7 General Reinforcement Definition

The general definition dialog allows single bars or lines or arc of bars to be defined.

Type

The type is where either individual bars, lines of bars or arcs of bars is specified.

First Bar, Intermediate Point, Last Bar

The coordinates of a single bar, or the coordinates at the start and end of a line or arc. When an arc of bars is to be defined an intermediate point has to be defined through which the arc passes but this need not be the position of any bar.

No of Bars and Bar Diameter or Area

When the reinforcement is for a line or arc of bars the number of bars must be specified. In all cases either the bar diameter or area must be specified. Where an area is specified this is converted into an equivalent diameter.

Rebar

The rebar material to be used for the reinforcement This can be either a standard rebar material or a user defined rebar material.

Prestress

Prestress can be applied to bars either as a force or as a strain. If force is selected the force per bar is specified. In all cases the prestress must not exceed the elastic limit of the material.

A negative prestress will result in tension in the steel.

5.7 Analysis

For both ULS and SLS analysis the analysis cases must be selected. Only the cases relevant to the particular analysis type and section will be offered.

In the case of a ULS analysis there is a choice of Strength or Loads analysis. A ULS Strength analysis determines the moment capacity of the section under the given axial load and moment angle. A ULS Loads or SLS analysis analyses the section under the specified loading – the main difference being in the stress-strain curves used in the analysis.

The [Analysis Control](#) option gives scope to adjust the parameters which control the analysis

For certain codes further parameters may need to be adjusted. These are detailed below:

BS5400 and related codes: Nominal cover, Crack width equation, M_q/M_g ratio

For BS5400 analysis some extra parameters are required. The crack width equation to be used must be specified: this can be either BS5400: Equation 24 or Equation 26. The M_q/M_g ratio is used in the crack width formula and to calculate the properties for a intermediate term analysis. The nominal cover is used to generate a perimeter for crack width calculations.

Eurocode and related codes: Distance from bar for crack width calc., Crack width equation, Duration factor

For Eurocode analysis some extra parameters are required. For UK variants the crack with can be reduced from the surface value based on a distance from the bar. The crack width equation to be used must be specified: this can be either EN1992 Equation 7.9 or Equation 7.18, of if PD6687 is used it will be PD6687:2006 2.17. codes the calculations BS5400 the nominal cover is used to

generate a perimeter for crack width calculations. As well as a creep factor there is a duration factor in Eurocode: this can be either Normal or Instantaneous

CSA S6: Reinforcement factors, Crack calculation

For CSA S6 analysis some extra parameters are required. The reinforcement factors k_b depending on the type of bar and ϵ_c depending on load controlled or deformation controlled. The crack calculation can follow the code directly or use a calculation based on a local interpretation of the code.

5.8 Advanced Analysis Control

The Advanced Analysis Control allows the details of the solution to be modified.

Iteration Control

As the AdSec solution is iterative it can be difficult at times to converge on a solution. The maximum number of iterations will allow the solution to terminate if the solution is not converging. Applying limits to each step may help the solution to converge where the stress-strain curves have significant slope discontinuities which can have an adverse effect on quasi-Newton solution methods. The perturbation factor allows a small perturbation to be introduced into the solution which can prevent the solution from getting "stuck".

Result Control

The default is to report strain and stress at the section nodes (vertices). It can be convenient to get the stresses and strains in more details, so a stress/strain perimeter interval can be specified and stress and strain values will be calculated and reported at these points. For some codes cracks are calculated around the perimeter. The distance between crack calculations should be specified.

Interrupt Solution

This option allows the solution to be monitored (and adjusted) during the solution. This should only be used in extreme cases.

5.9 Chart Analysis Dialogs

AdSec offers various charting options for both ULS and SLS cases.

See also

[N/M Chart Analysis](#)

[Myy/Mzz Chart Analysis](#)

[M/Curvature and M/EI Chart](#)

5.9.1 N/M Chart

The N/M Chart Analysis dialog determines the parameters for which the axial load / moment interaction chart will be generated.

Name

The name is used as an identifier on the chart.

Moment Angle

The section is analysed for an applied axial load combined with a moment at a particular moment angle, typically 0° for bending of a beam about its major axis.

Included strains

Applied creep and shrinkage strains can be included in the generation of the N/M chart by specifying the load case which describes them.

Analysis control

The analysis control option allows control of items such as the maximum number of iterations, the solution perturbation factor and the option to interrupt the solution.

5.9.2 Myy/Mzz Chart

The M_{yy}/M_{zz} Chart Analysis dialog determines the parameters for which the moment interaction chart will be generated.

Name

The name is used as an identifier on the chart.

Points per curve

This specifies the number of points on each M_{yy}/M_{zz} curve.

Axial Loads

A number of curves can be generated on a single graph for different values of axial load. These can be added singly or for a range of axial load values using the from, to step options.

Included strains

Applied creep and shrinkage strains can be included in the generation of the M_{yy}/M_{zz} chart by specifying the load case which describes them.

Analysis control

The analysis control option allows control of items such as the maximum number of iterations, the solution perturbation factor and the option to interrupt the solution.

5.9.3 M/Curvature and M/EI Chart

The moment-curvature (M/κ) and moment-stiffness (M/EI) chart analysis dialog determines the parameters for which the moment curvature and moment stiffness charts will be generated. The stiffness is the secant stiffness.

Name

The name is used as an identifier on the chart.

Points per curve

This specifies the number of points on each M/Curvature curve.

Axial Loads

A number of curves can be generated on a single graph for different values of axial load. These can be added singly or for a range of axial load values using the from, to step options.

Included strains

Applied creep and shrinkage strains can be included in the generation of the M/Curvature chart by specifying the load case which describes them.

Analysis control

The analysis control option allows control of items such as the maximum number of iterations, the solution perturbation factor and the option to interrupt the solution.

5.10 Miscellaneous Dialogs

Various dialogs are used in AdSec to control or manipulate AdSec data. In many cases these dialogs are self explanatory but details for other are given below.

See also

[Stress-strain Chart](#)

[Expand Section](#)

[Label and Display Options](#)

[Chart Style](#)

[Additional Points on N/M and Myy/Mzz Charts](#)

5.10.1 Stress-strain Chart

The stress-strain chart gives a quick way to see what the stress-strain curves look like for any of the materials specified. Choose the material and the grade and the ULS and SLS stress strain curves are displayed. Two SLS curves are displayed for concrete – one for short term behaviour and the other for long term behaviour. The creep factor can be adjusted to show how the stress-strain curve is adjusted to account for creep. The chart can be modified to show only ULS and/or SLS curves and Compression and/or tension.

5.10.2 Expand Section

This allows sections to be expanded. There are three aspects to expanding the section

- Converting the section to a perimeter
- Expanding template reinforcement
- Expanding line, arcs and circles of reinforcement

Converting a section to a perimeter loses the section shape so any template reinforcement must also be expanded into general reinforcement with points, lines, arc and circles. The option to expand lines, arc and circles is provided mainly to allow bar conflicts to be resolved through the "Where bars overlap" option.

The template reinforcement can be expanded into general reinforcement without converting the section to a perimeter but not vice versa.

5.10.3 Label and Display Options

This allows control over what information is displayed in the graphics including labeling and colour options.

Display Options

The display options allow the sections and bars to be coloured in various different ways. The sections can be coloured:

- Solid areas – where the solid areas are shaded
- By Section – where different component sections are shaded in different colours
- By Material – where different material types are shaded in different colours
- None – where the sections are drawn in outline only

The bars can be coloured:

- By Type/Prestress – where the bars are drawn in different colours if general or template reinforcement and if prestressed or not
- None – where the bars are drawn in black

Labels

The labels allows different attributes of the section to be annotated.

Print Scaling

The print scaling allows the printed output of the section to be scaled to a particular scale rather than autoscaled. This value is not saved from one session to the next.

5.10.4 Chart Style

The chart style dialog allows curve and graph settings to be adjusted allowing control over the display of graphs.

Curve Settings

The curve settings allows control over the display of graphs. Individual curves can be selected and the colours, line styles and symbols adjusted.

Graph Settings

This related to the 'Label and Display Options' but gives some control over the layout — legend, graph style (border, grid lines and axes) and notes.

5.10.5 Additional Points on N/M and M_{yy}/M_{zz} Charts

The N/M and M_{yy}/M_{zz} charts display a capacity envelope for the section. It can be useful to check particular stress conditions (forces and moments) to ensure that they lie within the envelope.

The additional point option allows the user to superimpose force/moment (N/M) or moment (M_{yy}/M_{zz}) coordinates on the charts. The label option is used to identify particular points.

5.10.6 Output Specification

The output specification allows control of the tabulated output. The specification items are divided into various sections only some of which will be active. These are

- Input Data – all the data modules
- Analysis Results – results from the analysis
- General – results applicable to all analysis options
- ULS Loads Analysis – extra results for ULS loads analysis
- SLS Load Analysis – extra results for SLS analysis
- Chart Results – chart coordinates
- N/M Chart – force/moment interaction results
- Myy/Mzz Chart – moment interaction results
- M/Kappa Chart – moment curvature and moment stiffness results

The Preferred and Reset options set the results selected to those set in the Preferences or those on entry to the Output Specification respectively.

See also

[Preferences](#)

5.11 Preferences

Preferences are settings that are stored for the user. These do not affect the data, only the way it is presented.

Miscellaneous

This controls miscellaneous preferences associated with AdSec.

Startup – controls check and dialogs on starting the program

Table – choosing how the selection fields are display in AdSec

Graphics – adjusting of aspects of the graphics – useful in adjusting for a particular graphics card.

Code Options

Allows setting of a default design code, and default units

Advanced

Solver – allows default values to be set for solver control parameters

Data – allows long + short analysis to be switched off (for codes that permit it)

Compound sections – sets an upper limit on the number of components in a section

some of the less frequently used options to be hidden from the user.

Output

Numeric format – which control the way numbers are output. The options are

- engineering – numbers expressed in exponent form where the exponent is a multiple of 3.
- decimal – numbers expressed to a fixed number of decimal places.
- scientific – numbers expressed in exponent form.

Crack width output – option is to control the output of crack width details.

Page Setup – default setup of pages for text and graphic output

Company Info – how the company name appears on output

Language – the language used for output

Eurocode 2

Loading coefficient – the loading coefficient use to determine the distribution coefficient

Crack calculation – option to calculate crack widths based on surface or bar strains

BS5400

Cover options – maximum cover zone

AS3600

Strength reduction factor – basic for strength reduction calculation

Part VI

6 Output Options

Output from AdSec can be presented both graphically and in tabular form.

More

[Graphical Output](#)

[Tabular Output](#)

6.1 Graphical Output

The graphical output falls into two categories:

- a graphical representation of the section showing the location of bars and some basic results.
- a graphical plot of moment/force, moment/moment or stress/strain relationships

Section View

The section view displays the current section – both concrete and reinforcing bars. Post analysis the view displays

- the reference point
- the neutral axis, indicating the compression and tension sides
- crack widths and position of maximum crack width and the bar controlling the maximum crack width
- labelling of bar, concrete point and crack numbers

Graph Views

Graph views are used where these provide the most convenient way of displaying data or results.

Graphs of the stress-stress curves for both concrete and reinforcement are available.

For N/M, M_y/M_{zz} , M/curvature and M/EI chart analysis the results are most easily understood as diagrams showing the interaction of these effects. The charts also allow for easier identification of additional information such as the no tension point, balanced yield point, tension and compression plateau points on N/M charts.

6.2 Tabular Output

Input Data

All input data is available as tabular output. In addition to the basic input data expanded section properties and bar locations are available.

Results

The following results are available depending on the Analysis option:

	ULS Capacity Analysis	ULS Loads Analysis	SLS Loads Analysis	N/M Chart	M_y/M_{zz} Chart	M/Kappa Chart
Input Data	✓	✓	✓	✓	✓	✓
Cases Analysed	✓	✓	✓			
Analysis Summary	✓	✓	✓			
Total Loads	✓	✓	✓			
Strain Planes	✓	✓	✓			
Material Strains/ Stresses	✓	✓	✓			
Bar Strains/Stresses	✓	✓	✓			
Moment Summary		✓	✓			
Moment Summary for SLS Axial Loads			✓			
Crack Width Summary			✓			
Crack Width Details			✓			
N/M Chart Coordinates				✓		
M _y /M _{zz} Chart coordinates					✓	
M/Kappa Chart coordinates						✓

Part VII

7 Command File

The command file allows access to certain (basic) AdSec operations by specifying these in a text file called a command file, identified by a ".adcmd" or ".cmd" extension. The only difference in how these command file types are handled is how the program displays the splash screen. If running a ".cmd" command file, the splash screen will show when AdSec is running whereas it will be hidden when running a ".adcmd" command file.

The command file can then be run from the Windows "Run" command by typing

```
adsec <filename>
```

On machines that have access to several versions of AdSec it is necessary to be explicit in specifying the version of AdSec that is to be run. For example:

```
"c:\program files\oasys\adsec 8.4\adsec" c:\data\test.adcmd
```

The double quotes are necessary when the path includes spaces.

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.ads")

7.1 Command File Commands

The command file commands are

open(filename)

Open an AdSec file (binary or text)

filename – the file to be opened including the file path and extension

analyse(option,cases)

Carry out analysis for the specified option - if no option and cases are specified the current analysis task is assumed

option – analysis option

uls – ULS strength analysis

uls_load – ULS loads analysis

sls – SLS loads analysis

cases – the analysis analysis to analyse

analyse_nm(name,angle,strain,phi)

Carry out an force moment (N/M or P/M) chart analysis

name – name for chart

angle – moment angle

strain – include creep and shrinkage (concrete-only) strains for load case (0 for none)

phi – strength reduction factor (for ACI and AS codes) set to 0 for code specified factors and 1 for nominal strength

analyse_mymz(name,points,loads,strain,phi)

Carry out a moment interaction (My/Mz) chart analysis

name – name for chart

point – number of chart points

loads – the axial loads to be included in the analysis

strain – include creep and shrinkage (concrete-only) strains for load case (0 for none)

phi – strength reduction factor (for ACI and AS codes) set to 0 for code specified factors and 1 for nominal strength

analyse_mkappa(name,points,loads,angle,max,term,creep)

Carry out a moment curvature (M/kappa) chart analysis

name – name for chart

point – number of chart points

loads – the axial loads to be included in the analysis

angle – moment angle

max – the maximum moment for the chart

term – the load term

long – long term

short – short term

creep – creep factors for load term case (0 for non-concrete sections)

analyse_mei(name,points,loads,angle,max,term,creep)

Carry out a moment secant stiffness (M/EI) chart analysis

name – name for chart

point – number of chart points

loads – the axial loads to be included in the analysis

angle – moment angle

max – the maximum moment for the chart

term – the load term

long – long term

short – short term

creep – creep factors for load term case (0 for non-concrete sections)

param(iter,limit,perturb)

Set analysis parameters

iter – number of iterations

limit – limit the change in a single time step ('limit' or 'no_limit')

perturb – perturbation factor [0:1]

save()

Save the AdSec file

saveas(filename)

Save the AdSec file with the specified name

filename – the file to be saved including the file path and extension

export_text(filename,options)

Export data and/or results

filename – the file for the export including the file path and extension

txt	– text file (tab delimited)
csv	– text file (comma delimited)
html	– html file
rtf	– rich text file

options – the required output

spec	– specification data	
gen_spec	– general specification	[spec]
history	– file history	[spec]
task	– analysis task	[spec]
section	– section data	
defn	– section definition	[section]
nodes	– section nodes	[section]
cover	– section cover	[section]
sec_prop	– section properties	[section]
mat_prop	– section material properties	[section]
rebar_prop	– section rebar material properties	[section]
loads	– load data	
ref_pt	– reference point	[loads]
forces	– section forces and strains	[loads]
general	– general results	
cases	– analysis cases	[general]

anal_sum	– analysis summary	[general]
total_load	– total applied load	[general]
strain_planes	– strain planes	[general]
mat_strains	– material stress/strain	[general]
rebar_strains	– rebar stress/strain	[general]
uls	– ULS results	
uls_sum	– ULS summary	[uls]
sls	– SLS results	
sls_sum	– SLS summary	[sls]
crack_sum	– SLS crack summary	[sls]
cracks	– SLS crack widths	[sls]
crack_face	– SLS crack widths per face	[sls]
crack_max	– SLS crack widths (max)	[sls]
nm	– N/M chart points	
mymz	– My/Mz chart points	
mkappa	– M/ κ chart points	

export_graphics(filename,option)

Export an SVG graphics file

filename – the file for the export including the file path and extension

svg – scalable vector graphics file

options – the required output

– section view

nm – N/M (P/M) chart

mymz – My/Mz chart

mkappa – M/ κ chart

close()

Close the file

exit()

Exit AdSec

Part VIII

8 AdSec Text File

Details of the keywords are available in the AdFile.html file in the AdSec Docs folder.

Index

- A -

About 8
 ACI 9
 Additional points
 Charts 59
 AdSec 8
 Advanced preferences 60
 Analysis 26, 29, 40, 41, 54, 55, 57
 Cases 16
 Control 55
 Intermediate term 40, 41
 Long and short term 40, 41
 Long term 40, 41
 M/curvature chart 57
 M/Curvature Charts 29
 M/EI chart 57
 M/EI Charts 29
 Myy/Mzz chart 26, 56
 N/M chart 26, 56
 Serviceability Limit State 29, 40, 41
 Short term 40, 41
 Staged analysis 31
 Stress interval 55
 Ultimate Limit State 26, 40, 41
 Analysis cases 16, 19, 40, 41
 Analysis Dialogs 55
 AS 9
 Axial strain 39

- B -

Beam reinforcement 52
 Bend Radius 35
 Bending Axes 34, 45
 Biaxial 34, 45
 Bottom reinforcement 52
 BS 9
 BS5400 8
 Crack width equation 40, 41
 Mq/Mg Ratio 40, 41
 Nominal Cover 40, 41

 Specification 34
 BS8110 8, 34
 Hong Kong 34
 Specification 34
 BS8666 35
 Build
 Compound Section 51

- C -

Cases 16
 Analysis 16, 19
 Load 16
 Cast
 General Section Wizard 45
 Chart
 Stress-strain curve 58
 Charts 26, 29
 Additional points 59
 M/Curvature 29
 M/EI 29
 Myy/Mzz 26
 N/M 26
 Style 59
 Code
 Design 9
 Colour 58
 By material 58
 By section 58
 By type/prestress 58
 Column Reinforcement 52
 Command File 66
 Commands 66
 Component
 Strain 39
 Compound
 Section 51
 Compound Section 15, 19, 37
 Definition 51
 Concrete 35
 Limiting strain 35
 Concrete Only Strain 39
 Concrete Properties 35
 Control
 Analysis 55
 Convert to perimeter 58
 Country 34, 45

Cover
 Minimum 34
 Creating and editing sections 19
 Creep 39, 40, 41
 CSA 9
 Current section 19
 Curvature 39
 Chart specification 57

- D -

Definition
 General reinforcement 53
 General Section Wizard 46
 Section 38
 Design Codes
 Concrete 9
 Section 45
 Specification 34
 Dialogs 44
 Dimensions 38
 Display
 Options 58

- E -

EC2 8
 Specification 34
 Eurocode 9
 Expand section 58
 Expanding line arcs and circles 58

- F -

Features 8
 Fibre Reinforced Polymer 37
 Fill 45
 Find 25
 Find replace go to and modify in tables 25
 FRP 37
 Limiting strain 37
 FRP Material Properties 37

- G -

Gateway 21

General reinforcement
 General Section Wizard 46
 General Reinforcement Definition 53
 General Section Wizard 44
 Definition 46
 Design Option 45
 Reinforcement 46
 Titles 44
 Wrap/Cast 45
 General Specification 34
 Go to 25
 Graphical Output 63

- H -

Hong Kong 9

- I -

Input data 63
 Interaction Charts 26, 29
 Interactive Solution 57
 M/curvature chart 57
 M/EI chart 57
 Myy/Mzz chart 56
 N/M chart 56
 Intermediate term analysis 40, 41
 IRC 9
 IRS 9
 IS 9

- J -

Job Number 44

- L -

Label and Display Options 58
 Limiting strain
 Concrete 35
 FRP 37
 Rebar 35
 Steel 36
 Load 39
 Cases 16
 Loading 39
 Reference point 40

Long term analysis 40, 41
 Long term load factor 40, 41

- M -

M/curvature Chart Analysis 57
 M/Curvature Charts 29
 M/EI Chart Analysis 57
 M/EI Charts 29
 Material
 Concrete 35
 FRP 37
 Rebar 35
 Section 51
 Steel 36
 Stress-strain curve 58
 Material Properties 34
 Miscellaneous Dialogs 57
 Miscellaneous preferences 60
 Modify 25
 Moment Angle 56
 Multi-page tables 24
 Myy/Mzz Chart Analysis 56

- N -

N/M Chart Analysis 56
 Nominal Cover 40, 41

- O -

Output
 Graphical 63
 Tabular 63
 Output Options 63
 Output Specification 60

- P -

Partial safety factors
 Concrete 35
 FRP 37
 Rebar 35
 Steel 36
 Preferences 60
 Prestress 40, 41, 53

Program Data 33
 Properties
 Concrete 35
 FRP 37
 Rebar 35
 Steel 36

- R -

Rebar 35
 Limiting strain 35
 Rebar Material Properties 35
 Reference Point 40
 Reinforcement 19, 38
 Beam and Column 52
 General 53
 General Section Wizard 46
 Reinforcement Wizard 51
 Circular/Elliptical Reinforcement 53
 Links and Cover 52
 Main Reinforcement 52
 Rectangular Column Reinforcement 53
 Side Reinforcement 53
 Template definition 52
 Replace in tables 25
 Results 63

- S -

Secant Stiffness 29
 Section 14, 15, 19, 37, 38
 Builder 51
 Catalogue 48
 Compound 15, 19, 37, 51
 Creating 14, 15
 Creating and editing 19
 Current 19
 Definition 50
 Dimensions 38
 Expand 58
 General 37
 General Section Wizard 46
 Material 38, 51
 Perimeter 50
 Simple 14
 Standard shapes 48
 Type 47

Section 14, 15, 19, 37, 38
 Wizard 47
 Section Definition 38
 Section view 22
 Section Wizard 47
 Catalogue Section 48
 General 44
 Perimeter section definition 50
 Section definition 50
 Section type 47
 Standard shapes 48
 Sections 47
 Serviceability Limit State 29
 Settings 59
 Curve 59
 Graph 59
 Short term analysis 40, 41
 Shrinkage 39
 Side reinforcement 53
 Sign convention 21
 Simple Section 14
 Single and multi-page tables 24
 Single page tables 24
 Slab/wall 34
 SLS
 Analysis 54
 Analysis cases 40, 41
 Cases 19
 Specification 34, 57
 General 34
 M/curvature chart 57
 M/EI chart 57
 Myy/Mzz chart 56
 N/M chart 56
 Output 60, 63
 Units 34
 Staged analysis 31
 Steel 36
 Limiting strain 36
 Steel Material Properties 36
 Step by Step 14
 Stiffness 29
 Chart specification 57
 Strain 39
 Concrete only 39
 Whole Section 39
 Strain Discontinuities 30

Strain plane 39
 Stress-strain chart 58
 Surface tolerance 34

- T -

Table 23, 24
 Adjusting display 24
 Data editing 24
 Table view 23
 Tables 25
 Find replace and go to and modify in 25
 Single and multi-page 24
 Tabular Output 60, 63
 Analysis results 60
 Chart results 60
 Input data 60
 Template 52
 Circular column 53
 Cover 52
 Expand 58
 Links 52
 Main reinforcement 52
 Rectangular column 53
 Reinforcement 52
 Side reinforcement 53
 Titles 33
 General Section Wizard 44
 Tolerance 34
 Top reinforcement 52

- U -

ULS
 Analysis 54
 Analysis cases 40, 41
 Capacity 40, 41
 Cases 19
 Loads 40, 41
 Ultimate Limit State 26
 Uniaxial 34, 45
 Units 34
 Using AdSec 19

- V -

View 63

View 63
Graph 63
Section 63

- W -

Welcome to AdSec 14
Whole Section Strain 39
Wizards 44, 47
Working with 19, 23
 Gateway 21
 Section view 22
 Using AdSec 19
Wrap/Cast
 General Section Wizard 45

- Z -

Zeta min 51

Endnotes 2... (after index)

Back Cover