AdSec

Version Version 8.2

Oasys

Oasys Ltd

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.comcom
Website: http://www.oasys-software.com/

Oasys AdSec

Printed: July 2010

Table of Contents

Foreword

This is just another title page
placed between table of contents
and topics

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.

Tel: +44 (0)20 8996 9001

email:cservices@bsi-global.com

1.1 Features

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

- **ACI 318-02** (Building Code Requirements for Structural Concrete (ACI 318-02))
- **ACI 318M-02** (Building Code Requirements for Structural Concrete (ACI 318M-02))
- **ACI 318-05** (Building Code Requirements for Structural Concrete (ACI 318-05))
- **ACI 318M-05** (Building Code Requirements for Structural Concrete (ACI 318M-05))
- **ACI 318-08** (Building Code Requirements for Structural Concrete (ACI 318-08))
- **ACI 318M-08** (Building Code Requirements for Structural Concrete (ACI 318M-08))
- **AS 3600—2001** (Australian Standard - Concrete Structures)
- **AS 3600—2009** (Australian Standard - Concrete Structures)
- **BS8110:1985** (BS8110-1:1985, BS8110-2:1985) (now withdrawn in the UK)
- **BS8110:1997** (BS8110-1:1997 incorporating amendments 1and 2, BS8110-2:1985)
- **BS8110:1997 (2005)** (BS8110-1:1997 incorporating amendments 1, 2 and 3, BS8110-2:1985)
- **EN 1992-1-1:2004** Eurocode 2
- **BS EN 1992-1-1:2004 / PD 6687:2006** Eurocode 2 (UK) with background document PD 6687
- **Hong Kong Code of Practice 1987** (Code of Practice for Structural Use of Concrete)

- **Hong Kong Code of Practice 2004** (Code of Practice for Structural Use of Concrete 2004)
- **IS 456 : 2000** (Indian Standard - Plain and Reinforced Concrete - Code of Practice)

and for bridges:

- **BS5400** (BS5400-4:1990)
- **HK Structures Design Manual - 2002** (Structures Design Manual for Highways and Railways 1997, Amended 2002)

Analysis

For the Ultimate Limit State (ULS) or strength analysis the options available are:

- The ultimate moment capacity/strength of the section
- Stresses from the ultimate applied load
- Ultimate resistance charts
- Ultimate resistance yy-zz moment interaction chart (for biaxial bending only)

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 FRP sections of either standard or user defined shape. Steel sections can be selected from UK, European, American and Australian steel catalogues. Reinforcement can be defined for concrete sections.

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

Reinforcement can be pre-stressed 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 Abbreviations

ACI – American Concrete Institute

AS – Australian Standard

BS – British Standard

BSI – British Standards Institution

CEN – European Committee for Standardization

CFRP – carbon fibre reinforced polymer

EC2 – Eurocode 2

EN – European Standard

FRP – fibre reinforced polymer

GFRP – glass fibre reinforced polymer

HK – Hong Kong

IS – Indian Standard

N/A – neutral axis

Step by Step Guide

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.

- Creating a Simple Section
- Creating a Compound Section
- Load Cases and Analysis Cases

The details of the working with the program and the program data are give in separate chapters.

2.1 Welcome to AdSec

When launching AdSec the "Welcome to AdSec" dialog is displayed. The "Did you know..." section offers various AdSec tips.

The options offered are:

Create a new section

This will open the General Section Wizard guiding the user through the definition of the titles, specification and section.

Work on your own

Expert users this 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.

2.2 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.

When creating a new section job titles and details are entered here and then the design code is chosen. In most cases the design code by itself determines all the required parameters, however Eurocode 2 (EC2) allows for variations from one country to another so in this case the country must be selected from the list. The countries that can be selected depends on the Eurocode database supplied with AdSec and may vary from time to time. In all cases there is a generic option which uses the Eurocode recommended values. 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. Uni-axial bending 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 (eg angles restrained by walls or slab) and for standard rectangular beams to EC2 to allow crack widths to be calculated. For other cases a bi-axial 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.

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).

Any bars defined are shown on the section. The reinforcement is coloured as follows:

- **bright green** — general reinforcement
- **red** — general reinforcement with a pre-stress force
- **blue** — general reinforcement with an initial strain
- **green** — template reinforcement
- **yellow** — selected general reinforcement

(In the section view these bar colours can be selected although the default is dark grey.)

General Reinforcement

In the General Reinforcement Definition the section extents are displayed as a guide for placing the reinforcement. General reinforcement defines records of single bars, lines of bars, arcs/circles of bars or perimeters of bars (depending on the section shape). 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.

Pre-stress in bars is specified here as part of the bar definition. Pre-stress force is specified as a force in each bar.

The modify allows the reinforcement definition to be edited while the shift option allows the reinforcement to be repositioned. The shift option allows a record of reinforcement to be shifted - this can be defined explicitly or by the bar diameter + clearance, typically to produce a second row of bars. A context sensitive menu allows for copy/paste options so that bar records can be copied to/from a text file or spreadsheet.

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. Template reinforcement can be defined for the following section shapes:

- Rectangle
- Circular
- Tee & inverted tee
- Tapered tee
- I section

- General I section
- Taper
- Channel
- Angle
- Elliptical
- Recto-circular

The section is now complete.

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

2.3 Creating a Compound Section

A compound section is one built from a number of component sections. This can be used to look at sections where a slab may be cast after a beam is in place and where slab and beam may be of different strengths, to model confined zones within a section or for sections of different materials such as concrete cased steel sections or 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 the 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 wrap 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 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 has the correct intersection with the reference section.

Create Confined + Unconfined Section

This option allows a section to be split into confined and unconfined components of a compound section. The original section is split into a pair of sections: a confined section bounded by the links and an unconfined section outside the links. The wizard allows new materials to be defined for these sections. For ACI these will typically be defined by Mander curves.

This option is only available for sections defined by a template.

2.4 Load Cases and Analysis Cases

It is important to understand the difference between load cases and analysis cases. A load case is used to group together a set of loads that will always act together. An analysis cases specifies the loading that applies for this particular analysis so can include more than one load cases. So for example load case 1 may be dead load and load case 2 imposed load. then the analysis case could consider combined dead and imposed loads (e.g 1.4L1 + 1.6L2. for BS8110).

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.

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

If the loading is dead load the 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 loadfing 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 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 Mq/Mg.

When starting an analysis (either ULS or SLS) you are given 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.

Working with AdSec

Part III

3 Working with AdSec

AdSec is a general non-linear section analysis program.

Output from AdSec includes:

- Section properties
- Ultimate resistance of irregular sections (reinforced and pre-stressed) N/M and Myy/Mzz interaction charts
- Crack widths and cracking moment
- Flexural stiffness (*EI*) of section (M/Curvature and EI/Moment charts)

AdSec can provide different serviceability limit state (SLS) material models for concrete eg for BS8110:

Compression models vary from linear elastic, through bi-linear and recto-parabolic stress-strain curves (BS8110-1, Figure 2 / BS5400-4, Figure 1 / EN 1992-1-1, Figure 3.3 and 3.4) to more realistic stress-strain models (BS8110-2, Figure 2.1 and EN 1992-1-1, Figure 3.2)

Tension models vary from no tensile strength, to model which allow for tension strength and cracking (BS8110-2, Figure 3.1, ICE Tech Note 372 and EN 1992-1-1, 7.4.3 interpolated model)

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

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) = $\varepsilon_x + \kappa_y z + \kappa_z y$
- Loops until a plane is found that satisfies three conditions

These steps are described in more detail in the Theory section.

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. This section details:

- Creating and Editing Sections
- Reinforcement
- Compound Sections
- Current Section
- Analysis Cases
- Sign Convention

3.1.1 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 "Sections" tab on the Gateway gives access to all aspects of the section. Select 'Dimensions' if you want to change its dimensions. Only select 'Definition' if you wish to change the section shape and re-create the reinforcement.

3.1.2 Reinforcement

General reinforcement (ie single bars, lines, arcs and circles of bars) can be placed for any section. The option of perimeter is also available for a selection of section shapes.

Template reinforcement (ie beam and column arrangements) are available for several 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.

3.1.3 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.

3.1.4 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.

3.1.5 Analysis Cases

Analysis cases contain data to be used in a ULS Strength, ULS Loads, or SLS Loads 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.

See also:

Load Cases and Analysis Cases

3.1.6 Sign Convention

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

Vertical axis is labelled Z, positive upwards

Applied Forces

Axial Load is positive for compression.

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

Axial strain is positive for compression.

κ_{yy} is the curvature about the Y axis, positive for compression on the Top of the section.

κ_{zz} is the curvature about the Z axis, positive for compression on the Right of the section.

The strain at a position in the section is

$$\varepsilon = \varepsilon_{ax} + \kappa_{yy}.z' + \kappa_{zz}.y'$$

Note: Moment angles and neutral axis (N/A) angles are measured positive anticlockwise from the positive Y axis. A negative concrete-only strain models shrinkage in the concrete. A negative pre-stress gives tension in the steel.

3.2 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.

The Gateway has two tabs: **Sections** for access to the section information and **General** which gives access to general data editing.

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 **Sections** tab in the Gateway organises the section data for each component section. The items are updated dynamically as new sections are created.

The **General** tab in the Gateway organises the data into several categories. Against each item is reported the number of records currently specified for the data module.

3.3 Working with the Material View

The Material View is the entry point for Materials in AdSec. This list all the materials in a tree style under the main material types headings:

- Concrete
- Rebar
- Steel
- FRP — *not available for AS3600 or IS 456*

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.

A context sensitive menu is available to allow viewing and editing of materials.

When a new model is created this will list only the standard materials for the selected design code. Materials not referenced in a section are shown with an "open" icon and those which are referenced use a "closed" icon. New materials can be added from the right-click menu. These have a dark red icon so can be distinguished from the standard materials which have a dark blue icon.

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 any of DXF, JPEG, PNG and WMF formats.

See also:

Graphical Output

Label and Display Options

3.5 Working with Table Views

Most input 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. These are highlighted in detail below:

- Single and multi-page tables
- Tabular data entry and editing
- Basic operations in tables
- Find, replace, go to and modify in tables
- Copying to and from spreadsheets
- Adjusting data display

3.5.1 Single and multi-page tables

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

In other cases the data to be displayed is more complex or several data modules are related so it is convenient to display the data on more than one page of a table. For example there are standard and user defined concrete materials – these are displayed on separate pages in the table.

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

3.5.2 Tabular data entry and editing

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

Changes made in a table are immediately reflected in other tables 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.

3.5.3 Basic operations in tables

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

Navigation

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

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

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

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

Editing

The following actions change a cell to edit mode:

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

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

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

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

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

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.

3.5.4 Find, replace, go to and modify in tables

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.

To find a specific **record**, use the 'Edit | Go To' (Ctrl+G) menu command or the 'Go To' button on the Data toolbar.

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.

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).

3.5.5 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.

3.5.6 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.

3.6 Ultimate Limit State Analysis Options

ULS Strength Analysis

An ultimate strength 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. For ACI and AS analysis the user can specify the strength reduction factor to be used or leave the value as zero for AdSec to determine an appropriate value.

ULS Loads Analysis

The program finds the state of strain corresponding to the input (factored) force and moments. Where appropriate (i.e. European and Hong Kong codes) factored material curves are used. The resulting stresses and strains can be viewed.

Charts

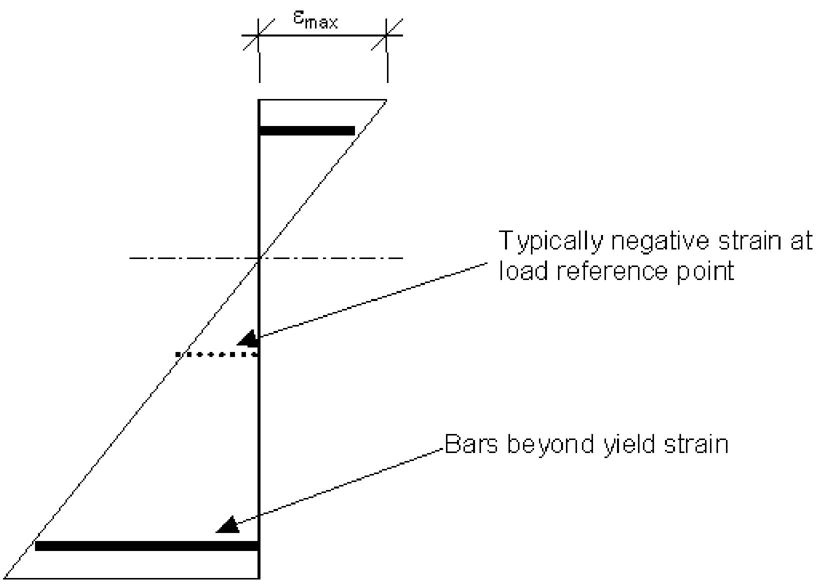
In addition to analysis of particular load cases there are chart options for:

- Force/Moment Interaction Charts

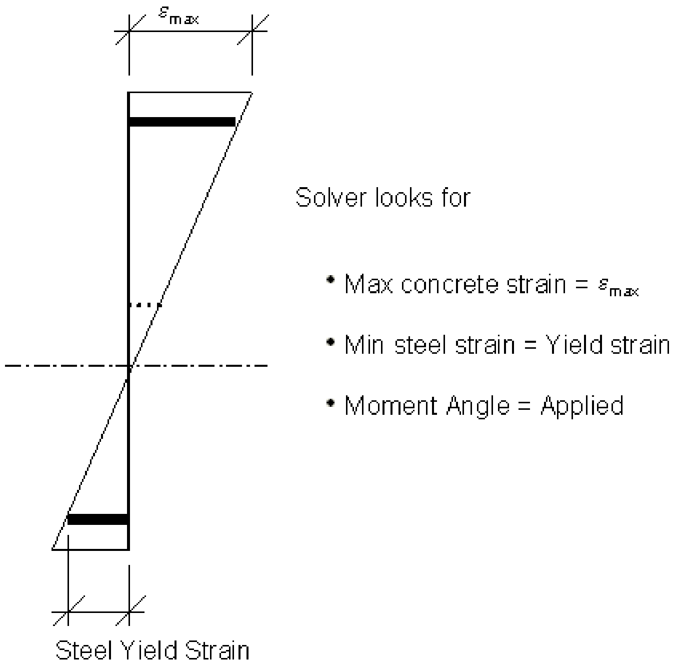
- Moment Interaction Charts

3.6.1 Force/Moment Interaction Charts

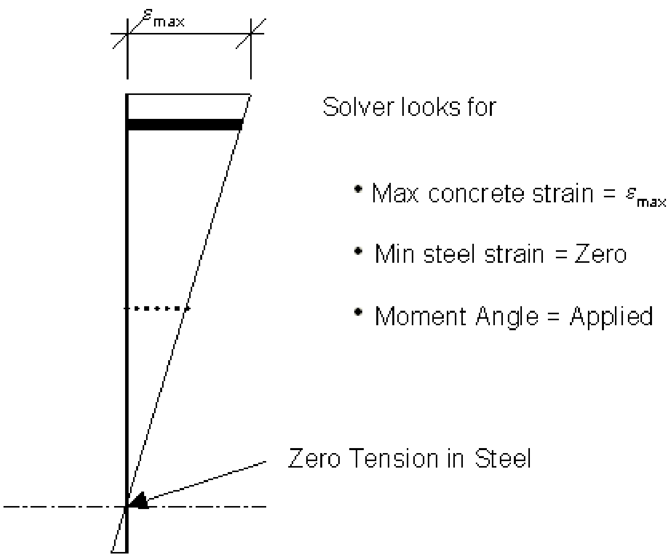
The Force/Moment Interaction (N/M) chart gives a strength envelope for a given moment orientation. The *no tension* and *balanced yield* values are found for the input value of moment orientation. An ultimate strength analysis is performed to generate ultimate moments for a range of axial force values 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



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 by using the "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.

3.6.2 Moment Interaction Charts

The Moment Interaction (M_{yy}/M_{zz}) charts give a capacity/strength envelope for a given axial force. The moment orientation is varied from 0º to 360º. 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_{yy},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 Program Data and Theory sections.

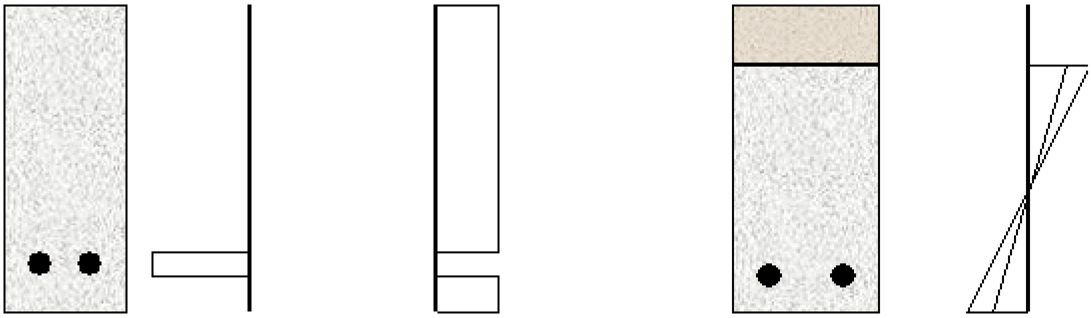
Charts

In addition to analysis of particular load cases there are chart options for Moment/Curvature (M/κ) and Moment/Stiffness (M/EI) charts. These 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 pre-stress, creep and shrinkage or locked in strain planes in a compound section.



Example of strain discontinuities

Pre-stress

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 pre-stress 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 programme using the creep factor and need not be modelled by concrete only strain planes.

Part IV

4 Program Data

This chapter describes the different types of data that can be used to describe the model. The data is organised in modules and displayed for input and editing in a number of dialogs and tables. 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 UK, Hong Kong, European, USA and Australian codes of practice is available. Code-specific defaults are set by the program on selection of a code of practice.

Ultimate strength analysis will give limiting moments based on load factors and limiting concrete and steel strains. Material factors or strength reduction factors are applied according to the selected code of practice.

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 crackwidth which complies to BS5400.

4.1 Titles

The titles view contains the job details. This data is for information only and has no affect 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.

Bitmap

The user can place a graphic image of the structure in this field. This is useful when viewing the file in Columbus.

4.2 Specification

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

- General Specification
- Units

4.2.1 General Specification

Design Code and Country

In the general specification the user chooses the design code – at present the options for structures are:

- **ACI 318-02** (Building Code Requirements for Structural Concrete (ACI 318-02))
- **ACI 318M-02** (Building Code Requirements for Structural Concrete (ACI 318M-02))
- **ACI 318-05** (Building Code Requirements for Structural Concrete (ACI 318-05))
- **ACI 318M-05** (Building Code Requirements for Structural Concrete (ACI 318M-05))
- **ACI 318-08** (Building Code Requirements for Structural Concrete (ACI 318-08))
- **ACI 318M-08** (Building Code Requirements for Structural Concrete (ACI 318M-08))
- **AS 3600—2001** (Australian Standard - Concrete Structures)
- **AS 3600—2009** (Australian Standard - Concrete Structures)
- **BS8110:1985** (BS8110-1:1985, BS8110-2:1985) (now withdrawn in the UK)
- **BS8110:1997** (BS8110-1:1997 incorporating amendments 1and 2, BS8110-2:1985)
- **BS8110:1997 (2005)** (BS8110-1:1997 incorporating amendments 1, 2 and 3, BS8110-2:1985)
- **EN 1992-1-1:2004** Eurocode 2
- **BS EN 1992-1-1:2004 / PD 6687:2006** Eurocode 2 (UK) with background document PD 6687
- **Hong Kong Code of Practice 1987** (Code of Practice for Structural Use of Concrete)
- **Hong Kong Code of Practice 2004** (Code of Practice for Structural Use of Concrete 2004)
- **IS 456 : 2000** (Indian Standard - Plain and Reinforced Concrete - Code of Practice)

and for bridges:

- **BS5400** (BS5400-4:1990)
- **HK Structures Design Manual - 2002** (Structures Design Manual for Highways and Railways 1997, Amended 2002)

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.

Minimum cover

Minimum covers can be specified. These are only used to check the location of the bars. AdSec does not incorporate the code detailing rules about covers and bar spacings.

For a BS5400 analysis a nominal cover is specified for crack width calculations and the bar covers may be checked against this value.

Surface tolerance

The surface tolerance 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 (eg 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, rebar, steel and fibre reinforced polymer (FRP) materials and these are defined separately.

- Concrete Properties
- Rebar Material Properties
- Steel Material Properties
- FRP Material Properties

4.3.1 Concrete Properties

Several standard concrete types are offered which cannot be edited. New, user defined, concrete types can be created either by copying a standard type in the concrete material wizard or by defining the properties explicitly. Concrete properties defined are short-term and unfactored. 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, and creep coefficients, where defined by the code.

Name

A name is used to identify the concrete material.

Strength Parameters

The concrete strength is characterised by the compressive strength (cube strength for BS, HK and IS design codes and cylinder strength for Eurocode 2, ACI and AS design codes) and the tensile strength. The stiffness is characterised by the short term Young's modulus. The Poisson's ratio and coefficient of thermal expansion are not used at present. The proeprties can be calculated according to the design code or user specified values can be used.

Normal/Light Weight

Either normal weight or lightweight concrete can be specified. For lightweight concrete a density must also be specified.

Serviceability Curves

Two serviceability stress-strain curves are specified – one for compression and one for tension. The stress-strain curves are described in detail in the Theory section.

Ultimate Curves

Two ultimate stress-strain curves are specified – one for compression and one for tension The tension curve is always assumed to be "no-tension". The stress-strain curves are described in detail in the Theory section.

Partial Safety Factors

Where the design code makes use of material partial safety factors (γ_m) these are specified for the ultimate analysis .

Limiting Strain

Depending on the design code selected different strain limits may be required. Typically the maximum strain (ε_{max}) is where the concrete fails in compression. The value is normally set out in the code, for example it is 0.0035 for BS8110 but must be reduced for high strength concrete, while ACI uses a value of 0.003. The user may override these code values.

Confining Stress

Where a confined concrete model is selected (EC2 and ACI only) the confining stress must be specified. For ACI analysis the Mander option assists in setting up the stress-strain curve.

User Stress/Strain Curves

The user defined material curves are represented by a list of points in the form (σ,ε) (σ,ε) ... The *manipulate curve* option allows for

- Factoring the curve by a value (scaling)
- Switching between compression and tension (it can be easier to define tension curves in the compression zone).
- Convert to true stress and strain from engineering stress and strain as follows

The relationship between true stress/strain and engineering stress/strain is as follows:

$$\sigma_t \to \sigma_e (1+\varepsilon_e)$$
$$\varepsilon_t \to \log(1+\varepsilon_e)$$

where the subscripts t and e refer to true and engineering respectively.

4.3.2 Rebar Material Properties

A number of standard steel and FRP reinforcement types are available. New, user defined, rebar materials can be created either by copying a standard type in the rebar material wizard or by defining the properties explicitly.

For UK design codes the bend radius of the rebar is to BS8666:2000. This is used when calculating the link profile when positioning template reinforcement. This value is ***not*** appropriate for GFRP reinforcement.

Name

A name is used to identify the rebar material.

Stress/Strain Curves

Stress-strain curves are specified for the rebar. The same curve is used for both ultimate (strength) and serviceability analysis. These basically represent either normal reinforcement or pre-stressing strands. The stress-strain curves are described in detail in the Material Curves section of the Technical Notes. If the explicit option is used then user defined material curves must be supplied. In accordance with current guidance, FRP rebar is deemed to have no strength or stiffness in compression.

Strength and Stiffness

The rebar strength is characterised by the tensile strength. The stiffness is characterised by the Young's modulus.

Partial Safety Factors

Where the design code makes use of material partial safety factors (γ_m) these are specified for the ultimate analysis.

Limiting Strain

This is the strain (ε_{max}) at which the rebar is deemed to have failed.

FRP Factors

For ACI with FRP reinforcement an environmental factor must be specified which modifies the stress-strain curve. See ACI 440.1 Table 7.1 for details.

User Stress/Strain Curves

The user defined material curves are represented by a list of points in the form (σ,ε) (σ,ε) ... The *manipulate curve* option allows for

- Factoring the curve by a value (scaling)
- Switching between compression and tension (it can be easier to define tension curves in the compression zone).
- Convert to true stress and strain from engineering stress and strain as follows

The relationship between true stress/strain and engineering stress/strain is as follows:

$$\sigma_t \to \sigma_e(1+\varepsilon_e)$$
$$\varepsilon_t \to \log(1+\varepsilon_e)$$

where the subscripts t and e refer to true and engineering respectively.

4.3.3 Steel Material Properties

A number of standard steel types are available. New, user defined, steel materials can be created either by copying a standard type in the steel material wizard or by defining the properties explicitly.

Name

A name is used to identify the steel material.

Strength and Stiffness

The steel stiffness is characterised by the Young's modulus. The strength is characterised by the tensile strength.

Partial Safety Factors

Where the design code makes use of material partial safety factors (γ_m) these are specified for the ultimate analysis .

Limiting Strain

This is the strain (ε_{max}) at which the rebar is deemed to have failed.

4.3.4 FRP Material Properties

A number of standard FRP (Fibre Reinforced Polymer) types are available. New, user defined, FRP materials can be created either by copying a standard type in the FRP material wizard or by defining the properties explicitly.

Name

A name is used to identify the FRP material.

Strength and Stiffness

The FRP stiffness is characterised by the Young's modulus. The strength is characterised by the tensile strength. FRP is conservatively deemed to have no strength or stiffness in compression.

Partial Safety Factors

Where the design code makes use of material partial safety factors (γ_m and γ_{me}) on strength and elastic modulus these are specified for the ultimate analysis. FRP is anisotropic so the relevant FRP properties for AdSec analysis are those perpendicular to the plane of the section.

Bond Strain

This is the limiting strain in the FRP to avoid separation from the parent material or de-bonding failure. Default bond strain limits may not be the most appropriate values to use. For ACI analysis the bond strain limit should be a function of the FRP thickness and for other design codes it will vary depending on the loading conditions and/or the strain in the rebar. Use to bond options for more help in setting bond strain limits.

Strength Reduction Factor

This is a factor used in ACI 440 to reduce the strength and stiffness of FRP material. This is similar to the reciprocal of the partial safety factor in other codes.

4.4 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 – 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.

Load Titles

Loads

Reference Point

4.4.1 Load Titles

Load case & Title

This allows a title to be associated with a particular load case. This is not used directly by AdSec it is provided for the convenience of the user.

4.4.2 Loading

Load case

The load case is 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: pre-stress is defined in the Reinforcement Table. If anchorage and curvature forces have been included in the loading as a section force, ensure that this is registered in the reinforcement wizard as "Applied load which ***includes*** pre-stress anchorage and curvature forces" or pre-stress effects will be accounted for twice over. If the option "applied loads ***exclude*** prestress ancorage and curvature forces" is selected in the reinforcement wizard, the prestress is treated as a strain discontinuity.

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.

4.4.3 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

The 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.

Centroid

The geometric centroid is really only useful for homogeneous sections. A number of other centroids could be defined.

The effective centroid is found by applying a constant strain over the section and converting this to stresses in the concrete and steel using the current stress-strain assumptions. The resulting force acts through the effective centroid.

The plastic centroid is the centroid of stress when the strain acting over the section equals the limiting compressive strain of the material.

4.5 Analysis case

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

ULS Analysis Cases

SLS Analysis Cases

4.5.1 ULS Analysis Cases

To perform a ULS or strength analysis it is necessary to define one or more ULS analysis cases.

Name

A name used to identify the analysis case.

Description

This is where the loading is described. The description syntax is of the form

a_1Lc_1 + a_2Lc_2 + …

where a_i is the factor that applies to the load defined in load case c_i.

Pre-stress Factor

The pre-stress is applied to the reinforcement, defined for the section. For analysis this pre-stress may be factored as required.

Strength reduction factor

The strength reduction factor applies only to ACI and AS codes. If this is set to zero (the default value) the factor is calculated by AdSec in accordance with the design code. This code value can be overridden by specifying a non-zero strength reduction factor. For a nominal analysis the strength reduction factor can be set to 1

Note: if no ULS analysis cases exist, the programme will generate a set of unfactored analysis cases corresponding to each load case.

4.5.2 SLS Analysis Cases

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

Name

A name used to identify the analysis case.

Analysis Type

The analysis 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 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 and Additional Short Term Load

This is where the loading is described. The description syntax is of the form

a_1Lc_1 + a_2Lc_2 + …

where a_i is the factor that applies to the load defined in load case c_i.

Prestress Factor

The pre-stress is applied to the reinforcement, defined for the section. For analysis this pre-stress may be factored as required.

Creep Coefficient

This specifies the creep coefficients that modify the concrete properties so that for linear stress-strain curves

$$E_{long} = \frac{E_{short}}{(1 + creep\, coeff.)}$$

The theory section describes how creep is applied to other concrete curves.

Depending on the design code other fields may be required.

BS5400

Nominal Cover

For BS5400 the nominal cover is used to generate a perimeter for crack width calculations.

Crack Width Equation

For BS5400 analysis the crack width equation to be used must be specified. This can be either BS5400: Equation 24 or Equation 26.

Mq/Mg Ratio

For BS5400 analysis the M_q/M_g ratio is used in the crack width formula to calculate the strain (equation 25) and to calculate the properties for a intermediate term analysis.

Eurocode & PD 6687

Cover

For the version of UK version Eurocode and PD 6687 the cover can be specified at which crack widths are reported.

Crack Width Equation

For Eurocode analysis the crack width equation to be used can be specified. This can be either EN 1992-1-1 Equation 7.9 or Equation 7.18. In PD 6687 a different equation is given for calculation of crack widths in Section 2.17.

Duration

The duration is only relevant for short term analysis. Various factors in the analysis depend on a distinction between normal short term and "instantaneous" short term.

	normal	instantaneous	
kt	0.4	0.6	7.3.4 (2)
β	0.5	1.0	Equation (7.19)

4.6 Sections

A section is defined in of two parts – the concrete section definition and the reinforcement. More than one section can be included in a model to facilitate the analysis of compound sections (or composite sections), or sections belonging to a family with similar geometry or different design options subjected to similar load.

4.6.1 Definition

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

Name

All sections have a name used to identify that section.

Definition and Dimensions

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.

Material

A particular material is associated with a section. This consists or a material type (eg concrete) and a material grade (eg C50) These may be either standard or user defined concrete, steel or FRP materials.

4.6.2 Reinforcement

Reinforcement can be defined in two ways

General Reinforcement

General reinforcement is defined a single bars, lines, arcs, circles or perimeters of reinforcement at explicit positions. This is defined in the Reinforcement Wizard.

Add

This allows new reinforcement definitions to be added via the general reinforcement definition dialog.

Modify

Reinforcement records can be selected for modification via the general reinforcement definition dialog. This allows any aspect of the reinforcement to be adjusted.

Shift

The bars can be shifted. This is useful when one record is a copy of the other or where the dimensions have been adjusted and the record needs to be shifted to a new position in the section. The option to shift by bar diameter + clearance allows for bars positions top be adjusted to provide a second row of reinforcement.

Copy, Paste and Paste Special

Records of reinforcement can be copied and pasted. The paste special allows for the reinforcement to be shifted before pasting.

Delete

This deletes records of reinforcement.

Import and Export

These work in a similar way to copy and paste but read from / write to a file rather than use the clipboard.

Template Reinforcement

Template reinforcement is defined with respect to the section for either a beam or column. This means that the reinforcement is repositioned if the section dimensions are adjusted.Template reinforcement cannot be pre-stressed.

4.6.3 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.

4.6.4 Cover

There are a number of factors which contribute to the cover in AdSec. What is defined here is the environmental cover but bar size and aggregate will also affect the required cover. Cover is only used in AdSec to locate the template reinforcement.

Cover

The cover can be **uniform** on all faces or **variable**. Variable cover allows different cover to be set on different faces. The Code cover option will give access to code specific cover options for different types of situation.

Allowance for transverse steel

This is not used in AdSec at present.

Dialogs and Wizards

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 interdependencies a wizard is provided to lead the user through the various steps.

Wizards exist for:

- 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.

The steps in the general section wizard are:

- 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

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

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. For most codes the appropriate country will be displayed. For EC2 the country is selected from a list with known national parameters.

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 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.

Surface tolerance

The surface tolerance is used when dealing with sections with curved edges. In the analysis these are represented as polygons. The number of facets are determined so that the analysed surface never deviates from the actual surface by more than the surface tolerance.

5.1.3 General Section Wizard : Wrap/Cast Section

In many situations a compound section is constructed by wrapping or casting around (or inside) anotehr 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 cretaed 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** – advanced option only
- **FRP** – advanced option only

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.

Aggregate Size

For concrete sections the cover and bar spacing can be affected by the aggregate size. Leaving the aggregate size as zero means that these adjustments cannot be made when position bars in template reinforcement.

5.1.5 General Section Wizard : Reinforcement

If the section is defined with a concrete material type 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.

A context menu provides access to these functions and also a

Paste Special – allows bars to be shifted before pasting

Copy + Shift – copies and shifts existing records of reinforcement

Import – import of reinforcement in text or csv format

Export – export of reinforcement in text or csv format

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 section shapes can be defined.

The wizard takes the user through the following pages:

- Section Wizard : Section type
- Section Wizard : Catalogue Section
- Section Wizard : Standard shapes
- Section Wizard : Perimeter section definition
- Section Wizard : Section definition

Note that not all section options are available for all materials

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 Access 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.

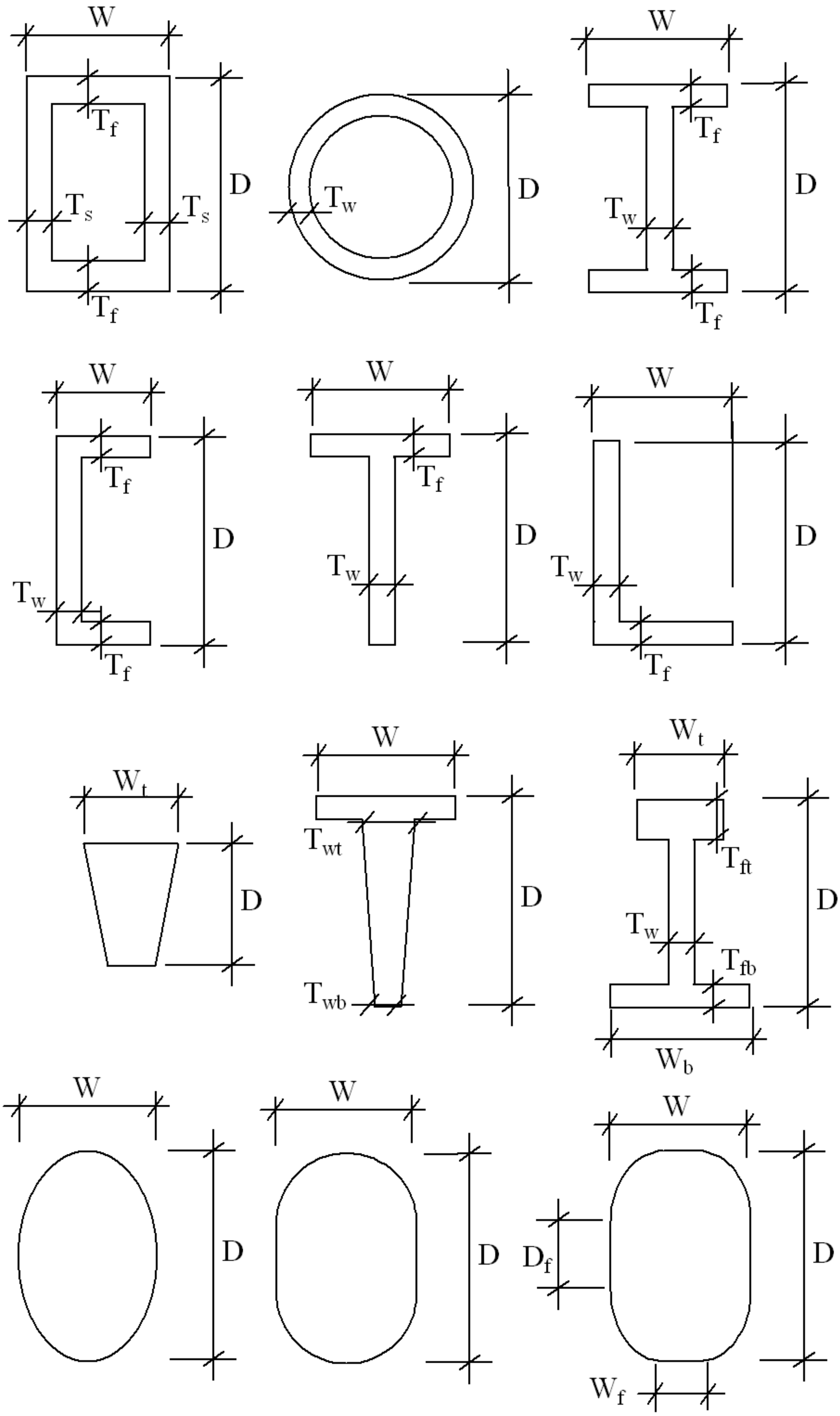
Include superseded sections

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.


W
T_f
T_s
T_s
T_f
D
T_w
D
W
T_f
T_w
T_f
D
W
T_f
D
T_w
T_f
W
T_f
T_w
D
W
D
T_w
T_f
W_t
D
W
T_{wt}
D
T_{wb}
W_t
T_{ft}
T_w
T_{fb}
D
W_b
W
D
W
D
W
D_f
D
W_f

Ellipse index (n)

This field is only available for the ellipse section type. It allows super-ellipses 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 sectionto 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 summarises 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 means no previous cracking.

5.4 Compound Sections

Compound section give powerful additional capabilities to AdSec by allowing section with different materials and/or properties. A number of tools are available to help construct and manipulate compound sections:

- Compound Section Definition
- Wrap/Cast Section - described separately
- Section Builder
- Create Confined + Unconfined Section

These tools create section geometry. AdSec does not assess the level of confinement achievable by wrapping/casting in steel tubes or through the rebar.

If the resulting compound section will result in confinement of the concrete, the user will need to assess the confinement pressure/strength. A confined concrete model can be assigned to the section in order to take advantage of this behaviour.

5.4.1 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 and D. Component sections B, C and D 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.

5.4.2 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.

5.4.3 Create Confined + Unconfined Section

Where concrete in a section is confined its behaviour is quite different to unconfined concrete. This option allows a concrete section with template reinforcement to be split into its confined and unconfined components along the line of the links.

Compound Section

This is a name to identify the new section.

Confined and Unconfined Section

This allows the confined and unconfined parts of the section to be assigned a name and material.

Number of segments

The number of segments is used only in sections with circular or elliptical links where these will be faceted.

Add material

This allows, if required, new materials to be defined for the confined and unconfined parts of the section

5.5 Reinforcement Wizard

Template Reinforcement

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.

- 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

General Reinforcement allows bars to be located by coordinates inside the section.

- General Reinforcement Definition

General and template reinforcement can both be used in the same section.

5.5.1 Reinforcement Wizard : Template Definition

Template Reinforcement Definition

The basic choice when defining reinforcement using a template is between beam and column arrangements or for the cases of slab/wall sections a slab/wall arrangement.

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 depends on the shape of the section. For rectangular sections a standard bar pattern is chosen and bar sizes and covers specified. For circular sections and those based on circles a pattern of rings is assumed and the user gives details of the bars in the rings.

The slab/wall arrangement allows for top and bottom bars to be specified with a spacing rather than a number of bars and takes into account that the section represents a typical strip. Checks are made that the reinforcements spacing fits the width of the section.

Spiral Reinforcement

For ACI analysis the strength reduction factors depend on the type of reinforcement. This adjustment is made if the spiral reinforcement flag is checked.

5.5.2 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. This is the cover required for environmental factors. Additional cover may be required depending on the bar size and aggregate size but this is determined by AdSec.

Links (stirrups & ties)

The diameter of the bars and the type of rebar is specified. The rebar can refer to a standard or user defined rebar materials. In the case of beam sections with tapered side the user can select links with are parallel to the sides or vertical.

Links Shape

For tapered sections link shape can be either rectangular or trapezoidal. This governs the shape of the links in the web part of the section.

Aggregate Size

The cover and bar spacing can be affected by the aggregate size. Leaving the aggregate size as zero means that these adjustments cannot be made when position bars in template reinforcement.

5.5.3 Reinforcement Wizard : Main Reinforcement

Top

By checking the include option reinforcement bars can be placed in the top of the section.

This is specified as a number of bars in each row, the number of rows of bars and the distance between the rows. The *auto* option allows AdSec to select the spacing between rows to be based on the code minimum spacing. Either single bars or pairs of bars can be placed. Where pairs of bars are used the bars are assumed to be in bundles. The diameter of the bars and the type of rebar is also specified. The rebar can refer to a standard or user defined rebar material.

Bottom

By checking the include option reinforcement bars can be placed in the bottom of the section.

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

5.5.4 Reinforcement Wizard : Side Reinforcement

Side Bar Arrangement

This specifies the number or pitch of the side bars. When the pitch is specified the bars can then be located relative to the top or bottom main steel. Where a section is tapered the bars will be sloping or vertical depending on the option previously selected for the links.

Side Bars

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

5.5.5 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. The *auto* option allows AdSec to select the spacing based on the code minimum spacing.

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 rebar can refer to a standard or user defined rebar material.

5.5.6 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. The *auto* option allows AdSec to select the spacing based on the code minimum spacing.

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.

5.5.7 General Reinforcement Definition

The general definition dialog allows bars to be placed at explicit position in a section. The section description and section extents are displayed to aid location of the reinforcement.

Definition Type

The type is one of the following:

- **single** – a single bar at a position
- **line** – a line of bars defined by a start and end position
- **arc** – an arc of bars defined by a start and end position and a point on the arc
- **circle** – a circle of bars defined by the position of the first bar and the centre of the circle
- **perimeter** – a set of bars specified by a uniform cover. This is midway between general and template reinforcement.

Bar Diameter or Area

Either the bar diameter or area must be specified. Where an area is specified this is converted directly into an equivalent diameter.

Bar Bundle and Number of Bars

Normally single bars are placed in the section but there is the option of choosing pairs of bars, or bundles of three or four bars. For line, arc, circle and perimeter definitions the number of bars (or bundles of bars) must be specified.

First Bar, Last Bar, Point on Arc, Centre of Circle and Cover

The coordinates or cover used to define the bar positions. The relevant coordinates depend on the type as described above. Note that the cover specified for perimeter option is not checked against the environmental cover, bar size, etc.

Reinforcement

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

Pre-stress

Pre-stress 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 pre-stress must not exceed the elastic limit of the material.

A negative pre-stress will result in tension in the steel.

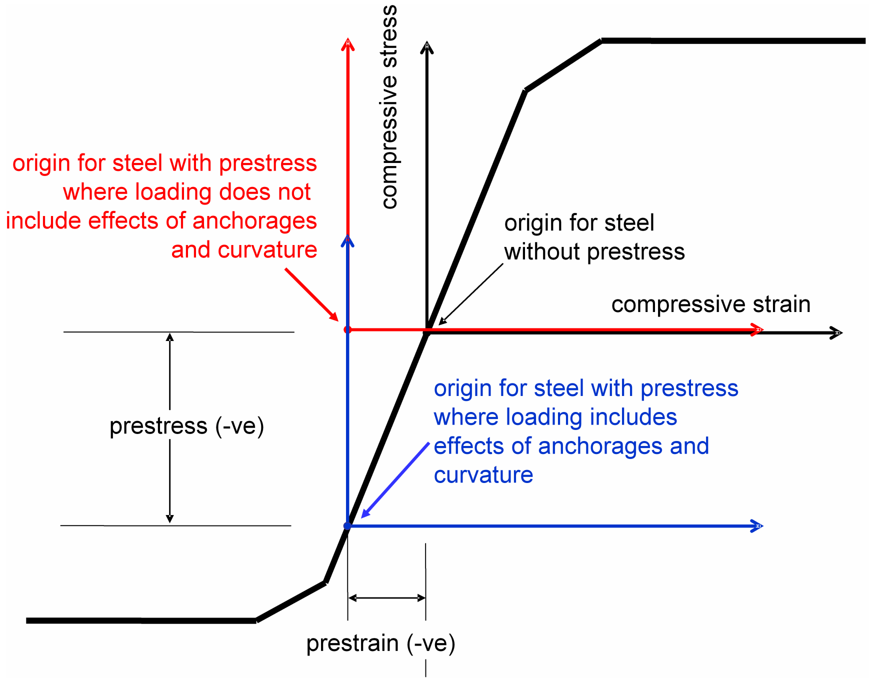
There are two ways in which engineers consider prestressing.

Situations where the external loads do not include the effects of anchorages and curvature

For structural elements such as prestressed precast floor or bridge units with straight tendons, usually stressed before the concrete is cast, it is usually more convenient to consider only the forces and moments applied to the element as a whole. AdSec will generate the prestress force in the concrete. Applying a prestress this way is equivalent to shifting the origin of the stress-strain diagram of the tendon or bar along the tensile strain axis, as shown in red on the diagram below.

Situations where the external loads include the effects of anchorages and curvature

For structural elements such as post-tensioned floors or bridge beams, it is usually more convenient to consider the forces exerted on the structure by the prestressing tendons or bars in addition to the forces and moments applied to the element as a whole. The forces exerted by the tendons or bars will be applied at their anchorages, where they deflect or curve and where frictional losses occur. AdSec will not generate any additional prestress force in the concrete. Applying a prestress this way is equivalent to shifting the origin of the stress-strain diagram of the tendon or bar along the tensile stress and strain axes, as shown in blue on the diagram below.



5.6 Chart Analysis Dialogs

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

- ULS and SLS Analysis
- N/M Chart Analysis
- Myy/Mzz Chart Analysis
- M/Curvature and M/EI Chart

5.6.1 ULS and SLS Analysis

There are three basic analysis option in AdSec for ULS and SLS analysis.

ULS Strength Analysis

In a ULS strength analysis AdSec looks at the axial force and moment applied to the section. The axial force and moment angle are held constant and AdSec determines the maximum (ultimate) moment that can be sustained by the section. The ULS material properties are used in this analysis.

ULS Loads Analysis

A ULS loads analysis uses the the same loading and material properties as the ULS strength analysis, but instead of increasing the moment to the maximum for the section the loading is kept constant and AdSec determines the corresponding strain plane and the section stresses and strains.

SLS Analysis

In an SLS analysis the SLS material properties are used and AdSec find the corresponding strain plane, and material stresses and strains. In an SLS analysis various other checks are carried out, such are cracking moments and, where appropriate, crack width calculations.

5.6.2 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.

Strength reduction factor

For ACI and AS analysis the user can select how the strength reduction factor will be used. The options are:

- **from code** – calculates the strength reduction as specified by the code
- **ignore** – excludes the strength reduction from the calculation. This gives a nominal strength analysis
- **user specified** – the user specified strength reduction factor is used throughout.

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.

Interactive Solution

The interactive solution option allows the user to control the solution during the iteration by adjusting the solution parameters and convergence tolerances. This is intended for expert users as inappropriate use may lead to solution instabilities.

5.6.3 M/Curvature and M/EI Chart

The M/Curvature (or M/κ) and 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.

Interactive Solution

The interactive solution option allows the user to control the solution during the iteration by adjusting the solution parameters and convergence tolerances. This is intended for expert users as inappropriate use may lead to solution instabilities.

5.6.4 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.

Strength reduction factor

For ACI and AS analysis the user can select how the strength reduction factor will be used. The options are:

- **from code** – calculates the strength reduction as specified by the code
- **ignore** – excludes the strength reduction from the calculation. This gives a nominal strength analysis
- **user specified** – the user specified strength reduction factor is used throughout.

In an ACI analysis for design strength (i.e. including the strength reduction factor) the axial force cutoff is included in the calculation.

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.

Interactive Solution

The interactive solution option allows the user to control the solution during the iteration by adjusting the solution parameters and convergence tolerances. This is intended for expert users as inappropriate use may lead to solution instabilities.

5.7 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:

- Stress-strain Chart
- Mander Curve
- Expand Section
- Transform Section
- Label and Display Options
- Chart Style
- Additional Points on N/M and Myy/Mzz Charts
- Extract Analysis Strain Planes
- Output Specification

5.7.1 Stress-strain Chart

The stress-stran 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 using the default creep factor associated with the selected design code.

5.7.2 Mander Curve

This allows the parameters to be set to generate a Mander curve as user defined concrete curve.

Confined / Unconfined

There are two Mander curves one for confined concrete and the other for unconfined.

Strength and strain parameters

The particular Mander curve is a function of the concrete strength, the confined strength and the strain behaviour of the concrete.

5.7.3 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. Typically this is used where larger "bottom" bars and smaller "side" bars have been defined using the same reference points – this allows the smaller bars to be removed to avoid overlap of bars.

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

5.7.4 Transform Section

Sections can be translated and rotated using this option. The section can be either a simple section or compound section. All reinforcement is transformed as well.

5.7.5 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:

- **Solid** areas – where bars are shaded in uniform colour
- **By type/pre-stress** – where the bars are drawn in different colours if general or template reinforcement and if prestressed on 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.

5.7.6 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 as required. The "Update Curve" applies these changes.

Legend

The legend can be displayed or omitted. The width of the legend panel can be adjusted if not all of the legend is visible or if more space is required for the chart.

Border & Grid Lines

This controls if the graph displays a border around the graph area and grid lines within the graph area.

Axes

The axes can be labelled or left blank. The axes can be specified to include units by including the unit description in the title within angle brackets, for example "Moment <MOMENT>".

Note

This allows a note to be associated with the graph

5.7.7 Additional Points on N/M and Myy/Mzz 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 force/moment interaction charts or moment interaction charts. The label option is used to identify particular points.

5.7.8 Extract Analysis Strain Planes

This reports the strain planes from analysis and allows these to be selected and extracted to become part of the data for subsequent analysis.

This is useful for staged loading / construction.

5.7.9 Output Specification

Specification selection

The output specification allows control of the tabulated output. The specification items are arranged in a tree which varies depending on the state of the model and/or the analyses which have been carried out. These fall into the main categories of

- Specification
- Section
- Loads
- Results – depending on the analysis

The Reset option sets the output to that selected in the Preferences.

Case list

The results are output for all cases by default. This can be changed by specifying a list of cases for which output is required.

Output table/summary

This allows full tables of results to be output and/or just summary values.

See Also:

Preferences

5.8 Preferences

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

Miscellaneous

Numeric format

This controls various preferences which mainly allow the user control over what AdSec outputs. The main option is the 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.

Startup

This controls various options when AdSec is started, and allows file locations for reference files to be specified.

Code Options

This allows the default design code and units to be specified. This is then used by default when creating a new model, but can be overridden in the new model/section wizard.

Advanced

This offers two options. The data option allow some of the less frequently used options to be hidden from the user. The solver option allows the number of iterations to be set. It can be useful to increase this if analyses are failing to converge.

The graphics line width option allows different default line widths to allow for variations between different monitors.

Output

This allows the user control over what is output. The main option is to control the output of crack width details.

BS5400

This allows BS5400 specific changes for the nominal cover.

AS3600

This allows different option for consideration of the strength reduction factor, either following the rules as outlined in AS3600 or using a weighted reinforcement strain.

Output Options

Part VI

6 Output Options

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

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
- contours of stress output

Graph Views

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

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

For N/M, M_{yy}/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_{yy}/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_{yy}/M_{zz} Chart coordinates					✓	
M/Kappa Chart coordinates						✓

Part VII

7 Theory

This describes in more detail the calculation methods used in AdSec. The details of the calculations depend on the assumptions made in the material models and in particular the concrete material models that are recommended in the design codes.

For concrete sections linear material models are generally inadequate, so it is important to have material models and analysis methods which allow for cracking to develop. The models and methods used for the non-linear analysis are described in this section.

More:

Solution Method

Concrete Materials

Rebar Materials

Stiffness (EI)

Cracking

7.1 General Solution Method

For a given analysis the program selects three criteria to govern the solution and assigns them target values from the data. The program then chooses a strain plane to act over the section with the form:

$$\varepsilon = \varepsilon_{ax} + \kappa_{yy}.z' + \kappa_{zz}.y'$$

This plane is then used to generate the stress distribution over the section in accordance with the acting material stress-strain assumptions and locked in strain data (eg pre-stressing). From this, values for the criteria are calculated and compared with the target values. If the difference between the target and calculated values for the criteria is greater than the program's tolerance another strain plane has to be chosen.

The above process is repeated until a strain plane is found which generates values of the criteria which are within the program's tolerance of the target values. Each new strain plane is selected by calculating (using a finite differences approximation) the partial differentials of the criteria with respect to the strain plane parameters.The various search conditions depend on the type of analysis selected.

The solution process can be viewed by selecting the *Interactive Solution* option in the Analysis Wizard.

The two broad categories and strength calculations, looking at the ultimate strength of the section and serviceability calculations looking at the section in its normal working state.

The strength analysis either uses material factors (e.g. BS 8110, BS 5400, Hong Kong and EC2) or strength reduction factors (ACI 318 and AS 3600)

The serviceability analysis allow for both long term and short term loading with long term loads considering the effect of creep. In addition for BS 5400 analysis the is an intermediate term option and for a subset of design loads there is an option for long and short term analysis.

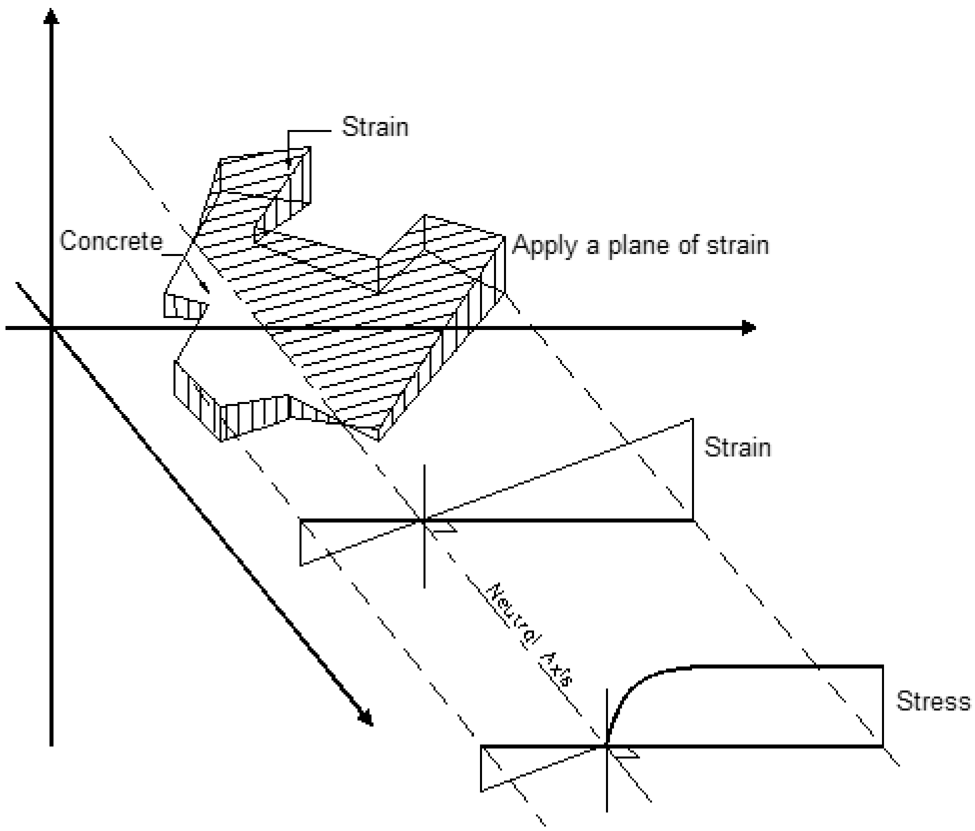
7.2 Search Conditions

The following table shows the criteria chosen for each type of analysis available in AdSec.

Analysis	**Solution criteria**		
	1	*2*	*3*
Ultimate Capacity	Axial force	Moment angle	Maximum concrete or steel strain : failure ratio
N/M Charts – no tension	Moment angle	Strain condition	Maximum concrete strain : failure ratio
N/M Charts – balanced yield	Moment angle	Strain condition	Maximum concrete strain : failure ratio
N/M Charts – chart coordinate	Moment angle	Dummy variable	Maximum concrete or steel strain : failure ratio
SLS & ULS Loads	Axial force	Moment angle	Applied moment
Cracking Moment	Axial force	Moment angle	Cracking strain

Sections can be analysed either as subject to biaxial bending, or restrained to bend uniaxially. Conversion is allowed between the two within the program. ULS M_{yy}/M_{zz} charts are only available for biaxially bending sections.

Uniaxially bending sections are analysed with the neutral axis angle fixed parallel to the Y-axis. Bending moments about the Z-axis are ignored (eg. downstand 'L' beam which is monolithic with the slab). Asymmetric, unrestrained sections should be analysed as biaxially bending even if the applied moments are about the Y-axis only. Analysis with axial force is optional for beams and columns.

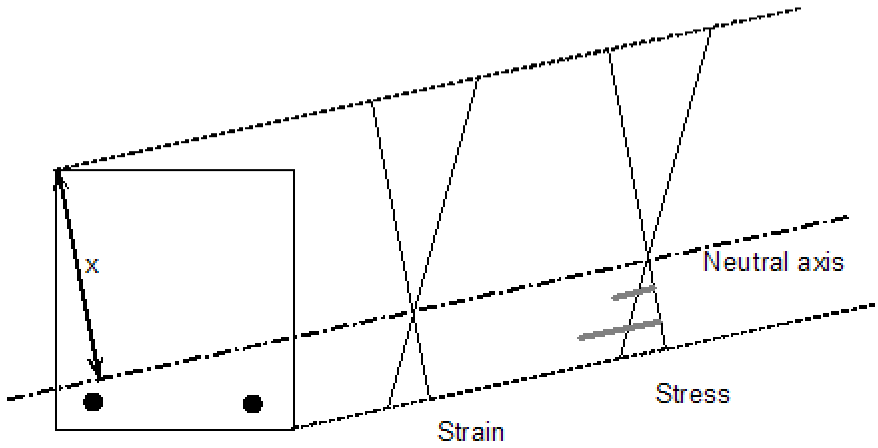


Finding a strain plane

SLS Loads Analysis

The material properties are typically linear (compression) and ICE TN 372 (tension). AdSec iterates until:

- axial force = N
- applied moment = M
- resultant moment orientation = θ



Stress and strain conditions for a loads analysis

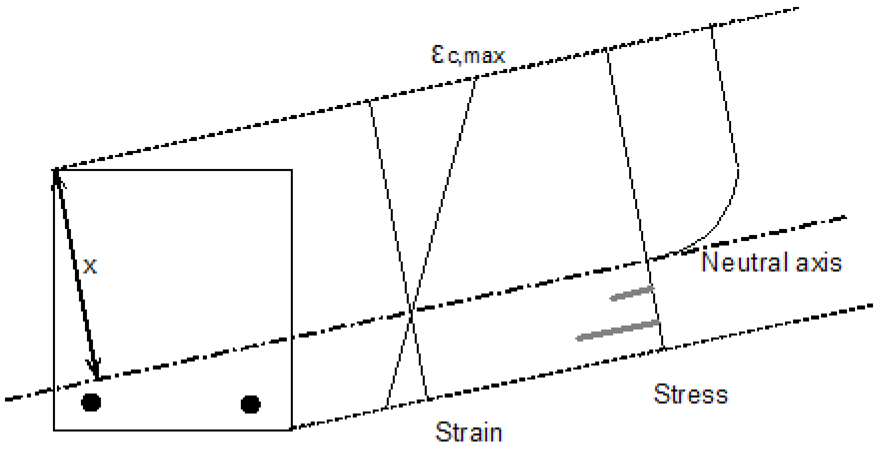
ULS Strength Analysis (concrete governing)

The material properties are typically recto-parabolic (BS8100, BS5400), parabola-rectangle (EC2) or rectangular (ACI and AS) (compression) and no tension. AdSec iterates until:

- axial force = N
- resultant moment orientation = θ
- strain ratio = 1

where the strain ratio is defined as

$$\frac{\varepsilon_{c,max}}{\varepsilon_{limit}}$$



Stress and strain conditions for a capacity analysis

7.3 Long and Short Term Analysis

"Long and short" term analysis is an option in AdSec to understand the serviceability behaviour of sections. This is available for BS8110 and Hong Kong codes of practice.

Loading is defined in two stages. Firstly long term loading, combined with a creep factor and then an additional short term loading.

In some circumstances, the long term loading is a permanent or quasi permanent loading, and the short term loading is an extreme event that happens after an extended period of time. However in many cases short term loading will occur intermittently throughout the life of the section. The long and short term analysis option in AdSec will model the second case.

- Firstly the cracking moment is calculated assuming the total long & short term axial load and moment direction, and short-term material properties.
- Secondly the strains and stresses are found for the long term loads, and long term material properties. These strains are used to calculate the creep effects of long term loads where creep strain is

$$\varepsilon_{creep} = -\varepsilon_{long} \frac{\phi}{(1+\phi)}$$

In this analysis the BS8110 Pt2 tension curve will use 0.55N/mm^2 as the maximum stress, if the section was deemed cracked under the total load. This will model the conservative assumption that, if cracked, this happened at an early stage of the section's life.

- The cracking moment is then recalculated, for the total load including the creep strain in the concrete calculated above. This will have the effect of slightly reducing the cracking moment if a compressive force has been acting on the section for a long time. This is the case, because the stress in the concrete will have reduced as the concrete creeps and more stress is transferred to the reinforcement.
- Finally a short term analysis is performed for the total loads, using short term material properties and the calculated creep strain to include for the long term effects.

Note that if the same process is followed manually using sequential AdSec analyses the initial cracking moment will be calculated from the long term load only. This will give different results than the automated AdSec 'long and short' term analysis in a small number of cases. The cases affected are where the BS8110 Pt2 tension curve is selected, and the section is cracked under total load, but uncracked under the long term load, and the stress under long term load is between 0.55 and 1.0 N/mm^2 at the centroid of tension steel.

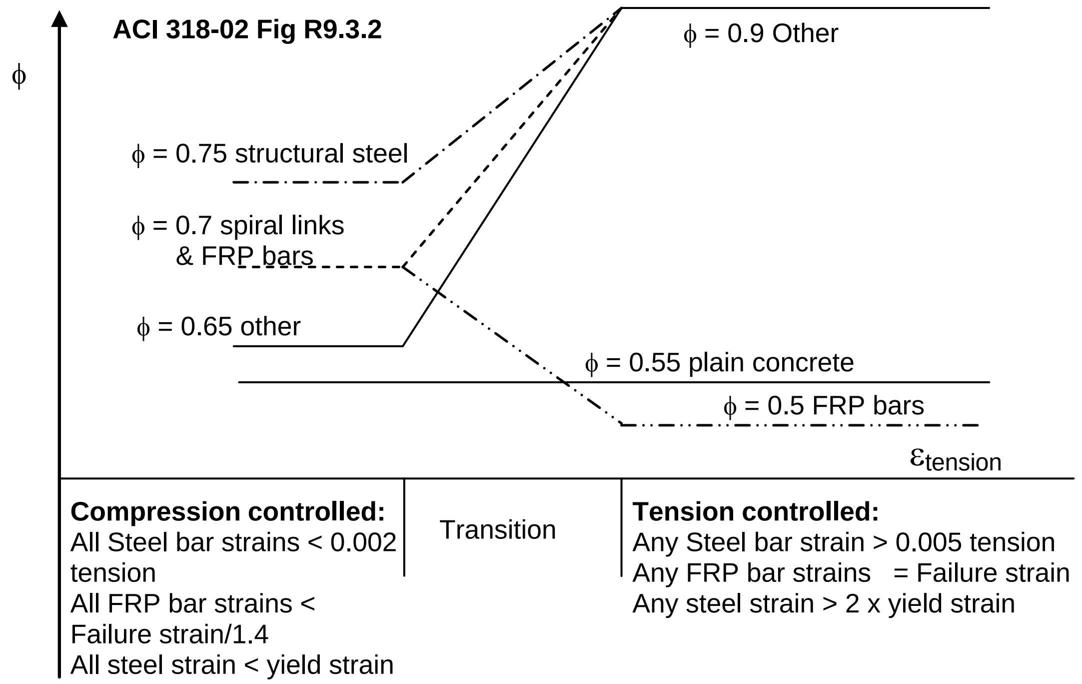
7.4 Strength Reduction Factors

The ACI and AS design codes use strength reduction factors (ϕ) rather than partial safety factors.

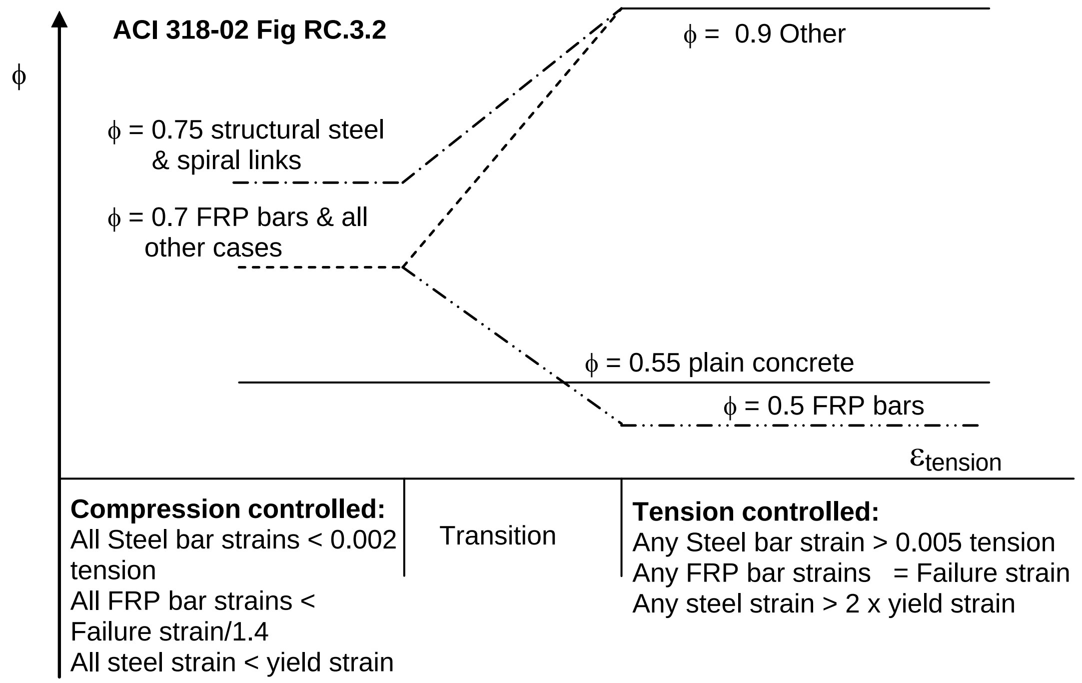
ACI Strength Reduction Factors

For ACI there are two alternative methods of calculating ϕ: there is the method set out in section 9.3.2 and a different method set out in C.3.2. For sections with FRP ACI 440.2 R2 sets out an alternative method.

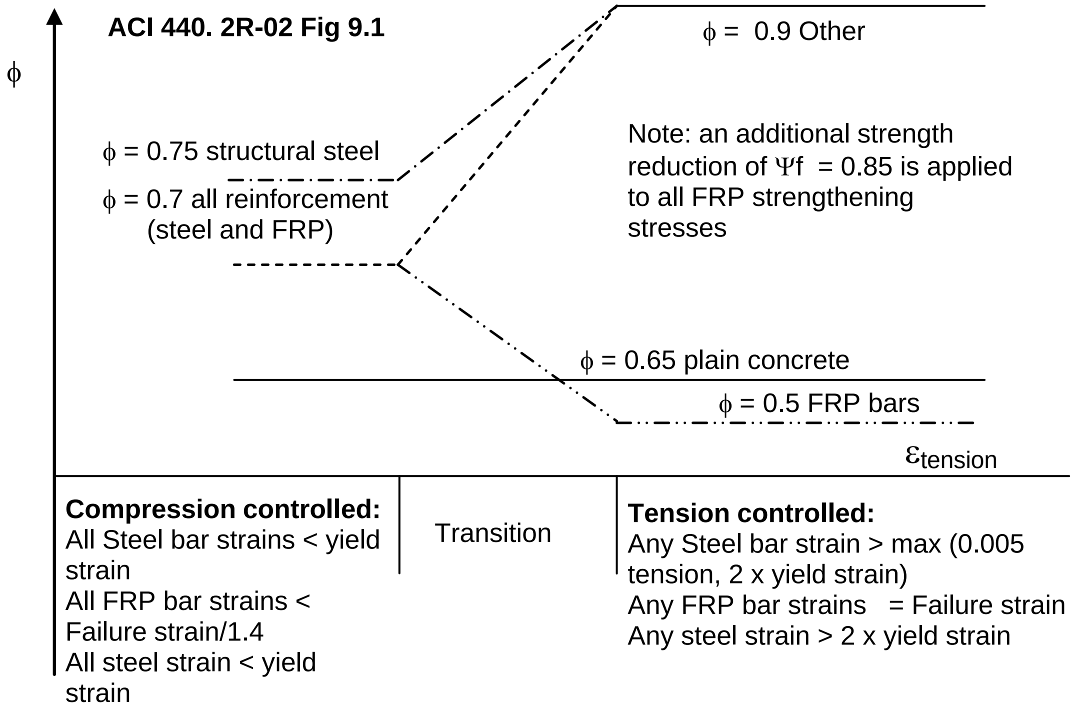
The following diagrams show the curves use in AdSec.



This is the standard curve used in ACI 318.



This is an alternative to the curve above but noted that this should be used in conjunction with different factored loads.



AS3600 Strength Reduction Factors

The strength reduction factors are given in Section 2.3 and Table 2.3. Where reinforcement controls the failure

$$\phi = 0.8$$

where compression controls the failure

$$\phi = 0.6$$

For general sections with combined (biaxial) bending and compression a general rule is needed. The code rules are generalised as follows:

$$r = \frac{\sum F_i(\varepsilon_i / \varepsilon_{yi})}{\sum F_i}$$

where

F_i = force in bar

ε_i = strain in bar

ε_{yi} = yield strain in bar

Only bars in tension are included in the calculation. Then ϕ is calculated as follows:

$$\phi = 0.6 \qquad r \le 1$$
$$\phi = 0.8 \qquad r \ge 2$$
$$\phi = 0.6 + 0.2(r - 1) \qquad 1 < r < 2$$

So ϕ = 0.6 for plain concrete.

7.5 Concrete Materials

There are a range of material models available to the AdSec user. The particular material models offered depend on the currently selected design code and material type (concrete, rebar, etc.). Where a suitable material curve does not exist the user has the option of defining a stress-strain curve directly.

Strength analysis

For strength (ULS) analysis AdSec offers a number of stress/strain options. The stress/strain options offered depend on the design code selected: compression and tension curves are specified separately. The main stress.strain curves offered are:

- recto-parabolic (compression) – BS and HK , (BS8110-1 Figure 2.1 and BS5400BS5400-4 Figure 1)
- parabola rectangle compression – EC2 (similar to recto-parabolic)
- rectangular (compression) – ACI and AS
- bi-linear (compression) – EC2
- no-tension – all

but other curves can be selected.

Serviceability analysis

For the serviceability limit state (SLS) there is a choice of compression curves (depending in the design code) from linear, BS8110-2 Figure 2.1 and BS8110-1 Figure 2.1 recto-parabolic; and tension curves from BS8110-2 Figure 3.1, ICE Note 372 and zero tension.

Often concrete serviceability analysis is based on elastic section properties of the cracked or concrete section, and the results are factored by (1+ ϕ) to take account of creep. This is a simplistic approach as

- concrete behaves as a non-linear material in both tension and compression
- concrete has different stiffness behaviour in tension and compression
- concrete elements are generally loaded with a combination of long term, medium term and short term loading and unloading stages

The following sections look at the available models for concrete stiffness in tension and compression, and how they can be interpreted and used.

AdSec approaches serviceability stiffness analysis by searching for a strain plane. The strain plane is used to calculate stresses based on the stress strain relationships below. These are integrated to give forces and moments. When the resulting forces and moments are in equilibrium with the analysis task set, the solution is said to have converged.

If concrete is subjected to sustained loading (long term loading), it will creep. The steel is assumed not to creep. This means that the strain will increase without the stress increasing. The amount of creep depends on a number of factors, and creep calculations are covered in most concrete codes of practise. The amount of creep is quantified using a creep factor, ϕ, and this is used in calculations to calculate total stiffness, strain, curvature or deflections by factoring by

$$(1+\phi)$$

AdSec adjusts the concrete stress-strain relationships to include the effect of creep before it starts the iterative search process. Creep can also be modelled in AdSec using strain planes applied to the concrete alone.

7.5.1 Concrete Compression Curves

The main stress-strain relationships for concrete in compression included in AdSec:

- linear elastic stress strain relationship
- recto-parabolic stress strain relationship
- rectangular curve
- bi-linear (Eurocode 2 only)
- peaked curve

See also:

BS8110 Concrete and Reinforcement Materials

BS5400 Concrete and Reinforcement Materials

EC2 Concrete Materials

7.5.1.1 Linear Elastic Material

Here the stress, *f*, in the concrete is proportional to strain, ε.

$$f = E\varepsilon$$

Values for E are given in the codes of practice. This is a secant modulus approximately equal to the initial modulus of the peaked curve/1.12. Using these values of E will not give good accuracy above a stress of 0.4 f_{cm} where f_{cm} is the peak stress of the peaked curve.

The following formulae can be used to calculate the short term modulus in GPa from the concrete strength in MPa

In BS8110 (Ref: BS8110-2 equation 17)

$$E_{28} = 20 + 0.2 f_{cu,28}$$

In BS5400 values are given in Table 3 with no formulaic version. An approximate fit to the values in the table gives

$$E = 19 + 0.3 f_{cu}$$

This is used in AdSec.

The Hong Kong 2004 code gives the relationship (3.1.5)

$$E = 3.46\sqrt{f_{cu}} + 3.21 \quad \text{in MPa}$$

In EC2 the modulus is given as (Ref Table 3.1)

$$E_{cm} = 22\left(\frac{f_{cm}}{10}\right)^{0.3}$$

$$f_{cm} = f_{ck} + 8$$

where f_{ck} =cylinder strength

In ACI the modulus is given in 8.5.1 as

$$E_c = w_c^{\ 1.5} 33\sqrt{f_c'} \quad \text{in psi}$$

$$E_c = w_c^{\ 1.5} 0.043\sqrt{f_c'} \quad \text{in MPa}$$

In AS the modulus is given in 6.1.2 as

$$E_c = \rho^{1.5} 0.043\sqrt{f_{cm}} \quad \text{in MPa}$$

To modify these curves for lightweight concrete:

- BS8110 – multiply E by $(\rho /2400)^2$ ref BS8110 Pt 2, 7.2
- BS5400 – multiply E by $(\rho/2300)^2$ ref BS5400 Pt 4, 4.3.2.1
- EC2 – multiply E by $(\rho/2200)^2$ ref EC2 11.3.1.2 equation 11.2

In all the codes above, the change in stiffness under sustained loading can be dealt with by reducing the modulus of elasticity as follows

$$E' = \frac{E}{1 + creep\,coeff.}$$

This approach for assessing serviceability behaviour is acceptable in BS8110, BS5400 and EC2.

In BS5400, an intermediate value of E is recommended. This will be between the short term and the crept value of E depending on the ratio of long and short term loading.

Ref BS5400 Pt 4 clause 4.3.2.1 and Table 3. This is included in AdSec as:

$$E_{inter} = E_{short}\left[\frac{M_q}{M_g}\right] \Big/ \left[1 + \frac{M_q}{M_g}\right] + E_{long}\, 1 \Big/ \left[1 + \frac{M_q}{M_g}\right]$$

$$E_{short} = E$$

$$E_{long} = \frac{E}{[1+\phi]} = \frac{E}{2}$$

7.5.1.2 Recto-Parabolic Material

This curve is given in a number of codes for use in ultimate design calculations.

This is typified by:

- Figure 2.1 in BS8110 Pt 1, transition strain 2.44e-4$\sqrt{(f_{cu}/\gamma_m)}$, plateau stress 0.67 f_{cu}/γ_m
- Figure 1 in BS5400 Pt 4, transition strain 2.44e-4$\sqrt{(f_{cu}/\gamma_m)}$, plateau stress 0.67 f_{cu}/γ_m
- Figure 3.3 in EC2, transition strain 0.002 (but see below), plateau stress $f_{cd} = \alpha_{cc} f_{ck}/\gamma_c$ For serviceability analysis $f_{cd} = f_{cm}$.

For BS8110 and BS 5400 the recto-parabolic curve is modified for high strength concrete (f_{cu}>60) according to Concrete Society Technical Report TR49 'Design Guidance for high strength concrete'. This has the effect of changing the maximum strain limit.

Specifically : 3.1.3 TR49 for f_{cu}>60N/mm² $\varepsilon_{cu} = 0.0035 - (f_{cu} - 60) / 50000$

EC2 modifies the curve in 3 ways when strength exceeds 50N/mm². Namely modifying the power of the initial curve, the transition strain from curve to plateau, and the ultimate strain. (EC2 Table 3.1)

It should be noted that this curve was developed for ultimate strength analysis and is therefore designed for use when the strain at the extreme compressive fibre approaches the ultimate strain limit (0.0035 for BS8110). This curve has a similar area underneath it as more accurate models of concrete in compression, so it is a useful simplification in this situation. It is not recommended in codes of practice for serviceability analysis, but may be a useful approximation when investigating the behaviour of concrete elements approaching capacity if, for example, there is a problem with convergence using the peaked curve.

7.5.1.3 Rectangular Stress Block

This curve is given in a number of codes for use in ultimate design calculations. The assumption is that a uniform stress exists over most of the distance from the extreme fibre to the neutral axis. The remaining part of the sections sees zero stress. This is characterized by a stress-strain curve which has zero strength over the first part of the curve with a step change at a transition strain which will be determined differently for each code.

This stress strain relationship is commonly used for strength analysis but is aimed at hand calculation for simple section shapes. It is likely to be a less reliable choice for more complex sections shapes or compound sections.

This is the stress-strain implied by Figure 3.3 in BS8100, Figure 3.5 in EC2, Clause 10.2.7 in ACI318 and Clause 8.1.2.2 in AS3600.

7.5.1.4 Bi-linear Material

This curve is given in EC2 for use in ultimate design calculations.

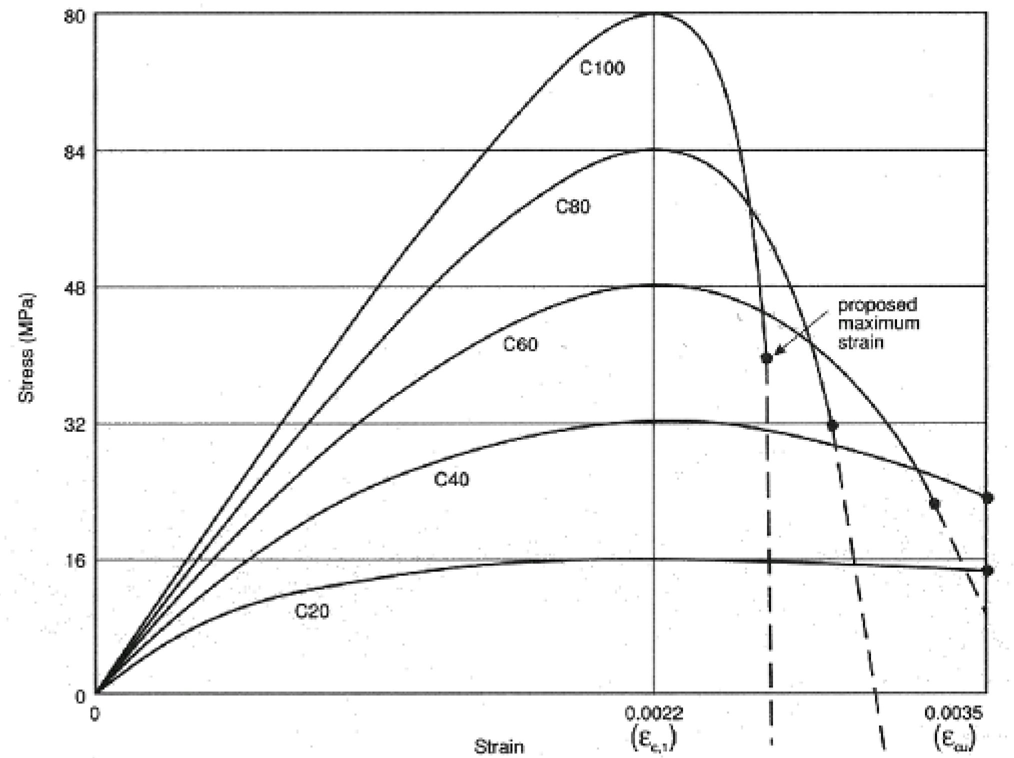
This is typified by Figure 3.4 in EC2, with a transition strain of 0.00175 (but see below), plateau stress $f_{cd} = \alpha_{cc} f_{ck}/\gamma_c$ For serviceability analysis $f_{cd} = f_{cm}$.

EC2 modifies the curve in two ways when strength exceeds 50N/mm². Namely modifying the transition strain from curve to plateau, and the ultimate strain. (EC2 Table 3.1)

It should be noted that this curve was developed for strength analysis and is therefore designed for use when the strain at the extreme compressive fibre approaches the ultimate strain limit (0.0035 for BS8110). This curve has a similar area underneath it as more accurate models of concrete in compression, so it is a useful simplification in this situation. It is not recommended in codes of practice for serviceability analysis, but may be a useful approximation when investigating the behaviour of concrete elements approaching capacity if, for example, there is a problem with convergence using the peaked curve.

7.5.1.5 Peaked Curve

Many codes of practice include a stress strain curve for serviceability analysis which peaks at approximately 0.0022 strain and then falls away towards the ultimate value.



Concrete stress-strain curve for serviceability analysis

The form of the curve presents particular difficulties for direct integration. In the windows version of AdSec, the curve is approximated to 6 parabolic curved segments. The peak stress point is explicitly used in the derivation of these segments. Beyond the ultimate strain value, the curve is treated as a horizontal line, to allow the iterative procedure to work. However final solutions which take the strain beyond the ultimate strain will result in a flag that the design is not safe.

BS8110

An example is the curve in BS8110-2 Figure 2.1. In BS8110 the peak strain is set at 0.0022.

There are 2 limits to the applicability of the curve given in BS8110

- k must be >1 where $k = 1.4\varepsilon_{c1} E / f_{cu}$
- strain must be < $\varepsilon_{cu} = \varepsilon_{c1} [k + 2 + \sqrt{(k^2 + 4k - 4)}] / 4$ or the ultimate strain for the concrete

These limits are not given in BS8110 but are numerical limitations of the equations presented. If these conditions are not met AdSec will not allow this curve to be used.

To allow for creep under sustained loading two changes are needed.

$$E = E_0/(1+\phi)$$

and

$$\varepsilon_{c1} = \varepsilon_{c1}(1+\phi)$$

Eurocode 2

This form of curve is copied in 3.1.4 EC2 fig 3.2 and in this case the peak strain varies with the concrete grade from 0.0018 to 0.0029. (ref Table 3.1 'Strength and deformation characteristics for normal concrete'). The max stress at the peak of the EC2 curve is $f_{cm} = f_{ck} + 8$. This is because serviceability analysis is generally concerned with mean effects rather than characteristic values.

7.5.1.6 Confined and Explicit Materials

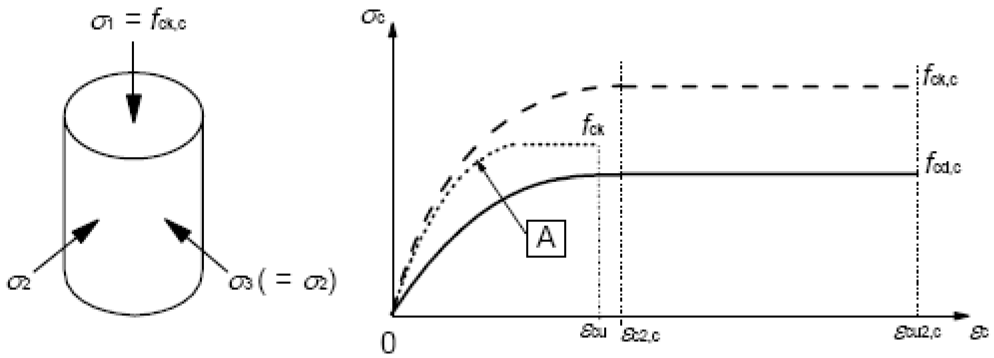
The effective confinement of the concrete is achieved by either confining the whole section with steel or FRP, or, by taking into account of the confinement generated by the three-dimensional arrangement of links and main bars. The concrete ‘arches’ between the bars.

Confined compression concrete models in AdSec can be specified in three ways. The choices available depend on the selected code of practice. The relevant confined curves are:

- EC2 confined concrete
- Explicit stress strain coordinates
- Mander curves.

EC2 Confined concrete

EC2 has a confined concrete model which is a modification of the recto-parabolic curve. The effect of the confinement is to increase the effective concrete strength and the strain to failure as show in the figure below (EC2 Figure 3.6) where A is the unconfined curve.



The relationship between confined and unconfined curves is:

$$f_{ck,c} = \begin{cases} f_{ck}(1+5\sigma_2/f_{ck}) & \sigma_2 \leq 0.05 f_{ck} \\ f_{ck}(1.125+2.5\sigma_2/f_{ck}) & \sigma_2 > 0.05 f_{ck} \end{cases}$$

and

$$\varepsilon_{c2,c} = \varepsilon_{c2}\left(f_{ck,c} / f_{ck}\right)^2$$
$$\varepsilon_{cu2,c} = \varepsilon_{cu2} + 0.2\sigma_2 / f_{ck}$$

Explicit stress-strain coordinates

Explicit stress-strain coordinates can be entered for both SLS and ULS analysis. These are stored as a table of values and are normally displayed a list stress-strain values (ε_1,σ_1), (ε_2,σ_2), ...

Mander Curves

Confined and unconfined concrete stress strain relationships are proposed by Mander et al in 'Theoretical stress-strain model for confined concrete' J.Structural Engineering, ASCE, 1988 114 (8), p 1804-1849

Much of the Mander paper is concerned with defining the effectiveness of the confinement for rectangular or circular link arrangements. This needs to be calculated separately because AdSec does not have the full 3D geometry of reinforcement, being concerned only with the section analysis.

However, given a few parameters it can generate the stress strain curve in the Mander paper. One difficulty is predicting the ultimate strain ε_{cu}. It has been found that typically ε_{cu} is in the range 0.04 to 0.05. However the UK Concrete Society Technical report 55 Chapter 8 suggests that the value should be limited to 0.01. It states *"It should be noted, however, that at concrete compressive strains of over approximately 0.01, the concrete will have been crushed and lost all cohesion"*. Other sources limit the strain to 0.02.

When the strains in the section pass a certain value ε_{sp} the unconfined concrete in the cover zone will spall off. AdSec models this by including an *unconfined Mander curve*, in which the stress drops to zero at the point of spalling, and then continues at zero until it reaches the ultimate value for the confined zone. This model will then not limit the calculation of ULS strength, to the strain in the cover zone.

Unconfined Mander Curve

The following variables are needed to define an unconfined Mander curve:

		default
Cylinder strength	f_{co} (in American terminology, f_{ck} in EC2 terminology)	
Unconfined strain at peak	ε_{co}	0.002 (< ε_{sp})
Spalling strain	ε_{sp}	0.005 (< ε_{cu})
Confined ultimate strain	ε_{cu}	0.01

The stress strain curve is then generated using the Mander relationship given below using the following substitutions for $\varepsilon < 2\,\varepsilon_{co}$.

$$\varepsilon_{cc} = \varepsilon_{co}$$
$$f_{cc} = f_{co}$$

From ε_{co} the relationship is linear to the point (ε_{sp}, 0). From then on the stress is constant and zero until $\varepsilon = \varepsilon_{cu}$

Confined Mander Curve

The following variables are needed to define a confined Mander curve:

		default
Cylinder strength	f_{co} (in American terminology, f_{ck} in EC2 terminology)	
Confined strength	f_{co}	(> f_{co})
Unconfined strain at peak	ε_{co}	0.002 (< ε_{sp})
Confined ultimate strain	ε_{cu}	0.01

$$\varepsilon_{cc} = \left[5\left(\frac{f_{cc}}{f_{co}} - 1\right) + 1\right]\varepsilon_{co}$$

$$E_{sec} = \frac{f_{cc}}{\varepsilon_{cc}}$$

$$r = \frac{E_c}{E_c - E_{sec}}$$

E_c is the normal tangent modulus (eg ref ACI 318 Section 8.5.1)

Then for varying strain ε, the stress f_c can be calculated as follows:

$$x = \frac{\varepsilon}{\varepsilon_{cc}}$$

$$f_c = \frac{f_{cc} x r}{r - 1 + x^r}$$

7.5.2 Concrete Tension Curves

Concrete exhibits a 4 phase behaviour in response to tension stresses.

- Low tension stress – concrete tension stiffness similar to compression
- Cracking starts – stiffness drops off as cracks form
- Cracks formed, cracks open up – stiffness drops off more rapidly as cracks open up
- Fully cracked – no residual stiffness left

This behaviour is complex as it is controlled by the reinforcement. The simplified means prescribed to deal with these phenomena vary from code to code.

All codes state that ultimate strength analysis and design should ignore the tension stiffening from the concrete. All codes will accept fully cracked section properties as a lower bound on stiffness.

Serviceability analysis is usually performed for stiffness, stress/strain checks, or crack width checks. Some codes imply a different tension stiffening method for crack width as opposed to the other checks. This may lead to a disparity in AdSec results between the "cracking moment" and the moment at which the crack width becomes > zero.

The code rules are developed for a rectangular section with uniaxial bending and one row of tension steel. However the rules are not extended to irregular sections made up of various zones of concrete, some with locked in strain planes. Because the tension stiffening is a function of the amount of 'damage' / cracking in the section, adjoining tensile zones need to be considered in evaluating the tension strength of a zone, as these may contain steel which will control the cracking.

BS8110 Pt 2 presents a stress/strain 'envelope' which provide means of calculating an effective tensile Young's modulus for a linear tension stress/strain curve.

ICE Technical note 372 presents a more sophisticated envelope approach than BS8110 and is offered as an option in AdSec.

BS5400 presents the same approach as BS8110 in Appendix A for stiffness calculations. But this is rarely used. Instead the main body of the code gives a crack width formula based on strains from an analysis with no tension stiffening. The crack width formula itself includes some terms to add back in an estimate of the contribution from tension stiffening. Ref BS5400 5.8.8.2 equation 25.

EC2 proposes 2 analyses, one with full tension stiffness and one with none. The final results are an interpolation between these results.

Recent research about the cracked stiffness of concrete has shown that the tension stiffness measured in the laboratory can only be retained for a very short time. This means that both the tension stiffening given in BS8110 and TN 372 is un-conservative for most building and bridge loadings. AdSec includes these findings for BS8110 and will give a smaller tension stiffness than previous versions.

This research is outlined in th UK Concrete Society TR 59 "Influence of tension stiffening on deflection of reinforced concrete structures". This report also compares UK, American and European codes.

See also:

BS8110 Pt2 Tension Stiffening

BS5400 Tension Stiffening

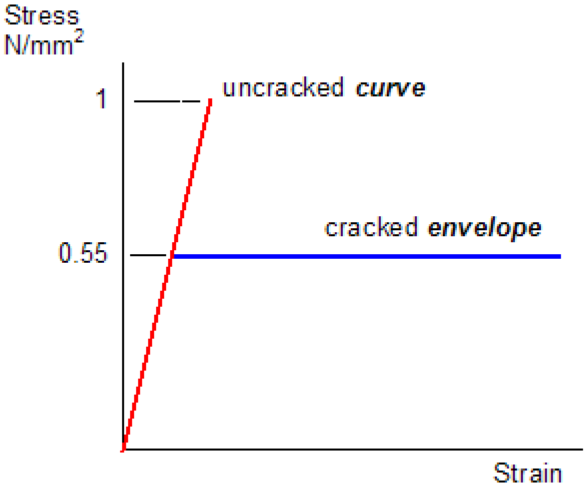
ICE Technical Note 372 Tension Stiffening

EC2 Tension stiffening & crackwidth calculation

ACI Tension Stiffening

7.5.2.1 BS8110-2 Tension Stiffening

BS8110-2 Figure 3.1 shows tension stiffening assumptions for calculating curvatures. This is described in BS8110-2, clause 3.6 (4). The tension stiffening is based on the concrete strain at the level of the centroid of the tension steel. The concrete strain will ignore prestress in the steel, but will include concrete only strain planes (eg shrinkage, creep). The concrete is assumed to have the same modulus for tension and compression until a limiting strain is reached. Then the maximum concrete stress at the centroid of tension steel remains constant, regardless of the strain. The shape of the stress block over the tension zone is triangular. AdSec implements this relationship by calculating an effective tensile Young's modulus for every strain plane on every zone during the iterative solution process as described below.



BS8110-2 stress-strain relationship

AdSec includes an interpretation of the BS8110 Pt 2 envelope which takes account of the latest research about cracked stiffness, combined with a new interpretation of the centroid of tension steel for composite sections.

This research is outlined in th UK Concrete Society TR 59 "Influence of tension stiffening on deflection of reinforced concrete structures". This report also compares UK, American and European codes.

The cracking moment is calculated using the total axial force and moment angle from short and long term loading and the short term concrete modulus. The cracking moment is the moment at which the maximum strain at the centroid of force of tension steel is 1.0 /E_0. This strain condition is checked on a component by component basis.

For composite sections, once the cracking moment calculation has converged, the strain conditions are rechecked. If the centroid of force of tension steel is not in the zone which it is controlling a warning message is generated "WARNING: AdSec had to extrapolate BS8110-2 tension stiffening rules for this section and loading. Please review results". This does not necessarily mean that the result is wrong. But *does* mean that the stress distribution across the section should be reviewed and engineering judgement used to decide if it is rational.

If the total moment (long and short term) is greater than the cracking moment, the limiting strain in the section is calculated from 0.55N/mm² for long or short term analysis. This is because the stress of 1.0N/mm² is achievable for only a very short time.

If the total moment is less than the cracking moment, the effective tensile modulus is the same for tension as compression. The change in stiffness under sustained loading is dealt with by reducing the modulus of elasticity of the initial slope of the envelope as follows

$$E = E/(1+\phi)$$

For composite sections, once the serviceability analysis has converged, the strain conditions are rechecked. If the centroid of force of tension steel is not in the zone which it is controlling a warning message is generated "WARNING: AdSec had to extrapolate BS8110-2 tension stiffening rules for this section and loading. Please review results"

The Part 2 tension curve in BS8110-2 was devised for typical beam bending situations. The assumption is that the centroid of tension steel is closer to the extreme tensile fibre of the concrete than it is to the neutral axis. If a section has an axial force present, or steel in the centre of a thin section, these assumptions are not always appropriate. The curve has been adjusted for these circumstances as follows

No tension steel found

If no tension steel is found, but there is some curvature and concrete in tension, the BS8110-2 tension stress strain 'envelope' is used as a stress strain relationship. This means that the maximum tension stress achievable in the tension zone is 1.0N/mm² (0.55N/mm² if cracked or long term). The stress will increase linearly up to this value and then remain constant.

Steel very close to neutral axis

If tension steel is found but the neutral axis is very close to the centroid of tension steel, it is possible for stresses at the extreme fibre of the section to be higher than the tension capacity of the concrete.

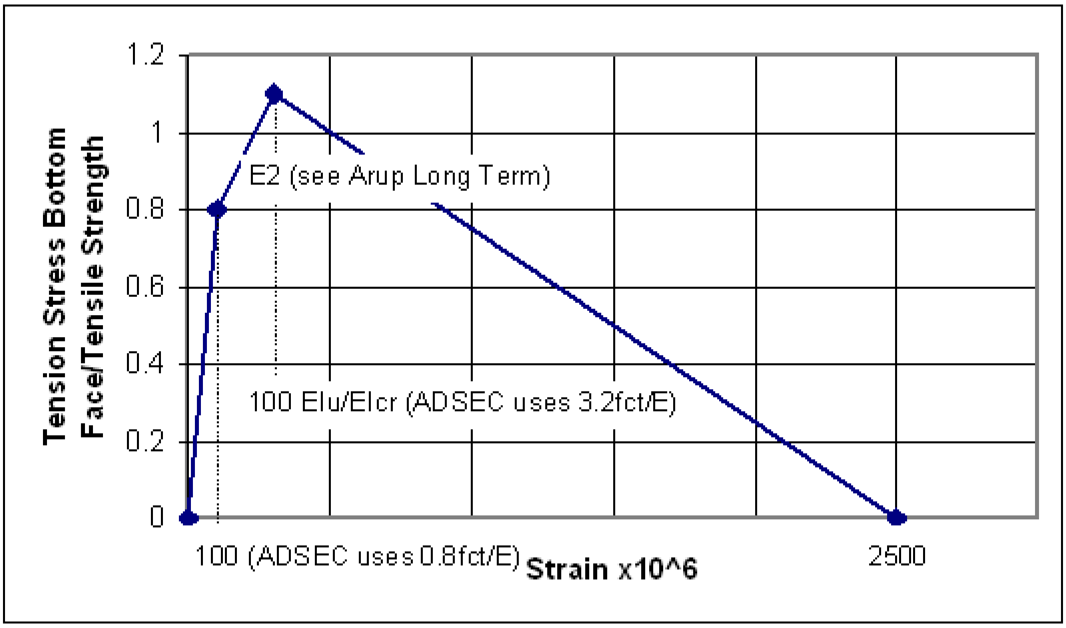
In these circumstances, AdSec may not be able to converge, as the neutral axis may move over the tension steel, resulting in large and sudden changes in stiffness. If convergence is hard to achieve, try using one of the other tension stiffening models, which may be more appropriate.

When convergence is achieved with the BS8110-2 curve, AdSec will check the extreme fibre of all the component sections. If the extreme fibre stress is greater than 1.25N/mm² a warning message is generated. The user will be prompted to use an alternative tension stiffness relationship.

7.5.2.2 BS5400 Tension Stiffening

The BS8110 tension stiffness envelope described above is valid for BS5400 ***stiffness*** analysis, but not for ***crack-width*** checks. For this reason, it is not normally used. To calculate crack-widths to BS5400 a fully cracked section is assumed in the calculation of curvature and strain. The BS5400 crack-width formula includes explicit terms which add back in an approximation for the effect of tension stiffening. Therefore in AdSec 'No Tension' should be selected in the materials table when calculating crack-widths to BS5400.

7.5.2.3 ICE Technical Note 372 Tension Stiffening

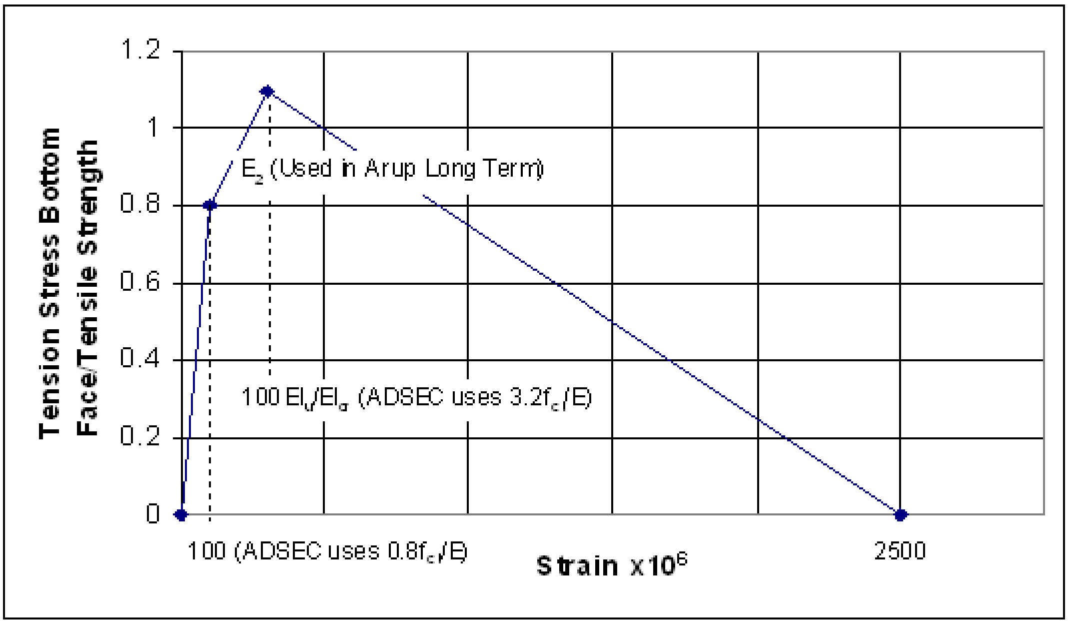


ICE Technical Note 372

Durham University developed a more sophisticated approach to the BS8110 tension stiffening envelope. This is described in the Institution of Civil Engineers Technical note 372. This is referred to in AdSec as TN372. The curve shows the 4 stages of cracking described above.

The envelope differs from the BS8110 approach as it is based on the strain at the most tensile fibre of the section rather than the centroid of tension steel. The *envelope* above gives the relationship between stress and strain at the extreme fibre of the section or zone. The *distribution* of stress from the neutral axis to the extreme fibre is then assumed to be linear (giving a triangular stress block).

This basic curve has been developed by further research which is reported in the Concrete Society TR 59. The developments in TR 59 have been adopted into AdSec and are shown in the following figures:



ICE 372 Short-term tensile stress at tension face related to strain

f_{ct} = concrete ultimate tensile stress

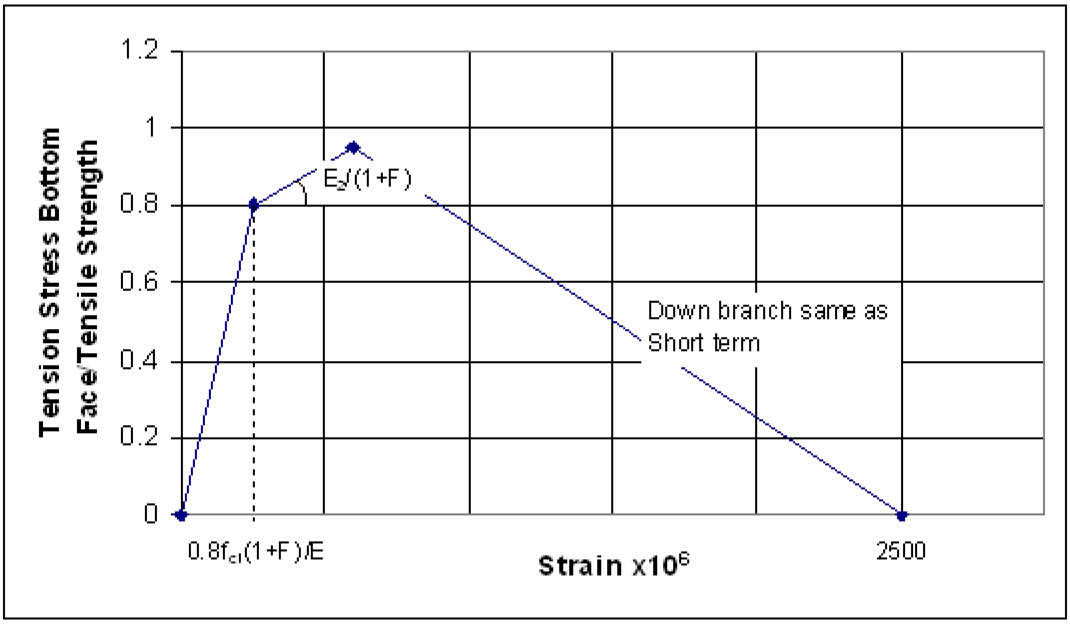
E = instantaneous Young's Modulus

E_2 = effective stiffness between first and last crack

EI_u = stiffness of un-cracked section

EI_{cr} = stiffness of fully cracked section

The approach appeared to address the shortcomings of the BS 8110 approach and was therefore incorporated as an option into AdSec. The paper did not discuss long-term properties and so the following relationship was developed within Arup to address this:



Modified ICE 372 Long-term tensile stress at tension face related to strain.

ϕ = Creep coefficient.

7.5.2.4 EC2 Tension Stiffening & Crackwidth Calculation

EC2 tension stiffening is described in Eurocode 2 section 7.4.3 equation 7.18. EC2 differs from BS8110 in that it does not have a specific tension stiffening relationship used in analysis. Instead, two analyses are carried out assuming cracked and uncracked stiffness values, and the actual curvature and stiffness is an interpolation between the two results based on the amount of cracking predicted.

The cracking moment, M_{cr}, is defined as the moment when the stress in the outer most tensile element of an uncracked concrete section has reached f_{ctm}

The tension stiffening options offered for EC2 in AdSec are zero tension, linear tension, and interpolated. 'Zero-tension stiffness' will give a conservative, fully cracked lower bound. The 'linear-tension stiffening' uses the Young's modulus of the concrete to produce a linear stress–strain relationship. This is for checking of the other results only and it is not appropriate to use this beyond the cracking moment. Note that the values in EC2 for servicability are based on mean concrete properties rather than the characteristic values used for ultimate analysis and design. The interpolation depends on the amount of damage sustained by the section. This is calculated by AdSec based on the proximity of the applied loading to the cracking moment. But for sections which have been cracked in a previous load event the minimum value of ζ for use in equation 7.18 can be input. The default value of ζ_{min} is 0. To take account of the fast drop in tension stiffening following cracking, the value of β in equation 7.19 defaults to 0.5

AdSec does not use equation 7.19 to calculate the damage parameter ζ. Instead ζ is calculated from the cracking strain

$$\varepsilon_{cc} = \frac{f_{ctm}}{E_{cm}}$$

and the most tensile strain ε_{uncr} in the section under an uncracked analysis under full applied load. The *E* used for to determine ε_{uncr} is short term. For composite sections ζ_{comp} is calculated for each component using the component material properties for ε_{cc}, and the most tensile strain on the component for ε_{uncr}. The highest value of ζ_{comp} will be used for ζ in stiffness and cracking calculations.

Note: engineering judgement should be used to assess if this approach fits the particular situation.

$$\zeta = 1 - \beta\left(\frac{\varepsilon_{cc}}{\varepsilon_{uncr}}\right)^2 \qquad \varepsilon_{uncr} > \varepsilon_{cc}$$
$$\zeta = \zeta_{min} \qquad \varepsilon_{uncr} < \varepsilon_{cc}$$

If the EC2 interpolation is selected for the tension stiffness at serviceability, the properties which depend on the average behaviour along the element (eg stiffness, curvature and crack widths) are based on the interpolated strain plane. However for moments greater than M_{cr}, the ***stresses*** output by AdSec for the interpolated tension stiffness are from the fully cracked analysis, because these represent the maximum stresses which occur at crack positions.

7.5.2.5 ACI Tension Stiffening

For ACI, two analyses are carried out assuming cracked and uncracked stiffness values, and the actual curvature & stiffness is an interpolation between the two results based on the amount of cracking predicted.

Clause 9.5.2.3 in ACI318 gives a method to compute a value of the second moment of area intermediate between that of the uncracked, I_g, and fully cracked, I_{cr}, values. The expression is:

$$I_e = \left(\frac{M_{cr}}{M_a}\right)^3 I_g + \left[1 - \left(\frac{M_{cr}}{M_a}\right)^3\right] I_{cr}$$

where M_{cr} is the cracking moment and M_a is the applied moment.

AdSec replaces expression (9–8) of ACI 318 with a first principles approach based on strain

$$x = \zeta x_{II} + (1-\zeta) x_I$$

where *x* is a strain function and x_{II} and x_I are the strain functions calculated for uncracked and fully cracked conditions respectively.

This is similar to the approach in EN1992 (7.18) which gives a strain plane intermediate to the uncracked and fully cracked strain planes.

It should also be noted that I_g is defined as the value of second moment of area ignoring the reinforcement. It is assumed that this definition was made for simplicity; AdSec's approach *includes* the reinforcement.

Let $\alpha = (M_{cr}/M_a)^3$, the uncracked curvature be κ_I and the fully cracked curvature be κ_{II},

For ACI318, the interpolated curvature

$$\kappa = 1 \Big/ \left[\alpha \big/ \kappa_I + (1-\alpha) \big/ \kappa_{II} \right]$$

The aim is to make this equivalent to

$$\kappa = \zeta \kappa_{II} + (1-\zeta)\kappa_I$$

Equating these two expressions gives:

$$\alpha\zeta\frac{\kappa_{II}}{\kappa_I} + \alpha(1-\zeta) + (1-\alpha)\zeta + (1-\alpha)(1-\zeta)\frac{\kappa_I}{\kappa_{II}} = 1$$

Which can be re-arranged to give

$$\zeta = 1 \Big/ \left[1 + \frac{\kappa_{II}/\kappa_I}{(1/\alpha - 1)} \right]$$

The ratio κ_I/κ_I is appropriate for uniaxial bending. For applied loads (N, M_y and M_z), and uncracked and fully cracked strain planes (ε_I, κ_{yI} and κ_{zI}) and (ε_{II}, κ_{yII} and κ_{zII}) respectively, κ_{II}/κ_I is replaced by the ratio

$$\frac{(N\varepsilon_{II} + M_y\kappa_{yII} + M_z\kappa_{zII})}{(N\varepsilon_I + M_y\kappa_{yI} + M_z\kappa_{zI})}$$

In the absence of axial loads, this ensures that the curvature about the same axis as the applied moment will comply with ACI318. In the absence of moments, the axial strain will follow a relationship equivalent to that in ACI318 but using axial stiffness as imposed to flexural stiffness. The proposed ratio is independent of the location chosen for the reference point.

The ratio (M_{cr}/M_a) is also inappropriate for general loading. For the general case, it is proposed that it is replaced by the ratio (f_{ct}/σ_{tI}), where f_{ct} is the tensile strength of the concrete and σ_{tI} is the maximum concrete tensile stress on the uncracked section under applied loads.

Summary:

$$\zeta = 1 \Big/ \left[1 + \frac{(N\varepsilon_{II} + M_y\kappa_{yII} + M_z\kappa_{zII}) / (N\varepsilon_I + M_y\kappa_{yI} + M_z\kappa_{zI})}{\left[(\sigma_{tI}/f_{ct})^3 - 1\right]} \right]$$

7.6 Rebar Materials

AdSec offers both steel and GFRP reinforcement.

Steel Rebar

There is less variation in the modelling of rebar compared with the modelling of concrete, reflecting the simpler stress-strain behaviour of steel. Most rebar stress-strain curves are bi-linear or tri-linear in form.

The standard steel rebar stress-strain curves for BS and Hong Kong code are BS8110-1 Figures 2.2 and 2.3 and BS5400-2 Figure 2. EC2 uses a similar curve in Figure 3.8 which allow the choice of elastic-perfectly plastic or elastic-linear hardening. ACI 318 defines a bilinear (elastic-perfectly plastic) curve in 10.2.4.

An f_y value must be entered for standard types. For BS5400 the expression for compressive stress limits the allowable value of f_y.

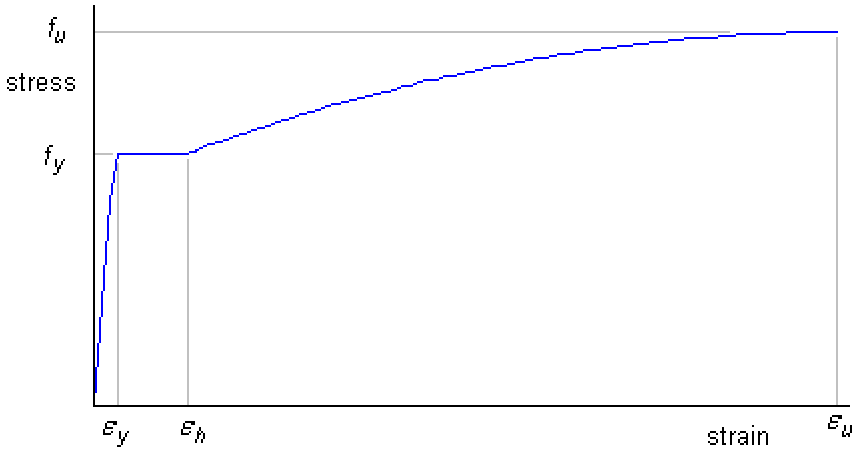
For BS and Hong Kong code there are also a pair of default pre-stressing steel types. These are designated "1770" & "1860". These describe a 15.7mm nominal diameter superstrand, with a standard stress-strain curve to BS8110 Pt 1 Figure 2.3; f_{pu} = 1770 N/mm^2 & 1860 N/mm^2 and E=195 kN/mm^2 or BS5400 Figure 3 f_{pu} = 1770 N/mm^2 & 1860 N/mm^2 and E = 200KN/mm^2.

GFRP & CFRP Rebar

Material properties for GFRP and CFRP are taken from the manufacturers published data.

Park Material Model

AdSec implements a modified Park model. This is linear elastic to yield (ε_y,f_y), followed by a perfectly plastic phase to (ε_h,f_y) and finally a parabolic curve up to failure at (ε_u,fu) . This curve is represented below.



Modified Park Model

Thus

$$f = E\varepsilon \qquad \varepsilon \le \varepsilon_y$$

$$f = f_y \qquad \varepsilon_y < \varepsilon \le \varepsilon_h$$

$$f = f_y + \frac{(\varepsilon - \varepsilon_h)^2}{(\varepsilon_u - \varepsilon_h)^2}(f_u - f_y) \qquad \varepsilon > \varepsilon_h$$

7.7 Steel and FRP Materials

Steel

Steel materials are assumed to be elastic-perfectly plastic with an initial slope equal to the Young's modulus (E) and constant stress once yielded.

FRP

FRP materials are provided for strengthening of sections rather than analysis of FRP sections. FRP materials are assumed to be linear to failure. For codes which use material partial safety factors there are two for FRP materials - one which modifies the strength and another that modifies the stiffness. In addition to the FRP material properties the behaviour is also governed by the bond strength between FRP and concrete.

FRP materials are anisotropic. The relevant properties for section analysis are those perpendicular to the section.

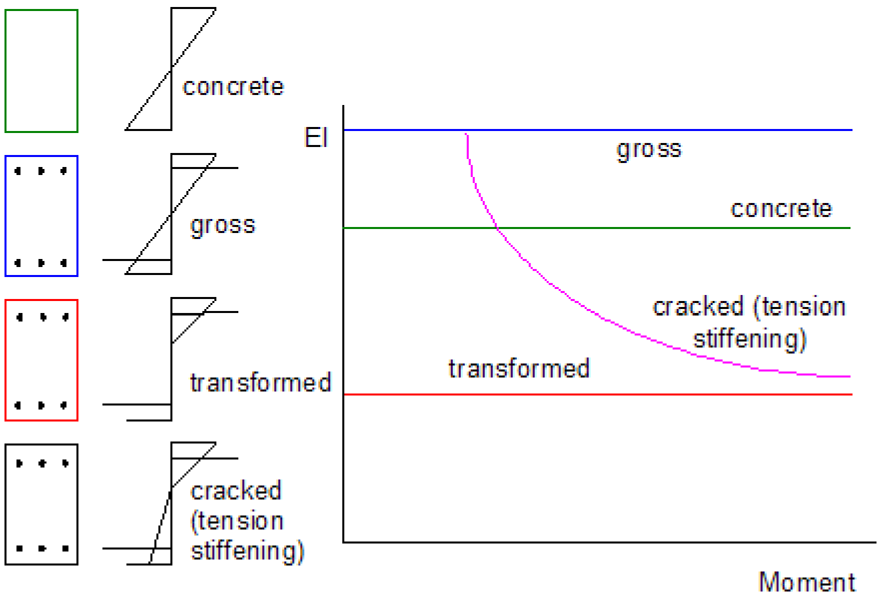
7.8 User Defined Stress Strain Curves

To implement material models not described above users can implement user defined stress-strain curves. These are input as a table of stress values as a function of strain. These are stored as a table of values and are normally displayed a list stress-strain values (ε_1,σ_1), (ε_2,σ_2), ...

At present these are only available for concrete and rebar materials.

7.9 Stiffness

AdSec operates on strain, using non-linear materials. AdSec will show how the stiffness of the section changes with load and the effect of non-linear material behaviour. There are a number of ways in which the stiffness of a reinforced concrete section can be approximated. These are show in the diagram below. This diagram plots AdSec results along with the approximate stiffness values for comparison.



Stiffness variations for different material assumptions

For a symmetric section, symmetrically loaded, stiffness can be expressed as:

$$EI = \frac{M}{\kappa}$$

but if there is an axial force, locked in strain plane, or in some cases of prestress, there will be a residual curvature at zero moment. This curvature can be called κ_0 so AdSec uses:

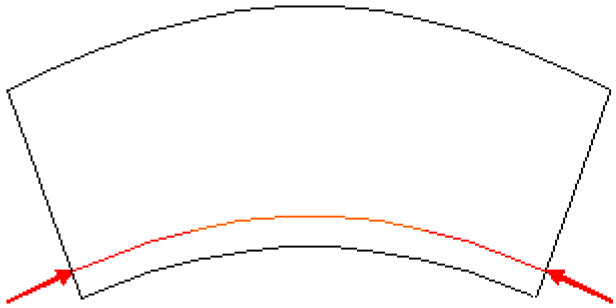
$$EI = \frac{M}{(\kappa - \kappa_0)}$$

The curvature at zero moment may not be in the same direction as the applied moment angle. To allow for this, the formula is further modified to give

$$EI = \frac{M}{[\kappa - \kappa_0(\alpha_{appl} - \alpha_{NA})]}$$

Where α_{appl} is the angle of the applied moment and α_{NA} is the neutral axis angle from the κ_0 calculation.

The following diagram of a pre-stressed beam illustrates the issue when the applied loads ***exclude*** pre-stress and anchorage forces.



Curvature of the section with no applied moment due to prestress

7.10 Cracking

AdSec offers two features for investigating cracking.

Cracking moment

This is an explicit analysis task in AdSec. The programme searches for the strain plane at point of first crack, and then integrates the stresses to calculate the cracking moment. All cracking moment calculations use short term material properties as the crack will form the instant the load is applied.

Crack-width

This is a post-processor performed after searching for the state of strain under applied loading. The result will include the long or short-term material properties dependent on the duration of load specified. Crack widths are not calculated for ACI and AS codes.

Cracking behaviour is complex as it is controlled by the reinforcement. Calculations to predict these phenomena vary from code to code. Most codes have developed their cracking rules for very simple situations. This means that the rules are impossible to 'codify' for a computer programme for all possible geometries.

Some codes imply a different tension stiffening method for crack width as opposed to the other checks. This may lead to a disparity in AdSec results between the 'cracking moment' and the moment at which the crack width becomes > zero.

The code rules are generally developed for a rectangular section with uniaxial bending and one row of tension steel. To include cracking in AdSec the rules need to be interpreted to sections made up of various zones of concrete, some with locked in strain planes, bending in different directions.

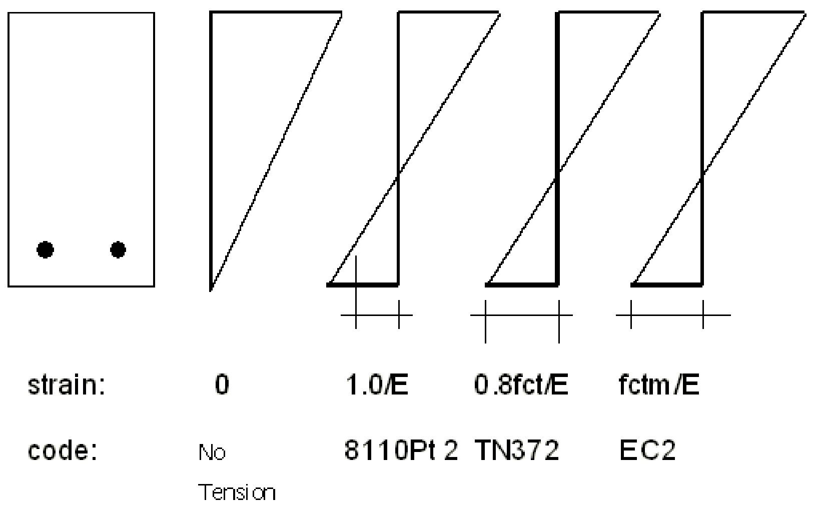
The approach in the development of AdSec has been to give as many of the 'judgement' calls about interpretation of the code back to the user. This is helped by the use of graphics to show the user what has actually been calculated. ***All crack width calculations*** should be reviewed (at least graphically) for correct engineering interpretation of the code.

For EC2 the rules in the code are contradictory when taken beyond the simple design case of a uniaxially bending rectangular beam with the neutral axis in the section. Therefore crack width calculations are **only** available for this case in EC2. Sections need to be defined with beam template reinforcement and constrained to bend uniaxially, otherwise no crack width will be output. Where crack widths are calculated to EC2, a constant value will be output for the tension face of the beam.

Cracking moment and crack width calculations can be performed using the full range of stiffening options for the compression zone given in the code.

7.10.1 Cracking Moment Calculations

The programme calculates the total axial force and moment from the loads in the analysis case definition. It uses the axial force and the angle of the applied moment to define the cracking moment analysis task. The programme then searches for a strain plane that gives the cracking strain as shown below.

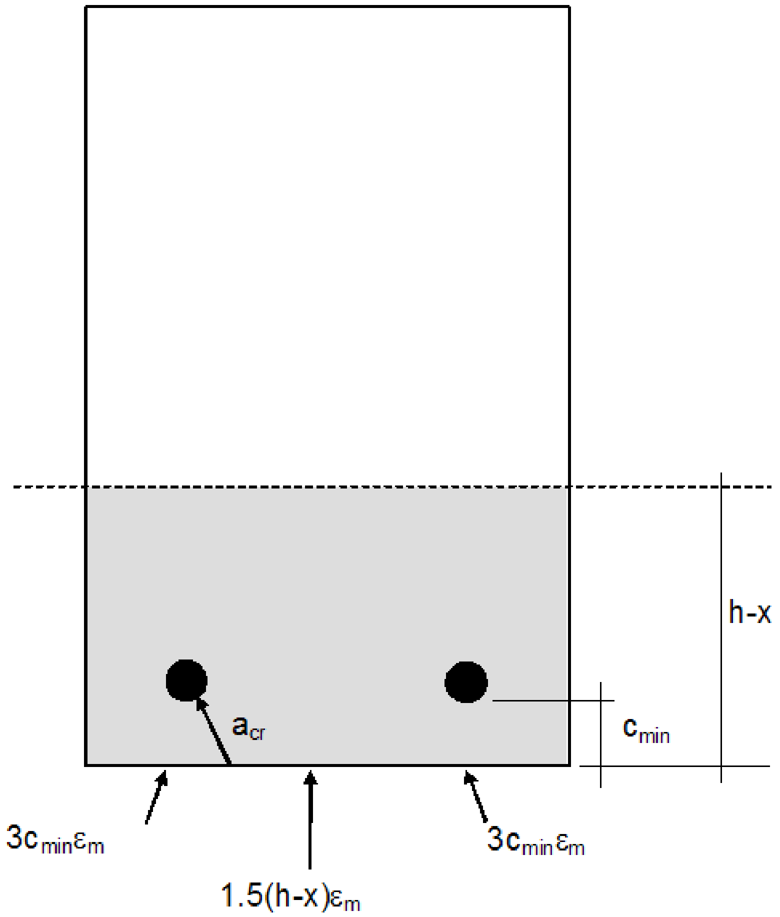


Code cracking assumptions

Integrating the stresses from this strain plane over the section will give an axial force equal to the applied force, and a moment which is parallel to the applied moment. The value of this moment is the cracking moment. Short-term material properties are always used for the cracking moment calculation.

7.10.2 Crack-width

Crack width formulae to BS8110, BS5400, BS EN 1992 : PD 6687 and IS456 are based on a weighted interpolation between two effects. Close to a bar, crack width is a function of the bar cover, c_{min}. Between bars, it is a function of the depth of tension zone, (h - x).



Code cracking assumptions

A different approach is used in EC2 for tension stiffening & crackwidth calculations.

7.10.2.1 BS8110,BS EN 1992 : PD 6687 and IS 456

Crack widths can be calculated to BS8110 and IS456 using any of the available tension stiffening options.

After calculating the cracking moment, AdSec will search for a strain plane which gives forces and moments within tolerance of the applied forces and moments. The resulting strain distribution is used to calculate the crack width.

The maximum crack width output is related to the given resultant moment orientation. This is particularly important for circular sections, as the maximum strain may not occur between two bars. This would give a lower crack width value than may occur in reality.

The sides of the section are divided into small segments and the crack width calculated for each segment. The crack width formula

$$crack\ width = \frac{3a_{cr}\varepsilon_m}{1+2\left(\frac{a_{cr}-c_{min}}{h-x}\right)}$$

is given in BS8110, section 3.8, equation 12. All the results are displayed graphically and in tabular form for checking. The maximum crack-width is available in a summary table.

Crack width calculations involve a large amount of engineering interpretation for faceted sections, sections with voids, sections with re-entrant corners, and multi-zone sections. Depending on the situation, a different definition of 'cover' is required. The programme stores the minimum cover to each bar and uses this in the calculation of crack width. This means that the cover used in the calc may not relate to the side being checked (it will always give a conservative result). The reason for this approach is that curved sections are analysed as a multifaceted polygon, and there may be no bars present parallel to a small facet. This is because the number of facets may be greater than the number of bars.

The crack width calculation is done on a component by component basis using the component strain plane (resulting strain plane + component strain plane + concrete only component strain plane). This component strain plane applied to the whole section is used to calculate neutral axis depth and section height (x and h) relative to the whole section – using all the section coordinates.

The crack width includes the term (a_{cr}–c_{min}). If c_{min} is smaller than a_{cr} the crackwidth is increased. For each division on the concrete outline the closest bar is found (min a_{cr}). For a re-entrant corner, and a bar which is on the 'outside' of the section with ref to the side being checked a warning flag is generated, A conservative crack width can still be calculated using the minimum cover to the bar.

If the cover is greater than half the depth of the tension zone, the crack width in both codes in invalid. The term for concrete only cracking should be used instead. This is $1.5(h - x)\varepsilon_m$. This is included in AdSec.

These are warning in the crack width calculation. They do not necessarily mean that the answer is wrong. But do mean that the graphical results should be checked for engineering interpretation

- C_{min} < controlling bar diameter – crack width not valid
- Controlling bar is remote from crack location
- Controlling bar and crack are located on either side of re-entrant corner
- Cover to controlling bar measured to different side from crack location

7.10.2.2 BS5400

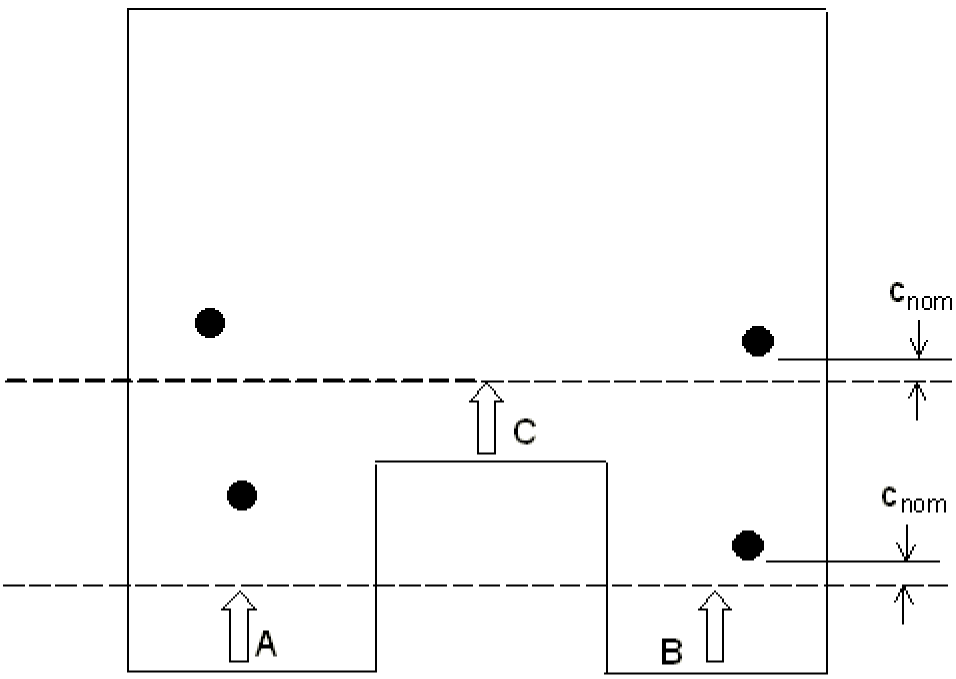
The BS8110 specification above is valid for BS5400 plus some additional points.

The BS5400 includes a fudge for effective tension stiffening. So for crack widths to be correct using the BS5400 formula, SLS analysis must use no tension stiffening in calculation of the strains around the section.

This fudge requires calculation of 'the level of tension steel'. This is re-calculated on a component by component basis using a similar method to BS8110-2 tension stiffening. The tension steel is identified as the steel which is in the tension zone when the zonal strain plane is extended across the whole section (the zonal strain plane is the resulting strain plane + zonal locked in plane + zonal concrete only plane). From the steel bars identified, the centroid of steel force is calculated using the actual stress in the bars ignoring prestress, and ignoring any bars in compression.

Once the level of centroid of tension steel is found, the width term bt needs to be calculated. This includes interrogation of all the section coordinates (as for x and h). If more than 2 sides cross the level of centroid of tension steel, the width is taken as the distance between the two extreme dimensions. This needs to be checked for sections with large voids, or channels with thin legs, as the term b_t x h is used to make an approximation for the force in the tension zone and assess the area of tension steel vs the area of tension concrete. It may be appropriate to substitute a smaller value of bt.

BS5400 includes a notional surface a distance c_{nom} from the bars. AdSec will look at all bars to define this surface excluding any with 'negative cover'. It plots the assumed check surface with the results. This should be reviewed, particular for sections with sharp acute angles and re-entrant corners. In the example below the adjustment to sides A and side C may not be the adjustment that would be chosen by engineering judgement. In this situation, the cracking parameters output for the relevant sides can be extracted from the output, and the results recalculated, by the user, substituting the corrected values.



Cover to bars

The crack width equations in BS5400-4 are either equation 24

$$Design\,crack\,width = \frac{3a_{cr}\varepsilon_m}{1+2(a_{cr}-c_{nom})/(h-d_c)}$$

where the strain ε_m is given by equation 25

$$\varepsilon_m = \varepsilon_1 - \left[\frac{3.8 b_t h (a' - d_c)}{\varepsilon_s A_s (h - d_c)}\right]\left[\left(1 - \frac{M_q}{M_g}\right)10^{-9}\right]$$

or the alternative equation 26

$$Design\,crack\,width = 3a_{cr}\varepsilon_m$$

are offered by AdSec.

7.10.2.3 EC2 Tension Stiffening and Crackwidth Calculation

Tension stiffening — Eurocode 2 - 7.4.3 equation 7.7

EC2 differs from BS8110 in that it does not have specific tension stiffening relationship used in analysis. Instead, analyses are carried out assuming cracked and uncracked stiffness values, and the actual strain plane/ stiffness is an interpolation between the two results based on the amount of cracking predicted..

Cracking — Eurocode 2 - equation 7.3.4

Cracking in Eurocode is calculate from a strain and spacing in equation (7.8).

$$w_k = s_{r,max}(\varepsilon_{sm} - \varepsilon_{cm})$$

There are two options for calculating the strain in the above equation. This strain can be calculated either from Equation (7.9) or with the option labelled Equation (7.18). In the second approach the strain is calculated by looking at the interpolated strain plane from the SLS analysis and using this strain to calculate the crackwidth

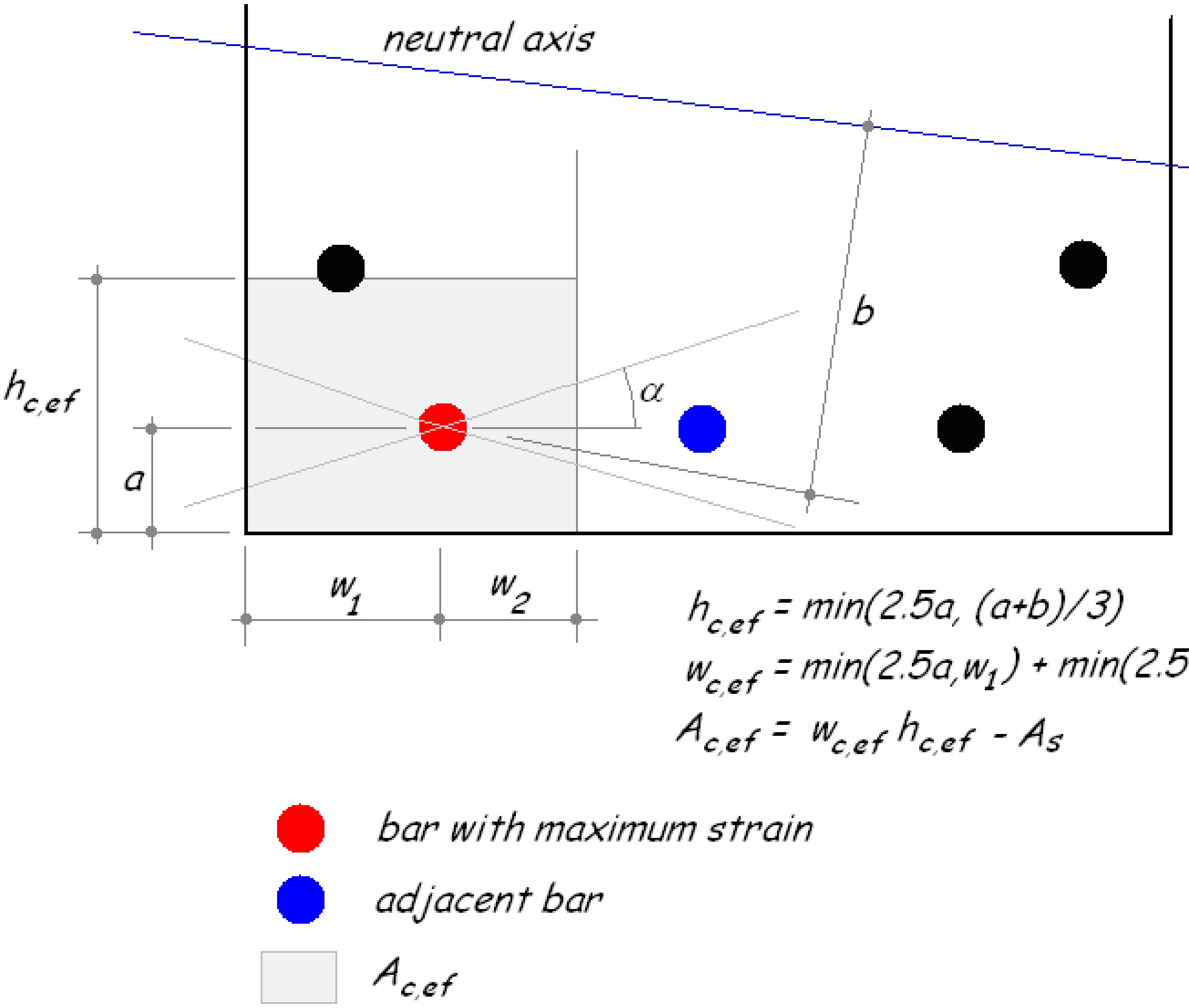
Code option

Crack widths to Eurocode 2 for the code option are only performed on rectangular sections with beam template reinforcement and uniaxial bending results. This is due to inherent problems in Eurocode 2.

The crack width calculation approach is also different in that instead of traversing the section looking at cover and bar spacing, one crack width calculation is performed for the concrete component using the strain results. A maximum crack width is calculated according to the size and distribution of tension steel and the area of concrete in the tension zone giving $\rho_{p,eff}$.

Approximate method

Where the code option cannot be applied the approximate method is used instead. This is illustrated below:



A search is made for bar with the largest tensile strain and the calculation is carried out for area around this bar. The Area of the bar is A_s and an associated concrete are $A_{c,eff}$ is calculated. The width $w_{c,ef}$ is calculated by looking at the distance from the bar to the nearest adjacent bar or the edge of the section and this is limited to 2.5a where a is the distance from the surface to the centre of the bar. Only bars within an zone defined by an angle $\pm\alpha$ to the edge of the element are considered.The height $h_{c,ef}$ of the block is governed by the cover and the distance to the neutral axis. Inner rows of bars are ignored. The concrete associated with the bar is assumed to be a rectangle defined by this width and height, less the area of the bar.

This gives $\rho_{p,eff}$ needed to calculate the strain.

Eurocode UK Options

The UK national annex and PD 6687 outline modifications to the standard Eurocode crack calculations these are summarized in the table below.

Buildings	Bridges
UK National Annex **NA to BS EN 1992-1-1:2004** Table NA.4 Note – modification of surface crack widths $\frac{c_{min,dur}+\Delta c_{dev}}{c_{nom}}$	**UK National Annex** **NA to BS EN 1992-2:2005** –
PD 6687:2006 2.17 Crack widths for non-rectangular tension zones $wk = \{3a_{cr}\varepsilon_m/[1+2(a_{cr}-c)/(h-x)]\}$ d) the concrete stress varies linearly from zero at the neutral axis to 0.7N/mm^2 at the extreme fibre in tension	**PD 6687-2:2008** 8.2.2 – modification of surface crack widths $\frac{c_{min,dur}+\Delta c_{dev}}{c_{nom}}$

Part VIII

8 Technical Notes

This section describes some of the material curves used in AdSec.

Material Curves

8.1 Material Curves

The material curves used in AdSec are derived from the relevant design codes.

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

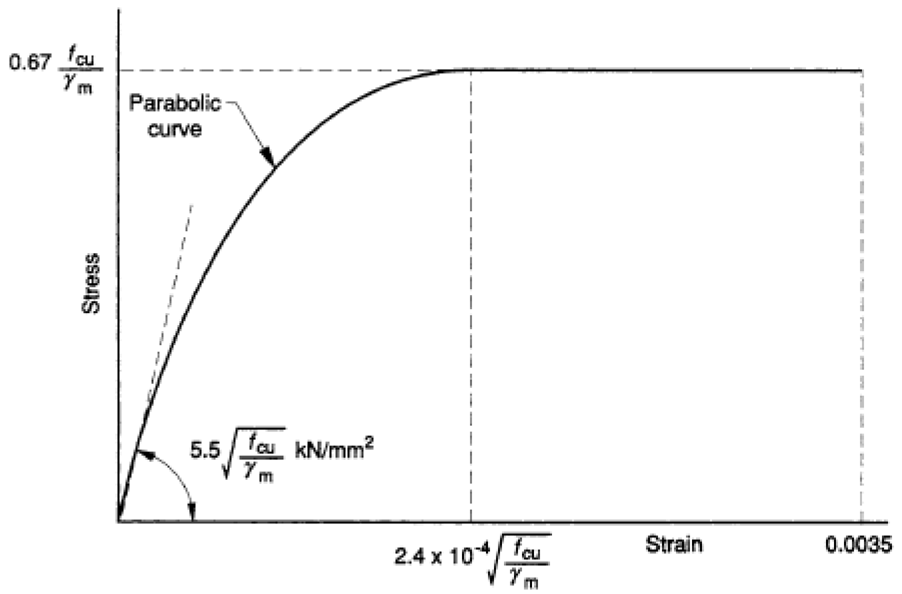
BSI Customer Services,
389 Chiswick High Road,
London W4 4AL.

Tel: +44 (0)20 8996 9001

email: cservices@bsi-global.com

8.1.1 BS8110 Concrete and Reinforcement Materials

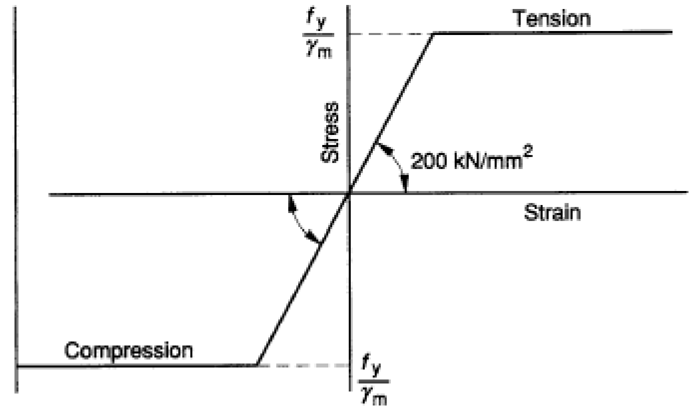
Material curves reproduced with permission from BSI



NOTE 1 0.67 takes account of the relation between the cube strength and the bending strength in a flexural member. It is simply a coefficient and not a partial safety factor.

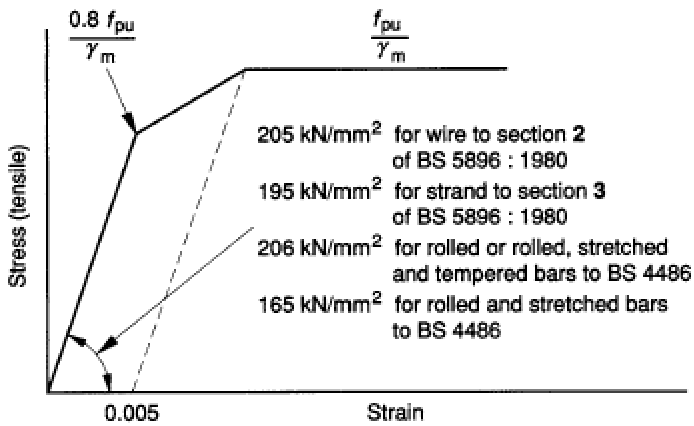
NOTE 2 f_{cu} is in N/mm².

Figure 2.1 — Short term design stress-strain curve for normal-weight concrete



NOTE f_y is in N/mm².

Figure 2.2 — Short term design stress-strain curve for reinforcement



NOTE f_{pu} is in N/mm².

Figure 2.3 — Short term design stress-strain curve for prestressing tendons

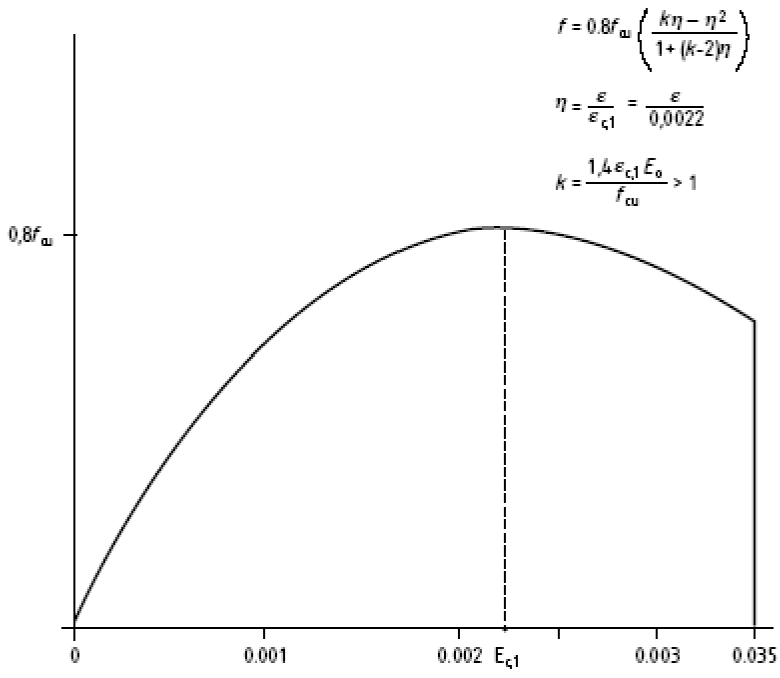


Figure 2.1 — Stress strain curve for rigorous analysis of non-critical sections

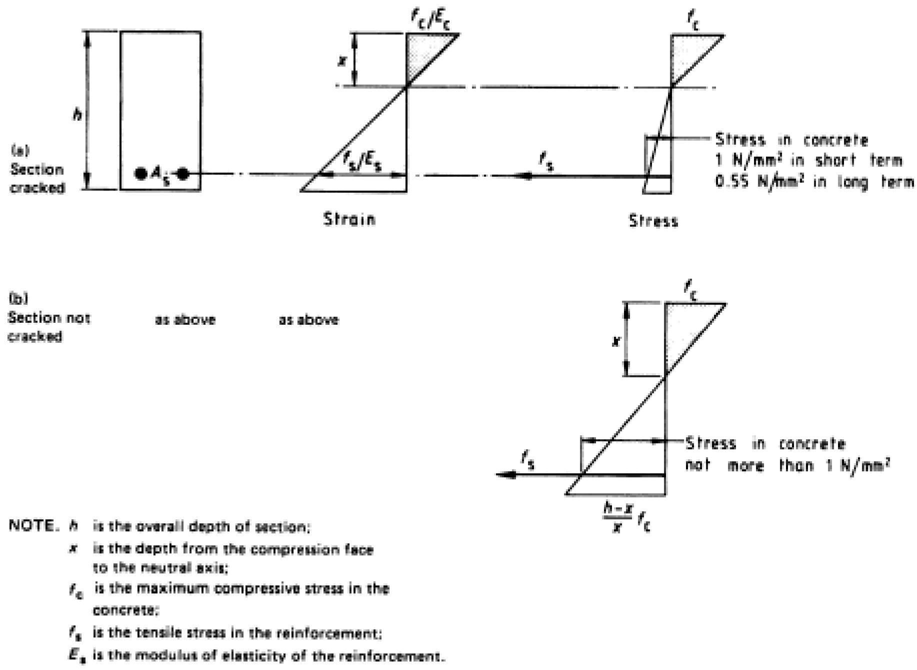
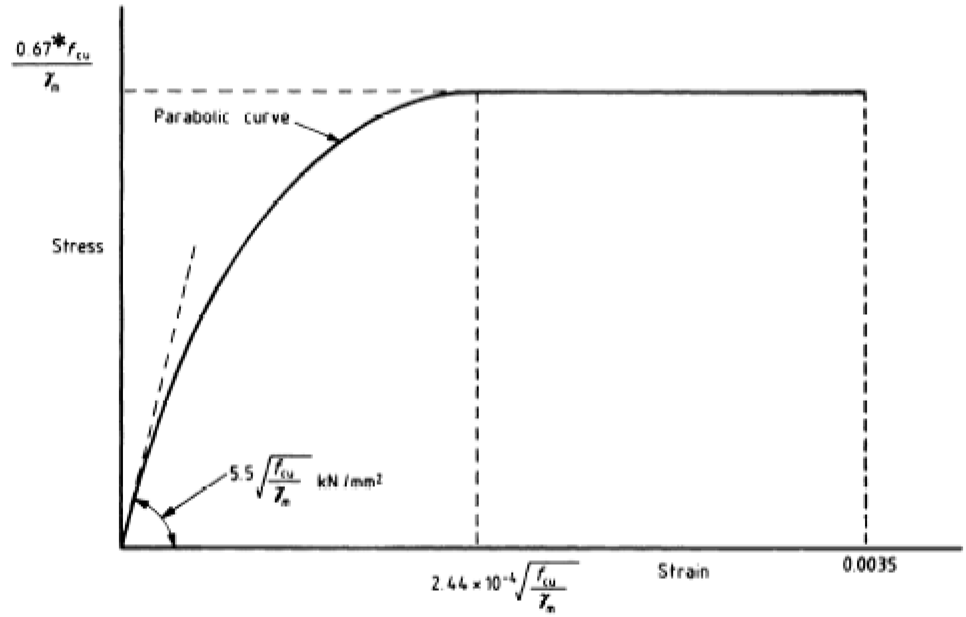


Figure 3.1 — Assumptions made in calculating curvatures

8.1.2 BS5400 Concrete and Reinforcement Materials

Material curves reproduced with permission from BSI



*0.67 takes account of the ratio between the characteristic cube strength and the bending strength in a flexural member.

NOTE 1. f_{cu} is in N/mm².

NOTE 2. The equation for the parabolic curve between $\epsilon = 0$ and $2.44 \times 10^{-4}\sqrt{f_{cu}/\gamma_m}$ may be taken as:

$$f = \left(5500\sqrt{\frac{f_{cu}}{\gamma_m}}\right)\epsilon - \left(\frac{5500^2}{2.68}\right)\epsilon^2$$

where

f is the stress;

ϵ is the strain.

Figure 1 — Short term design stress-strain curve for normal weight concrete

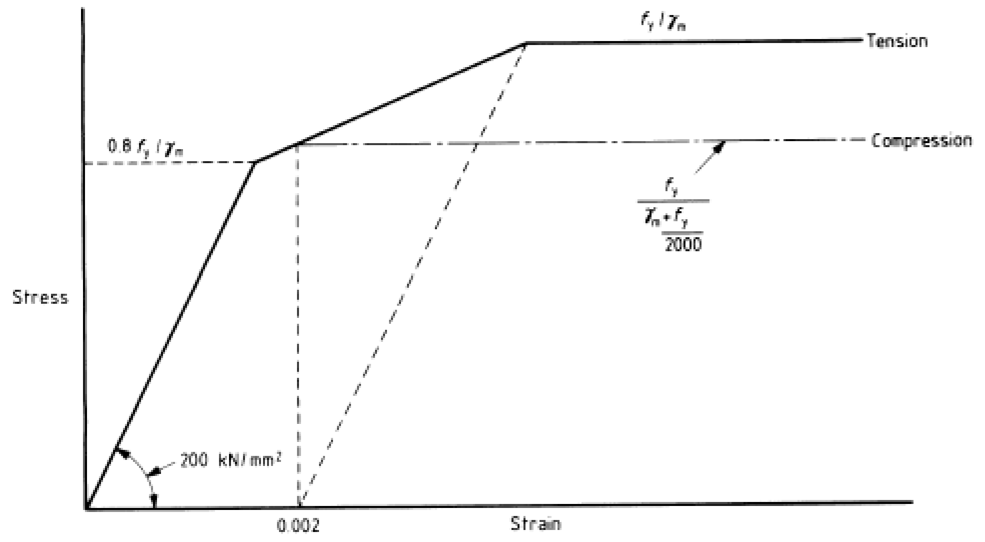


Figure 2 — Short term design stress-strain curve for reinforcement

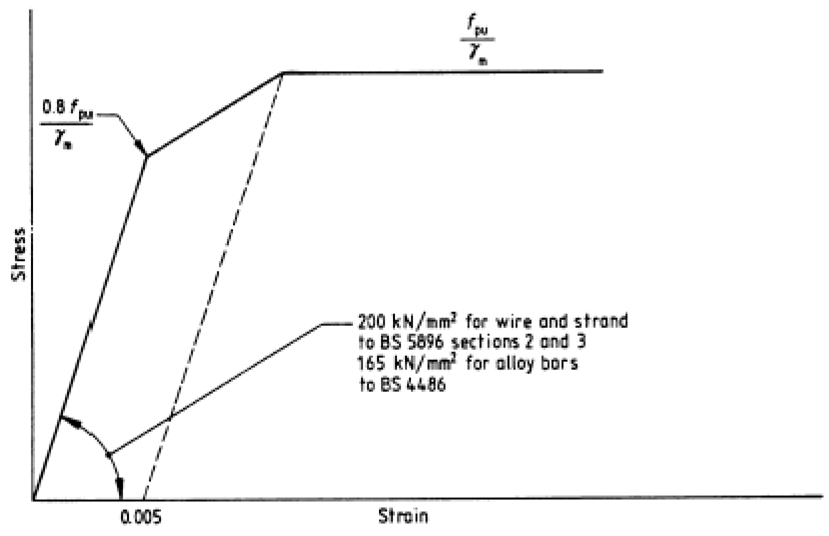


Figure 3 — Short term design stress-strain curve for normal and for low relaxation products

8.1.3 EC2 Concrete Materials

Material Curves reproduced with permission from BSI

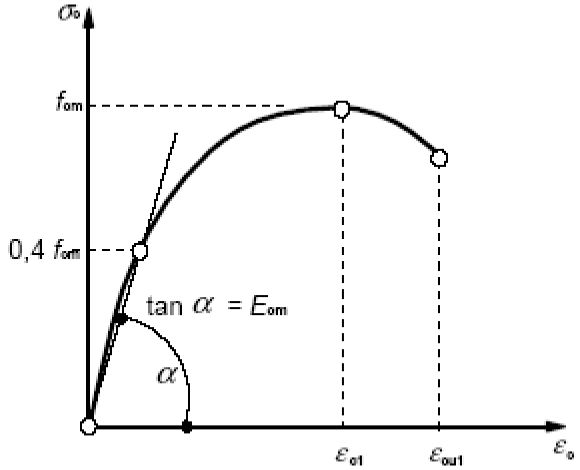


Figure 3.2 **Schematic representation of the stress-strain relation for structural analysis.**

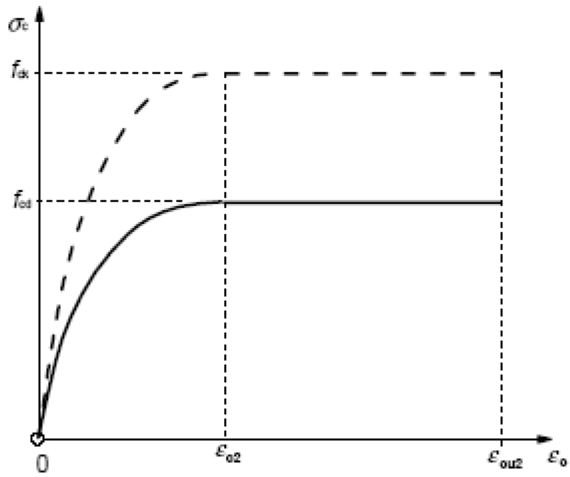


Figure 3.3. Parabola-rectangle diagram for concrete under compression.

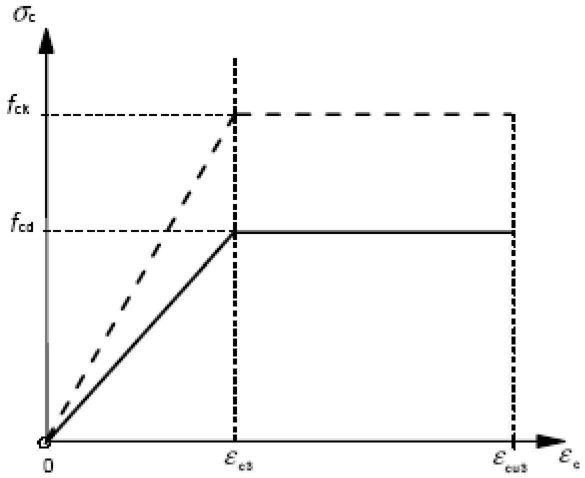


Figure 3.4. Bi-linear stress-strain relation.

Part IX

9 AdSec Text File

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

Part X

10 Bibliography

1. BS 8110-1:1985 – Structural use of concrete, Part 1. Code of practice for design and construction, BSI, 1985
2. BS 8110-1:1997 – Structural use of concrete code of practice for design and construction (incorporating amendments 1 and 2), BSI, May 2002
3. BS 8110-1:1997 – Structural use of concrete code of practice for design and construction (incorporating amendments 1, 2 and 3), BSI, Nov 2005
4. BS 8110-2:1985 – Structural use of concrete code of practice for special circumstances (incorporating amendments 1 and 2), BSI July 2001
5. BS 5400-4:1990 – Steel, concrete and composite bridges code of practice for design of concrete bridges, BSI, 1990
6. Eurocode 2: Design of concrete structures – Part 1.1 General rules and rules for buildings, CEN, 2004
7. Code of Practice for Structural Use of Concrete 1987, Buildings Authority, Hong Kong, 1987
8. Code of Practice for Structural Use of Concrete 2004, Buildings Department, Hong Kong, 2004
9. AS3600 – 2001 Australian Standard, Concrete Structures, Standards Australia, 2001
10. AS3600 – 2009 Australian Standard, Concrete Structures, Standards Australia, 2009
11. Building Code Requirements for Structural Concrete (ACI 318-02) and Commentary (ACI 318R-02), ACI, 2001
12. Building Code Requirements for Structural Concrete (ACI 318M-02) and Commentary (ACI 318RM-02), ACI, 2001
13. Building Code Requirements for Structural Concrete (ACI 318-05) and Commentary (ACI 318R-05), ACI, 2005
14. Building Code Requirements for Structural Concrete (ACI 318M-05) and Commentary (ACI 318RM-05), ACI, 2005
15. Building Code Requirements for Structural Concrete (ACI 318-05) and Commentary (ACI 318R-05), ACI, 2008
16. Building Code Requirements for Structural Concrete (ACI 318M-05) and Commentary (ACI 318RM-05), ACI, 2008
17. Structures Design Manual for Highways and Railways, Highways Department, Hong Kong, 1997
18. IS 456 : 2000, Indian Standard, Plain and Reinforced Concrete – Code of Practice (Fourth Revision), Bureau of Indian Standards, November 2005
19. Background paper to the UK National Annexes to BS EN 1992-1 (PD 6687:2006), BSI 2006

Index

A

B

C

J

L

M

N

O

P

Endnotes 2... (after index)