Application Manual
(Bridge, Civil & Structural)

Version 15.2 Issue 1

LUSAS
Forge House, 66 High Street, Kingston upon Thames,
Surrey, KT1 1HN, United Kingdom

Tel: +44 (0)20 8541 1999
Fax +44 (0)20 8549 9399
Email: info@lusas.com
http://www.lusas.com

Distributors Worldwide

Table of Contents

Introduction

Software Products Covered

This manual covers application-specific modelling and results facilities for LUSAS Bridge and LUSAS Civil & Structural software products.

Menu entries for these application products are inserted onto the main menu based upon the software key you have installed on your system or, if you are using LUSAS Academic software, by the selection of a software product during the set-up procedure. Occasionally some facilities for LUSAS Bridge and LUSAS Civil & Structural software products may also appear on the general Modeller menu.

This manual describes the options available for the following menu items:

- **Bridge menu**
- **Civil menu**

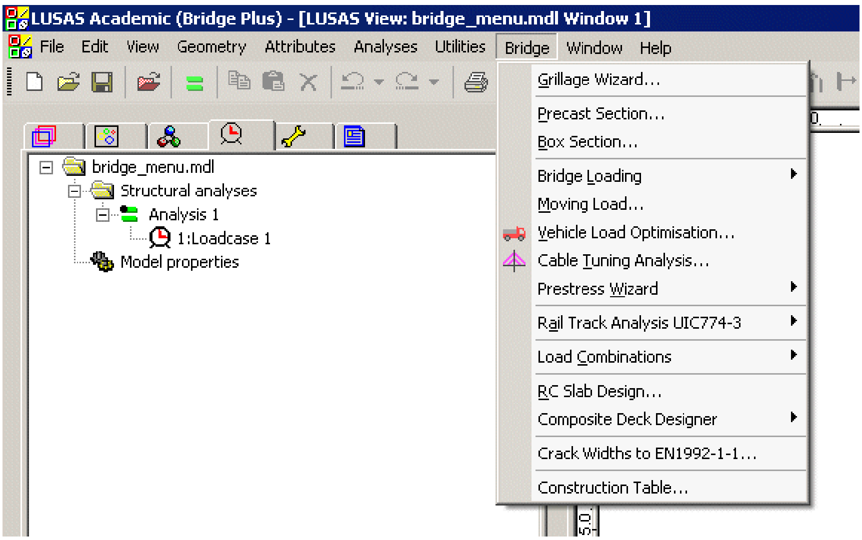
More detailed on-line help information on each of the application specific features described can be accessed from the Help button on the relevant dialog.

The Bridge Menu

The Bridge menu item will appear between the Utilities and Window menu items. It provides access to the following bridge-specific facilities:

- **Grillage Wizard** - Enables orthogonal, skewed and curved grillages to be generated from user-defined data.
- **Precast Section** - Produces 2D cross-sectional models of country-specific precast concrete beams with or without a top slab.
- **Box Section** - Calculate general box section properties from user defined dimensional data.
- **Bridge Loading** - Gravity, Surface loading, and Static vehicle loads
- **Moving Load** - Used to track the path of a static vehicle load (or a set of vehicles) across a structure
- **Vehicle Load Optimisation** - Identifies the most onerous vehicle loading patterns on bridges for a chosen design code.
- **Construction Tables** - Camber, Displacement History, and Incremental Displacement tables of results for selected locations (points) on the model.

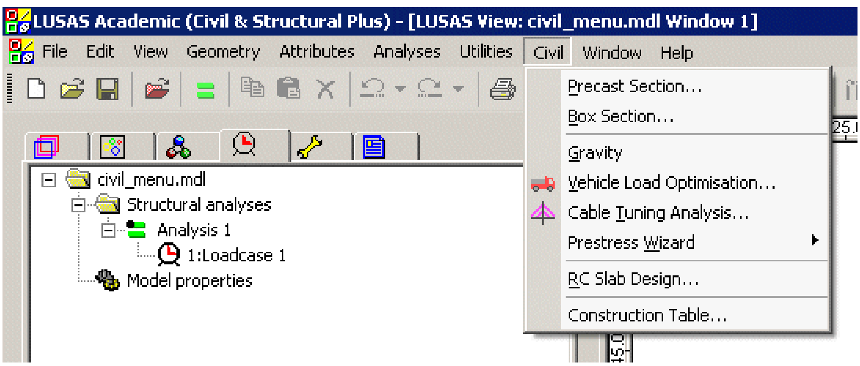
- **Cable Tuning Analysis** - Calculate load factors for selected lines in a model that represent cables in order to achieve defined target values set for various components or features
- **Prestress Wizard** - Single and multiple tendon prestress options calculate equivalent nodal loading due to tendon prestressing or post tensioning
- **Rail Track Analysis UIC 774-3** Enables track/structure interaction analysis to the International Union of Railways Code UIC774-3.
- **Load Combinations** - Pre-defined load combinations for selected design codes
- **RC Slab Design** - Enables the reinforcement required in slabs and walls to be computed in accordance with chosen design codes
- **Steel and Composite Deck Designer (PontiEC4)** - Export force and moment data from Modeller for selected sections, for use in design code checking by PontiEC4 software.
- **Crack Widths to EN1992-1-1** - Enables plotting of contours of design crack widths in accordance with the EN 1992-1-1 design code.
- **Construction Table** - Used to produce Camber, Displacement History, and Incremental Displacement tables of results for selected locations (points) on the model, and for any specified results loadcases



The Civil Menu

The Civil menu item will appear between the Utilities and Window menu items. menu items. It provides access to the following civil / structural specific facilities:

- **Precast Section** - Produces 2D cross-sectional models of country-specific precast concrete beams with or without a top slab.
- **Box Section** - Calculate general box section properties from user defined dimensional data.
- **Gravity** Adds a body force loading attribute corresponding to gravity to the Attributes Treeview.
- **Cable Tuning Analysis** - Calculate load factors for selected lines in a model that represent cables in order to achieve defined target values set for various components or features
- **Prestress Wizard** - Single and multiple tendon prestress options calculate equivalent nodal loading due to tendon prestressing or post tensioning
- **RC Slab Design** - Enables the reinforcement required in slabs and walls to be computed in accordance with chosen design codes
- **Crack Widths to EN1992-1-1** - Enables plotting of contours of design crack widths in accordance with the EN 1992-1-1 design code.
- **Construction Table** - Used to produce Camber, Displacement History, and Incremental Displacement tables of results for selected locations (points) on the model, and for any specified results loadcases



Availability of Bridge or Civil menu items according to software product.

Menu item	Bridge LT	Bridge	Bridge Plus	Civil LT	Civil	Civil Plus
Grillage Wizard	✓	✓	✓			
Precast Section		✓	✓		✓	✓
Box Section		✓	✓		✓	✓
Gravity				✓	✓	✓
Bridge Loading (inc gravity, and vehicle loads)	✓	✓	✓			
Moving Load	✓	✓	✓			
Vehicle Load Optimisation		✓	✓			
Construction Table		✓	✓			
Cable Tuning Analysis		✓	✓		✓	✓
Prestress Wizard		✓	✓		✓	✓
Rail Track Analysis UIC 773-3		▪	✓			
Pre-defined Load Combinations		✓	✓			
RC Slab Design		✓	✓		✓	✓
Steel and Composite Deck Designer (Ponti EC4)		✓	✓			
Crack Width Calculation to EN1992-1-1		✓	✓		✓	✓
Construction Table		✓	✓		✓	✓

Note

- Vehicle Load Optimisation, Rail Track Analysis UIC 773-3 and the Steel and Composite Deck Designer are software options and can only be accessed if the license key in use supports them.

Grillage Wizard

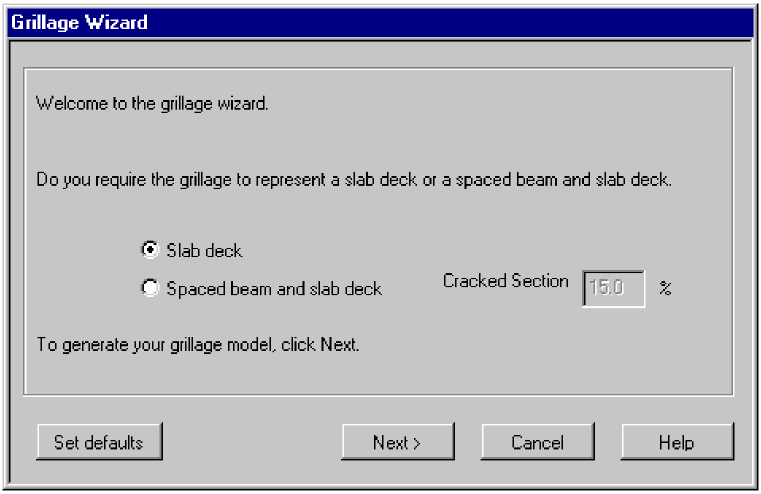
Overview

The grillage wizard comprises a series of dialogs and enables orthogonal, skewed and curved grillages to be generated from user-defined data.

When creating a grillage model some basic guidelines should be considered:-

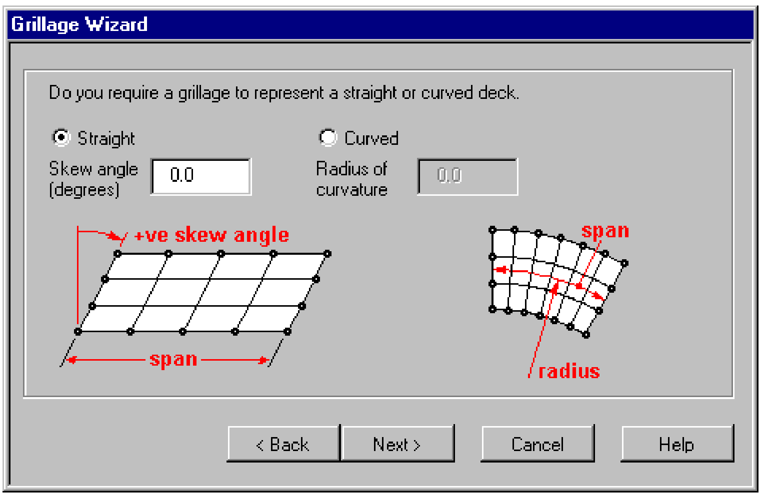
- Grillage wizard models are created with one line mesh division for each grillage element. Additional line mesh attributes can be created and assigned to the model should this ever prove to be required.
- Longitudinal grillage members should be placed along lines of design strength. For a slab model this could be where there was a concentration of reinforcement in the slab itself. For a composite bridge this could be at the location of a steel girder / prestress beam.
- The aspect ratio between the length of the transverse and longitudinal members should be set so that a good static distribution of loading is achieved. An aspect ratio of 1 to 1 is normally used but an aspect ratio of up to 1 to 3 is acceptable.
- Whenever possible it is recommended that a grillage model should have supports located at the intersection of longitudinal and traverse members.
- The grillage wizard automatically places rigid supports onto the model at bearing locations. These supports can be overwritten with new attributes of, say, an elastomeric bearing using a spring support. The supports should be chosen to as closely represent the actual structure as possible.
- If the results from a coarse grillage are in doubt for any reason a more refined grillage should be used to check the results.

Grillage Wizard : Step 1 - Grillage Type



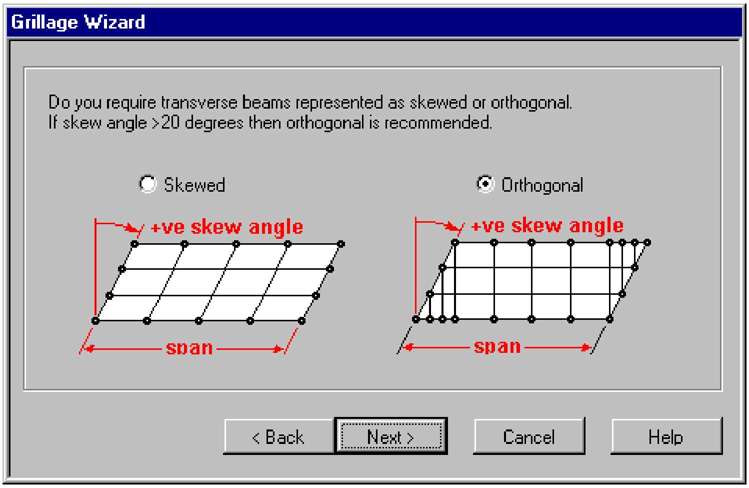
- Select the type of grillage model that you require. Both slab deck and spaced beam with slab deck constructions are supported and cracked section may be included.
- Spaced beam and slab deck models will automatically have elements defined for the allocation of cracked properties. By default 15% of the span (over internal supports) will be assumed to be cracked but this can be modified if required.
- The wizard accounts for multiple spans and applies support conditions. Groups are created to enable geometric and material properties to be easily assigned.

Grillage Wizard : Step 2 - Straight or Curved Deck



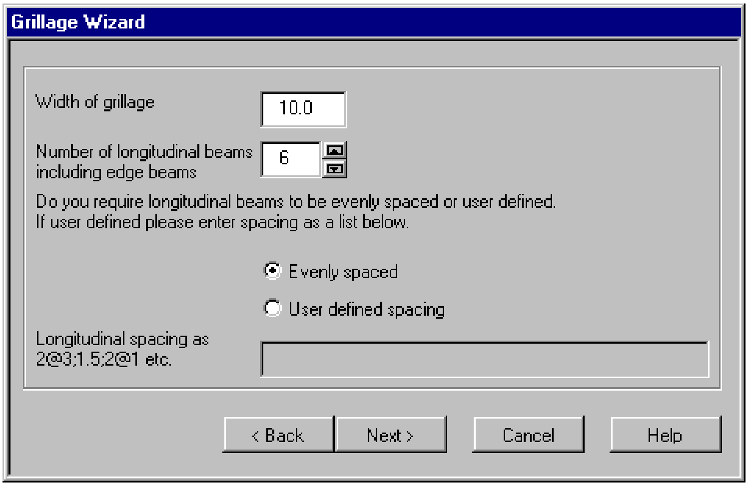
- Select whether a straight grillage or curved grillage is required. For a straight grillage a skew angle can be set. For a curved grillage a radius of curvature needs to be entered.
- For a straight grillage if any angle other than zero is entered, when the **Next** button is selected a dialog allowing a skewed or orthogonal transverse beam arrangement will be shown.

Grillage Wizard : Step 2a - Skewed or Orthogonal Beams



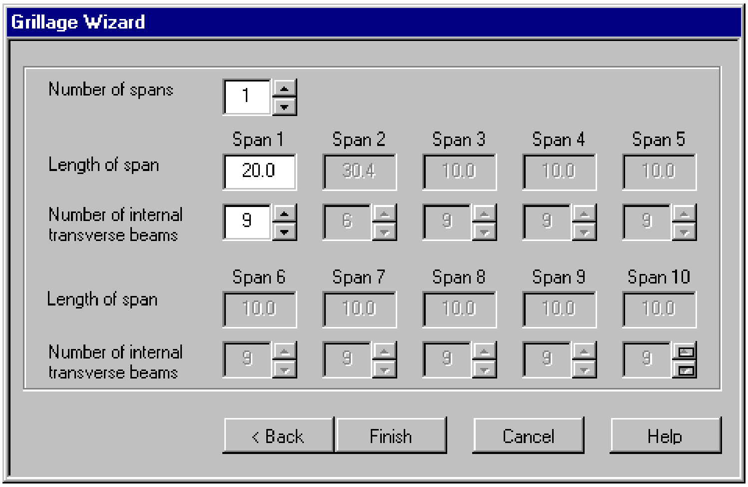
- If an angle has been entered for a straight grillage the geometry can be formed from skewed grillage elements or orthogonal grillage elements.
- If the skew angle is above 20 degrees then an orthogonal arrangement is recommended.

Grillage Wizard : Step 3 - Longitudinal Beam Detail



- The width of the grillage and the number of longitudinal beams can be set on this dialog
- The longitudinal beams can be evenly spaced or a user defined spacing can be set up allowing if services troughs dictate this is necessary.

Grillage Wizard : Step 4 - Span Details



- The number of spans, their lengths and the number of transverse elements can be set on this dialog.

- Clicking the **Finish** button creates the grillage model of the previously specified values.

Section generators and property calculators

In addition to the range of generally available standard section property calculators and the arbitrary section property calculator the following are available from the Bridge (or Civil) menu:

- **Precast Beam Section Generator**
- **Box Section Property Calculator**

Precast Beam Section Generator

The Precast Beam Section Generator is provided in Bridge and Civil & Structural software products only. It produces 2D cross-sectional models of country-specific precast concrete beams with a top slab. These models are typically used to calculate beam section properties for use in grillage and frame models.

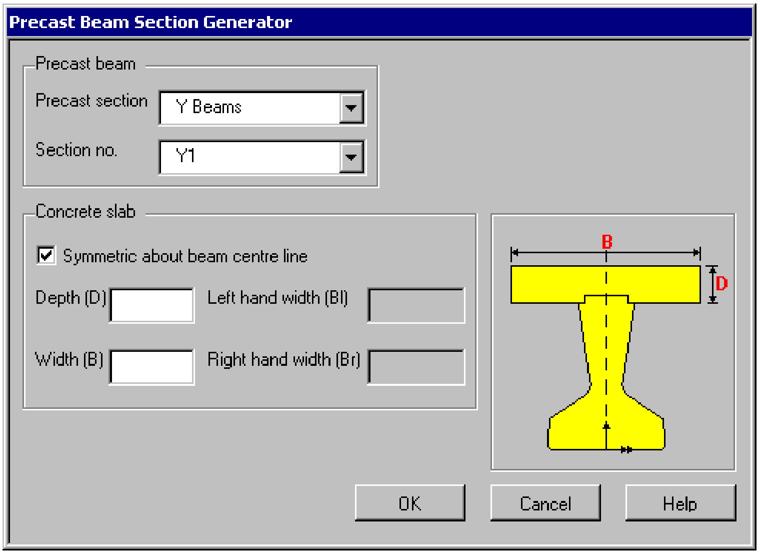
Precast concrete beam sections (with a slab) can be defined for the following countries:

- **UK beams: Y, YE, TY, TYE, SY, M, UM and U beam types**
- **US beams: AASHTO Types II to VI, Florida Bulb T72 and T78 beams, NU Girders and Texas DoT 'I' Girders, Northeast Bulb 'T'**
- **Australia and New Zealand beams: Super-T beams T1 to T5 (open and closed)**
- **Canada 'I' beams**

Note that these precast beam sections (of beams without a top slab) and other common shapes are held in the general **section library** for the countries stated. They can be accessed using the **Attributes > Geometric > Section Library** menu item.

Creating and saving a section

The Precast Beam Section Generator is accessed from the **Utilities> Section Property Calculator> Precast Section** menu item.



Precast beam

- **Precast section** allows selection of a range of precast beam types.
- **Section no** allows selection of a particular beam type.

Concrete slab

- **Symmetric about beam centre line** option, if ticked, creates a slab of a width that is centred about the beam centre. If unchecked, left and right hand widths for the slab can be entered.
- **Depth** and **Width** of top slab is required.

Once valid dimensions are specified and the **OK** button is pressed the defined section is drawn to the screen.

Calculating section properties

The arbitrary section property calculator must be used to calculate a full set of section properties for the shape and to save the section to a user-defined local or server library. Use the **Utilities> Section Property Calculator> Arbitrary Section** menu item to do this.

Using a section

To add the generated precast beam and geometry to the Attributes treeview select the **Attributes> Geometric> Section Library** menu item, select **User Sections,** then select **Local** or **Server** before choosing the section required from the list available. The geometric properties can then be **assigned** to the required Line(s) in the model.

Notes:

- Differences in concrete strengths between the beam and the slab are not considered. However, this can be emulated by altering the width and/or depth of the slab as required.
- Super-T precast sections are based upon Roads and Traffic Authority of NSW Drawing Number RTAB033 Issue 8 Jan 2007 entitled: “Standardisation of Super-T Girder Sections”.

Box Section Property Calculators

The Box Section Property Calculator is provided in Bridge and Civil & Structural software products only. It is accessed from the **Utilities> Section Property Calculator > Box Section...** menu item or from the Bridge menu.

The box section property calculators calculate general section properties from user defined dimensional data. The section shape can be defined either as a **simple box section** or as a **complex box section** created from as many points as are required to form a suitable representation of the true cross-sectional shape. A void can be included or excluded from a section. The sections generated are of a type typically used in precast and segmentally constructed bridges.

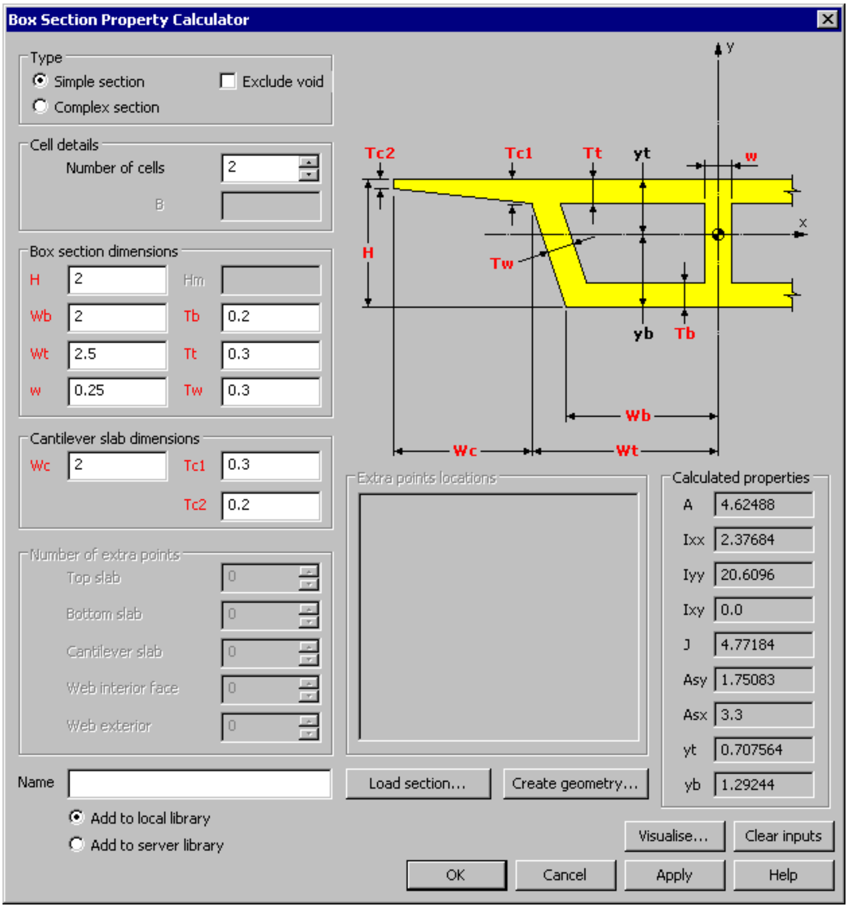
General section properties (area, moments of inertia and torsion constant etc.) of the section are computed automatically once a valid set of dimensions have been defined, using model units. Extreme fibre positions for use when plotting stresses on beams are also calculated. The resulting section can additionally be:

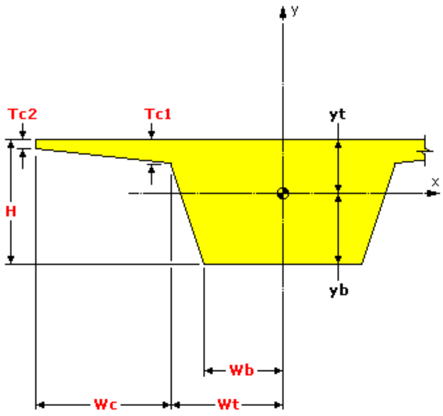
- Visualised to check for correct values being entered and to see the automatically defined fibre locations
- Converted into model geometry. This would typically be used, for example, if it was required to modify the generated section in some way inside LUSAS Modeller before re-calculating the new section properties of the edited section using the **Arbitrary Section Property Calculator**.
- Added to a local or server library to enable the section properties to be used on the current project or on other projects.

Note: The Box Section Property Calculator does not currently calculate the additional section properties (such as shear centres, warping, radius of gyration, plastic properties etc.) that are required for use with design calculations. Instead, after defining a section shape, the **Create geometry** button should be used to draw the section prior to using the **Arbitrary Section Property Calculator** which will calculate the full set of section properties for the shape.

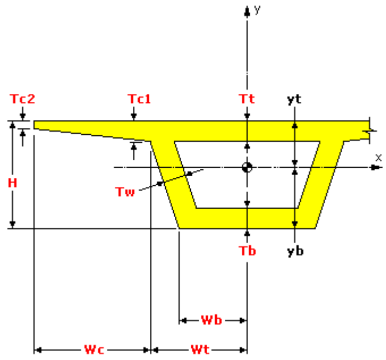
Simple Box Section

The simple box section calculator creates a box section of the style shown from user-defined values. No fillets can be specified internally when a void is present.

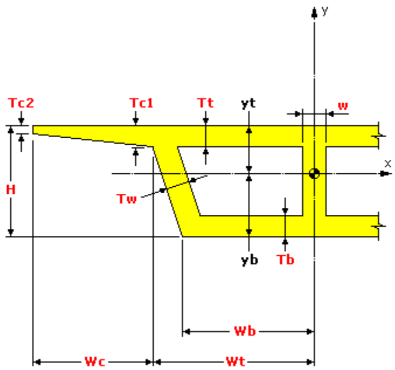




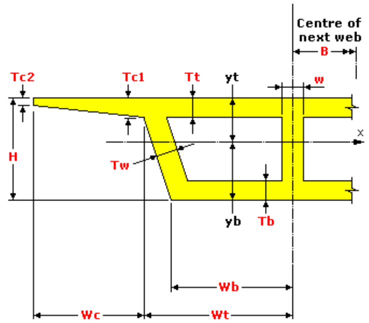
Simple box section without void



One cell simple box section



Two cell simple box section



Multi-cell simple box section

Dimensional data:

- **Cell details** Specify the number of cells, and the spacing between centre to centre distances of webs
- **Box section dimensions** These specify the overall height and width of the box, and the thicknesses of top and bottom slabs and side walls.
- **Cantilever slab dimensions** These specify the width and thickness of the cantilever.

Saving and using the defined section:

- The **Name** of the section must be entered.
- **Add to local library** Adds the calculated values to the local library when the Apply or OK button is selected.
- **Add to server library** Adds the calculated values to the server library when the Apply or OK button is selected.

- **Load section...** Displays a list of previously created and saved cross-sections that can be re-selected to populate the fields of the dialog. The list displayed is for the local or server library that is set.
- **Create geometry** Creates a 2D LUSAS model from the dimensional data. This would typically be used, for example, if it was required to modify the generated section in some way inside LUSAS Modeller before re-calculating the new section properties of the edited section using the **Arbitrary Section Property Calculator**. Additional options are available to position or orientate the section geometry that is created in a view window. By default a section created has its centre of gravity set at the view origin, that is 0,0,0.
- **Visualise...** Shows the cross-section that has been defined by the entered data. It also shows the fibre locations that are automatically created. Note that the axes shown on the visualisation are view axes. An additional option to add a picture of this visualised section to the Annotation layer is provided. Select Create Annotation to do this.
- **Clear inputs** Clears any entered or populated data.
- Use the **Apply** button to save a section to a library and continue to modify values or define another section using the same dialog. Use the **OK** button to save the defined section and close the dialog.

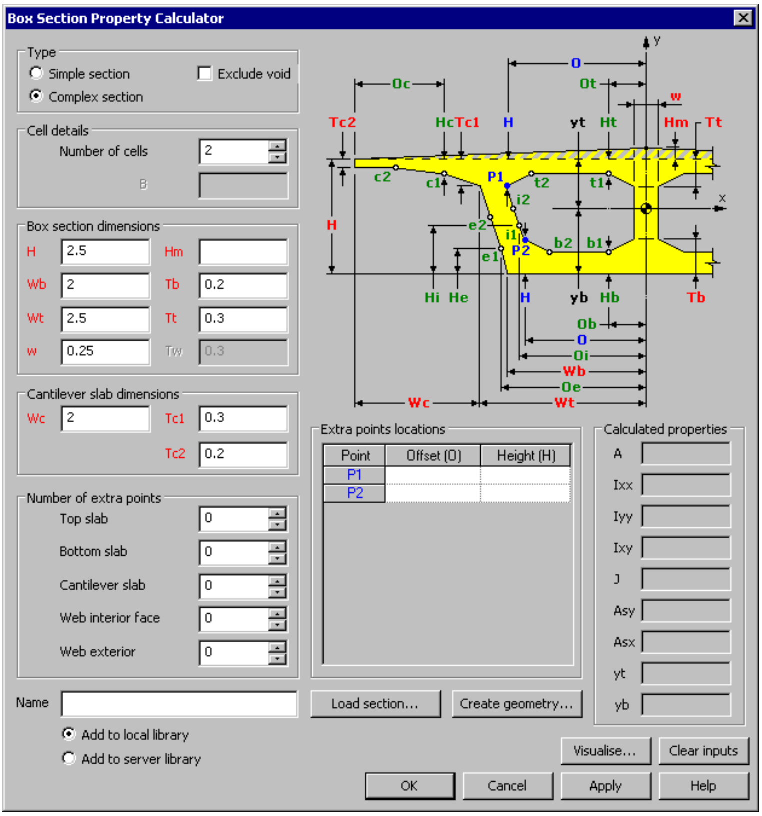
Using a defined simple box section

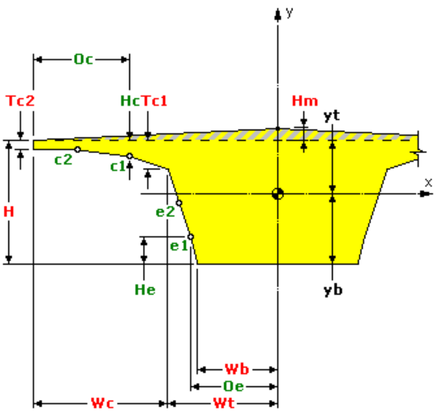
To use the computed section properties in a model the section must have been saved to a local or server library.

To add a library item to the Attributes treeview select the **Attributes**> **Geometric**> **Section Library** menu item, then select **User Sections,** then select **Local** or **Server** before choosing the section required from the list available. The geometric properties can then be **assigned** to the required Line(s) in the model. If the section shape is constant over a line feature a direct assignment to a line can be made. If the section shape varies over a line feature a set of pre-defined box sections can be used with the **Multiple Varying Sections** facility to create a multiple varying section line attribute for assignment to a line or lines on a model.

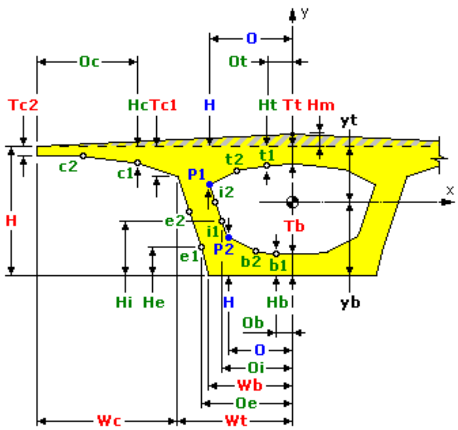
Complex Box Section

The complex box section calculator creates a box section of the style shown from user-defined values and as many additional points as are required to form the cross-sectional shape. A camber in the top slab can be optionally created for one and two celled sections. Fillets can be specified internally when a void is present.

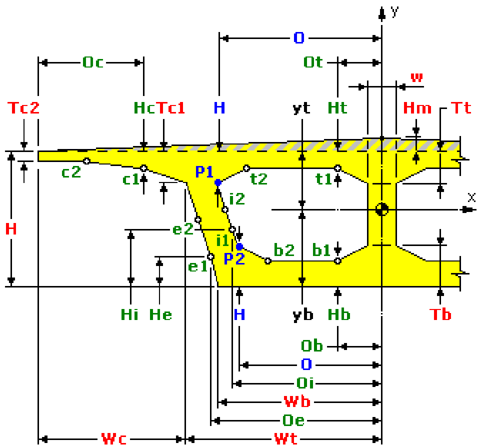




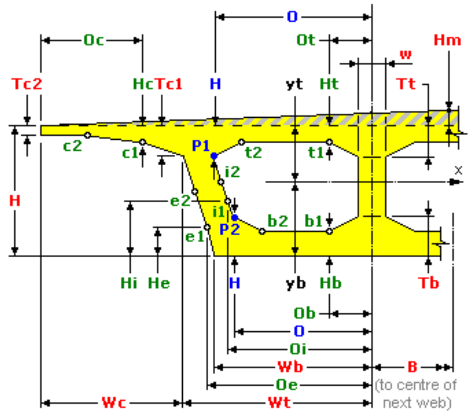
Complex box section without void



One cell complex box section



Two cell complex box section



Multi-cell complex box section

Dimensional data:

- ❑ **Cell details** Specify the number of cells, and the spacing between centre to centre distances of webs
- ❑ **Box section dimensions** These specify the overall height and width of the box and the thicknesses of the top and bottom slabs.
- ❑ **Cantilever slab dimensions** These specify the width and thickness of the cantilever.
- ❑ **No of extra points** These allow for any number of additional points to be defined in specified areas of the cross-section in order to accurately represent the true section shape.When extra points are added, additional dimensional data must be entered in the Points panel of the dialog.

Saving and using the defined section:

- ❑ The **Name** of the section must be entered.
- ❑ **Add to local library** Adds the calculated values to the local library when the Apply or OK button is selected.
- ❑ **Add to server library** Adds the calculated values to the server library when the Apply or OK button is selected.
- ❑ **Load sections...** Displays a list of previously created and saved cross-sections that can be re-selected to populate the fields of the dialog. The list displayed is for the local or server library that is set.
- ❑ **Create geometry** Creates a 2D LUSAS model from the dimensional data. This would typically be used, for example, if it was required to modify the generated section in some way inside LUSAS Modeller before re-calculating the new section properties of the edited section using the **Arbitrary Section Property Calculator**. Additional options are available to position or orientate the section geometry that is created in a view window. By default a section created has its centre of gravity set at the view origin, that is 0,0,0.

- **Visualise...** Shows the cross-section that has been defined by the entered data. It also shows the fibre locations that are automatically created. Note that the axes shown on the visualisation are view axes. An additional option to add a picture of this visualised section to the Annotation layer is provided. Select Create Annotation to do this.
- **Clear inputs** Clears any entered or populated data.
- Use the **Apply** button to save a section to a library and continue to modify values or define another section using the same dialog. Use the **OK** button to save the defined section and close the dialog.

Defining a complex box section

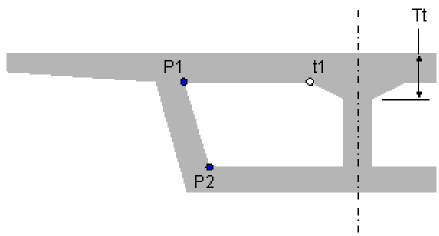
The main box cell, section and cantilever setting out dimensions for a complex box section are shown dimensioned with a height (H), width (W), slab thickness (T) and in the case of three or more celled sections, a breadth, (B). Slab thickness specifies the thickness of the slab at the location shown on the section dialog. Note that on multi-celled box sections, dependent upon the thickness specified, this may represent the distance from the top of the slab to the bottom of an internal fillet (Tt) or the distance from the bottom of the slab to the top of an internal fillet (Tb). In addition to these primary dimensions extra points can be defined to model a particular box section shape. Some extra points are mandatory as in the case of points P1 and P2. Others such as t1, b1, e1, i1, c1 etc. are optional.

Notes on defining a complex box section

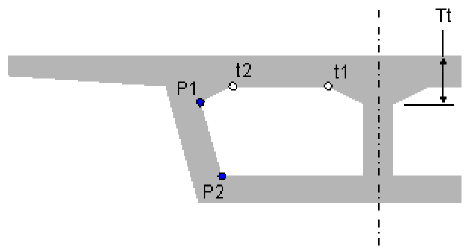
- Points P1 and P2 define the extent of the interior face of the external web. These points must be defined in the Extra Points grid in all cases, even when no fillets are required. Points P1 and P2 will lie at the corners of a void if no fillets are specified.
- Additional points can be defined to either model a particular cantilever shape, a side wall shape or to model simple fillets at top or bottom slab locations. Selecting the location and number of the points required in the No. of extra points cell will add fields to the Extra Points grid to allow offsets (O) and heights (H) for these points to be defined. As an example, to define fillets at the junction of the upper slab and the side and internal walls of a two cell box section, two internal points in the Top slab drop-down list would need to be specified (t1 and t2) in addition to specifying a thickness (Tt) from the top of the slab to the bottom of the fillet. Defining fillets for the lower slab/wall connections would be similar. See the Complex Box Section Examples for details.
- To model a curving external box section profile as many points on the web exterior face and cantilever slab can be defined as necessary to approximate the actual profile.
- A camber in the top slab can be optionally created by entering a value for the height of the midpoint of the slab (Hm).

Complex Box Section Examples

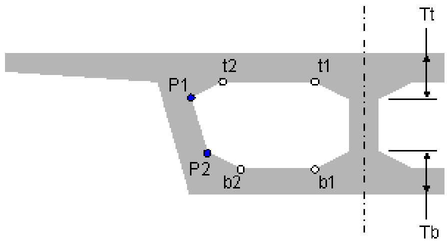
These examples show which extra points must be specified in addition to all other dimensions shown on the dialog in order to obtain the section shape shown. A two-cell box with no camber in the top slab is shown. The definition of extra points for single and multiple box sections is similar.



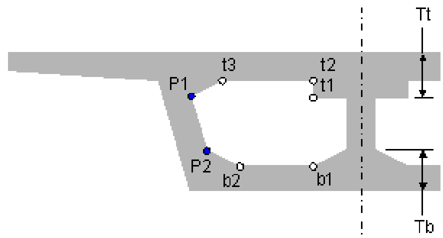
Upper slab fillet to internal wall only



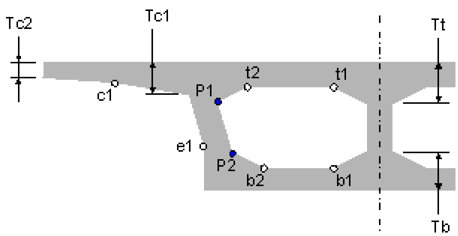
Upper slab fillet to all walls in section



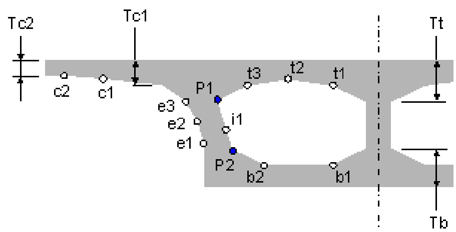
Upper and lower slab fillet to all walls



Custom connection to upper slab



Use of extra points to define outer wall and cantilever soffit



Use of multiple extra points

Complex section extra points explained

Extra points	Description	Mandatory / Optional
P1, P2	Define the extent of the interior face of the outer walls	Mandatory
t1, t2, t3...	Define upper slab fillets.	Optional
b1, b2, b3...	Define lower slab fillets	Optional
e1, e2, e3...	Define the shape of the external face of the outer walls	Optional
i1, i2, i3...	Define the shape of the interior face of the outer walls	Optional
c1, c2, c3...	Define cantilever soffit shape	Optional

Using a defined box complex box section

To use the computed section properties in a model the section must have been saved to a local or server library.

To add a library item to the Attributes treeview select the **Attributes**> **Geometric**> **Section Library** menu item, then select **User Sections,** then select **Local** or **Server** before choosing the section required from the list available. The geometric properties can then be **assigned** to the required Line(s) in the model. If the section shape is constant over a line feature a direct assignment to a line can be made. If the section shape varies over a line feature, a set of pre-defined box sections can be used with the **Multiple Varying Sections** facility to create a multiple varying section line attribute for assignment to a line or lines on a model.

Bridge Loading

Overview

Bridge loading types are accessed from the **Bridge > Bridge Loading** menu item.

The following bridge loading types and facilities are available:

- **Gravity**
- **Surfacing Loading**
- **Static Vehicle Loads**

Gravity Loading

By selecting the **Bridge or Civil> Gravity** menu item a body force loading attribute is added to the Attributes Treeview.

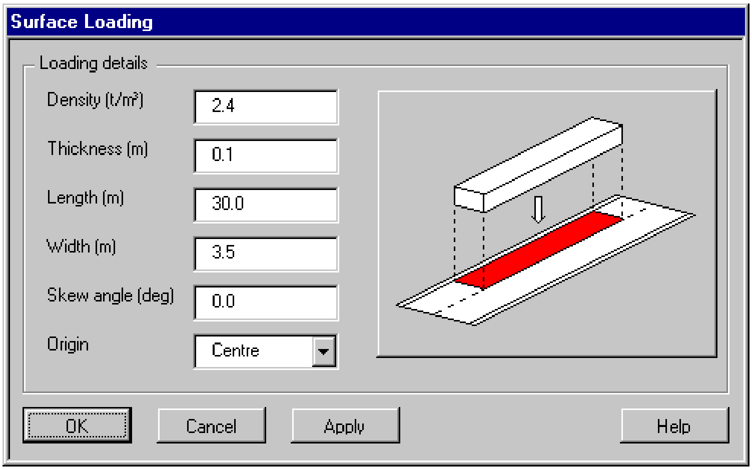
Gravity loading is defined in accordance with the vertical axis direction that was specified either initially on the New Model dialog or subsequently on the Vertical Axis dialog accessed using the **Utilities> Vertical Axis** menu item.

Gravity loading may also can be defined as a property of an analysis or loadcase, or by manually specifying a constant body force load.

Surface Loading

Surface loading can be defined by selecting the **Bridge > Bridge Loading > Surfacing** menu item.

Surface loading is computed for each lane from the density and thickness of the surfacing material. The density is defined in t/m^3 and the dimension of the lane and thickness of the surfacing are defined in metres. The surface loading is then computed in the units selected initially on the New Model dialog.



To use this loading type:

- **Density** - specifies the density of the surfacing in t/m^3
- **Thickness** - specifies the thickness of the surfacing
- **Length** - specifies the length of the surfacing
- **Width** - specifies the width of the surfacing
- **Skew** angle - specifies the skew angle
- **Origin** - specifies the origin of the surfacing load to be used to position the loading on the structure.

Static Vehicle Loads

Static vehicle loading types are accessed from the **Bridge > Bridge Loading** menu item.

A number of dialogs are available to simplify the input of bridge loading in accordance with regional codes of practice. These are continually being extended and currently include:

- **Australia**
- **Canada**
- **China**
- **Denmark**
- **Eurocode**
- **Finland**
- **India**
- **Israel**

- **Korea**
- **NATO**
- **New Zealand**
- **Norway**
- **Poland**
- **South Africa**
- **Sweden**
- **United Kingdom Vehicle Loading**
- **United Kingdom Train Loads**
- **United Kingdom Special Vehicle Loads**
- **United States of America**

Using static vehicle loadings

Vehicle loadings have a loading origin at their centre and, when relevant, have a direction of travel shown or stated on the dialog from which they are created.

Once a vehicle load type is selected or defined, discrete point and/or patch load attributes are added to the Attributes Treeview. For some loading types a compound discrete loading attribute (which comprises a set of discrete point or patch loads to define a loading type) may also be created to represent the static vehicle loading. The required discrete point, patch or compound loading must then be assigned to a point on a model (which need not form part of the model itself). During assignment a patch transformation can be used to mirror, rotate or otherwise transform the discrete patch load that represents the vehicle, to obtain a desired orientation of loading.

Notes

- Vehicle loads are defined in relation to the currently set Vertical axis (as set using the Utilities > Vertical Axis menu item). The longitudinal direction of the vehicle definition is assumed to be alomg the x-axis, unless the x axis is vertical in which case the vehicle load will be defined along the y axis. The vehcile is assumed to move forwards in the positive direction.
- A moving load generator can be used to track the path of a static vehicle load (or a compound set of vehicle loads) across a structure. See **Moving Loads** for details.
- As an alternative to using static vehicle loads, vehicle load optimisation software can be used to automatically identify the most onerous vehicle loading patterns on bridges for a supported design code and to apply these loading patterns to LUSAS models. See **Vehicle Load Optimisation Explained** for details.

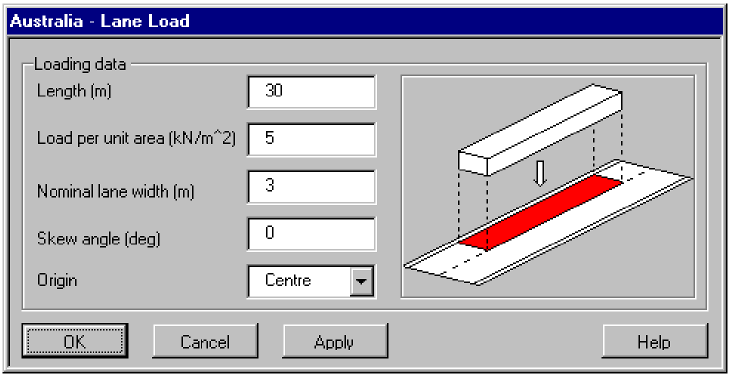
Australia Vehicle Loading

Australian vehicle loads are defined to the AUSTROADS Bridge Design Code HB77.2, AS 5100-2: 2004 : Bridge design - Design loads and AS 5100-7: 2004 : Bridge design - Rating of existing bridges.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.

Australia Lane Loads

The lane load generator produces a uniform patch load based on nominal lane width, loaded length and intensity. The lane load intensity is set to a default value but this can be modified to any value required.



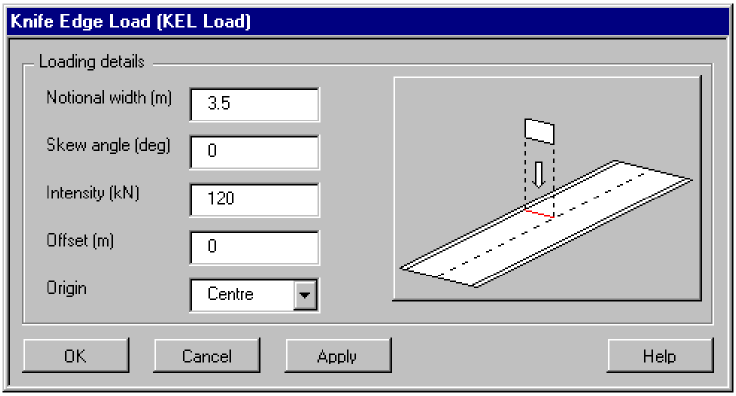
To use this loading type:

- Specify the loaded length.
- Specify the load per unit area.
- Specify the nominal lane width.
- Specify the skew angle to apply to the lane loading (clockwise positive).

- Choose the origin about which the load is to be generated.

Australia Knife Edge Loads (KEL) Loads

The KEL load generator produces a knife-edge load based on notional lane width and intensity. The intensity of the knife-edge load is set to a default value but can be modified to any value required.



To use this loading type:

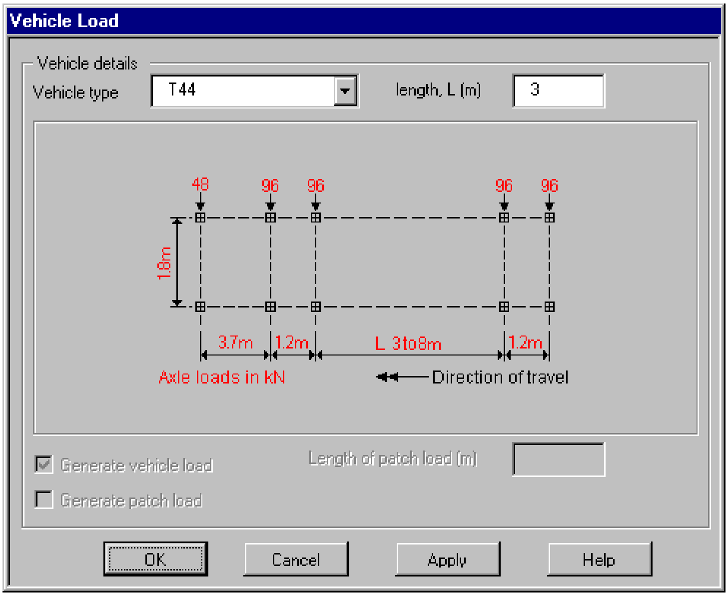
- Specify the notional width.
- Specify the skew angle to apply to the lane loading (clockwise positive).
- Specify the intensity.
- Specify the offset (longitudinal).
- Choose the origin about which the load is to be generated.

Australia Vehicle Load

For the vehicle loads the following truck types can be created: T44, HLP320, HLP400, W80, A160, M1600 and S1600.

For the truck types T44, HLP320, HLP400, M1600 and S1600 the variable axle spacing, L, can be set to the required value.

For the truck types M1600 and S1600 the vehicle load and the accompanying patch can also be created.



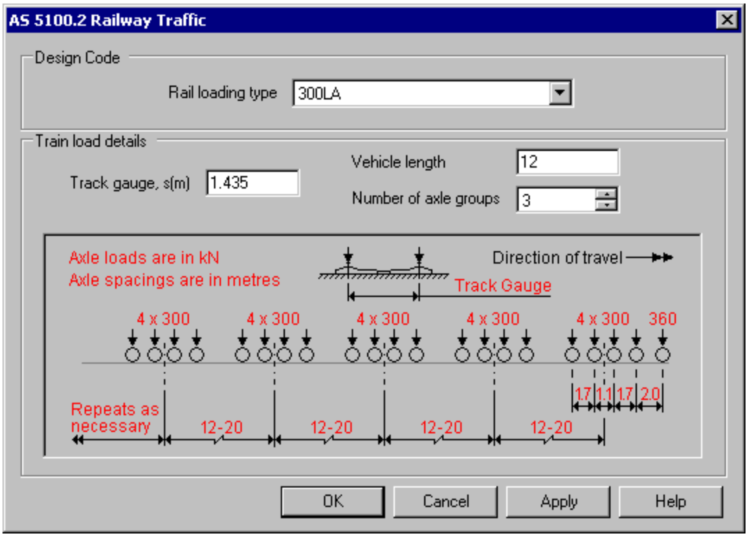
To use this loading type:

- Choose the vehicle type, and if necessary specify a value for the length.

Australia AS5100 Railway Loading

Australia Railway Traffic loading is accessed from the Australia Vehicle Loading dialog:

Three rail vehicle configurations are supplied to Australia Vehicle Loading in accordance with AS5100-2 representing vehicle 300LA, 300-A-12 and 300-A-12 (Single Axle). The 300LA vehicle is defined in accordance with the current standard and is the default. Vehicle 300-A-12 is added for assessment carried out in accordance with AS5100-7 “Bridge Design – Rating of Existing Structures.”

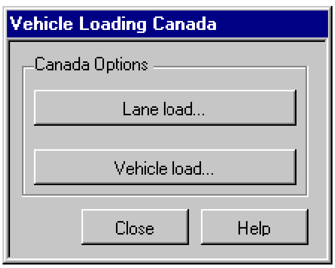


- **Rail loading type** Loading type 300LA is defined in accordance with AS5100-2 and is the default choice. Loading type 300-A-12 is for assessment carried out in accordance with AS5100-7 "Bridge Design – Rating of Existing Structures." Loading type 300-A-12 (Single Axle) provides a pair of loads equal to half the stated axle load.
- **Track gauge** Set to 1.435m, but can be user-defined.
- **Vehicle length** (for loading type 300LA only). Set to 12 by default.
- **Number of axle groups** Set to 3 by default

Canada Vehicle Loading

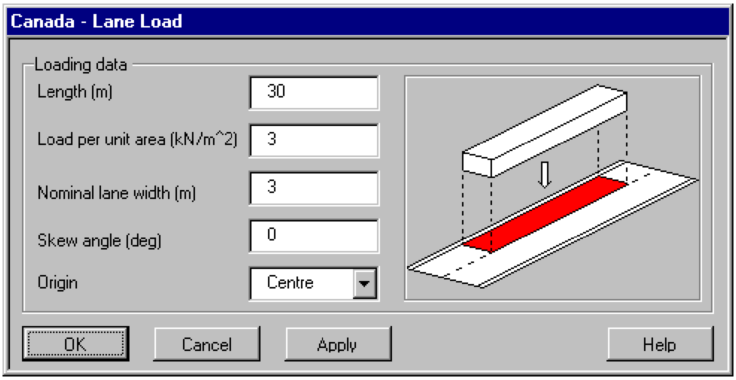
Canadian vehicle loads are defined to the Canadian Highway Bridge Design Code (CHBDC.)

See **Using static vehicle loadings** for general details regarding static vehicle loadings.



Canada Lane Loads

The lane load generator produces a uniform patch load based on the nominal lane width, loaded length and intensity. The lane load intensity is set to a default value but this can be modified to any value required.



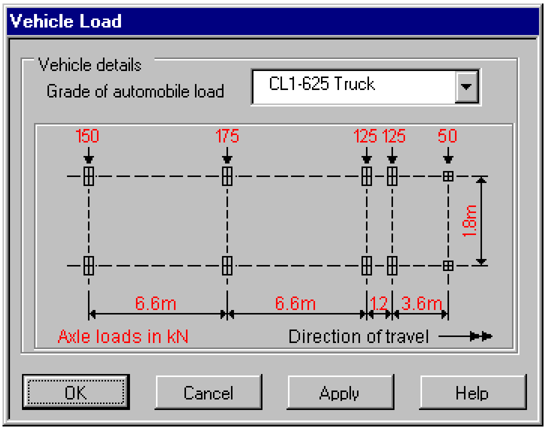
To use this loading type:

- Specify the loaded length.
- Specify the load per unit area.
- Specify the nominal lane width.
- Specify the skew angle to apply to the lane loading (clockwise positive).
- Choose the origin about which the load is to be generated.

Canada Vehicle Load

For the vehicle loads the following types can be created: CL1-625, CL2-625, CL3-625, CL1-8--, CL2-800, CL3-800 and CHBDC Maintenance. The load intensity maybe reduced to 80% for use with lane loading and the Ontario variations are provided.

The vehicle loads represent the CL-W truck. Two vehicle weights are available: 625 and 800. Three variations of each vehicle weight can be selected, namely, CL1, CL2 and CL3, representing five, four and three axles of the CL-W truck respectively. The load intensity of each vehicle may be reduced to 50% for use with lane loading and the Ontario variations of CL-625 truck are provided. The maintenance vehicle is defined in fig 3.4 of the design code is also available.

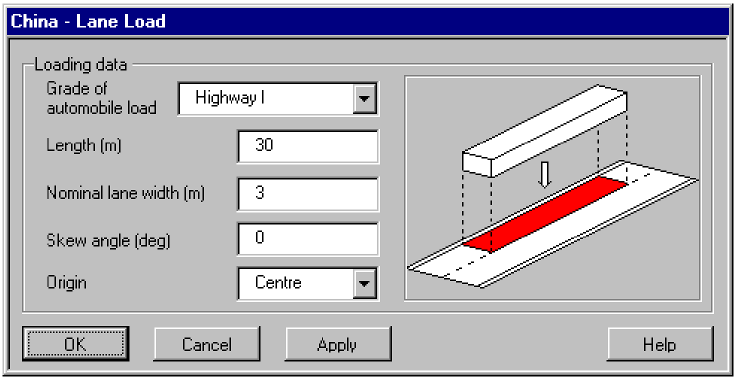


China Vehicle Loading

Chinese vehicle loads are defined to the JTG D60-2004 General Code for Design of Highway Bridges and Culverts.

China Lane Loads

The lane load generator produces a uniform patch load based on notional lane width, loaded length and grade of automobile loading (Highway I, Highway II and Highway II – fourth grade). The intensity of the patch is calculated based on the loaded length entered with shorter loaded lengths having higher intensity.

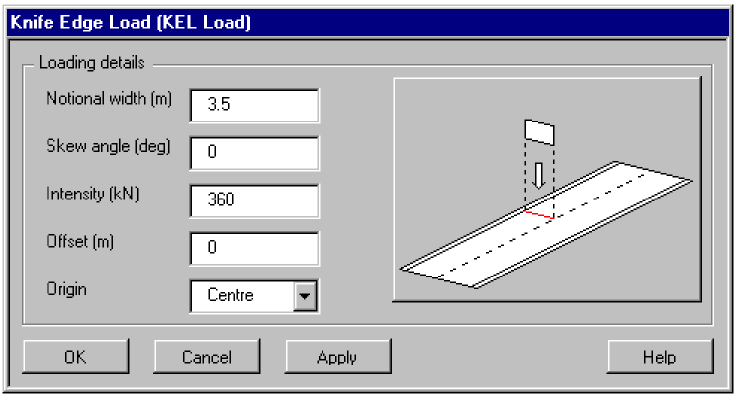


To use this loading type:

- Choose the grade of automobile load.
- Specify the length of loading.
- Specify the nominal lane width.
- Specify the skew angle to apply to the lane loading (clockwise positive).
- Choose the origin about which the load is to be generated.

China Knife Edge Loads (KEL) Loads

The KEL load generator produces a knife-edge load based on notional lane width and intensity. The intensity of the knife-edge load has a default value set but can be modified to any value required.



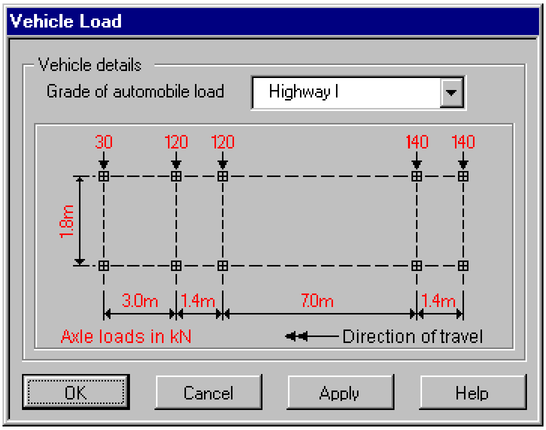
To use this loading type:

- Specify the notional width.

- Specify the skew angle to apply to the lane loading (clockwise positive).
- Specify the load intensity.
- Specify the offset.
- Choose the origin about which the load is to be generated.

China Vehicle load

The vehicle load generator produces truck loads based on grade of automobile loading (Highway I, Highway II and Highway II – fourth grade).



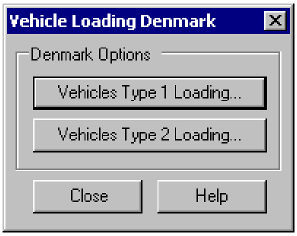
To use this loading type:

- Choose the grade of automobile required.

Denmark Vehicle Loading

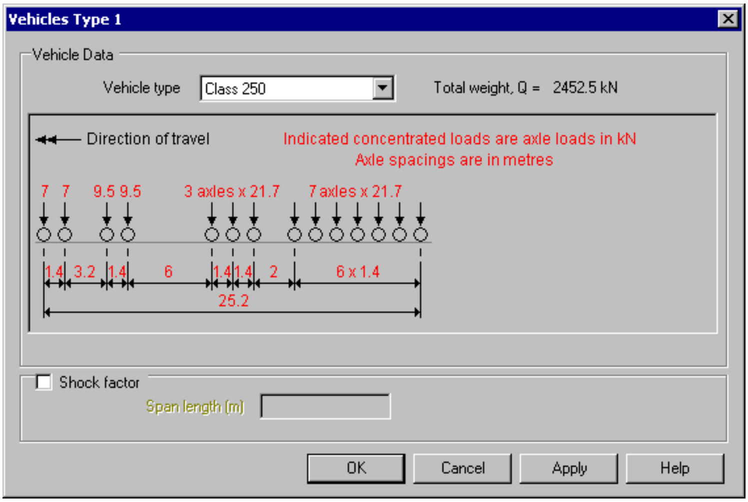
Denmark vehicle loads are defined in accordance with the special vehicles defined as Load Model 3 (LM3) in the Danish National Annex to EN1991-2. This comprises 22 vehicles of fixed axle weight and spacing that are denoted as “Type 1” and a further 8 vehicles that are denoted as “Type 2” for assessment purposes.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.



Denmark Vehicle Type 1 Loading

Vehicle type can be selected from a range that extends from Class 10 to Class 500. A shock factor can be optionally specified for a particular span length.

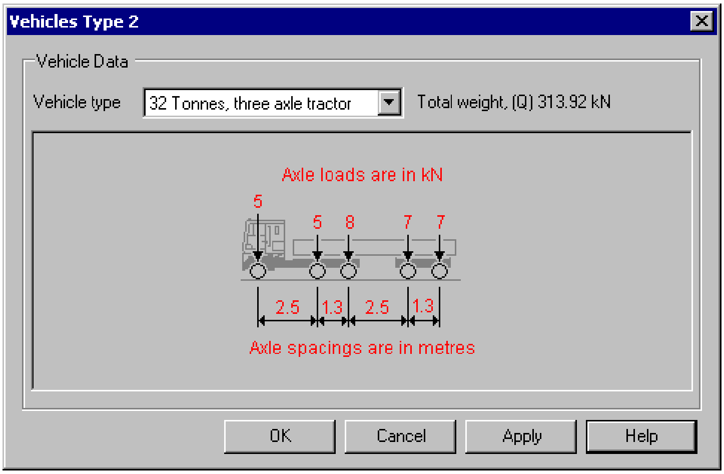


To use this loading type:

- Choose the vehicle type from the drop down list.
- Specify an optional Shock factor for a specified span length.

Denmark Vehicle Type 2 Loading

Vehicle type can be selected from a range that extends from 16 Tonnes to 48 Tonnes, with the 24 Tonnes and 32 Tonnes options including a two or three-wheeled trailer.



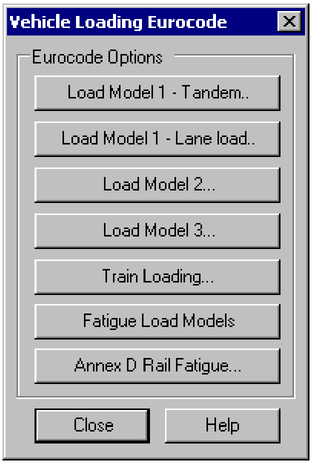
To use this loading type:

- Choose the vehicle type from the drop down list.

Eurocode Vehicle Loading

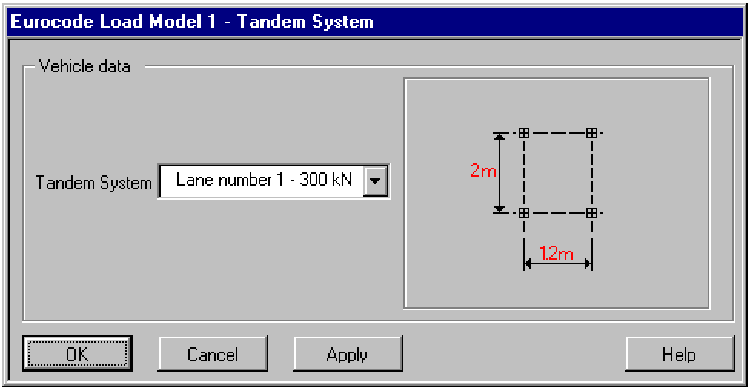
Eurocode vehicle loads are defined to EN1991-2:2003 Eurocode 1: Actions on structures - Part 2: Traffic loads on bridges. Note that any country-specific load models (incorporated as part of a country's National Annex to EN1991-2:2003) are documented in the static vehicle load types for that country.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.



Eurocode Load Model 1 (LM1) - Tandem

The Load Model 1 (LM1) - Tandem load generator produces a tandem system load comprising two axles of intensity 300kN, 200kN and 100kN, for use in Lane numbers 1, 2 and 3 as defined by the design code.

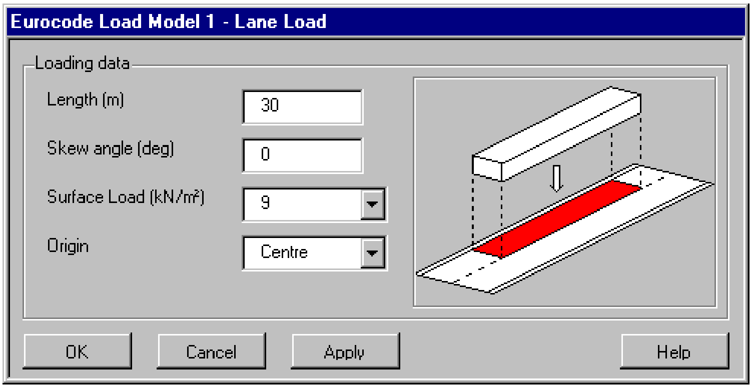


To use this loading type:

- Choose the tandem system from the drop down list.

Eurocode Load Model 1 (LM1) - Lane load

The Load Model 1 (LM1) – Lane load generator produces a uniform patch load based on loaded length and required intensity (9kN/m2 and 2.5kN/m2).



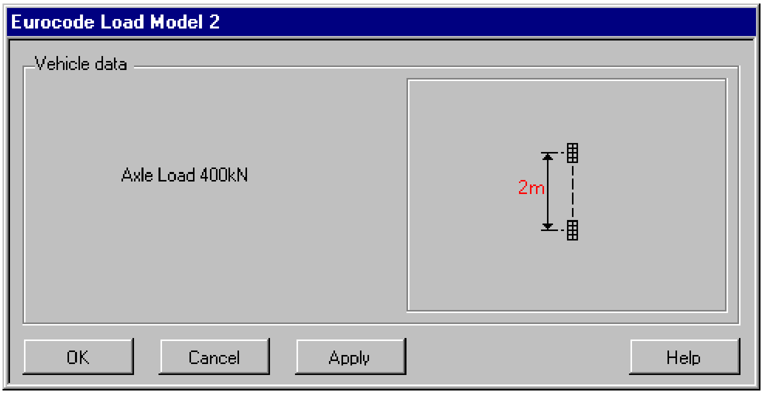
To use this loading type:

- Specify the length of lane load.
- Specify the skew angle to apply to the lane loading (clockwise positive).

- Choose the surface load.
- Choose the origin about which the load is to be generated about.

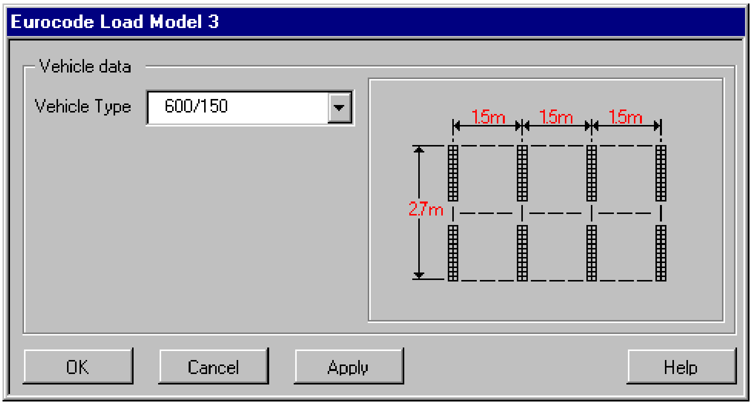
Eurocode Load Model 2 (LM2)

The Load Model 2 (LM2) load generator produces a single axle load of 400kN.



Eurocode Load Model 3 (LM3)

The Load Model 3 (LM3) load generator produces the following special vehicle loads, as defined in Annex A: 600/150, 900/150, 1200/150, 1200/200, 1500/150, 1200/200, 1800/150, 1800/200, 2400/200, 2400/240, 2400/200/200, 3000/200, 3000/240, 3000/200/200, 3600/200, 3600/240 and 3600/200/200.



To use this loading type:

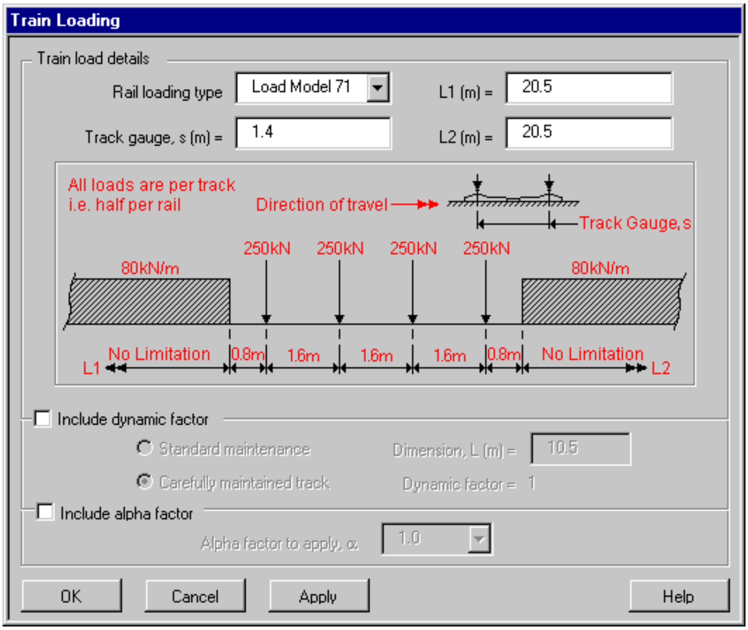
- Choose the special vehicle from the drop down list.

Classes of Special Vehicles

Total weight	Composition	Notation
600 kN	4 axle-lines of 150 kN	600/150
900 kN	6 axle-lines of 150 kN	900/150
1200 kN	8 axle-lines of 150 kN	1200/150
	or 6 axle-lines of 200 kN	1200/200
1500kN	10 axle-lines of 150 kN	1500/150
	or 7 axle-lines of 200 kN + 1 axle line of 100 kN	1500/200
1800 kN	12 axle-lines of 150 kN	1800/150
	or 9 axle-lines of 200 kN	1800/200
2400 kN	12 axle-lines of 200 kN	2400/200
	or 10 axle-lines of 240 kN	2400/240
	or 6 axle-lines of 200 kN (spacing 12m) + 6 axle-lines of 200 kN	2400/200/200
3000 kN	15 axle-lines of 200 kN	3000/200
	or 12 axle-lines of 240 kN + 1 axle-line of 120 kN	3000/240
	or 8 axle-lines of 200 kN (spacing 12 m) + 7 axle-lines of 200 kN	3000/200/200
3600kN	18 axle-lines of 200 kN	3600/200
	or 15 axle-lines of 240 kN	3600/240
	or 9 axle-lines of 200 kN (spacing 12 m) + 9 axle-lines of 200 kN	3600/200/200

Eurocode Train Loading

Eurocode train loading is accessed from the Eurocode Vehicle Loading dialog:



- ❑ **Rail loading type** Standard railway loading consists of Load Model 71, and Load Model SW/0 (for continuous bridges) to represent normal rail traffic on mainline railways and Load Model SW/2 to represent heavy loads. HSLM-A1 to A10 and HSLM-B loadings are High Speed Load Model (HSLM) to EN1991-2:2003. Section 6.4.6.

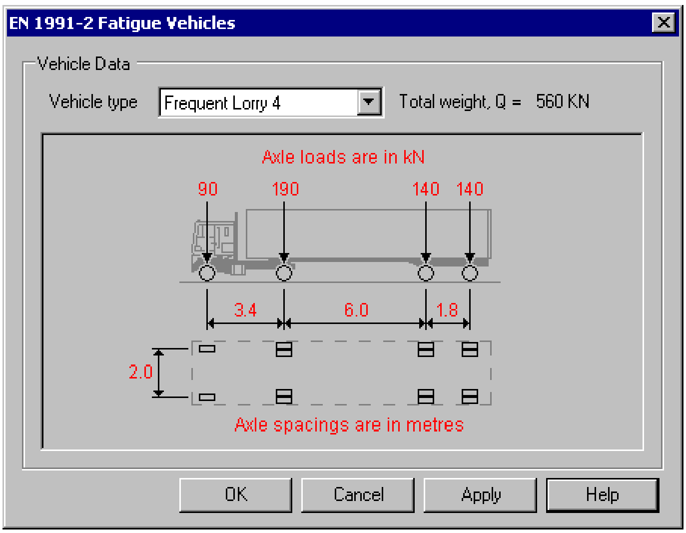
    - Load Model 71 represents the static effect of vertical loading due to normal rail traffic. It consists of four 250kN concentrated loads preceded, and followed, by a uniformly distributed load of 80kN/m.
    - Load Model SW/0 represents the static effect of vertical loading due to normal rail traffic on continuous beams. It consists of two uniformly distributed loads of 133kN/m, each 15m long and separated by a distance of 5.3m.
    - Load Model SW/2 represents the static effect of vertical loading due to heavy rail traffic. It is similar to SW/0 Loading, however, the uniformly distributed loads are 150kN/m, each 25m long and separated by a distance of 7m.
    - HSLM-A1 to A10 vehicles comprise a series of coaches of varying length and axle weights pulled by a standard power car. This load type is only applicable when carrying out a dynamic analysis.
    - HSLM-B loading varies depending on the supplied span length and is applicable for simply supported short spans of less than 7m. This load type is only applicable when carrying out a dynamic analysis.

- ❑ **Track gauge** must be entered in metres and has to be 1.4m or greater.

- **L1 and L2** are entered as appropriate for the load model chosen, as indicated on the dialog diagram. All dimensions are in metres.
- **Dynamic factor** The dynamic factor is applicable to Load Models 71, SW/0 and SW/2 and allows for impact, oscillation and other dynamic effects including those caused by track and wheel irregularities. The dynamic factor takes account of the dynamic magnification of stresses and vibration effects in the structure but does not take account of resonance effects. The factor used will vary according to determinant length (*L*) and whether the track receives standard maintenance or is carefully maintained. In deriving the dynamic factor, *L* is taken as the length (in m) of the influence line for deflection of the element under consideration. For non-symmetrical influence lines, *L* is twice the distance between the point at which the greatest ordinate occurs and the nearest end point of the influence line. In the case of floor members 3m should be added to the length of the influence line as an allowance for load distribution through track
- **Alpha factor** The characteristic values of Load Model 71 and SW/0 Loading shall be multiplied by a factor α, on lines carrying rail traffic which is heavier or lighter than normal rail traffic (Clause 6.4.5). When multiplied by the factor the loads are called "classified vertical loads" (Clause 6.3.2(3).

Eurocode Fatigue Load Models

Eurocode Fatigue Load Model loading is accessed from the Eurocode Vehicle Loading dialog:

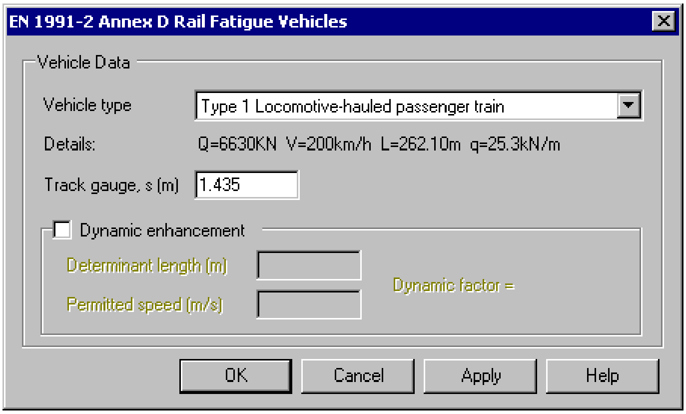


- **Vehicle type** Highway fatigue vehicles for Fatigue Load Models 2, 3 and 4 in accordance with section 4.6 of EN1991-2.
  - **Frequent Lorry (1 to 5)** Set of "frequent lorries" for Fatigue Load Model 2.

- **LM3 Single vehicle** Four axle loading in accordance with Fatigue Load Model 3.
- **Standard Lorry (1 to 5)** Set of "standard lorries" for Fatigue Load Model 4.

Eurocode Annex D Rail Fatigue Vehicles

Eurocode Rail Fatigue Vehicle loading is accessed from the Eurocode Vehicle Loading dialog:



- ❑ **Vehicle type** Rail fatigue vehicles in accordance with EN1991-2 Annex D. The rail fatigue vehicles comprise twelve different train type and loading arrangements.
  - **Type (1-12)** Set of twelve rail fatigue vehicles.
  - **Track gauge** Set to 1.435m but can be user-defined.
  - **Dynamic enhancement** Applies a dynamic factor to the static vehicle description. The dynamic factor is calculated from the determinant length and permitted speed in accordance with Annex D.1.

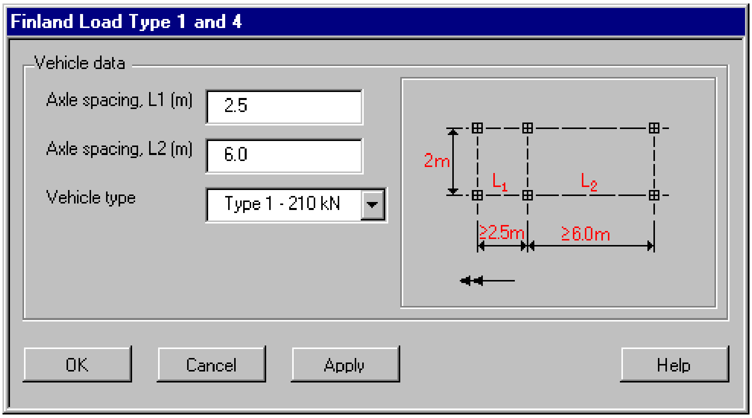
Finland Vehicle Loading

Finland vehicle loads are defined to the Finnish loading design code TIEL 2172072-99 and the National Annex EN1991-2

See **Using static vehicle loadings** for general details regarding static vehicle loadings.

Finland Load Types 1 and 4

The load generator for load types 1 and 4 produces vehicle loads based on the selection of Type 1 or 4. The variable axle spacings can be set to the required distance.

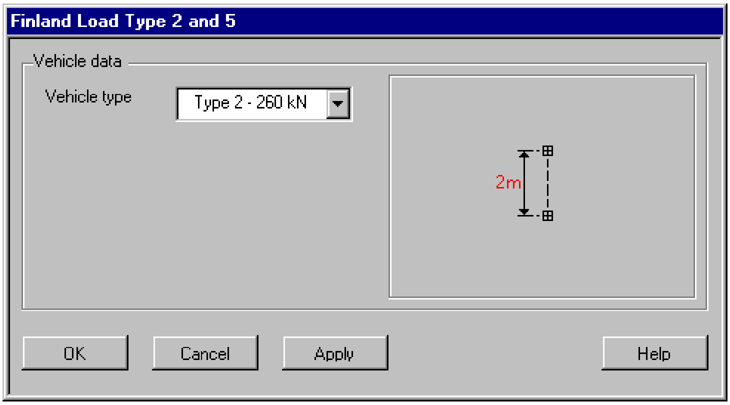


To use this loading type:

- Specify the axle spacings for L1 and L2. Note that L1 has to be greater than 2.5m and L2 greater than 6m.
- Choose the vehicle type from the drop down list.

Finland Load Types 2 and 5

The load generator for load types 2 and 5 produces vehicle loads based on the selection of Type 2 or 5.

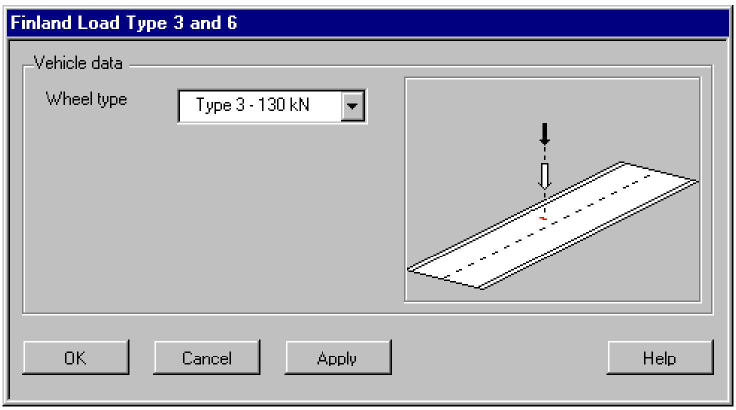


To use this loading type:

- Choose the vehicle type from the drop down list.

Finland Load Types 3 and 6

The load generator for load types 3 and 6 produces vehicle loads based on the selection of Type 3 or 6.

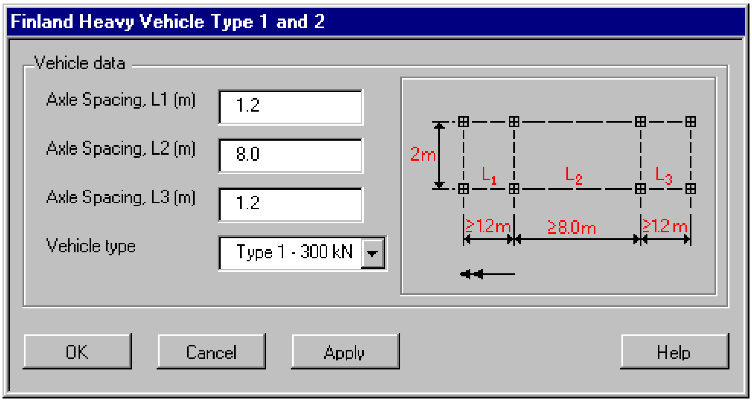


To use this loading type:

- Choose the vehicle type from the drop down list.

Finland Heavy Vehicle Types 1 and 2

The load generator for heavy vehicle types 1 and 2 produces vehicle loads based on the selection of Type 1 or 2. The variable axle spacings can be set to the required distance.

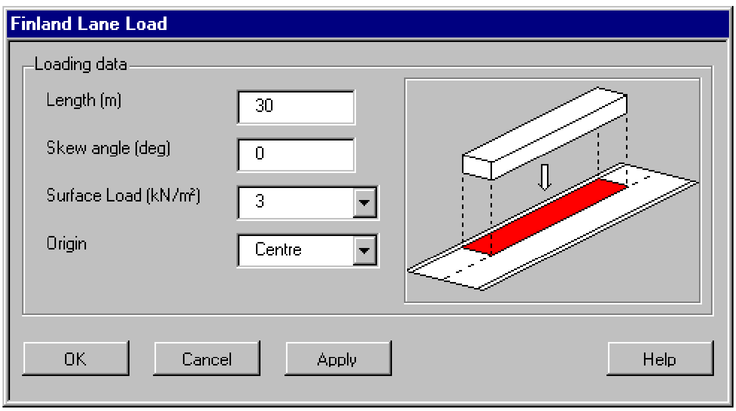


To use this loading type:

- Specify the axle spacings for L1, L2 and L3. Note that L1 and L3 must be greater than 1.2m, and L2 greater than 8m.
- Choose the vehicle type from the drop down list.

Finland Lane Loads

The lane load generator produces a uniform patch load based on loaded length and intensity. The intensity of the lane load is taken from Class 1 or Class II of Load Model 1. The lane width is 3m.



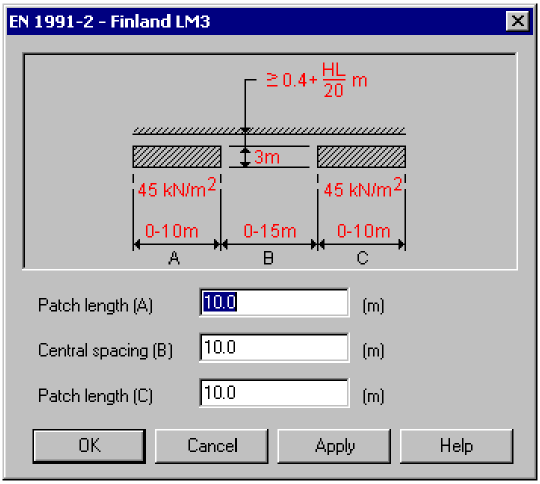
To use this loading type:

- Specify the length of lane load that you require to generate about the centre line of patch.
- Specify the skew angle to applied to the lane loading (clockwise positive).

- Choose the surface load intensity from the drop down list.
- Choose the origin that the load will be generated about.

EN 1991-2-Finland LM3

Special vehicle loading as defined as Load Model 3 (LM3) in the Finnish National Annex to EN1991-2. The loading comprises two patch loads separated by a stated distance (Clause 4.3.4).



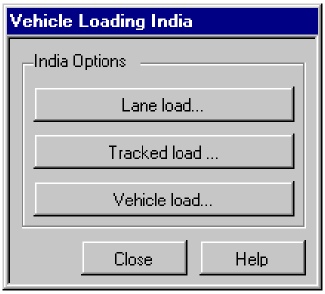
To use this loading type:

- Specify the patch length and central spacing.

India Vehicle Loading

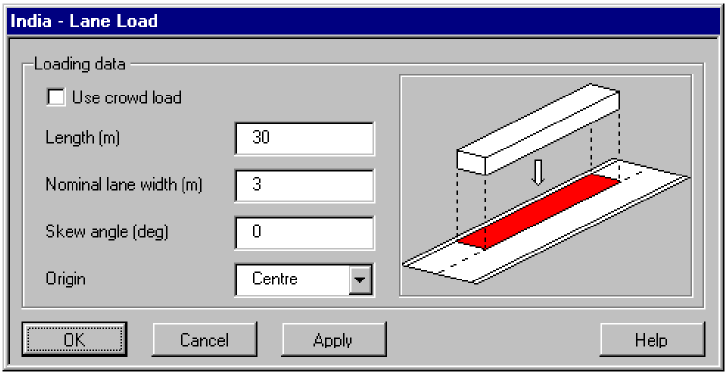
Indian vehicle loads are defined to IRC:6-2000 Section: II Loads and Stresses.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.



India Lane Load

The Lane load generator produces a uniform patch load based on loaded length and notional lane width. The option to include crowd load can be added to the loading.

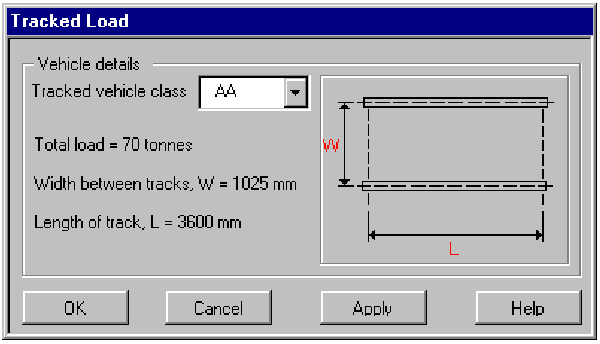


To use this loading type:

- Choose **Use crowd load** if applicable.
- Specify the length of lane load.
- Specify the nominal lane width in metres
- Specify the skew angle to apply to the lane loading (clockwise positive).
- Choose the origin about which the load is to be generated.

India Tracked Load

The tracked load generator produces the following tracked vehicle loads: AA, 5R, 9R, 12R, 18R, 24R, 30R, 40R, 50R, 60R and 70R.

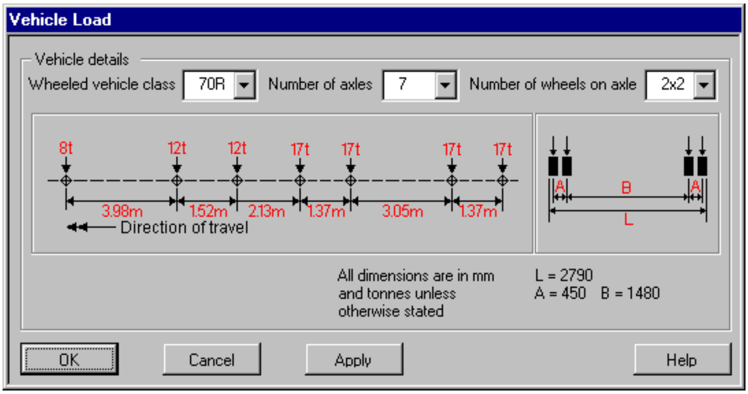


To use this loading type:

- Choose the tracked vehicle class.

India Vehicle Load

The India load generator produces the following vehicle loads: 3, 5R, 9R, 12R, 18R, 24R, 30R, 40R, 50R, 60R, 70R A, B and AA. Each of these vehicles can have many different configurations by setting the number of axles and wheels.



To use this loading type:

- Choose the wheeled vehicle class.
- Choose the number of axles.
- Choose the number of wheels on the axle.

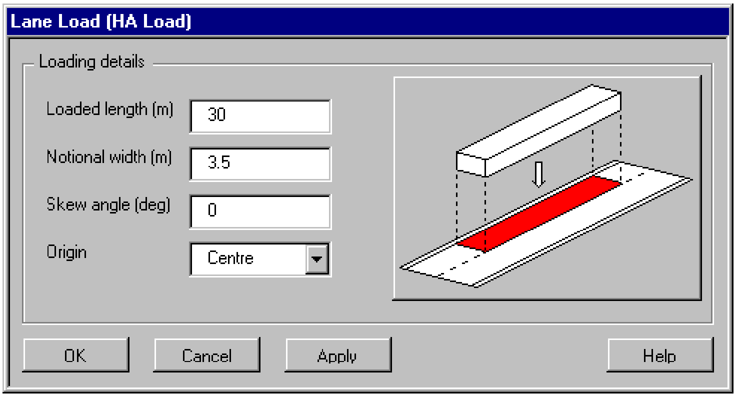
Israel Vehicle Loading

Israel vehicle loads are defined to the Israeli loading design code.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.

Israel Lane Load (HA Loads)

The HA load generator produces a uniform patch load based on notional lane width and loaded length. The intensity of the patch is calculated based on the loaded length entered with shorter loaded lengths having higher intensity

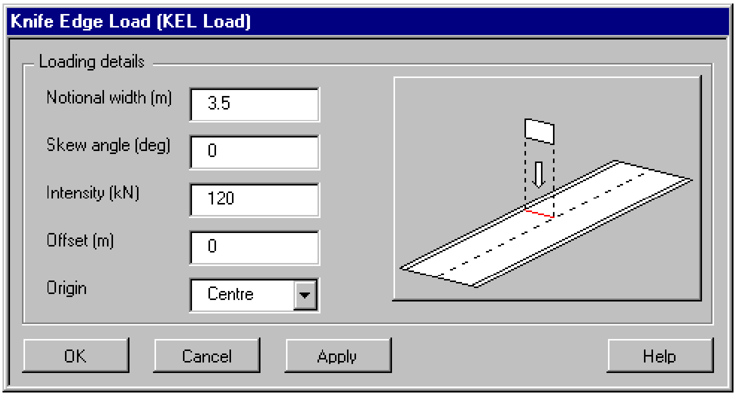


To use this loading type:

- Specify the loaded length.
- Specify the notional width.
- Specify the skew angle to apply to the lane loading (clockwise positive).
- Choose the origin about which the load is to be generated.

Israel Knife Edge Load (KEL Loads)

The KEL load generator produces a knife-edge load based on notional lane width and intensity. The intensity of the knife-edge load has a default value set but this can be modified to any value required.

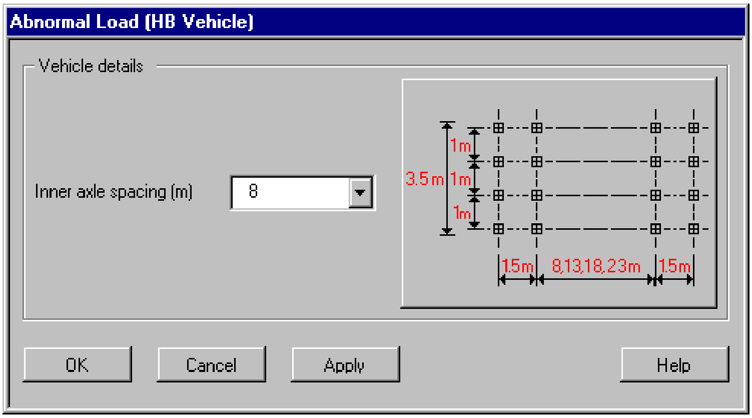


To use this loading type:

- Specify the notional width.
- Specify the skew angle to apply to the lane loading (clockwise positive).
- Specify the intensity
- Specify the offset.
- Choose the origin about which the load is to be generated.

Israel Abnormal Load Generator (HB Vehicle)

The HB load generator produces a HB vehicle by setting the inner axle spacing (8m, 13m, 18m, & 23m).

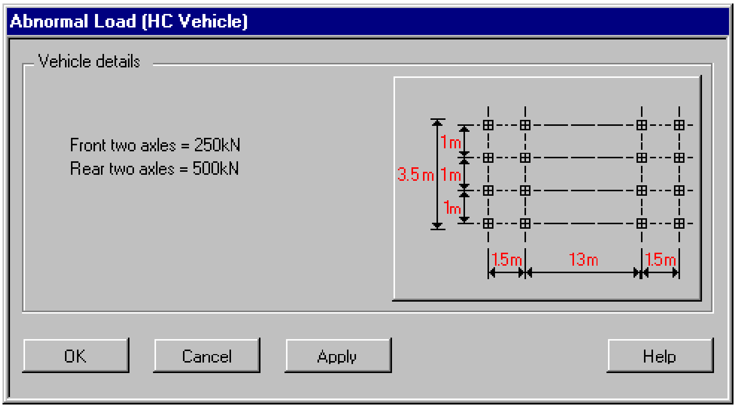


To use this loading type:

- Choose the inner axle spacing.

Israel Abnormal Load Generator (HC Vehicle)

The HC load generator produces a HC vehicle.



Korea Vehicle Loading

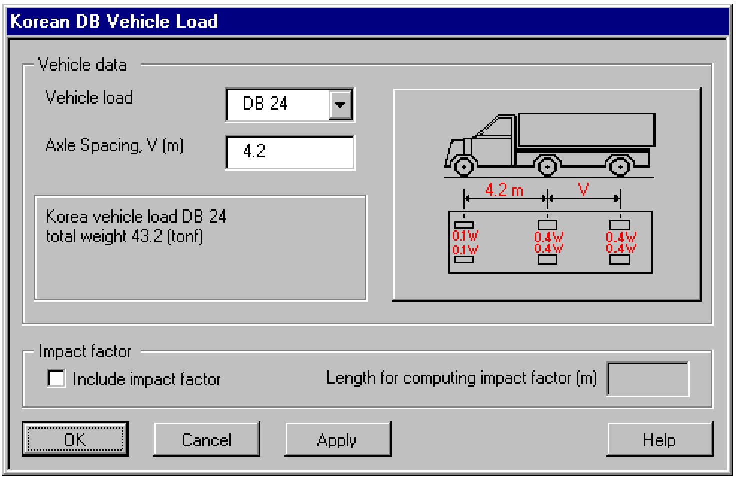
Korean vehicle loads are defined to the Korean loading code.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.

Korean DB Vehicle Load

The DB vehicle load generator can produce vehicle loads types DB24, DB18 and DB13.5 including the variable axle spacing which can be set between 4.2m and 9m.

For all the loads types the additional impact factor can be added to the loads based on loaded length.



To use this loading type:

- Choose which vehicle you require to generate as a vehicle load.
- Specify the variable axle spacing that you require for the vehicle.
- Choose whether you want to generate a vehicle to represent a forward movement, reverse movement or both directions (Forward is in the negative X direction with the cab at the front).
- If an impact factor is to be considered, select the check box and enter a length for computing the impact factor in the current length units. The impact allowance is a maximum of 30 % making the impact factor a maximum of 1.3. Assuming the model length units to be metres the impact factor is calculated from the equation below.

  $$I=15/(40+L)$$

  where L=length for computing the impact factor in metres

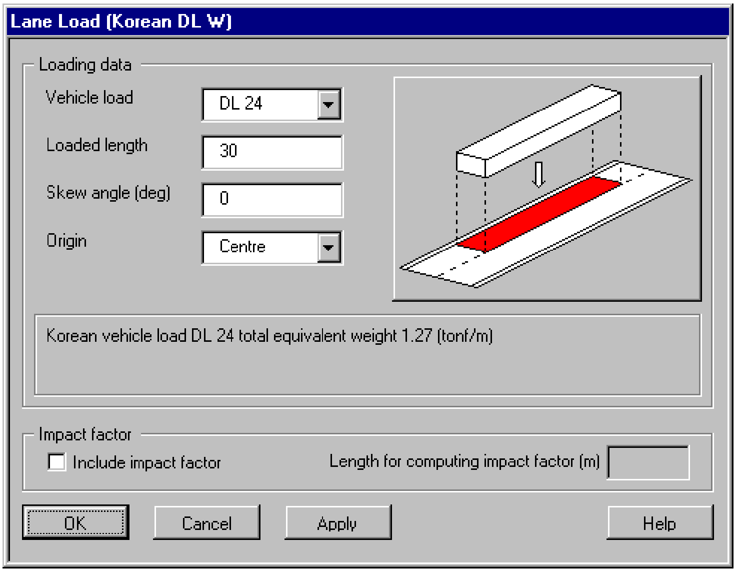
  This equation will automatically be adjusted to take in to account other model units if used. For example if the model length units are feet the impact factor is calculated from the equation below.

  $$I=(15*3.28)/((40*3.28)+L)$$

  where L=length for computing the impact factor in feet

Korea Lane Load (DL W)

The lane load generator can produce a uniform patch load based on load types DB24, DB18 and DB13.5 and loaded length. For all the loads types the additional impact factor can be added to the loads based on loaded length.



To use this loading type:

- Choose which vehicle you require to generate as a vehicle load.
- Specify the length of lane load that you require to generate.
- Specify the skew angle to apply to the lane loading. (clockwise positive)
- Choose the origin for which the load is to be generated about.
- If an impact factor is to be considered, select the check box and enter a length for computing the impact factor in the current length units. The impact allowance is a maximum of 30 % making the impact factor a maximum of 1.3. Assuming the model length units to be metres the impact factor is calculated from the equation below.

  $$I=15/(40+L)$$

  where L=length for computing the impact factor in metres

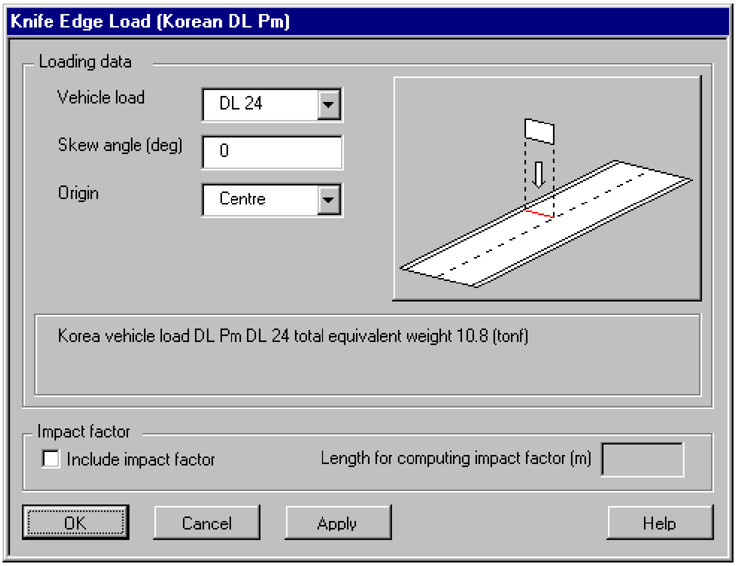
  This equation will automatically be adjusted to take in to account other model units if used. For example if the model length units are feet the impact factor is calculated from the equation below.

  $$I=(15*3.28)/((40*3.28)+L)$$

  where L=length for computing the impact factor in feet

Korea Knife Edge Load (DL Pm)

The knife edge load (Korean DL Pm) generator can produce a line load based on the following load types DB24, DB18 and DB13.5. For all the loads types the additional impact factor can be added to the loads based on loaded length.



To use this loading type:

- Choose which vehicle you require to generate as a vehicle load.
- Specify the skew angle to applied to the lane loading (clockwise positive).
- Choose the origin for which the load is to be generated about.
- If an impact factor is to be considered, select the check box and enter a length for computing the impact factor in the current length units. The impact allowance is a maximum of 30 % making the impact factor a maximum of 1.3. Assuming the model length units to be metres the impact factor is calculated from the equation below.

$$I=15/(40+L)$$

where L=length for computing the impact factor in metres

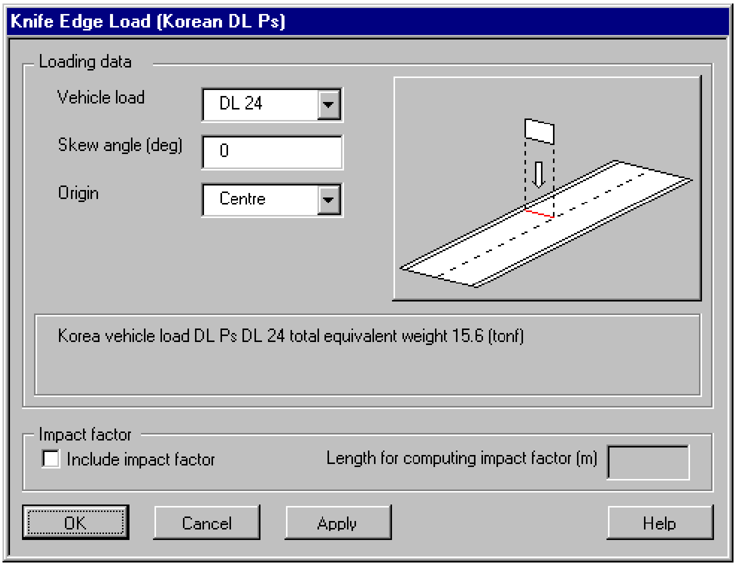
This equation will automatically be adjusted to take in to account other model units if used. For example if the model length units are feet the impact factor is calculated from the equation below.

$$I=(15*3.28)/((40*3.28)+L)$$

where L=length for computing the impact factor in feet

Korea Knife Edge Load (DL Ps)

The knife edge load (Korean DL Ps) generator can produce a line load based on the following load types DB24, DB18 and DB13.5. For all the loads types the additional impact factor can be added to the loads based on loaded length.



To use this loading type:

- Choose which vehicle you require to generate as a vehicle load.
- Specify the skew angle to applied to the lane loading (clockwise positive).
- Choose the origin for which the load is to be generated about.
- If an impact factor is to be considered select the check box and enter a length for computing the impact factor in the current length units. The impact allowance is a maximum of 30 % making the impact factor a maximum of 1.3. Assuming the model length units to be metres the impact factor is calculated from the equation below.

  $$I=15/(40+L)$$

  where L=length for computing the impact factor in metres

  This equation will automatically be adjusted to take in to account other model units if used. For example if the model length units are feet the impact factor is calculated from the equation below.

  $$I=(15*3.28)/((40*3.28)+L)$$

  where L=length for computing the impact factor in feet

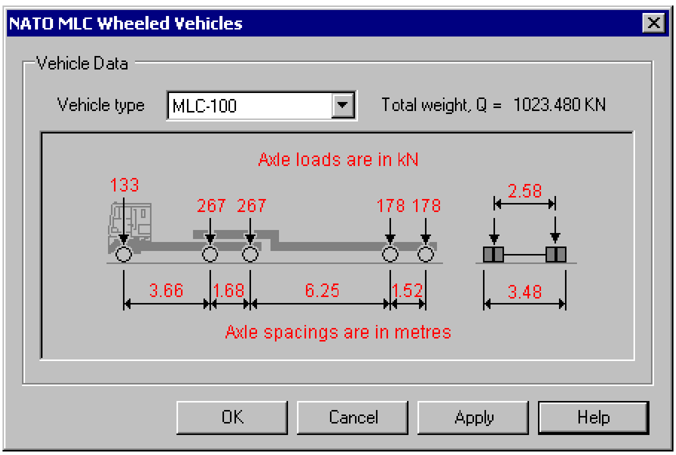
NATO Vehicle Loading

This comprises NATO vehicle loads as defined in annex A of STANAG 2021. Sixteen Military Load Classification (MLC) vehicle classes, each comprising a “Tracked” and corresponding “Wheeled” vehicle are listed , making thirty-two vehicles in total.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.

NATO MLC Wheeled Vehicles

A range of wheeled vehicles for classes MLC-4 to MLC-150 are supplied.

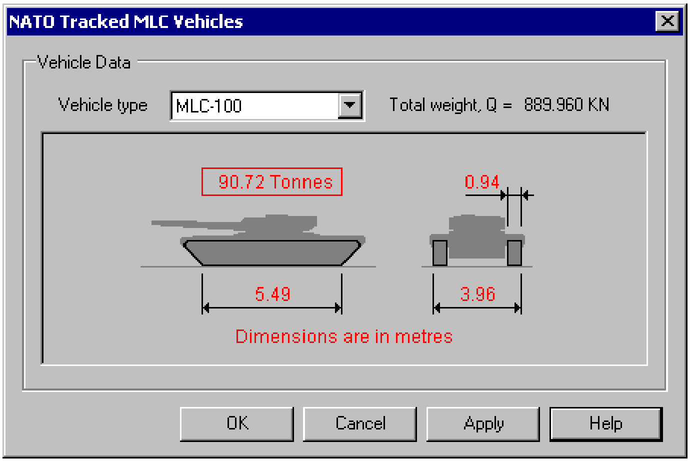


To use this loading type:

- Choose the vehicle type from the drop down list.
- A discrete point load attribute is generated in the Attributes Treeview with a point load at each wheel position. This discrete load should be assigned to a point on a model.

NATO MLC Tracked Vehicles

A range of tracked vehicles for classes MLC-4 to MLC-150 are supplied.



To use this loading type:

- Choose the vehicle type from the drop down list.
- A compound discrete load attribute is generated in the Attributes Treeview. The compound attribute combines two patch loads and should be assigned to a point on the model.
- A discrete patch load attribute is generated in the Attributes Treeview for each track, with each load being used in a Compound discrete load attribute that should be assigned to a point on a model.

Notes that military vehicles defined in Swedish code VVPubl. 2009:61 are identical to the vehicles defined here.

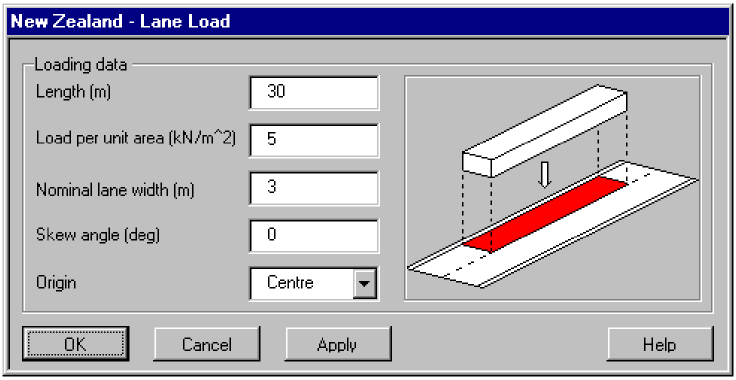
New Zealand Vehicle Loading

New Zealand vehicle loads are defined to the New Zealand loading code.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.

New Zealand Lane Loads

The lane load generator produces a uniform patch load based on notional lane width, loaded length and intensity. The intensity of the lane load has a default value set but can be modified to any value required.

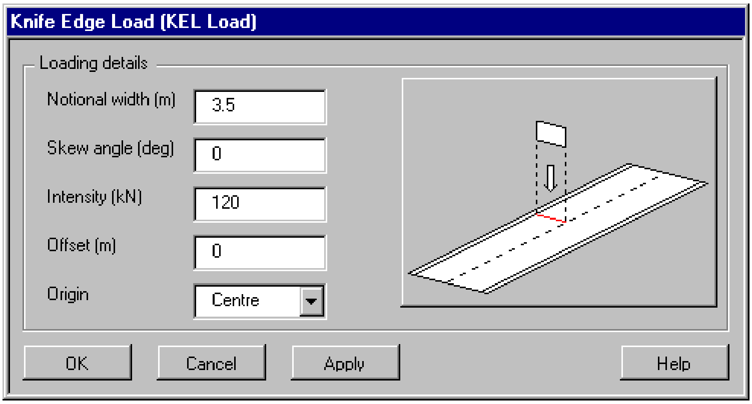


To use this loading type:

- Specify the loaded length.
- Specify the load per unit area.
- Specify the notional lane width.
- Specify the skew angle to apply to the lane loading (clockwise positive).
- Choose the origin about which the load is to be generated.

New Zealand Knife Edge Loads (KEL) Loads

The KEL load generator produces a knife-edge load based on notional lane width and intensity. The intensity of the knife-edge load has a default value set but can be modified to any value required.

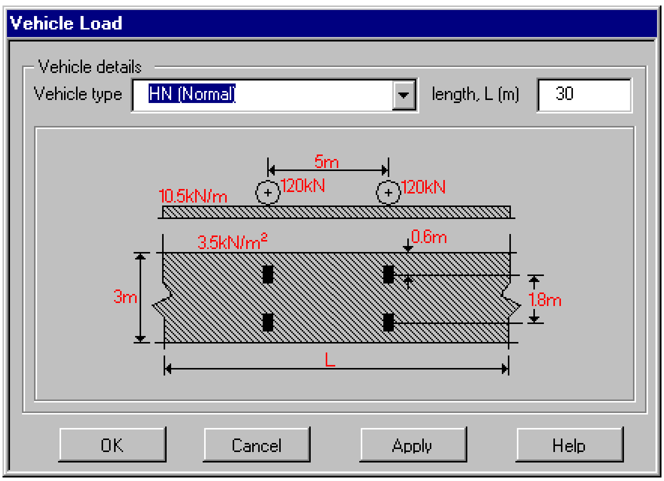


To use this loading type:

- Specify the notional width.
- Specify the skew angle to apply to the lane loading (clockwise positive).
- Specify the intensity.
- Specify the offset.
- Choose the origin about which the load is to be generated.

New Zealand Vehicle Load

The following truck types can be created: HN (normal) and HO (overload).



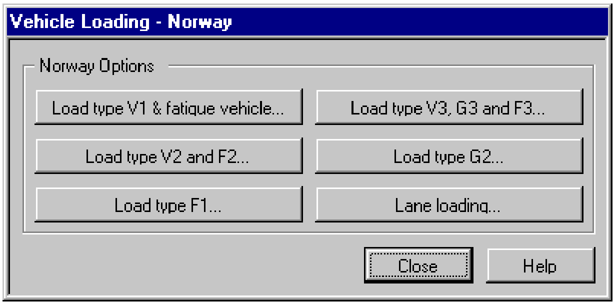
To use this loading type:

- Choose the vehicle type required, and specify the length if applicable.

Norway Vehicle Loading

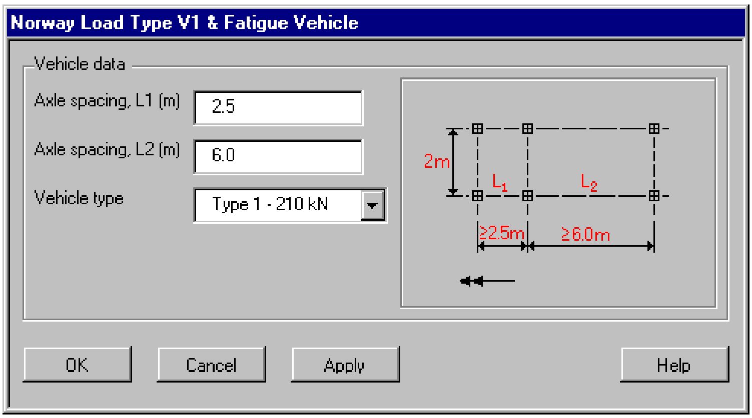
Norway vehicle loads are defined to the Norwegian loading design code.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.



Norway Load Type V1 & Fatigue Vehicle

The load type V1 & Fatigue Vehicle generator produces vehicle loads based on the selection of type V1 & Fatigue Vehicle. The variable axle spacings can be set to the required distance.

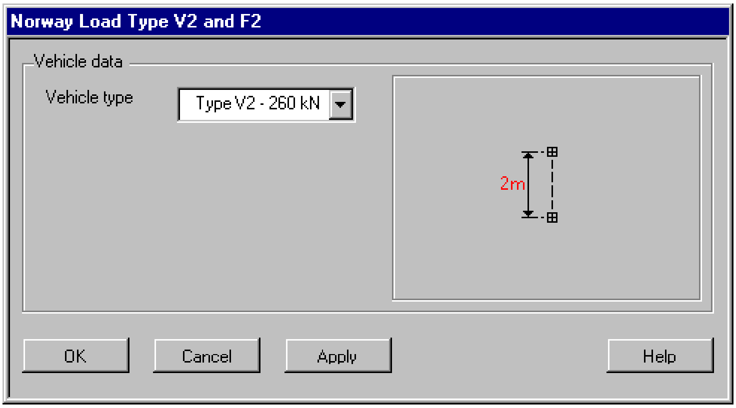


To use this loading type:

- Choose the axle spacings for L1 and L2 that you require for the vehicle. Note that L1 has to be greater than 2.5m and L2 greater than 6m. The vehicle will be generated about the second axle's centre point.
- Choose the vehicle type from the drop down list.

Norway Load Types V2 and F2

The load types V2 and F2 generator produce vehicle loads based on the selection of type V2 or F2.

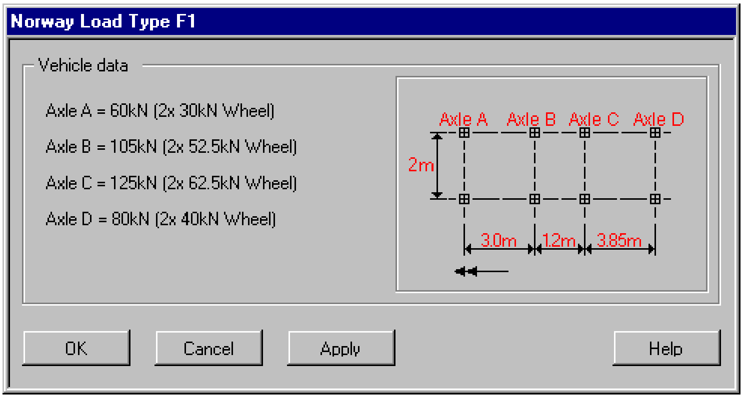


To use this loading type:

- Choose the vehicle type from the drop down list. The vehicle will be generated about the axle's centre point.

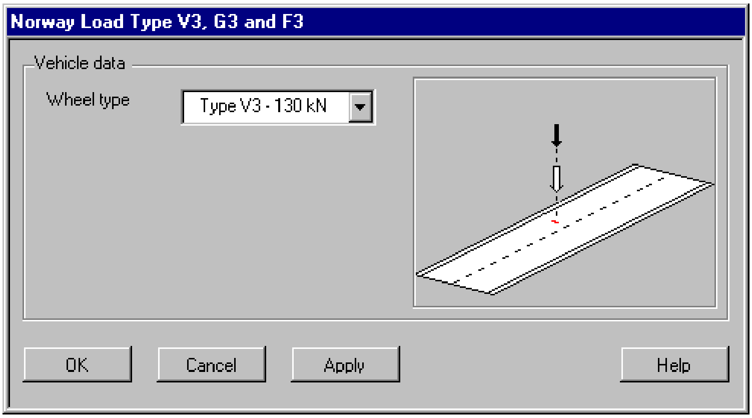
Norway Load Types F1

The load type F1 generator produces a vehicle load based on the selection of type F1.



Norway Load Types V3, G3 and F3

The load types V3, G3 and F3 generator produces vehicle loads based on the selection of type V3, G3 and F3 .

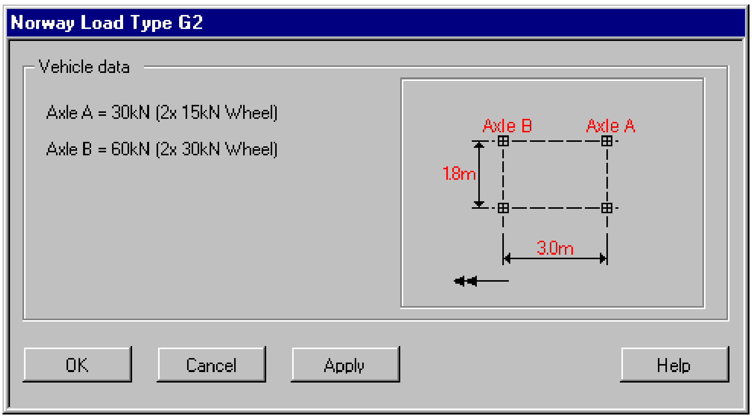


To use this loading type:

- Choose the vehicle type from the drop down list.

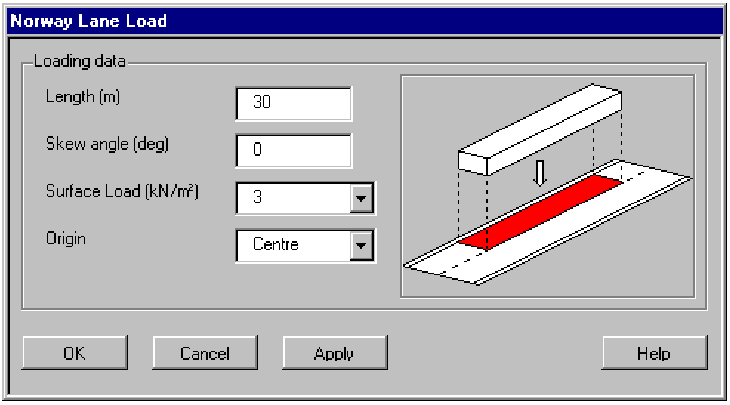
Norway Load Types G2

The load type G2 generator produces a vehicle load based on the selection of type G2.



Norway Lane Load

The lane load generator produces a uniform patch load based on loaded length and intensity. The intensity of the lane load has a default value set but can be modified to any value required.



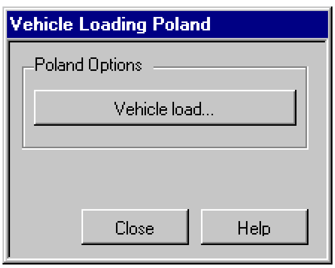
To use this loading type:

- Specify the length of lane load that you require to generate about the centre line of patch.
- Specify the skew angle to applied to the lane loading (clockwise positive).
- Specify the Surface load intensity from the drop down list.
- Choose the origin that the load will be generated about.

Poland Vehicle Loading

Poland vehicle loads are defined to the Polish loading code.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.

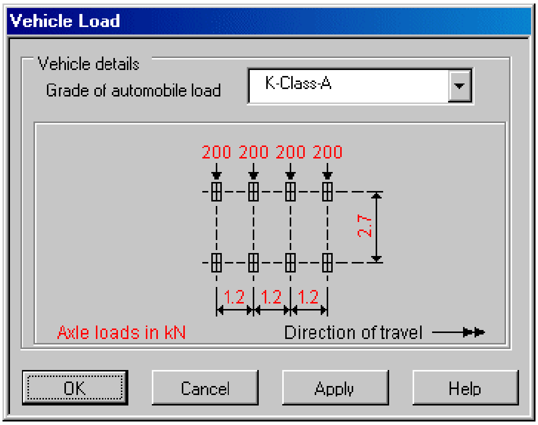


Poland Vehicle Loads

The Poland vehicle load generator produces the following grades of automobile loads:

- K-Class types A to E

- S-Class 123, 4 and 5



To use this loading type:

- Select a vehicle load type and press the OK or Apply button to add that loading to the Attributes Treeview.

Poland Military Vehicles

Military vehicles for Poland are not currently supplied but note that NATO military vehicles (which differ slightly from the Poland vehicles) are supplied. See **NATO Vehicle Loading** for details.

South Africa Vehicle Loading

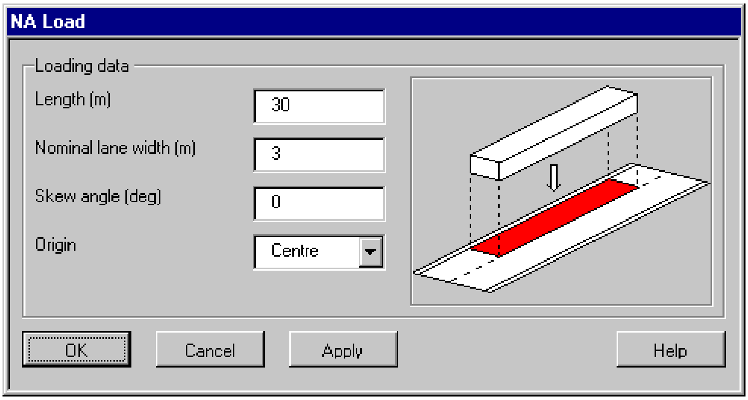
South Africa vehicle loads are defined to the TMH7 Code of Practice for the Design of Bridges and Culverts in South Africa 1981 with 1988 revisions.

Vehicle loads can be created for normal vehicle loading (NA), single abnormal vehicle loading (NB) and multi-wheeled trailer combination or multi-wheeled self-propelled vehicle loading (NC) loads with reference to TMH7 parts 1 and 2.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.

South Africa Lane Loads (NA Load)

For normal vehicle loading the lane load generator produces a uniform patch load based on a nominal lane width and loaded length. A skew angle can be defined and if done, should be defined clockwise positive.The intensity of loading is in accordance with the loading curve for NA loading as defined in the code.

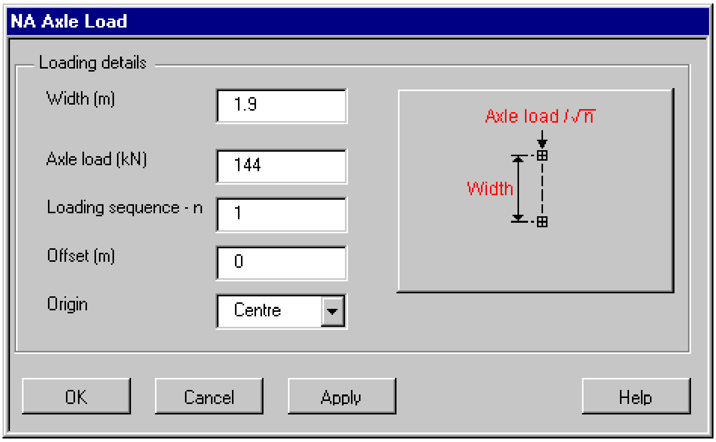


To use this loading type:

- Specify the loaded length.
- Specify the nominal lane width.
- Specify the skew angle to apply to the lane loading (clockwise positive).
- Choose the origin about which the load is to be generated.

South Africa Vehicle Load (NA Axle)

For NA axle loads a nominal axle load at a specified width can be defined. Two point loads are generated of an intensity computed in Kn from the formula (Axle load)/√n kN where n is the loading sequence number. For multiple applications of an axle load across more than one lane the loading sequence identifier should be incremented each time in accordance with the code. Setting an offset will position the loading away from an origin point that itself can be defined in one of three positions.

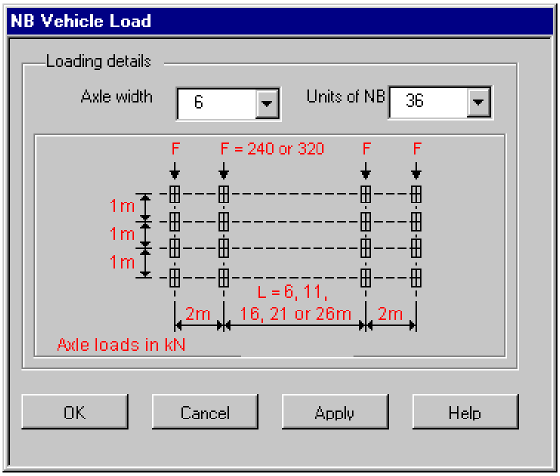


To use this loading type:

- Specify the width between wheels.
- Specify the axle load.
- Increment the loading sequence if loading is to be added to a different lane to any previously defined loading.
- Enter an offset (if any) and choose the origin about which the load is to be generated.

South Africa Vehicle Load (NB load)

NB loading is a notional load representing a single abnormal vehicle. For this loading pre-defined axle spacings and a choice of units of HB are available. The dialog allows selection of axle widths of 6, 11, 16, 21 or 26m and generates 24 or 36 units of NB loading where 1 unit is equal to 2.5kN per wheel.

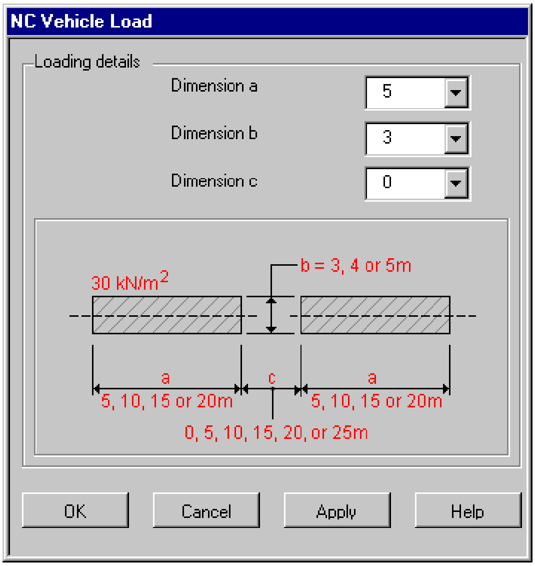


To use this loading type:

- Choose a longitudinal length between internal axles
- Choose the appropriate units of HB

South Africa Vehicle Load (NC load)

NC loading represents a multi-wheeled trailer combination (or self-propelled multi-wheeled vehicles) with controlled hydraulic suspension and steering intended to transport very heavy indivisible payloads. These loads are represented by a grid of point loads with a load intensity of 30kN/m^2. The dimensions a, b and c should be selected between the limits shown to have the most severe effect.



To create this loading type:

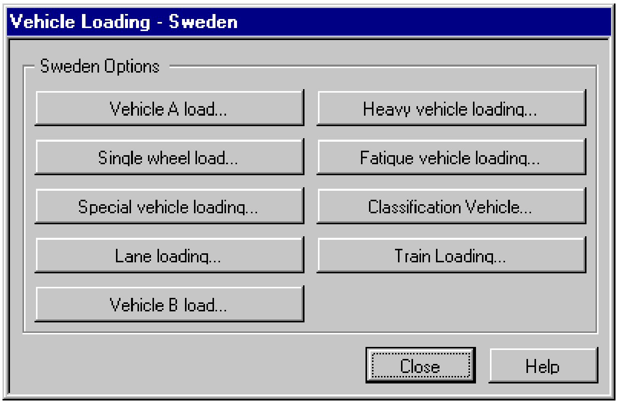
- Specify the extent of the loading by defining dimensions a, b and c.

For this loading a number of discrete load attributes will be created that are used to define a compound load. It is the compound load that should be assigned to the model.

Sweden Vehicle Loading

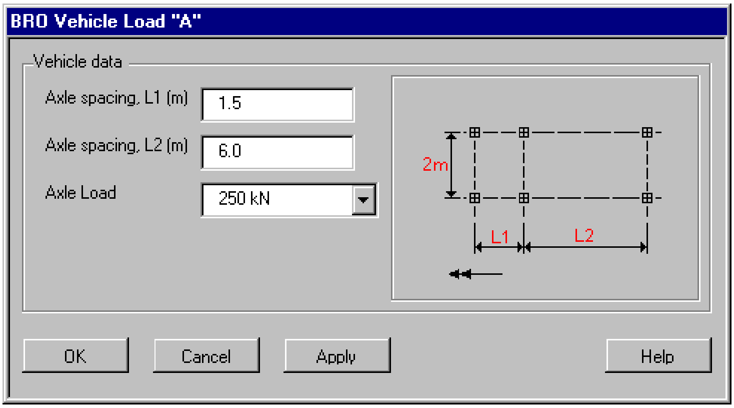
Swedish vehicle loads are defined to Swedish bridge BRO Classification loads.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.



Sweden BRO Vehicle Load A

The load type A generator produces a vehicle load based on axle weight (250kN or 170kN). The variable axle spacings can be set to the required distance.

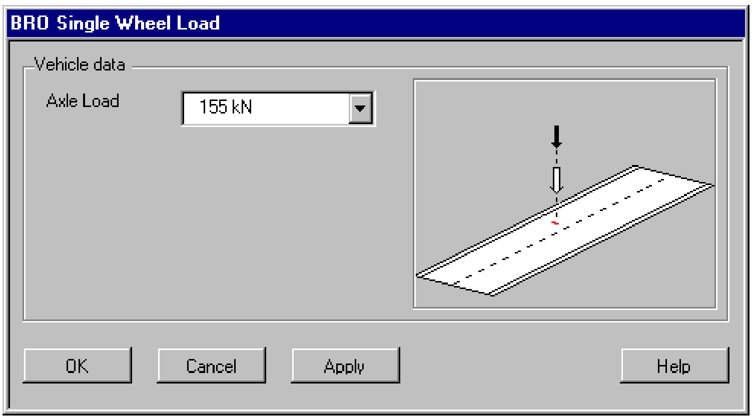


To use this loading type:

- Specify the front axle spacing that you require for the vehicle.
- Specify the back axle spacing that you require for the vehicle.
- Choose which intensity of axle load you wish to generate.

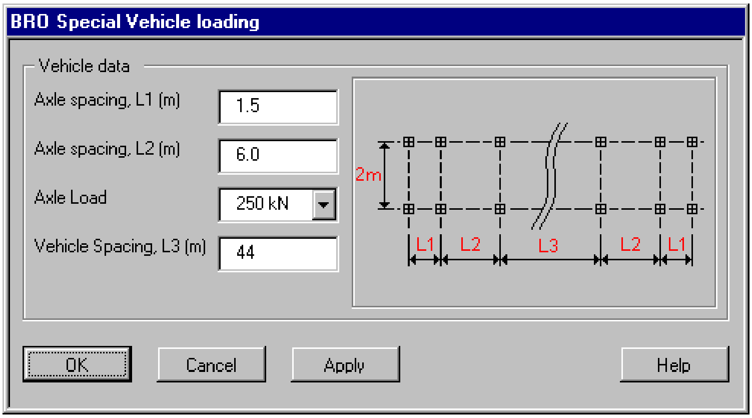
Sweden BRO Single Wheel Load

The load single wheel generator produces a point load of 155kN.



Sweden BRO Special Vehicle Loading

The special vehicle load generator produces a vehicle load based on axle weight (250kN or 170kN). The load represents two vehicles each with three axles, separated by a distance L3.

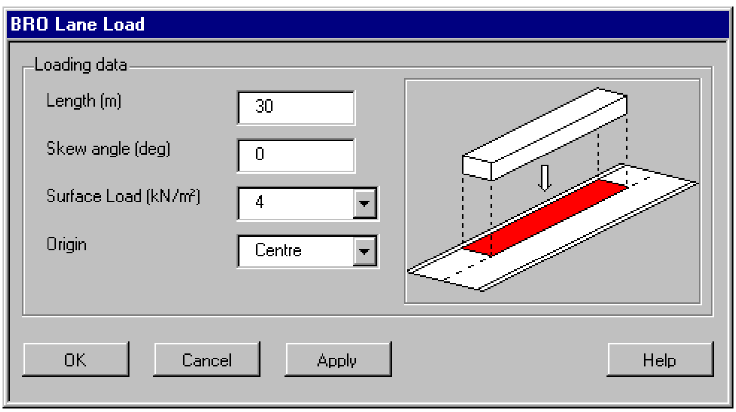


To use this loading type:

- Specify the front axle spacing that you require for the vehicle.
- Specify the back axle spacing that you require for the vehicle.
- Choose which intensity of axle load you wish to generate.
- Specify the distance between the vehicles.

Sweden BRO Lane Load

The lane load generator produces a uniform patch load based on loaded length and intensity.

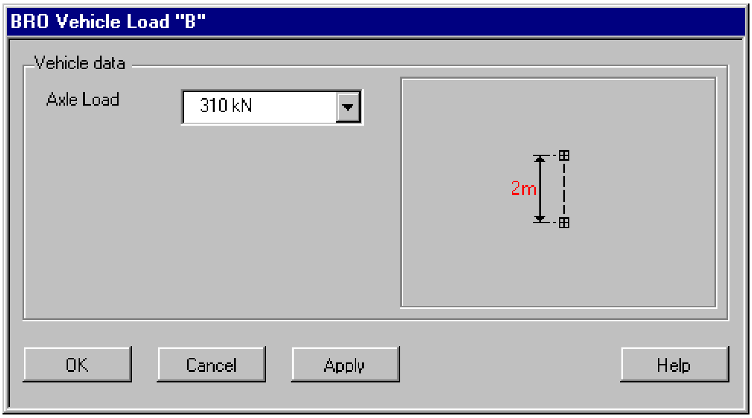


To use this loading type:

- Specify the length of lane load.
- Specify the skew angle to apply to the lane loading. (clockwise positive)
- Choose which intensity of load you wish to generate.
- Choose the origin for the about which the load is to be positioned.

Sweden BRO Vehicle Load B

The load type B generator produces a single axle load of weight 310kN or 210kN.

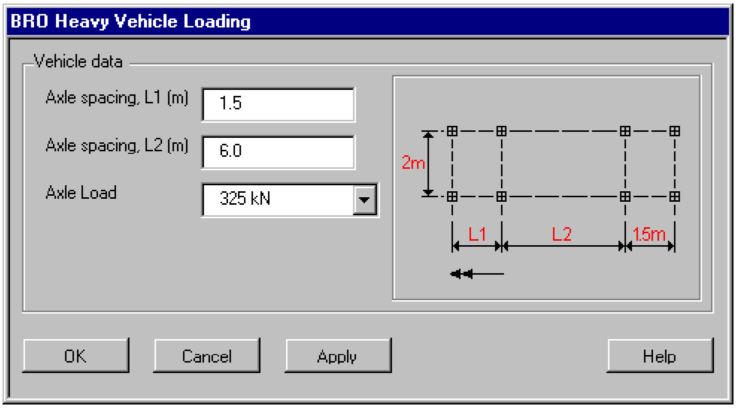


To use this loading type:

- Choose which intensity of load you wish to generate.

Sweden BRO Heavy Vehicle Load

The heavy vehicle load produces a three axle vehicle with an axle weight of 325kN.

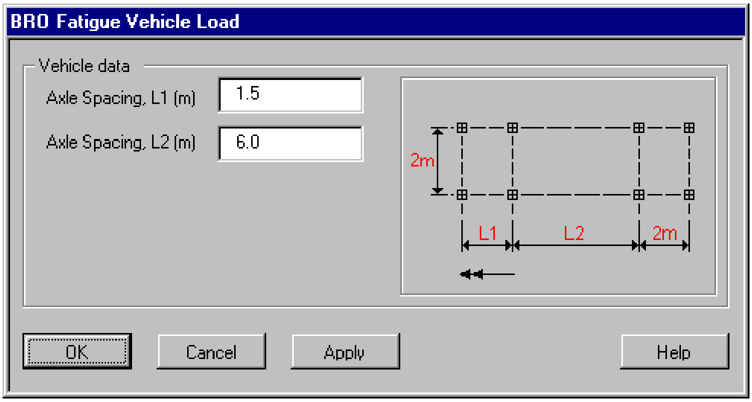


To use this loading type:

- Specify the front axle spacing that you require for the vehicle.
- Specify the back axle spacing that you require for the vehicle.
- Choose which intensity of load you wish to generate.

Sweden BRO Fatigue Vehicle Load

The fatigue load generator produces a vehicle load with four axles. The front two are 150kN, the rear two are 180kN each. The variable axle spacings can be set to the required distance.

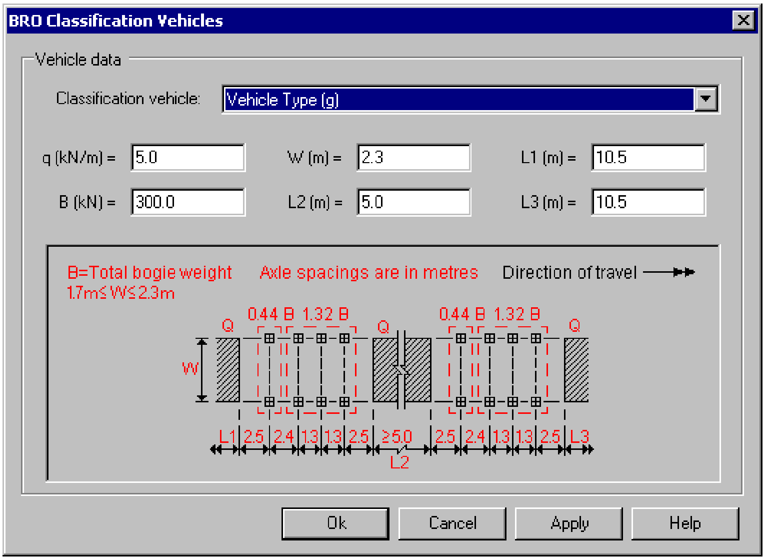


To use this loading type:

- Specify the axle spacing that you require for the vehicle.

Sweden BRO Classification Vehicles

The classification vehicle generator produces the following vehicle loads: Type a, b, c, d, e, f, g, h, i, j, k, l, m, n, military vehicle 45 ton and military vehicle 60 ton. Each of these vehicles can have many different configurations by setting the number of axle weights and spacings.



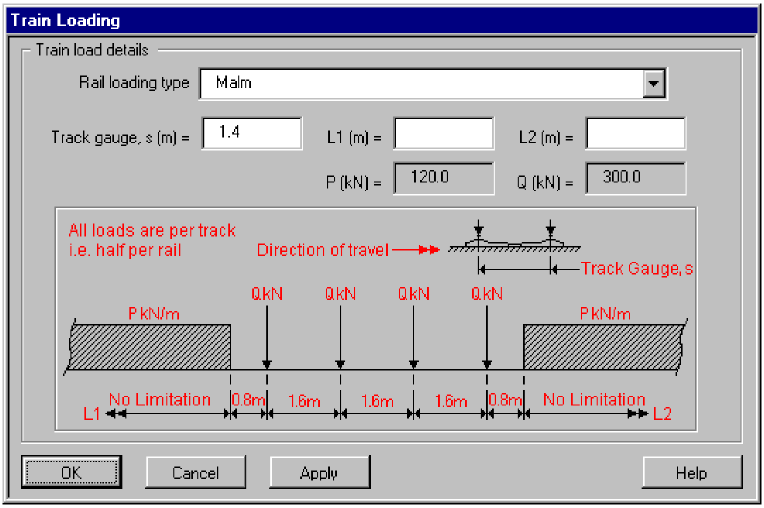
To use this loading type:

- Choose the classification vehicle required from the list.
- Specify the patch load intensity, Q, in kN/m^2
- Specify the bogey weight, B, in kN
- Specify the width of vehicle, W, in metres
- Specify the axle spacings; L1, L2 and L3, in metres

Notes

- Classification vehicles (g) to (l) and Military Vehicle 45 ton and Military Vehicle 60 ton will consist of several discrete load parts combined into a compound loading attribute.

Sweden BRO Train Loading



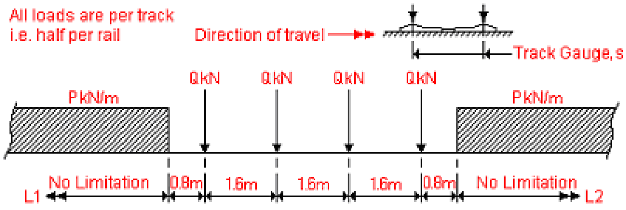
To use this loading type:

- Select the standard rail loading type from the drop down list.
- Enter the track gauge in metres. This has to be 1.4m or greater.
- Enter the dimension data required as appropriate for the chosen load to be generated. All dimensions are in metres and kN.

Notes

- The trainload will be generated about the loaded area's centre point.
- Load Model HSLM (High Speed Load Model) is only used in dynamic analysis and comprises of two separate Universal Trains with variable coach lengths, HSLM-A and HSLM-B.
- Limits of validity of Load Model HSLM are given in EN 1991-2 Annex E.
- Load Model HSLM is generally used to represent the loading from passenger trains at speed exceeding 200 km/h (dynamic analysis).
- The load HSLM-B should only be applied to simply supported plate bridges and simply supported beam bridges, or similar, with span lengths below 7m.
- Continuous bridges are not applicable for HSLM B.
- The definition of L is the span length.
- All other dynamic analyses on railway bridges should use the load HSLM-A.

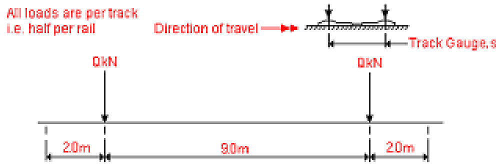
Rail Loading Type Malm, Malm 2000 and BV 2000



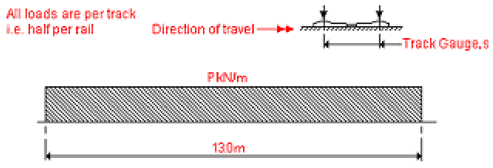
Malm, Malm 2000 and BV 2000

Train Load	P (kN)	Q (kN/m)
Malm	300	110
Malm 2000	350	120
BV 2000	330	120

Rail Loading Type RV-25 / RV-30



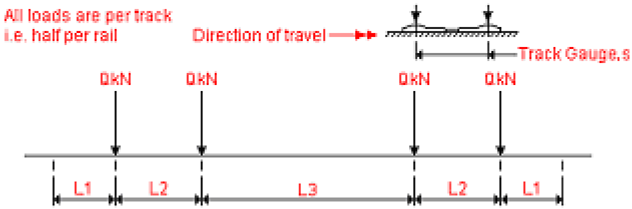
RV-25 / RV-30 Point load arrangement



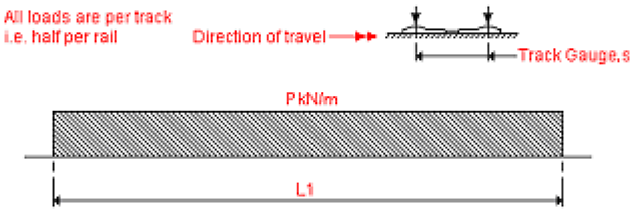
RV-25 / RV-30 UDL arrangement

Train Load		Load
RV25	Point, Q (kN)	250
	UDL, P (kN/m)	39
RV-30	Point, Q (kN)	300
	UDL,P (kN/m)	46

Rail Loading Type A to BV-4



Train loads A to BV-4 Point load arrangement

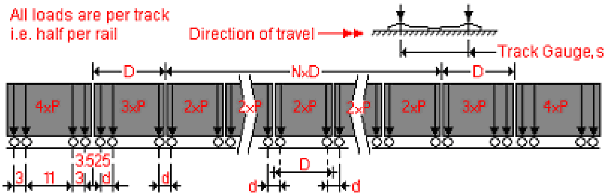


Train loads A to BV-4 arrangement UDL

Train Load		Load	L1 (m)	L2 (m)	L3 (m)
A	Point, Q (kN)	160	1.50	1.80	5.20
	UDL, P (kN/m)	50	12.80		
B1	Point, Q (kN)	180	1.50	1.80	7.80
	UDL, P (kN/m)	50	14.40		

B2	Point, Q (kN)	180	1.50	1.80	4.65
	UDL, P (kN/m)	64	11.25		
C2	Point, Q (kN)	200	1.50	1.80	5.90
	UDL, P (kN/m)	64	12.50		
C3	Point, Q (kN)	200	1.50	1.80	4.50
	UDL, P (kN/m)	72	11.10		
C4	Point, Q (kN)	200	1.50	1.80	3.40
	UDL, P (kN/m)	80	10.0		
D2	Point, Q (kN)	225	1.50	1.80	7.45
	UDL, P (kN/m)	64	14.05		
D3	Point, Q (kN)	225	1.50	1.80	5.90
	UDL, P (kN/m)	72	12.50		
D4	Point, Q (kN)	225	1.50	1.80	4.65
	UDL, P (kN/m)	80	11.25		
BV-2	Point, Q (kN)	250	1.50	1.80	7.30
	UDL, P (kN/m)	72	13.90		
BV-3	Point, Q (kN)	250	1.50	1.80	5.90
	UDL, P (kN/m)	80	12.50		
BV-4	Point, Q (kN)	300	1.50	1.80	5.40
	UDL, P (kN/m)	100	12.0		

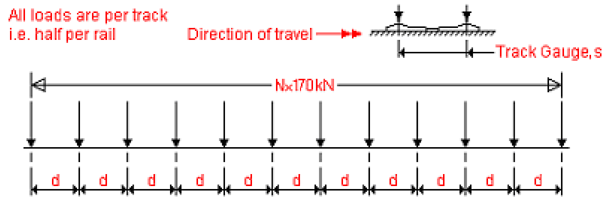
Rail Loading Type HSLM-A (A1 to A10)



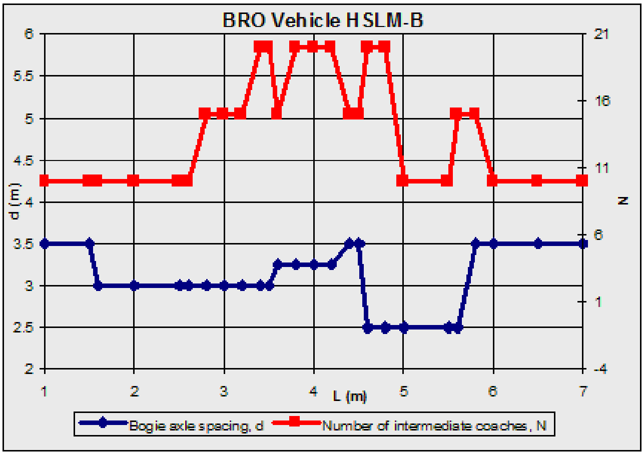
Train loads HSLM-A (A1 to A10)

Universal Train	Number of intermediate coaches (N)	Coach length D (m)	Bogie axle spacing d (m)	Point force P (kN)
A1	18	18	2.0	170
A2	17	19	3.5	200
A3	16	20	2.0	18
A4	15	21	3.0	190
A5	14	22	2.0	170
A6	13	23	2.0	180
A7	13	24	2.0	190
A8	12	25	2.5	190
A9	11	26	2.0	210
A10	11	27	2.0	210

Rail Loading Type HSLM-B



Train load HSLM-B



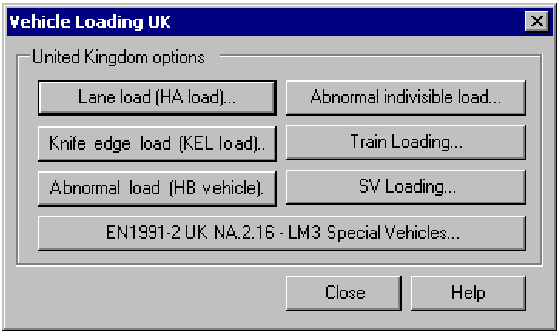
Sweden Military Vehicles

Vehicles defined in VV 2009-61 MB802 Barighetsutredning Av Byggnadsver (Swedish design code) are seen to be identical to the vehicles defined in STANAG 2012 Annex A. See **NATO Vehicle Loading** for details.

United Kingdom Vehicle Loading

The United Kingdom vehicle loads are defined to the UK bridge codes BS5400, BD37/88, BD37/01, BD21/97, BD21/01 and EN1991-2 National Annex 2.16

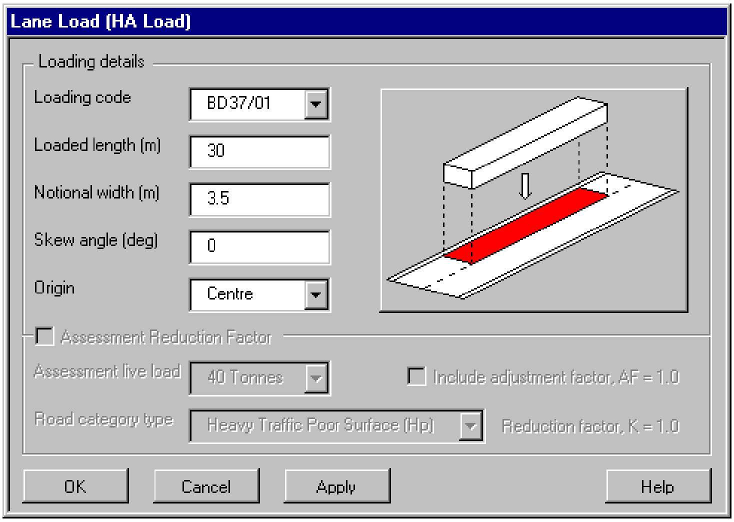
See **Using static vehicle loadings** for general details regarding static vehicle loadings.



United Kingdom HA Loads

The HA load generator produces a uniform patch load based on notional lane width and loaded length. The intensity of the patch is calculated based on the loaded length entered with shorter loaded lengths having higher intensity. The exact intensity will depend on the code being used.

For the assessment codes BD21/97and BD21/01 an additional reduction factor can be included. This is based on the assessment live load being used and the road category type.



To use this loading type:

- Choose the loading code to be used
- Specify the loaded length..
- Specify the notional width of lane load..
- Specify the skew angle to apply to the lane loading (clockwise positive).
- Choose the origin about which the load is to be generated.

Notes

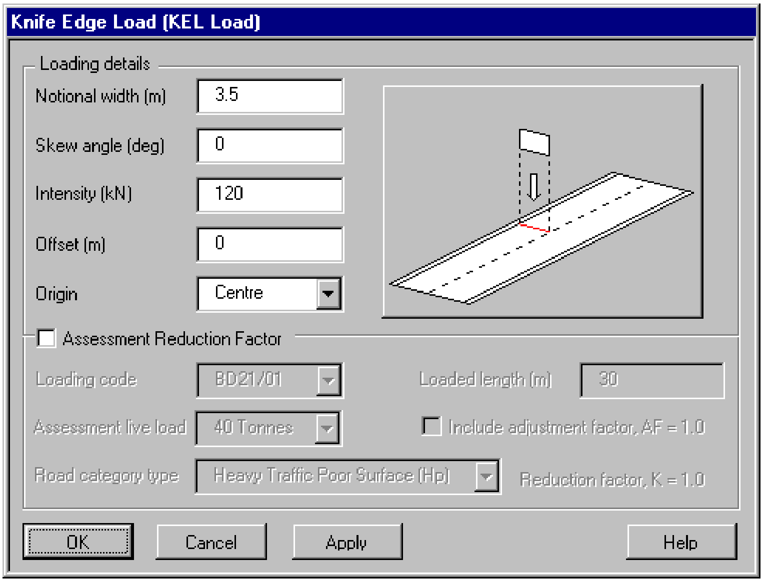
- If an assessment code (BD21/97 or BD21/01) has been selected from the design code list the option to apply an assessment reduction factor will be given. Switching this option off will generate a nominal unfactored assessment load.
    - Select the assessment live load vehicle type from the list
    - Select the road category type from the list
    - Choose whether to include the adjustment factor, AF, in the calculation.

- The calculate reduction factor will be displayed. Clicking the Apply button to generate the loading will also apply the calculated factor. For assessment loading the loaded length must be between 2m and 50m. For loaded lengths outside of this range the user should seek advice from the appropriate design code.

United Kingdom KEL Loads

The KEL load generator produces a knife-edge load based on notional lane width and intensity. The intensity of the knife-edge load has a default value set but can be modified to any value required.

For the assessment codes BD21/97and BD21/01 an additional reduction factor can be included. This is based on the assessment live load being used and the road category type.



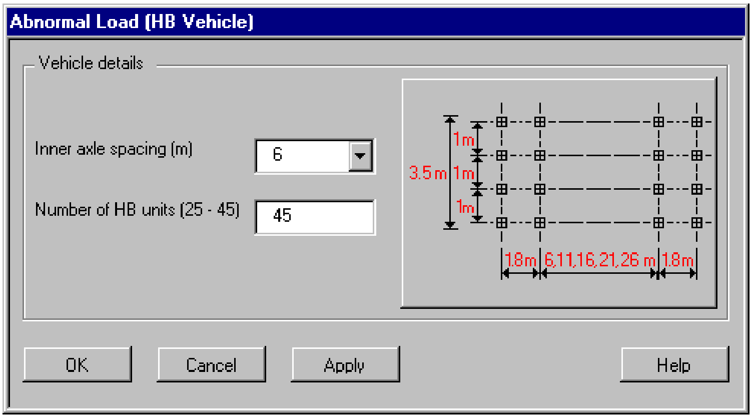
To use this loading type:

- Specify the width of lane load.
- Specify the skew angle to apply to the lane loading (clockwise positive).
- Specify the intensity of load.
- Specify the offset for the load.
- Choose the origin for the about which the load is to be generated.
- The option to apply an assessment reduction factor is given. Switching this option off will generate a nominal unfactored assessment load.
- Select the assessment live load vehicle type from the list

- Select the road category type from the list
- Specify the length of carriageway to allow the assessment reduction factor to be calculated.
- Choose whether to include the adjustment factor, AF, in the calculation.
- The calculate reduction factor will be displayed. Clicking the Apply button to generate the loading will also apply the calculated factor.

United Kingdom Abnormal Load Generator (HB Vehicle)

The HB load generator produces a HB vehicle by setting the inner axle spacing (6m, 11m, 16m, 21m, & 26m) and the number of HB units required (25-45).

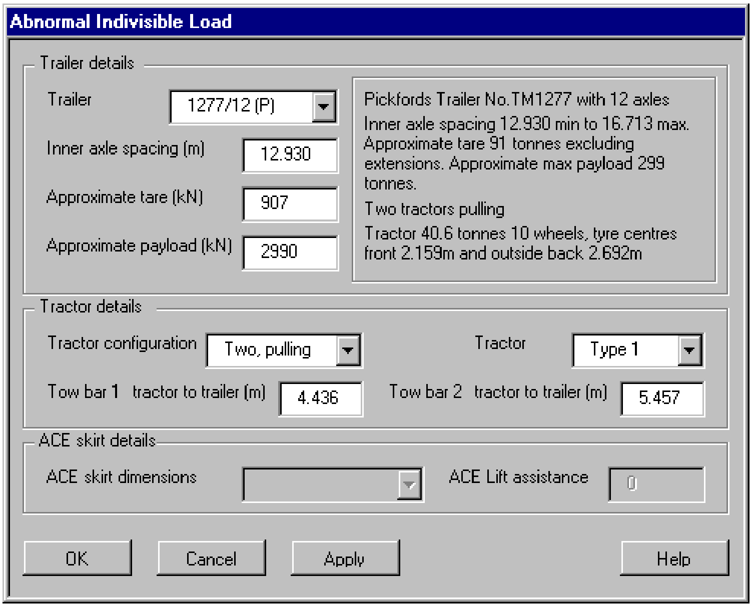


To use this loading type:

- Choose the inner axle spacing.
- Specify the number of HB units to be considered for the vehicle.

United Kingdom Abnormal Indivisible Loads

The abnormal indivisible load generator produces loads that represent heavy haulage vehicles. Many different configurations of trailer and tractor units can be created with user-defined loading intensities.



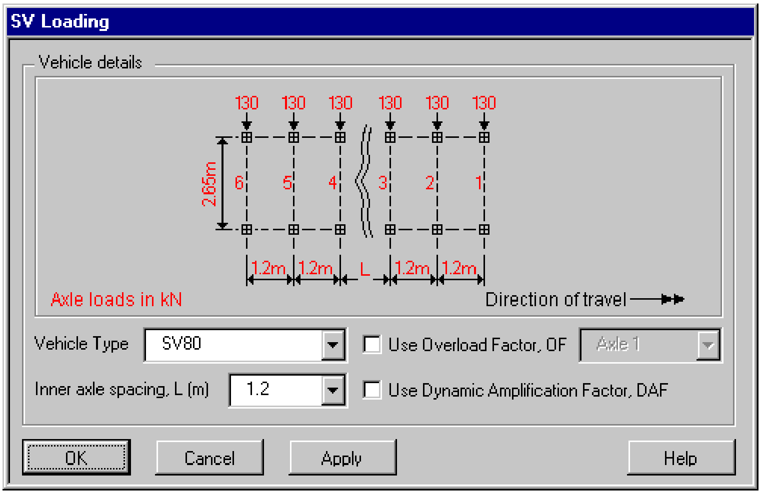
To use this loading type:

- Choose which trailer you require to generate as a vehicle load. A description is provided in the right hand panel.
- Specify the inner axle spacing for the trailer.
- Specify the approximate tare of the trailer.
- Specify the approximate payload of the trailer.
- Choose the tractor arrangement that is required for the vehicle.
- Choose the trailer type (a description is given in the panel above)
- Choose distance of tow to tractor (one or two)
- If a trailer with ACE has been chosen set up the details of the air skirt.

United Kingdom Special Vehicle Loads

The UK special vehicle loading dialog allows the creation of SV loading as defined in BD 86/11 “The Assessment of Highway Bridge and Structures for the Effects of Special Types General Order (STGO) and Special Order (SO) Vehicles”.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.



- ❑ **Vehicle Type** SV loading is defined in accordance to BD86/11 “The Assessment of Highway Bridge and Structures for the Effects of Special Types General Order (STGO) and Special Order (SO) Vehicles”. Five load models can be generated that simulate the vertical effects of different types of STGO vehicles with basic axle weights not exceeding 16.5 tonnes and military tank transporter vehicles with basic axle weights of up to 25 tonnes.

  - The SV80 vehicle is intended to model the effects of STGO Category 2 vehicles with a maximum gross vehicle weight of 80 tonnes and a maximum basic axle load of 12.5 tonnes.
  - The SV100 vehicle is intended to model the effects of STGO Category 3 vehicles with a maximum gross vehicle weight of 100 tonnes and a maximum basic axle load of 16.5 tonnes.
  - The SV150 vehicle is intended to model the effects of STGO Category 3 vehicles with a maximum gross vehicle weight of 150 tonnes and a maximum basic axle load of 16.5 tonnes.
  - The SV-Train is intended to model the effects of a single locomotive pulling a Category 3 trailer.
  - The SV-TT is intended to model the effects of a military tank transporter vehicles with a maximum basic axle load of 25 tonnes.

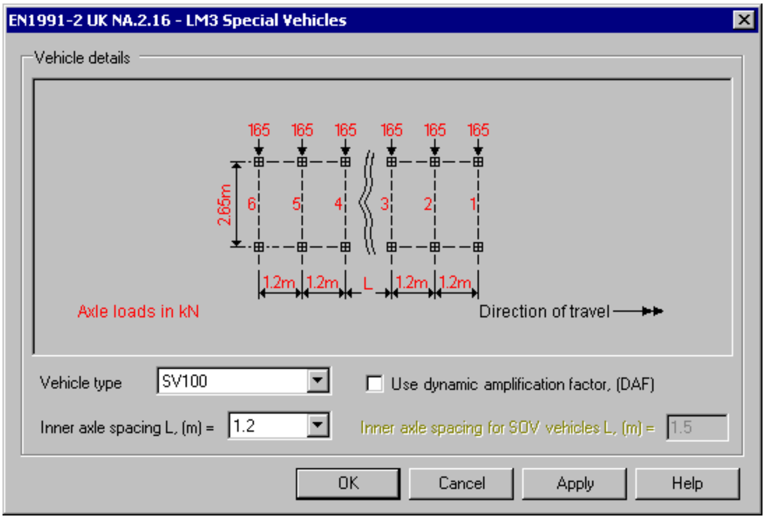
- ❑ Choose the **Overload Factor**, OF, to model SV vehicles in excess of the gross weight and axle weights notified by the hauliers to highway authorities. The Overload Factor shall be taken as 1.2 for the worst critical axle, chosen from the drop-down list, and 1.1 for all other axles.

- The **Dynamic Amplification Factor**, DAF, will factor each axle using the following equation:

  DAF = [1.7 X (basic axle load / 10) ^ -0.15] >= 1.05

United Kingdom EN 1991-2 UK National Annex 2.16 LM3 Special Vehicle Loads

Special vehicle types SV and SOV as defined as Load Model 3 in the UK National Annex to EN 1991-2:2003.



SV Vehicles

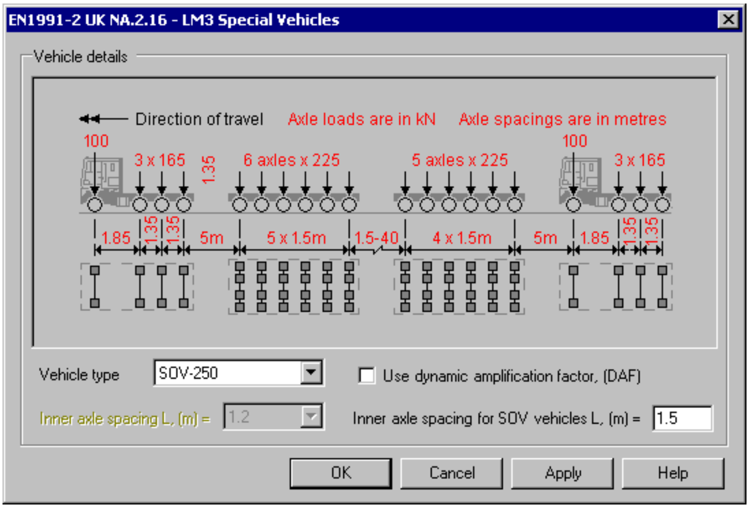
- **Vehicle Type** Three load models can be generated that simulate the vertical effects of different types of STGO vehicles with basic axle weights not exceeding 16.5 tonnes:

  - The SV80 vehicle is intended to model the effects of STGO Category 2 vehicles with a maximum gross vehicle weight of 80 tonnes and a maximum basic axle load of 12.5 tonnes.
  - The SV100 vehicle is intended to model the effects of STGO Category 3 vehicles with a maximum gross vehicle weight of 100 tonnes and a maximum basic axle load of 16.5 tonnes.
  - The SV196 vehicle is intended to model the effects of a single locomotive pulling a STGO Category 3 load with a maximum gross vehicle weight of 150 tonnes and a maximum basic axle load of 16.5 tonnes with the gross weight of the vehicle load train not exceeding 196 tonnes.

- The **Dynamic Amplification Factor**, DAF, will factor each axle according to values set out in Table NA 2 of the UK National Annex to EN 1991-2:2003, namely:

Basic axle load	Dynamic amplification factor
100kN	1.20
130kN	1.16
165kN	1.12
180kN	1.10
225kN	1.07

- Inner axle spacing can be set to 1.2, 5.0, or 9.0m



SOV Vehicles

- **Vehicle Type** Four load models can be generated to simulate vertical effects of Special Order Vehicles (SOV) with trailer weights limited to four different tonnages:

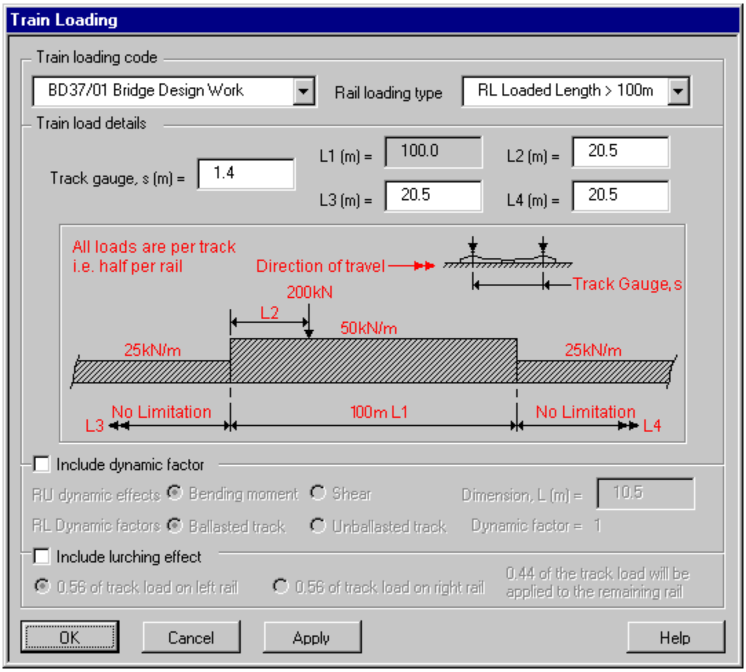
  - The SOV-250 vehicle models a maximum total weight of SO trailer units up to 250 tonnes.
  - The SOV-350 vehicle models a maximum total weight of SO trailer units up to 350 tonnes
  - The SOV-450 vehicle models a maximum total weight of SO trailer units up to 450 tonnes
  - The SOV-600 vehicle models a maximum total weight of SO trailer units up to 600 tonnes

- The **Dynamic Amplification Factor**, DAF, will factor each axle according to values set out in Table NA 2 of the UK National Annex to EN 1991-2:2003, as tabulated in the SV vehicles section of this topic.
- Inner axle spacing for SOV vehicles must be set between 1.5 and 40m..

United Kingdom Train Loading

The train load generator can produce loading types RU, SW/0, SW/2, RL, RA1, RT, Class 67 and Class 91.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.



- **Train loading code** Defines the design code to be used
- **Rail loading type** Standard railway loading consists at two types, RU and RL. RU loading allows for all combinations of vehicles currently running or projected to run on railways in the Continent of Europe, including the United Kingdom, and is to be adopted for the design of bridges carrying main line railways of 1.4m gauge and above.

    - RL loading is a reduced loading for use only on passenger rapid transit railway systems on lines where main line locomotives and rolling stock do not operate. The derivation of standard railway loadings is given in appendix D of BD37.

- Nominal type RU loading consists of four 250kN concentrated loads preceded, and followed, by a uniformly distributed load of 80kN/m.
- Nominal type SW/0 loading consists of a two uniformly distributed loads of 133kN/m, each 15m long and separated by a distance of 5.3m.
- Nominal type RL loading consists of a single 200kN concentrated load coupled with a uniformly distributed load of 50kN/m for loaded lengths up to 100m. For loaded lengths in excess of l00m the distributed nominal load shall be 50kN/m for the first l00m and shall be reduced to 25kN/m for lengths in excess of l00m. Alternatively, two concentrated nominal loads, one of 300kN and the other of 150kN, spaced at 2.4m intervals along the track, shall be used on deck elements where this gives a more severe condition. These two concentrated loads shall be deemed to include dynamic effects.
- The standard railway loadings RU and RL as specified in BD37 clause 8.2.1 and 8.2.2 (except the 300kN and 150kN concentrated alternative RL loading) are equivalent static loadings and shall be multiplied by appropriate dynamic factors to allow for impact, oscillation and other dynamic effects including those caused by track and wheel irregularities.

❑ **Include dynamic factor** In deriving the dynamic factor, *L* is taken as the length (in m) of the influence line for deflection of the element under consideration. For unsymmetrical influence lines, *L* is twice the distance between the point at which the greatest ordinate occurs and the nearest end point of the influence line. In the case of floor members. 3m should be added to the length of the influence line as an allowance for load distribution through track. The dynamic factors given below should be adopted, provided that maintenance of track and rolling stock is kept to a reasonable standard.

Dimension L	Bending Moment	Shear
up to 3.6m	2.0	1.67
from 3.6m to 67	$0.73 + 2.16/(L^{0.5} - 0.2)$	$0.82 + 1.44/(L^{0.5} - 0.2)$
over 67	1.0	1.0

The dynamic factor for RL loading. when evaluating moments and shears, shall be taken as 1.20, except for unballasted tracks where, for rail bearers and single-track cross girders, the dynamic factor shall be increased to 1.40. The dynamic factor applied to temporary works may be reduced to unity when rail traffic speeds are limited to not more than 25 km/h.

❑ **Include lurching effect** Lurching results from the temporary transfer of part of the live loading from one rail to another, the total track load remaining unaltered. The dynamic factor applied to RU loading will take into account the effects of lurching, and the load to be considered acting on each rail shall be half the track load.

Notes.

- The dynamic factor applied to RL loading will not adequately take account of all lurching effects. To allow for this, 0.56 of the track load shall be considered acting on one rail concurrently with 0.44 of the track load on the other rail. This distribution of load need only be taken into account on one track where members support two tracks. Lurching may be ignored in the case of elements that support load from more than two tracks.
- Train loads will be generated about the centre point.

United States of America Vehicle Loading

USA vehicle loads can be defined for:

- ❑ Truck loads **AASHTO LFD and LFRD loading and other state dependent design loading such as Oregon LRFR. and West Virginia Department of Transportation Truck Loading**
- ❑ Tandem loads **based on AASHTO LFRD**
- ❑ Uniform loads **based on AASHTO LFD and LRFD**
- ❑ Concentrated Pm loads **based on AASHTO LFD**
- ❑ Concentrated Ps loads **based on AASHTO LFD**
- ❑ Train loading **based on AREMA**

See **Using static vehicle loadings** for details regarding defining and assigning static vehicle loadings.

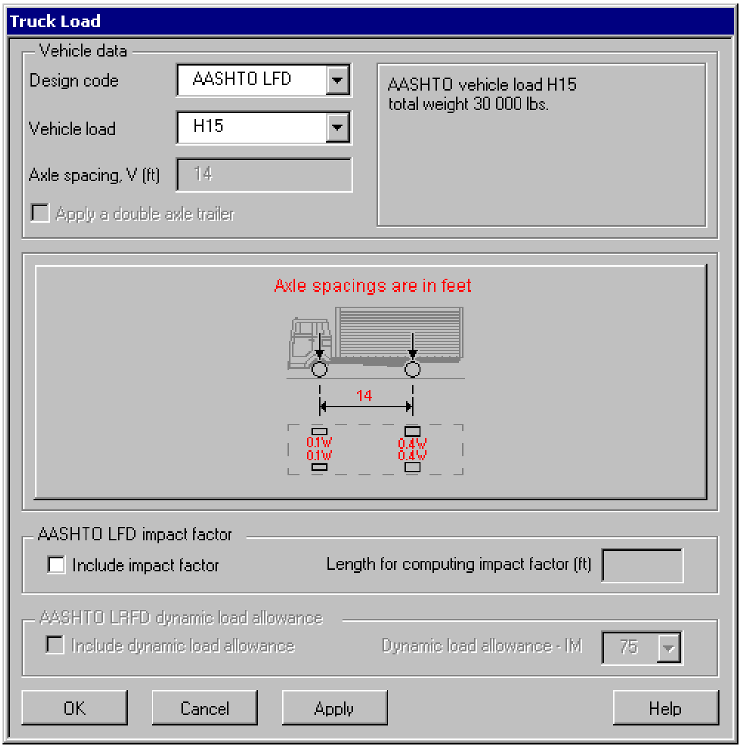
United States of America Truck Loading

The truck load generator produces vehicle loads based on:

AASHTO LFD and AASHTO LRFD codes and other state dependent codes such as Oregon LRFR and West Virginia.

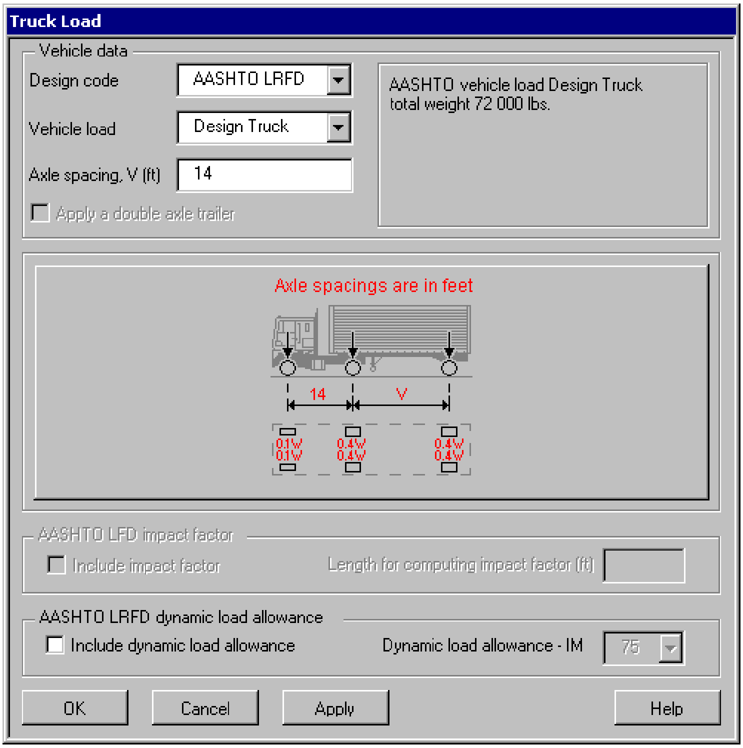
AASHTO LFD Truck Loading

For the AASHTO LFD code the following truck types can be created: H15, H20, HS15, HS20 and HS25. For the HS15, HS20 and HS25 the variable axle spacing can be set between 14ft and 30 ft. For all the trucks the additional impact factor can be added to the loads based on loaded length.



AASHTO LFD Truck Loading

For the AASHTO LRFD code the design truck that forms part of the HL-93 loading can be created. For the design truck the additional dynamic load allowance can be added to the loads based on the impact factor (IM).



To use either loading type:

- Choose the design code that the truck load is to be calculated from.
- Choose the vehicle type required to be generated. Note that if axle spacing is required this has to be between 14 and 30 feet for the **AASHTO LFD** vehicles and 14 and 32 feet for the **AASHTO LRFD** design truck. The vehicle will be generated about its centroid.

For **AASHTO LFD**'s **H20** and **HS20** the option to have a double axle trailer is possible. This is applicable for timber or orthotropic steel decks (excluding transverse beams). For these vehicles either one 32 000 lbs or two 16 000 lbs four feet apart axles can be used. The vehicle giving the most adverse effect should be used for design.

For **AASHTO LFD**, if an impact factor is to be considered, select the check box and enter a length for computing the **impact factor**. The impact allowance is a maximum of 30 % making the impact factor a maximum of 1.3. The impact factor is calculated from the equation below.

$$I = 50 / (L + 125)$$

where L = length for computing the impact factor in feet

For **AASHTO LRFD** what was known as impact in the Standard Specification is called **dynamic load allowance** (IM) in the LRFD Specification and is expressed as a percentage. The base dynamic load allowance factors are present in LRFD Table 3.6.2.1-1. Designers should note that the base values are reduced for buried components and for wood structures.

Oregon LRFR Truck Loading

See **Oregon Department of Transportation Truck Loading**

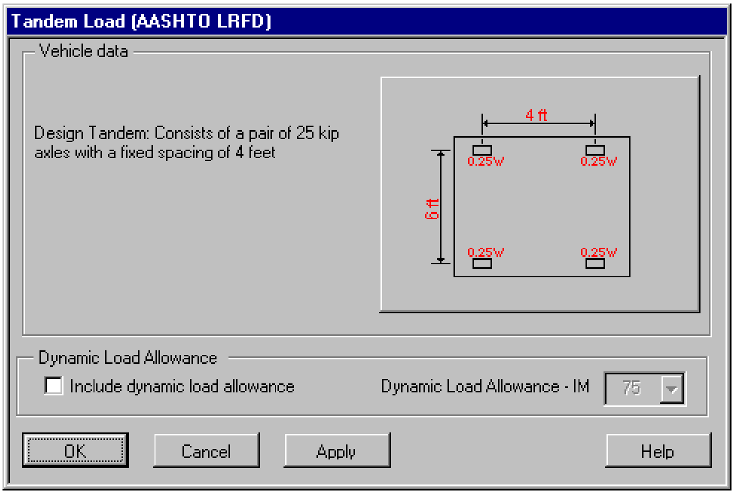
West Virginia Truck Loading

See **West Virginia Vehicle Loading**

United States of America Tandem Load (AASHTO LRFD)

The tandem load generator produces load groups based on the AASHTO LRFD code.

For the AASHTO LRFD code the tandem load that forms part of the HL-93 loading can be created. For the tandem load the additional dynamic load allowance can be added to the loads based on the impact factor (IM).



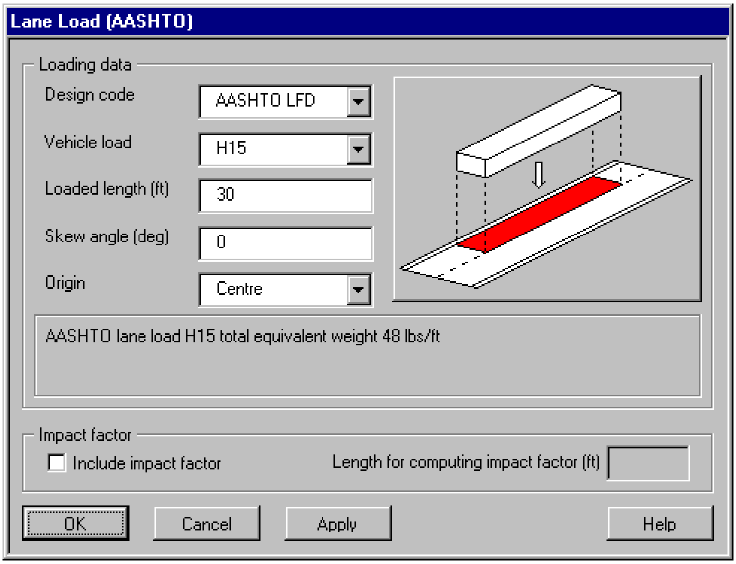
For AASHTO LRFD what was known as impact in the Standard Specification is called dynamic load allowance (IM) in the LRFD Specification and is expressed as a percentage. The base dynamic load allowance factors are present in LRFD Table 3.6.2.1-1. Designers should note that the base values are reduced for buried components and for wood structures.

United States of America Lane Load (AASHTO)

The lane load generator produces a uniform patch load based on the AASHTO LFD and AASHTO LRFD codes.

For the AASHTO LFD code the following load types can be created: H15, H20, HS15, HS20 and HS25. For all the load types the additional impact factor can be added to the loads based on loaded length.

For the AASHTO LRFD code the design lane load that forms part of the HL-93 loading can be created. For the design lane load the additional dynamic load allowance can be added to the loads based on the impact factor (IM).



To use this loading type:

- Choose the design code that the lane load is to be calculated from.
- Choose the patch type from the drop down list to specify the load intensity.
- Specify the length of lane load that you require to generate about the centre line of patch.
- Specify the skew angle to apply to the lane loading (clockwise positive).
- Choose the origin that the load will be generated about.

For AASHTO LFD, if an impact factor is to be considered, select the check box and enter a length for computing the impact factor. The impact allowance is a maximum of 30 % making the impact factor a maximum of 1.3. The impact factor is calculated from the equation below.

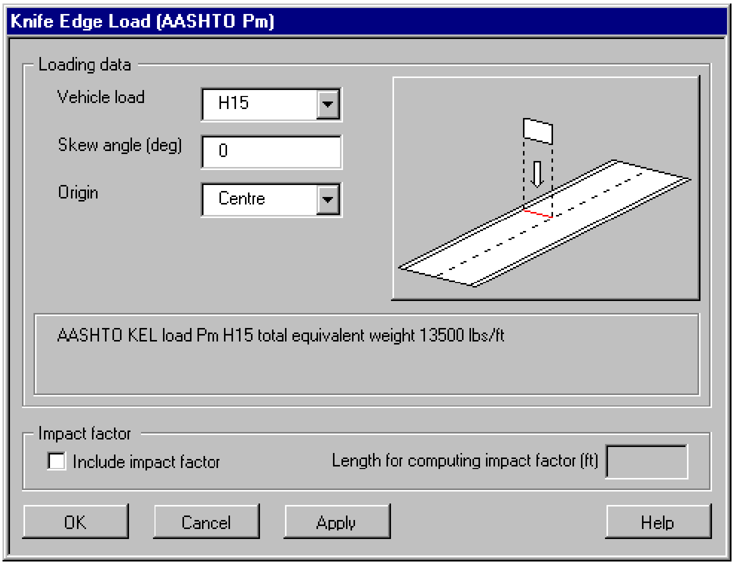
$$I = 50 / (L + 125)$$

where L=length for computing the impact factor in feet

United States of America AASHTO Knife Edge Load (Moment, Pm)

The knife edge load (moment, Pm) generator produces a line load based on the AASHTO LFD code.

For the AASHTO LFD code the following load types can be created: H15, H20, HS15, HS20 and HS25. For all the load types the additional impact factor can be added to the loads based on loaded length.



To use this loading type:

- Choose the vehicle load type from the drop down list.
- Specify the skew angle to apply to the KEL loading (clockwise positive).
- Choose the origin for which the load is to be generated about.
- If an impact factor is to be considered, select the check box and enter a length for computing the impact factor. The impact allowance is a maximum of 30 % making the impact factor a maximum of 1.3. The impact factor is calculated from the equation below.

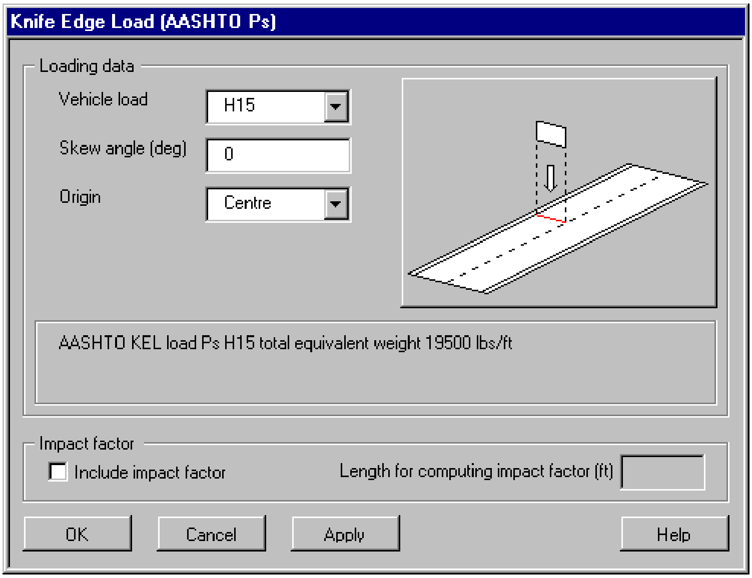
$$I = 50 / (L + 125)$$

where L=length for computing the impact factor in feet

United States of America AASHTO Knife Edge Load (Moment, Ps)

The knife edge load (moment, Ps) generator produces a line load based on the AASHTO LFD code.

For the AASHTO LFD code the following load types can be created: H15, H20, HS15, HS20 and HS25. For all the loads types the additional impact factor can be added to the loads based on loaded length.



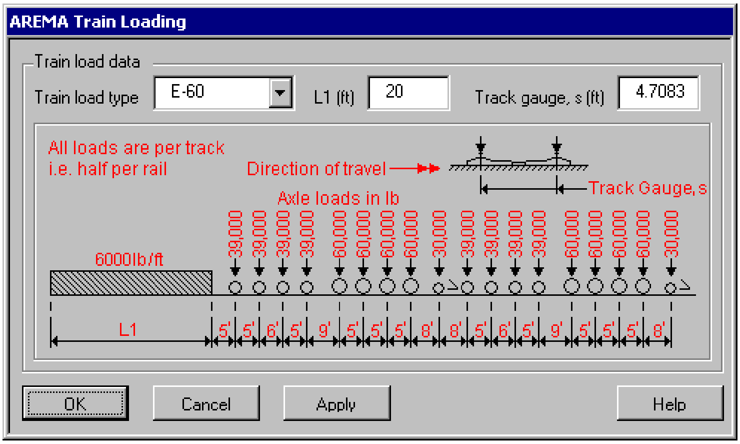
To use this loading type:

- Choose the vehicle load type from the drop down list.
- Specify the skew angle to apply to the KEL loading (clockwise positive).
- Choose the origin for which the load is to be generated about.
- If an impact factor is to be considered, select the check box and enter a length for computing the impact factor. The impact allowance is a maximum of 30 % making the impact factor a maximum of 1.3. The impact factor is calculated from the equation below.

$$I = 50 / (L + 125)$$

where L=length for computing the impact factor in feet

United States of America Train Loading – AREMA



To create this loading type:

- Select the standard rail loading type from the drop down list.
- Enter the dimensions L1 as appropriate for the load model chosen.
- Enter the track gauge.

For this loading a number of discrete load attributes will be created that are used to define a compound load. It is the compound load that should be assigned to the model.

Oregon Department of Transportation Truck Loading

Oregon vehicle loadings are accessed from the **Bridge** > **Bridge Loading** > **United States of America** > **Truck Load...** menu item.

Oregon vehicle loadings are defined to Oregon Department of Transportation specification (ODOT LRFR Manual, 2008). They include the following types:

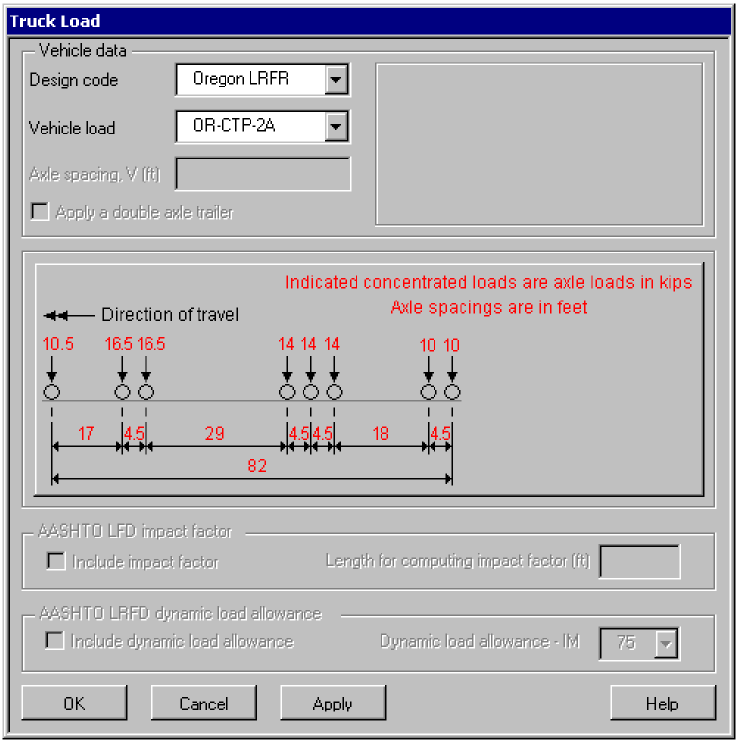
- **ODOT Continuous Trip Permit (CTP) Trucks**
- **ODOT Legal Trucks (LEG)**
- **ODOT Permit Load (PERMIT) Trucks**
- **ODOT Single Trip Permit (STP) Trucks**
- **Specialized Hauling Vehicles (Denoted as SU Trucks)**

For all vehicle types all axle loadings and axle spacings are set. No user input is required.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.

Continuous Trip Permit (CTP) Trucks

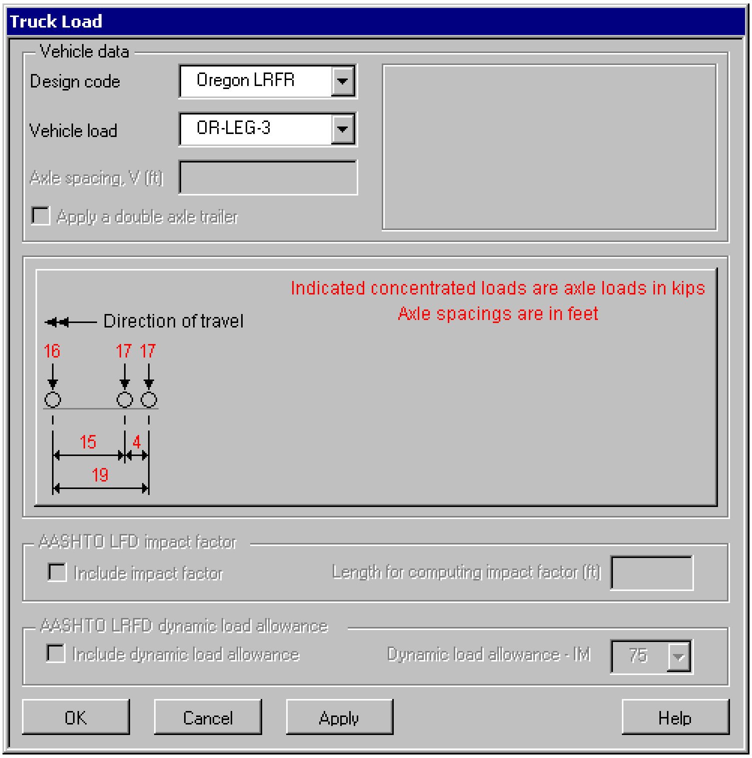
The designations for ODOT permit vehicles contain indicators of the type Continuous Trip Permit (CTP) or Single Trip Permit (STP), and the number of the MTCD weight table it represents. For example, "Type CTP-2A" indicates a Continuous Trip Permit vehicle that conforms to Weight Table 2.



The following Continuous Trip Permit (CTP) Trucks are provided:

- ❑ OR-CTP-2A
- ❑ OR-CTP-2B
- ❑ OR-CTP-3

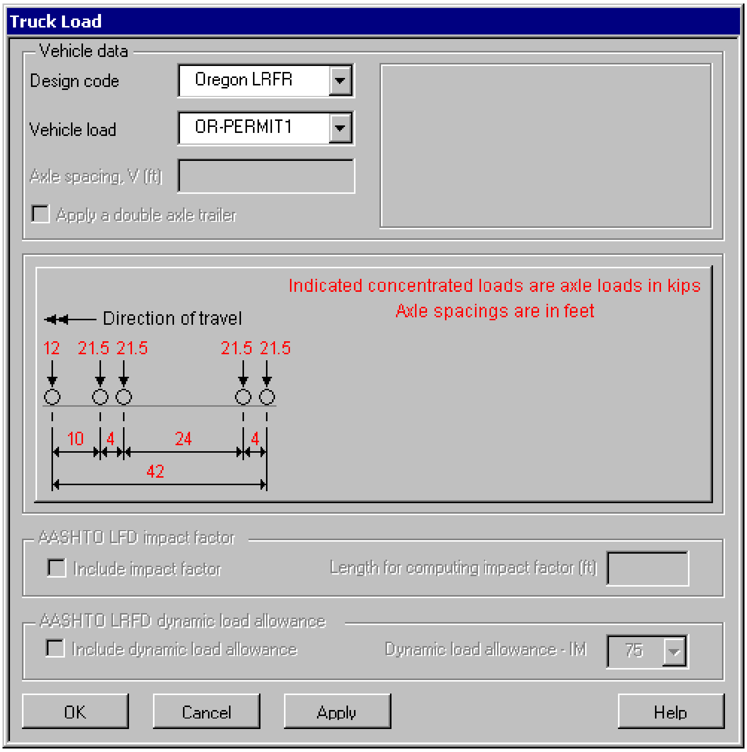
ODOT Legal Trucks (LEG)



The following Legal Trucks (LEG) are provided:

- OR-LEG-3
- OR-LEG-3-3
- OR-LEG-3-3 TRAIN
- OER-LEG-3S2

ODOT Permit Load (PERMIT) Trucks

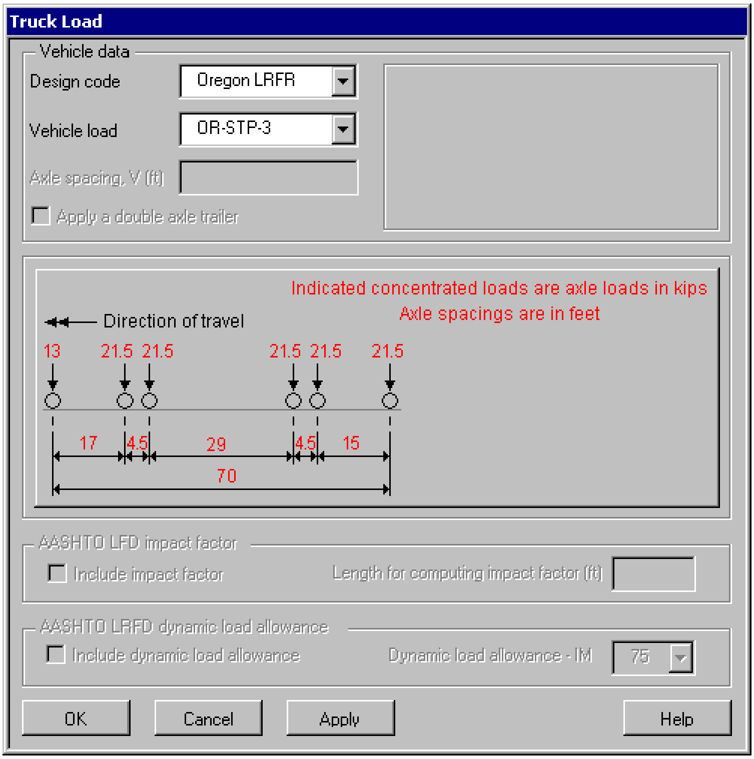


The following Permit (PERMIT) Trucks are provided:

- OR-PERMIT1
- OR-PERMIT2
- OR-PERMIT3
- OR-PERMIT4
- OR-PERMIT5
- OR-PERMIT6
- OR-PERMIT7

ODOT Single Trip Permit (STP) Trucks

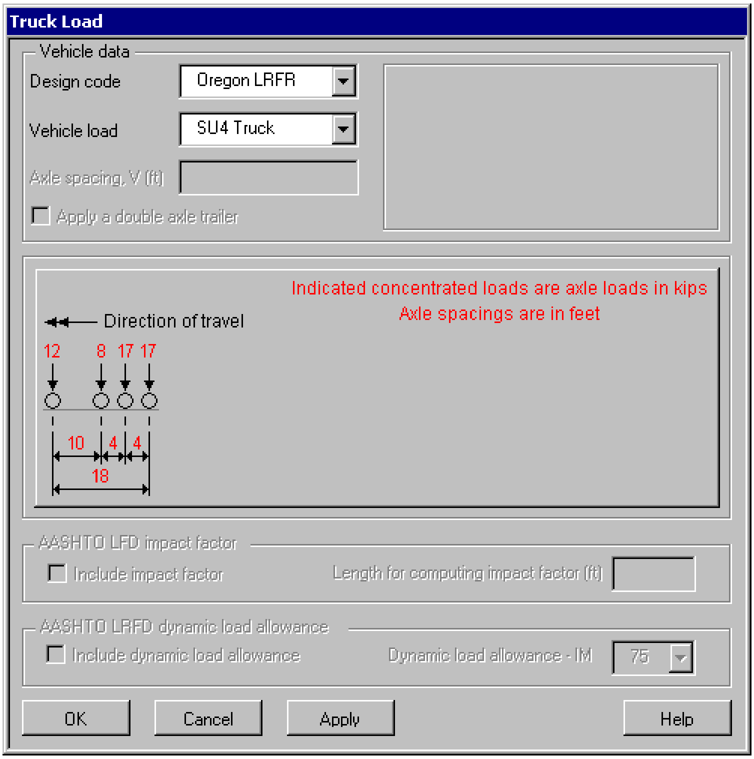
The designations for ODOT Permit Vehicles contain indicators of the type of Continuous Trip Permit (CTP) or Single Trip Permit (STP) and the number MTCD Weight Table it represents. For example, “Type STP-4A” indicates this is a Single Trip Permit vehicle that conforms to Weight Table 4.



The following Single Trip Permit (STP) Trucks are provided:

- OR-STP-3
- OR-STP-4A
- OR-STP-4B
- OR-STP-4C
- OR-STP-4D
- OR-STP-4E
- OR-STP-5BW

Specialized Hauling Vehicles (denoted as SU Trucks)



The following Specialized Hauling Vehicles (denoted as SU Trucks) are provided:

- OR-SU4 Truck
- OR-SU5 Truck
- OR-SU6 Truck
- OR-SU7 Truck

West Virginia Department of Transportation Truck Loading

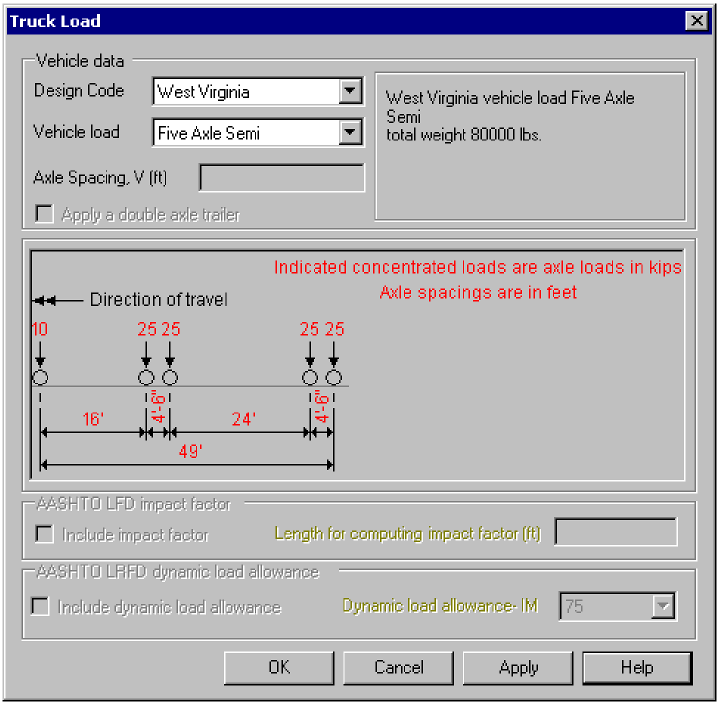
West Virginia vehicle loadings are accessed from the **Bridge** > **Bridge Loading** > **United States of America** > **Truck Load...** menu item. They include the following types:

- SU40 Truck
- SU45 Truck
- 3S55 Truck
- 3S60 Truck

- ❑ Tandem Truck
- ❑ Five Axle Semi Truck
- ❑ Six Axle Semi Truck
- ❑ T3 Truck
- ❑ 3S2 Truck

For all vehicle types all axle loadings and axle spacings are set. No user input is required.

See **Using static vehicle loadings** for general details regarding static vehicle loadings.

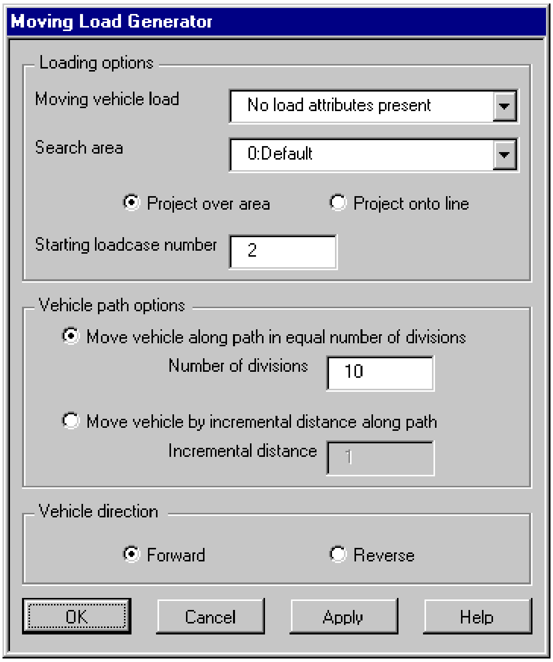


Moving Load Generator

Overview

The moving load generator is accessed from the **Bridge> Moving Load...** menu item. It is used to track the path of a vehicle (or a set of vehicles) across a structure by automatically setting up a number of static loadcases at prescribed locations along a selected line. These loadcases produced can then be enveloped to provide the maximum effect of the vehicle passing over the structure.

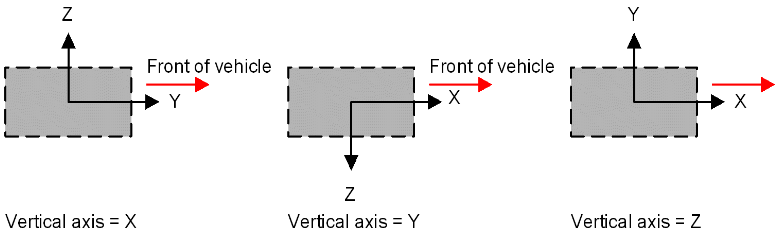
Prior to using the moving load generator a slab or grillage model that will allow the application of discrete loads must have been created. Ensure that the discrete load group representing the vehicle to be moved across the structure has been defined in the Treeview and that a line representing the vehicle path across the structure has been defined and selected. The line representing the vehicle path should not be one of the lines forming the model. If a combined line is selected then the generator will use the points joining the lines within the combined line as the locations for the application of the vehicle loads. For straight lines or arcs the generator will split the selected line representing the vehicle path into a number of segments and create a loadcase for the vehicle in each load position. Only the first line within the selection will be processed.



- **Moving vehicle load** Select the pre-defined vehicle load to be moved along the line
- **Search area** can used to restrict the area of application of point and patch loads to pre-defined part of the model. Generally, for grillage models, a search area is not required.
- **Project over area** needs to selected if the vehicle load is to be applied to a grillage or slab model (3D)
- **Project onto line** needs to be selected if the load is to be applied to a line beam model (2D)
- **Starting loadcase number** can be set to whatever loadcase number is required but by default will be set to be the next free loadcase number.
- **Vehicle path options** define whether the vehicle is to be moved across the structure by a equal number of divisions or by an incremental distance of the line representing the vehicle path.
- **Vehicle direction** allows for forward (defined by the geometric line direction) and reverse passing of the loading.

Load direction

When moving a discrete load where the load direction and the projection vector do not match care should be taken in the direction of the discrete load to ensure that the loading applies in the correct direction along the path. For all vehicles defined using the Bridge > Bridge Loading menu item the front of the vehicle and local axes are defined as follows:

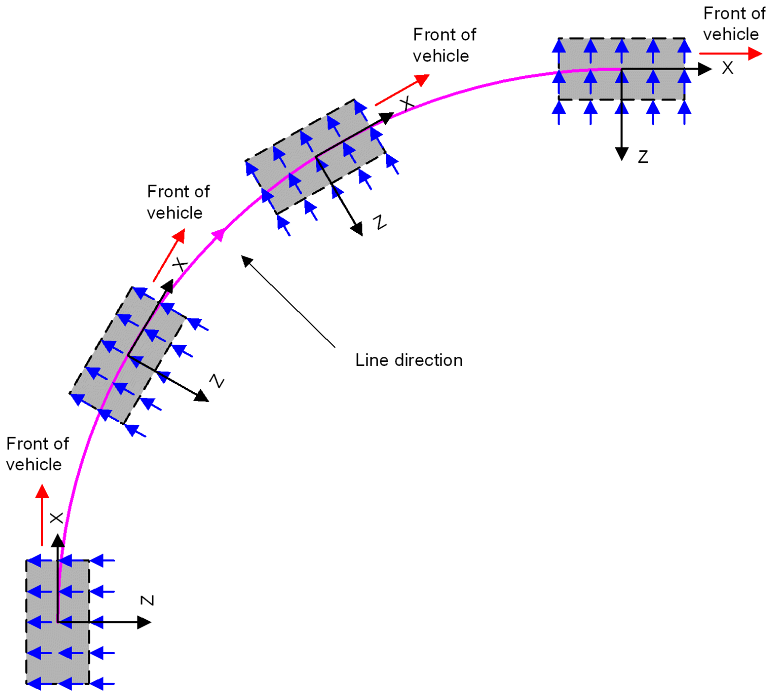


Centrifugal loading

When a centrifugal load is defined for use on a path represented by an arc the point or patch load direction and sign of the vehicle load should be set up to take account of the vertical axis of the model and of the local axes of the vehicle.

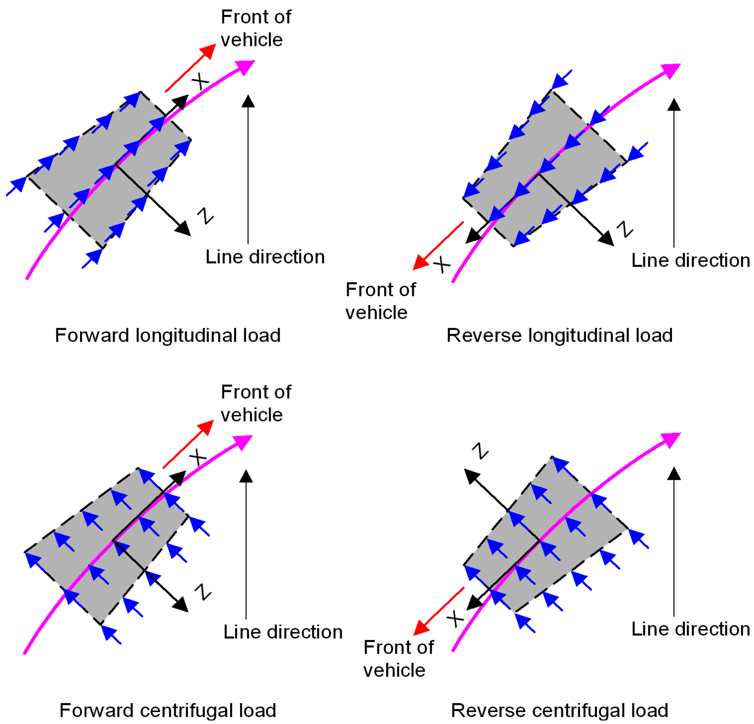
Patch loading

In the following patch load example the vertical axis of the model is set to the Y direction and a vehicle load is moved clockwise around an arc in the ZX plane with centrifugal loading defined as a negative value in the Z direction:



Reversing the vehicle direction by using the Reverse option on the Moving Load dialog will change the orientation of the vehicle configuration so it is rotated 180 degrees such that the vehicle will now pass along the path in the opposite direction with this new orientation. Loads defined in the lateral vehicle directions will be rotated ensuring that the lateral loads for centrifugal loading will be maintained in the correct direction as the vehicle passes along the path.

The following examples (using trapezoidal-shaped patch loading for clarity) show the effect of vehicle direction upon any horizontally defined longitudinal or centrifugal loads:



Point loading

For point loading similar care should be taken to ensure that the vertical axis of the model and the untransformed load directions for longitudinal or centrifugal loading are compatible. Reversing the vehicle direction by using the Reverse option on the Moving Load dialog will ensure that any lateral loads are rotated and, as for patch loading, the centrifugal loading will be maintained in the correct direction as the point load passes along the path.

Moving multiple loads

Discrete loads can be assigned to, and manipulated on a model as a load set or load train by creating a **compound discrete load.** See the Modeller User Manual for details.

Vehicle Load Optimisation

Overview

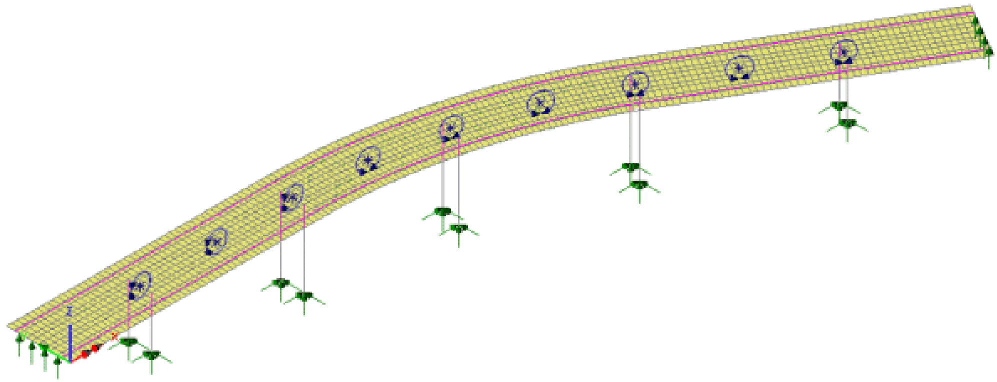
Vehicle load optimisation (VLO) makes use of influence surfaces and **influence analysis** to identify the most onerous vehicle loading patterns on bridges for a chosen design code and to apply these loading patterns to LUSAS models. It reduces the amount of time spent generating loadcases to replicate traffic and lane loading on models and leads to more efficient and economic design, assessment or load rating of bridge structures. It can be applied to grillage, line beam and plate/shell models.

Two influence analysis methods are available: the Reciprocal Method and the Direct Method. This general overview of VLO applies to both except where stated.

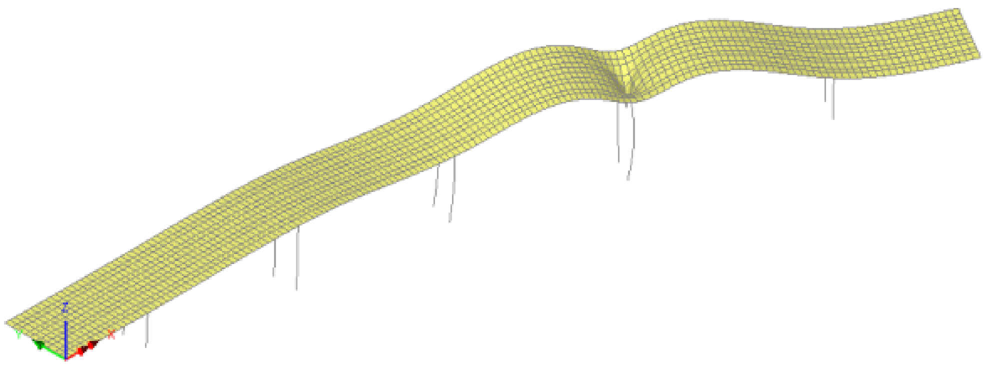
Performing a vehicle load optimisation analysis

To evaluate the effect of the most onerous vehicle loading patterns on a bridge using Vehicle load optimisation analysis is essentially a four stage process. A simple plate/shell model for analysing traffic loading to Eurocode EN-1991-2 for a number of indicative mid-span influence points is shown to illustrate the stages involved.

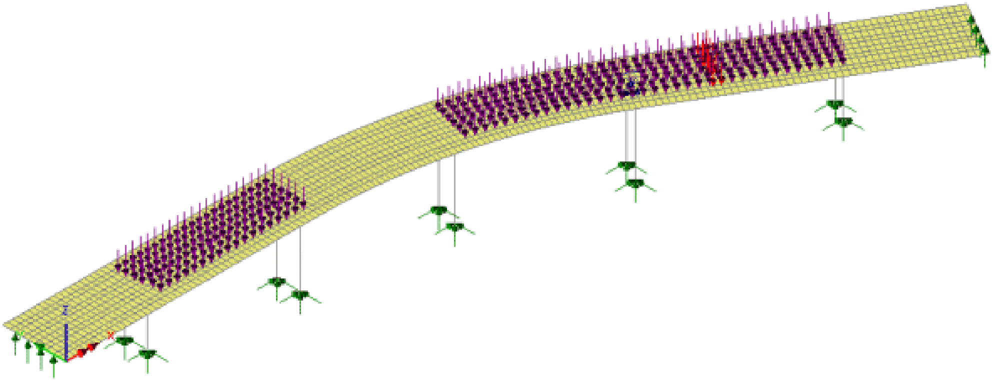
- **Stage 1 - Assign influence attributes.** Before running a Vehicle Load Optimisation analysis **influence attributes** must have been assigned to selected positions (nodes or points of interest) on a model. Kerb lines defining the extent of the carriageway need to be defined.



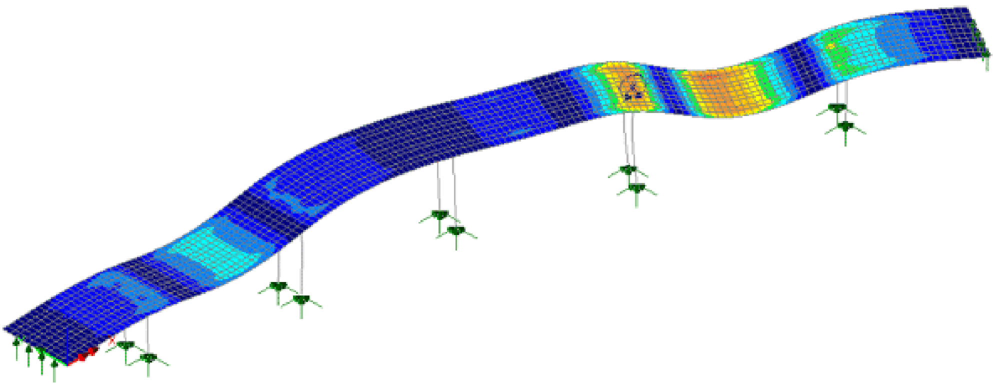
- **Stage 2 - Optional solve to view influence shapes.** Before running a Vehicle Load Optimisation analysis an **influence analysis** can be optionally run to investigate the influence lines or surfaces for the assigned influence attributes. An influence surface for each influence position is automatically calculated and can be optionally displayed. Note that influence shapes can also be viewed after a successful VLO analysis has been carried out.



- **Stage 3 - Run a VLO Analysis.** Running the Vehicle load optimisation facility interrogates each influence surface and calculates the critical loading pattern for the chosen effect. The critical loading pattern can be optionally displayed prior to calculating loading effects.



- **Stage 4 - Solve to calculate effects of traffic load patterns.** After running a Vehicle Load Optimisation analysis the model must be solved to calculate the effect of the critical loading patterns on the model. Critical loading patterns can be optionally superimposed on deformed or undeformed results plots.



Note: For a line beam model, where the geometric section represents a beam with a loadable top slab, a loadable grid of points is used to represent the slab. See **Direct Method Influence Attributes** for more details.

Assigning Influence Attributes to a model

See **Influence Attributes** for details.

Running a Vehicle Load Optimisation Analysis

Vehicle load optimisation is provided in Bridge software products only. It can be accessed from the **Analyses** > **Bridge** > **Vehicle Load Optimisation** menu items.

Before a Vehicle Load Optimisation analysis can be undertaken, **influence attributes** need to be assigned to the model and kerb lines defining the extent of the carriageway need to be specified. A successful analysis will add a Reciprocal Influence or a Direct Method Influence Analysis entry (depending upon influence attributes used) as well as a VLO Analysis entry (and VLO runs) to the Analyses Treeview. These entries permit viewing of influence shapes and traffic loading patterns prior to solving the model for those loading patterns.

Any number of VLO Analysis entries can be created in the Analyses Treeview. Each VLO Analysis entry can contain any number of VLO runs, with each run containing loadcases generated by the vehicle load optimisation software according to the chosen design code and associated settings chosen. Each loadcase comprises vehicle and lane loadings appropriate to the design code selected that, when solved, will generate the most adverse positive or negative loading effect for each of the influence points of interest, as defined on the model. If desired, it is possible to set-up one VLO run to investigate Positive effects and a separate run to investigate Negative effects, or they can be combined into one run, all within the same Analysis. VLO runs can be copied and pasted between different VLO analyses.

For a detailed explanation of how to use the Vehicle Load Optimisation facility see the **Vehicle Load Optimisation Wizard**

Viewing of influence shapes

After an influence analysis has been carried out the influence shape for each influence point of interest can be seen by setting each influence loadcase active, in turn, in the Analyses Treeview.

The influence shape can be viewed on the **Influence shape** layer in the Layers Treeview. Note that this layer replaces the Deformed mesh layer when an influence analysis loadcase is active. It cannot be added to the Layers Treeview manually.

By adding a Contours layer to the Layers Treeview contours of **Influence result** (Direct Method only) can be displayed. By referring to the contour key regions of the model where positive or negative loading effects take place can be seen.

Note: When a Direct Method influence analysis has been solved for one assigned attribute the effect of a unit load on any part of the structure can be seen immediately for any subsequently assigned influence attributes.

Viewing of vehicle loading patterns

By turning on the display of loading and setting each loadcase active in the Analyses Treeview, the vehicle loading pattern for each loadcase generated by the vehicle load optimisation wizard can be viewed.

By expanding each loadcase's loading entry, the individual loading types and their corresponding load factors that make-up each loadcase can be seen. Context menu items provide additional visualisation and other options.

Solving to calculate effects of traffic load positions

With influence result loadcases and loadcases representing the critical traffic loading patterns present in the Analyses Treeview, pressing the Solve button on the main toolbar menu will analyse the effect of these loadings on the structure.

It is also possible to the choose the **Solve Now** menu item on context menu of individual analysis entries. Loadcases can also be solved selectively by choosing the **Loadcase to Solve** context menu item.

Updating and re-running a vehicle load optimisation analysis

Editing an existing VLO Run entry in the Layers Treeview to include more influences will delete and re-create existing influences as well as add new results loadcases for the additional influences.

Superposition

Vehicle Load Optimisation software supported by LUSAS inherently uses the principal of linear superposition. In a linear static analysis this assumption holds true for all results components that are calculated by a LUSAS Solver, but note that it does not hold true for all results components that are post-processed by LUSAS Modeller. For example, Wood Armer calculations are not linear calculations, therefore superposition is not safe for these. Similarly, the definition of any user-defined results components (that can be used in the definition of a Direct Method influence) may include equations with terms or constants that are not scalable.

Viewing Results

See **Visualising the Results**.

Vehicle Load Optimisation Wizard

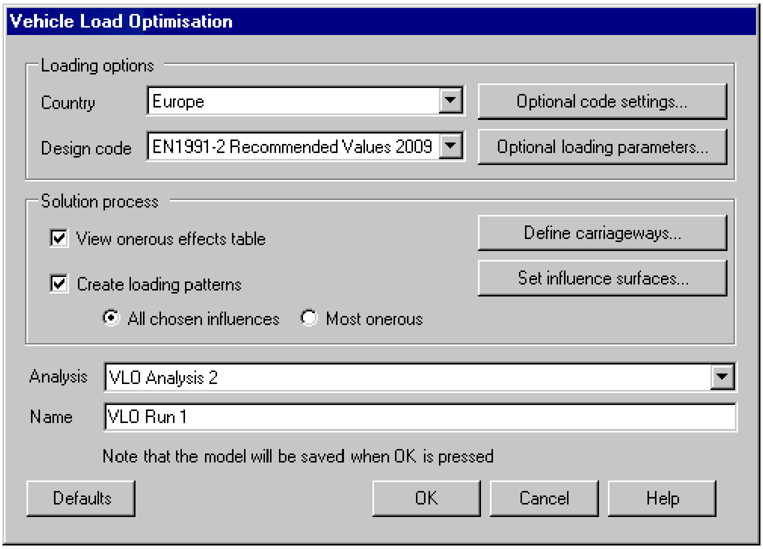
The vehicle load optimisation wizard can be accessed from the **Analyses > / Bridge > Vehicle Load Optimisation** menu items. It provides the means of defining parameters, for a particular design code, to generate the most critical traffic loading pattern for each influence shape under consideration.

The actual vehicle load optimisation software that is used to generate this loading (either LUSAS Traffic Load Optimisation, or Autoloader Vehicle Load Optimisation) depends upon the design code chosen. Please see the appropriate heading within this topic for details of which codes are supported and how each piece of software operates.

Note that prior to accessing the Vehicle Load Optimisation wizard to run an VLO analysis **influence attributes** must have been assigned to selected nodes or points of interest on a model. Kerb lines defining the extent of the carriageway also need to be defined or selected.

See **Vehicle Load Optimisation Explained** for a general overview of the vehicle load optimisation process. **Worked Examples** illustrate the steps involved in carrying out different types of vehicle load optimisation.

Design codes supported by LUSAS Traffic Load Optimisation (LUSAS TLO)



Country design codes currently supported by the LUSAS Traffic Load Optimisation (LUSAS TLO) software option include:

- **Australia** - **AS5100-2:2004** and **AS5100-7: 2004 (Austroads)**
- **Canada** - **CAN/CSA-S6-06 (Design)**
- **Europe** - Eurocode EN1991-2 Recommended values
- **Ireland** - **Eurocode EN1991-2**
- **Italy** - **Eurocode EN1991-2**
- **New Zealand** - Transit New Zealand Bridge Manual (SP/M/022 2nd Edition)
- **Poland** - **Eurocode EN1991-2**
- **Sweden** - **Eurocode EN1991-2 (2009)**
- **Sweden** - **Eurocode EN1991-2 (2011)**

- **United Kingdom** - **Eurocode EN1991-2**
- **United Kingdom** - Highways Agency Departmental Standard **BA34/90**“Design Manual for Roads and Bridges, Volume 3, Section 4, Part 17: BA34/90 (and BD34/90) Technical Requirements For The Assessment And Strengthening Programme For Highway Structures, last updated Sept 1990.
- **United Kingdom** - Highways Agency Departmental Standard **BD21/01** “Assessment of Highway Bridges and Structures”.
- **United Kingdom** - Highways Agency Departmental Standard **BD86/11** “Loads for Highway Bridges”, which incorporates BS5400 Part 2:1978.
- **United States of America** AASHTO LRFD (7th and 6th Edition), and AASHTO Standard Specifications (17th Edition).

Note: For the Eurocode EN1991-2 **Recommended values** option the values for Nationally Determined Parameters (α_Q, α_q, ψ) and the EN1991-2 informative Annex A Special Vehicles are used by default.

Optional Code Settings (LUSAS TLO)

Optional code settings can be specified for all design codes supported by LUSAS TLO. Click on the country links above for each country, see the online help dialogs for each country load type for details, or refer to the relevant help pages in the *Application Manual (Bridge, Civil & Structural).*

Optional Loading Parameters (LUSAS TLO)

For design codes supported by LUSAS TLO the following parameters are available for selection:

- **Longitudinal increment** - specifies the increment used when moving the abnormal vehicle along the carriageway. The smaller the increment, the more accurately the effects of the vehicle across the structure are calculated, giving more accurate results. A larger increment gives quicker TLO runs, but with less accuracy. A generally suitable default value is provided, but users can specify an alternative value based upon experience.
- **Transverse increment** - specifies the increment used when moving the vehicle across the carriageway. The smaller the increment, the more accurately the effects of the vehicle across the structure are calculated, giving more accurate results. A larger increment gives quicker TLO runs but with less accuracy. A generally suitable default value is provided, but users can specify an alternative value based upon experience.
- **Vehicle Direction** - specifies the vehicle direction which is used to calculate the effects of each vehicle. Vehicle axles are defined from the leading axle. **Forward** means the vehicle is run in a forward direction along the carriageway from the starting point of the carriageway. **Reverse** means the vehicle is run with the normally trailing axle leading along the carriageway from the starting point of the carriageway. **Both** means both cases are run. For an asymmetric vehicle, either forward or reverse could

produce the greatest effect, dependent upon the shape of the influence surface and the increments used. The default is both directions.

Defining Carriageways (LUSAS TLO)

For design codes supported by LUSAS TLO, lines defining the width of the carriageway must be present in the model and selected. Where lines already represent the edges of a structure, or the centreline of a carriageway, lines representing the width of a carriageway can be easily added using the **Geometric> Line> By offsetting** menu item. Note that these lines do not have to form a structural part of the model. Once done the following parameters are available for selection:

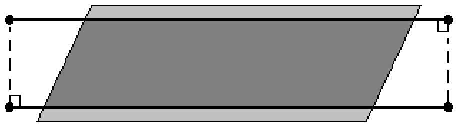
- **Kerb positions** - define the width of each carriageway. Note that for LUSAS TLO it is not possible to define kerb positions via tabular input.

    - **Kerb from Selection** - The kerb positions of carriageways are defined by selecting those lines defining the kerbs on either side of a carriageway in the view window prior to accessing the Vehicle Load Optimisation dialog. Straight lines, arcs or a combination of these two feature types can be used and selected to describe the extent of the carriageway. Multiple sets of carriageways can be defined by selecting multiple sets of those lines defining the extent of each carriageway.

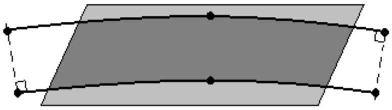
Permissible kerb arrangements (LUSAS TLO)

- Pairs of straight lines defining kerb positions must be parallel, and the start and end points of each kerb must be orthogonally opposite each other.
- Pairs of arcs defining kerb positions must be concentric and have the same subtended angle
- Consecutive straight lines and arcs can be used to define kerb positions but opposite pairs of lines that define the kerb positions must of the same type, that is, either both straight lines or both arcs, and not a mixture. Consecutive straight lines and arcs must be tangential. Using only consecutive straight lines to define kerb positions is invalid unless the intersection points are filleted.
- Kerb lines should lie in a common plane. For bridge decks with a variable longitudinal vertical profile (such as a vertically curved deck) a primary (X,Y), (Y,Z) or (Z,X) plane must be used. For bridge decks with a constant longitudinal vertical profile (such as a horizontal or inclined bridge deck) the kerb lines may lie in a primary plane or in the same plane as the bridge deck.
- Kerb lines must be transversely within the limits of any Direct Method Influence grid, if one is used during influence attribute assignment.

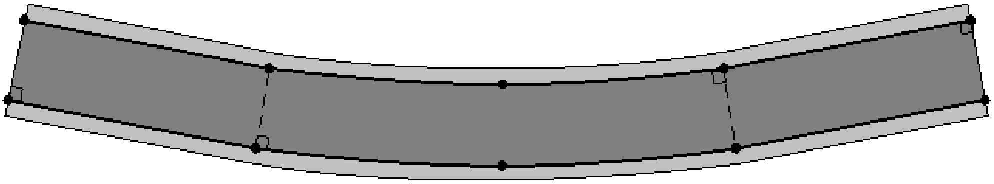
Valid kerb arrangements showing the resulting loadable regions in darker shading are shown:



Straight kerbs



Curved kerbs



Straight and curved kerbs - opposing pairs of lines/arcs are of the same type (LUSAS TLO only)

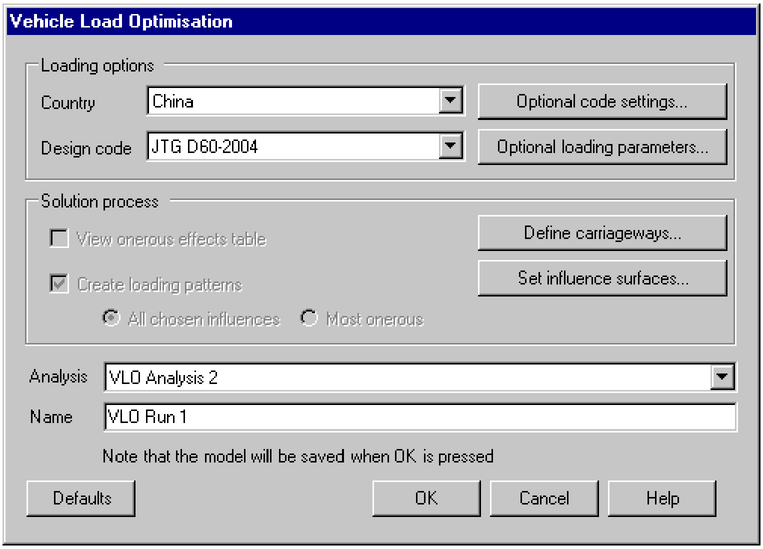
Definition of multiple sets of carriageways

General notes for design codes supported by LUSAS TLO:

- When using LUSAS TLO the model can be defined in any quadrant of the Modeller view window. For example in positive x, positive y, or in negative x, positive y.
- LUSAS TLO vehicle loads can be projected in either the X, Y or Z directions, meaning that models can lie in any of the primary (X,Y), (Y,Z) or (Z,X) planes, noting that non-planar decks are also supported.
- When using shells, quadratic elements with mid-side nodes are permitted.

See also **Setting-up Influence Surfaces** and **Solution Process Options** below.

Design codes supported by Autoloader Vehicle Load Optimisation (Autoloader)



Country design codes currently supported by LUSAS using the Autoloader vehicle load optimisation software option include:

- **China - JTG D60-2004**
- **Republic of South Africa** - **TMH7 Loading** standard.
- **United Kingdom**- Highways Agency Departmental Standard **BD37/01** "Loads for Highway Bridges".

To use retired Autoloader implementations of design codes for old models please contact LUSAS technical support.

Optional Code Settings (Autoloader)

Optional code settings can be specified for all design codes supported by Autoloader. Click on the country links above for each country, see the online help dialogs for each country load type for details, or refer to the relevant pages in the *Application Manual (Bridge, Civil & Structural).*

Optional Loading Parameters (Autoloader)

For vehicle load optimisation of design codes supported by Autoloader, the following parameters are available for selection:

- **Vehicle Library**- specifies an alternate file containing the Autoloader Vehicle Library. This enables the user to define another vehicle for a set of Autoloader runs. By default, Autoloader looks for autoload.vec in the working directory.
- **Vehicles** - defines a list of vehicles that can be chosen to calculate the greatest effect. Autoloader tests the vehicle chosen in this list. Each vehicle must be specified in the Autoloader vehicle library. If the field is not specified, HA or equivalent only loading is assumed.
- **Multiple Vehicles** - specifies whether a single vehicle is used or a list of vehicles. Each vehicle must be specified in the Autoloader vehicle library. To use this option select the check box next to the label and enter the list of vehicles in the text box with commas separating each name. To deselect this option ensure that the check box is not selected. By default this option will not be selected.
- **Vehicle Direction** - specifies the vehicle direction which is used to calculate the effects of each vehicle. Vehicle axles are defined from the leading axle. **Forward** means the vehicle is run in a forward direction along the carriageway from the starting point of the carriageway. **Reverse** means the vehicle is run with the normally trailing axle leading along the carriageway from the starting point of the carriageway. **Both** means both cases are run. For an asymmetric vehicle, either forward or reverse could produce the greatest effect, dependent upon the shape of the influence surface and the increments used. The default is both directions.
- **Longitudinal increment** - specifies the increment used when moving the abnormal vehicle along the carriageway. The smaller the increment, the more accurately the effects of the vehicle across the structure are calculated on the carriageway, giving more accurate results. A larger increment gives quicker Autoloader runs. A generally suitable default value is provided but users can specify an alternative value based upon experience. If an invalid value is specified, Autoloader gives a warning and uses the default value.
- **Transverse increment** - specifies the increment used when moving the vehicle across the carriageway. The smaller the increment, the more accurately the effects of the vehicle across the structure are calculated on the carriageway, giving more accurate results. A larger increment gives quicker Autoloader runs. A generally suitable default value is provided but users can specify an alternative value based upon experience. If an invalid value is specified, Autoloader gives a warning and uses the default value.
- **Edit advanced loading options** - further to the basic options provided on this dialog, Autoloader allows more advanced changes to be made. To access this functionality ensure that the check box next to the label is selected and click on the Advanced button. However, note that it is unlikely that these advanced options will be required for the majority of work.

Optional Loading Parameters (Advanced) (Autoloader)

For vehicle load optimisation of design codes supported by Autoloader the following advanced parameters are available for selection:

- **Use extra parameter file**- specifies another parameter file for Autoloader. This option allows the user to specify a generic set of parameters in another file and use that file in a series of Autoloader runs. This additional file can contain anything that the input file can contain. It is read after the entire input file is processed, and any parameter specified within this file will overwrite one specified in the original file. To use this option select the check box next to the label and specify the location of this extra file, to deselect this option ensure that the check box is not selected. By default this option will not be selected.
- **Use alternative KEL** - specifies an alternate value for the Knife Edge Load to use when applying KEL loading. To use this option select the check box next to the label and specify the new value for the KEL loading in the text box, to deselect this option ensure that the check box is not selected. By default this option will not be selected.
- **Use UDL Limit** - allows specification of a loaded length, over which the lane in question is not loaded. This can be used in conjunction with HA Alternative to create a situation such as in BD 21/97, where if the loaded length is below 2m, the lane is loaded with a Single Axle Load instead of a UDL + KEL. To use this option select the check box next to the label and specify the UDL limit in the text box, to deselect this option ensure that the check box is not selected. By default this option will not be selected.
- **Use Beta Lane Factors** - allows specification of alternate HA lane factors to use when calculating the effects of HA loads. The format is a list of factors. The first factor is used for the lane with the greatest effect, the second is used for the lane with the second greatest effect, etc. If there are more lanes than factors, then the last factor in the list is used for any lanes without corresponding factors. It should be noted that for the JKR standard the make up lane is also treated as an HA lane and the user should specify a factor for it (bearing in mind that it is quite likely to have the least effect). To use this option select the check box next to the label and specify the alternate HA lane factors in the text box, to deselect this option ensure that the check box is not selected. By default this option will not be selected.
- **Use Lane Modification factors** - allows specification of alternate lane modification factors to use when calculating effects. The format is a list of factors. The first value is used when there is one lane loaded, the second when there is two, etc. If there are more lanes than factors, then the last factor in the list is used. This option is intended when AUSTROADS, but can also be used within the other standards. To use this option select the check box next to the label and specify the alternate lane modification factors in the text box, to deselect this option ensure that the check box is not selected. By default this option will not be selected.
- **Use Dynamic Load allowance** - allows the dynamic load allowance for gamma factors. This is intended for AUSTROADS, but can also be used within the other standards. If an invalid value is specified, Autoloader gives a warning and uses the default value. To use this option select the check box next to the label and specify the dynamic load allowance for gamma factors in the text box, to deselect this option ensure that the check box is not selected. By default this option will not be selected.

- **Use Alternative loading intensity curve** - provides the ability to change the relationship between loaded length and intensity of the UDL applied. Each pair of comma separated values is a length and an intensity for that length. Autoloader does straight line interpolation between these values. If more accurate loading intensity is required the user should specify as many points as possible. To use this option select the check box next to the label and specify the relationship between loaded length and intensity of the UDL in the text box, to deselect this option ensure that the check box is not selected. By default this option will not be selected.
- **Use Alternative adjustment factors** - provides the ability to change the relationship between loaded length and the Adjustment factor for BD 21/97. Each pair of comma separated values is a length and a factor for that length. Autoloader performs linear interpolation between these values. If a more accurate value is required the user should specify as many points as possible. To use this option select the check box next to the label and specify the new relationship between loaded length and the Adjustment factor in the text box, to deselect this option ensure that the check box is not selected. By default this option will not be selected.

Defining Carriageways (Autoloader)

For vehicle load optimisation of design codes supported by Autoloader, lines defining the width of the carriageway can be optionally drawn in the model, and selected, or tabular coordinate input can be used. For either case, the following parameters are available for selection:

- **Carriageway Shape** - specifies whether the carriageway is **curved** or **straight** in plan view
- **Number of Carriageways** - specifies the number of carriageways to be placed on the deck. Autoloader assumes the kerbs of each carriageway are parallel.
- **Angle of carriageway** - is only required when the carriageway shape is straight. It specifies the angle of inclination in degrees of the carriageway in an anticlockwise direction relative to the positive x axis. The angles for multiple carriageways are entered as a comma separated list of values. This is the input for the CWDIR parameter for Autoloader as explained in the Autoloader Reference Manual.
- **Kerb positions - -**allows entering the positions of the kerbs on each carriageway using two methods:
    - **Kerb from tabular input** - if this option is selected, when the Apply button is clicked a tabular dialog is presented for entering the kerb positions in cartesian coordinates. When the carriageway shape is straight, two points per carriageway are specified. The first of these is the ‘base’ point and is on a kerb at the start of the carriageway. The other point is anywhere along the other kerb. When the carriageway shape is curved, four points per carriageway must be specified. The first of these is the ‘base’ point and is on a kerb at the start of the carriageway. The next two points are on the same kerb, and are used to calculate the centre of curvature and the radius of curvature. The final point is

anywhere along the other kerb. Autoloader uses this data to calculate the width of the carriageway.

- **Kerb from Selection**- The kerb positions of the carriageways may be defined by selecting lines defining the kerbs on either side of a carriageway on screen prior to displaying the Vehicle Load Optimisation dialog. Only straight lines or only arcs that describe the extent of the carriageway can be selected. Combined sequences of these feature types are not supported by Autoloader. Multiple sets of carriageways can be defined by selecting multiple sets of those lines defining the extent of each carriageway.

Permissible kerb arrangements (Autoloader)

- Pairs of straight lines defining kerb positions must be parallel, and the start and end points of each kerb must be orthogonally opposite each other.
- Pairs of arcs defining kerb positions must be concentric and have the same subtended angle
- Consecutive straight lines and arcs cannot be used to define kerb positions.

For valid and invalid kerb arrangements see **Permissible Kerb Arrangements (LUSAS TLO)**

General Notes for design codes supported by Autoloader

For vehicle load optimisation of design codes supported by Autoloader:

- Autoloader projects vehicle loads in the negative Z axis direction. As a result, models for use with Autoloader must be set up in the x,y plane to ensure that loading can be applied in this direction.
- When using Autoloader the model must be created in the positive x,y quadrant of the view window
- In certain loading situations an upward load may be applied by Autoloader. In the case of a UK design code, this can be a result of a HB loading overhanging a lane where HA loading is applied. The upward load is applied to cancel the excess of HA loading. The loading standard BD 37/88 "Loads for Highway Bridges", clause 6.4 deals with this loading situation in more detail.
- When using a grillage with the Vehicle Load Optimisation facility each of the lines representing a section of the deck must be meshed with one element only i.e. each bay of the grillage must have one element assigned. In addition, only lower order elements can be used. Quadratic elements with mid-side nodes are not permitted.
- Non-planar decks are supported.

- The use of Autoloader is described fully in the *Autoloader User Manual.*

Setting-up Influence Surfaces

Applicable to vehicle load optimisation of design codes supported by LUSAS TLO and Autoloader.

- The **List of available influences** shows all assigned **influence attributes** for the model. These can be included in a vehicle load optimistion analysis en-mass, or by individual selection.

The following additional settings can be made:

- **Include all influence surfaces** - selects all the influence surfaces in the model for use in the vehicle load optimisation.
- **Positive** and/or **Negative** Vehicle loading can be placed on either Positive or Negative areas by checking the Positive or Negative (or both) checkboxes for each of the included influences. Note that clicking in the header cell of a column will select the whole column so that checking (ticking) one entry in the column will check (tick) all entries in that column. Selection of multiple columns is also possible.
- **Alternative load pattern** (for AASHTO LFD Standard Specifications clause 3.11.3, AASHTO LRFD clause 3.6.1.3.1, and Korean lane loads on continuous spans only). When the column is checked, the influence surface referred to in that row of the grid will be treated as set out in the clauses mentioned above - an option suitable for negative moments between points of contraflexure or reactions of interior piers. Checking the box will use two concentrated (knife edge) loads per lane (LFD) or two trucks per lane (LRFD) in the load pattern generation (other rules in the referenced clauses relating to spacing and reduction factors are included, as appropriate also).
- **Increment for influence surfaces** (for vehicle load optimisation of **design codes supported by Autoloader only**) - Specify the increment interval used when interpolating influence values to obtain the influence lines for the centrelines of the notional lanes. A generally suitable default value is provided but users can specify an alternative value based upon experience. The smaller the increment, the more positions of the vehicle will be calculated along the carriageway, giving more accurate results. A larger increment gives quicker Autoloader runs but with potentially less accuracy. If an invalid value is specified, a warning will be given and the default value used. Length units as specified on the main dialog should be used.
- **Search Area to be used for reciprocal analysis** - The search area choice applies only to the assignments of reciprocal influences. Direct method influence assignments will automatically use the search area (if any) of the direct method influence analysis in which the influence attribute was assigned. Use of a search area will restrict the applied vehicle loading to a specified portion of the model. If a search area is not specified the generated vehicle loading will be projected onto the whole model ("Default"). For models where multiple intersections of the load projection occur it is necessary to restrict the loading to the required face using a search area.

Solution Process Options

Applicable to vehicle load optimisation of design codes supported by LUSAS TLO and Autoloader (except where noted)

- **View onerous effects table** (For design codes supported by LUSAS TLO only) Shows tabbed and sorted results for all chosen influences at nodes that are visible, with the most onerous result shown in the first row of the table. Displayed results can be filtered by subsequently selecting features or mesh objects of interest. For more information see **Onerous Effects Table** below.

- **Create loading patterns** (For design codes supported by LUSAS TLO only) To run a Vehicle Load Optimisation analysis with optimised loading select this check box, selecting either the **All chosen influences** option, which generates loads and loadcases for all chosen influences; or the **Most onerous** option, where for each influence attribute the loads and loadcases are only generated for the influence assignment that gives the most onerous effect. For both options, loading entries are created in the Attributes treeview and an entry is created in the Analyses Treeview under the corresponding VLO Run. To visualise the loading patterns a loadcase must be set active and appropriate **loading visualisation settings** made.

- The **Analysis** name provided creates an appropriate entry in the Analyses Treeview. Note that an analysis can contain up to 1000 VLO Runs. Separate user-defined analysis entries can be created to hold different VLO Runs using the 'New' analysis option on the drop-down menu for the Analysis entry, but for speed of solving, note that it is more beneficial to have all VLO Runs contained within one analysis.

- The Vehicle Load Optimisation **Name** is used to identify each VLO Run. Short names are recommended since VLO Run names are used to create filenames that also include the influence name and coordinate and element details, and if collectively this is too long, the Windows path limit of 260 characters may be exceeded. See **File and folder naming in LUSAS** for details.

- When the **OK** button is pressed LUSAS will carry out the influence surfaces analysis and run either LUSAS TLO / Autoloader vehicle load optimisation analysis software based on the country design code selected. The results from the LUSAS / Autoloader vehicle load optimisation analysis will be loaded into the LUSAS model in readiness for the model to be solved using the **Solve Now** button on the main toolbar.

When the model is solved for any selected onerous vehicle loading arranegments, the results from the analysis will be seen in the relevant Vehicle Load Optimisation entry in the Analyses Treeview.

Notes

- For models with numerous influence assignments of the same type, potentially unnecessary load generation can be avoided, and solution time can be saved, by ensuring that ‘Create loading patterns’ is not selected on the main VLO dialog, to allow for only the most onerous loading patterns to be created from the subsequent VLO onerous results table listings.
- The time taken to evaluate critical vehicle loading effects for a structure depends upon the design code in use; the types and numbers of vehicles to be considered; the number of lanes; whether any remaining areas exist after lane loading is positioned according to particular design codes; the number of spans; the mesh size; the values used for longitudinal and transverse load increments; whether one-way or two-way vehicle direction is specified; and the number of influence points to be evaluated. A good assessment of the time that will be required to evaluate a particular loading scheme for a large number of influence points can be achieved by initially timing how long it takes for just one influence point to be evaluated and factoring by the number of influences required.

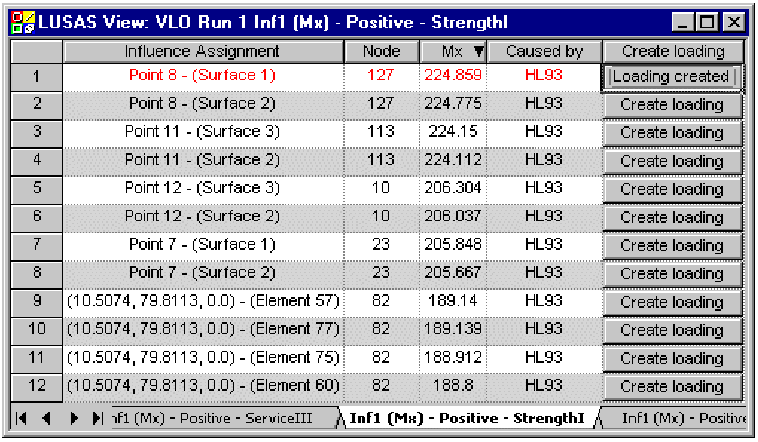
Defaults Buttons

- **Defaults** buttons that are present on the main and many of the associated Vehicle Load Optimisation dialogs will reset selected parameters and values to their default settings.

Onerous Effects Table

For design codes supported by LUSAS TLO only, and when a VLO analysis is run, a VLO onerous effects table can be optionally displayed which shows sorted results for all chosen influences at nodes that are visible, with the most onerous result shown in the first row of the table. For Direct Method Influences these values are due to traffic loading for the specified load effect of interest at specified locations on the model. That is, a single value direct from the LUSAS Traffic Load Optimisation facfility, without the need for a further static solution.

In the onerous effects table a results tab is present for each **<Influence attribute – Sign – Design Case>** group listed. This group is referred to as an 'Influence Design Loadcase'. Load patterns can be created on a case-by-case basis if not already created by means of the 'Create loading patterns' setting on the main VLO dialog.



LUSAS View: VLO Run 1 Inf1 (Mx) - Positive - StrengthI

	Influence Assignment	Node	Mx	Caused by	Create loading
1	Point 8 - (Surface 1)	127	224.859	HL93	Loading created
2	Point 8 - (Surface 2)	127	224.775	HL93	Create loading
3	Point 11 - (Surface 3)	113	224.15	HL93	Create loading
4	Point 11 - (Surface 2)	113	224.112	HL93	Create loading
5	Point 12 - (Surface 3)	10	206.304	HL93	Create loading
6	Point 12 - (Surface 2)	10	206.037	HL93	Create loading
7	Point 7 - (Surface 1)	23	205.848	HL93	Create loading
8	Point 7 - (Surface 2)	23	205.667	HL93	Create loading
9	(10.5074, 79.8113, 0.0) - (Element 57)	82	189.14	HL93	Create loading
10	(10.5074, 79.8113, 0.0) - (Element 77)	82	189.139	HL93	Create loading
11	(10.5074, 79.8113, 0.0) - (Element 75)	82	188.912	HL93	Create loading
12	(10.5074, 79.8113, 0.0) - (Element 60)	82	188.8	HL93	Create loading

nf1 (Mx) - Positive - ServiceIII | Inf1 (Mx) - Positive - StrengthI | Inf1 (Mx) - Positive

On each results page, the following column headings are present:

- **Influence Assignment** The location (coordinate and feature/object type) of the assigned DMI or Reciprocal Influence attribute.
- **Node** The node at which the assignment was made. Note that the **Find** context menu option for the cells in this column will, when clicked, highlight the corresponding node in the model View window by animating concentric shrinking squares. This easily identifies the assignment location.
- '**Chosen results effect**' For Direct Method Influence, this column is named after the result entity component selected in the DMI attribute (e.g. Mx, Fz, etc); the values in this column show the effect of the traffic loads at locations of interest. For reciprocal influences, the column is named according to the influence type (i.e. Shear/Moment/Reaction/Displacement).
- **Caused by** states the name of the loading type which produced the most onerous effect.
- **Create loading** Creates loading entries in the Attributes Treeview and a loadcase in the Analyses Treeview that cause the onerous effect listed.

If the option to 'Create loading patterns' was chosen on the main VLO dialog, Loading folders will additionally appear for each influence loadcase added to the Analyses Treeview, and the VLO results table will show 'Loading created' alongside all influence assignment entries. If 'Create loading patterns' was not selected on the main VLO dialog the most onerous loading effects can be created individually by using the 'Create Loading' button that appears instead of the 'Loading Created' one.

For the influences where no loading is produced by LUSAS TLO (because for the specified location there aren't any loads that can be placed to produce an adverse effect), the 'Chosen results effect' and 'Caused by' cells will be empty.

Viewing, filtering and saving onerous effects tables

VLO onerous effects results tables can be re-displayed for all influences at any time by selecting the **View table results...** context menu for a VLO Run entry in the Analyses Treeview. A table for a single particular influence can be created by double-clicking on the relevant onerous results table entry in the Analyses Treeview.

Results displayed in the tables can be filtered by selecting features or mesh objects of interest in the view window. So, for instance, with table results displayed in one view window, it is possible to select a line representing a span in a bridge in the model view window and have the results table only display the results for the nodes in the elements within that line. Similarly it is possible to select a surface representing a span of a slab and view only the results for the nodes in the elements within that surface. Use the Window > Tile facilities to see the onerous results table data and model view window side-by-side. Column data can be sorted by clicking on column headers.

Table results for an active table can be saved to a spreadsheet using the **File > Save as Microsoft Excel** menu item.

Usage

Once the onerous effects for each 'Influence Design Loadcase' have been obtained, and if the option to 'Create loading patterns' was not chosen on the main VLO dialog, the corresponding loading will need to be created on a case-by-case basis, if not already done. Whilst generated loading patterns can be visualised, results are available only after the VLO analysis has been solved. Once solved, the resulting loadcases should be combined with self weight and factored accordingly to create design combinations.

Worked examples

- The use of the LUSAS Traffic Load Optimisation facility is described in the worked example Bridge Slab Traffic Load Optimisation. See *Application Examples Manual (Bridge, Civil&Structural)*
- The use of the Autoloader Vehicle Load Optimisation facility is described in the worked example Grillage Load Optimisation.See *Application Examples Manual (Bridge, Civil&Structural)*
- The use of Direct Method Influence attributes is described in the worked example Vehicle Load Optimisation of a Box Beam Bridge. See *Application Examples Manual (Bridge, Civil&Structural)*

Australia AS5100-2:2004 Loading

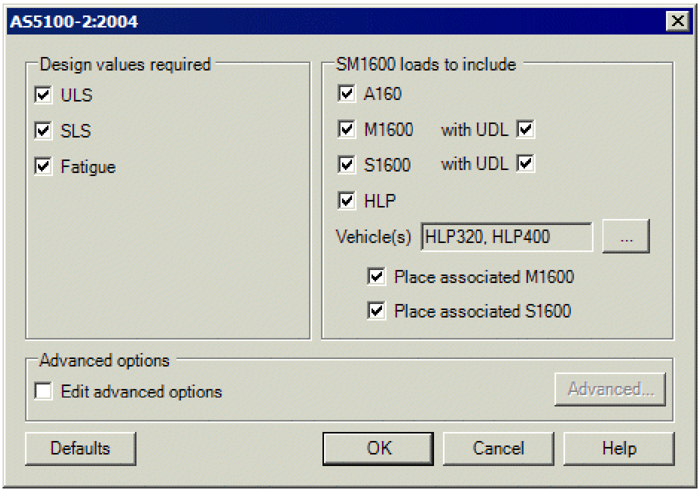
This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

When using the country "Australia" and Design code "AS5100-2: 2400" is selected on the main Vehicle Load Optimisation dialog, road traffic loading is generated with reference to: **AS5100-2: 2004** Australian Standard, Bridge design, Part 2: Design loads.

The scope of the vehicle load optimisation to AS5100-2:2004 is restricted to vertical highway traffic loading and global effects. Horizontal components, railway loads and pedestrian loads are not currently included, although fatigue loading (AS5100-2 clause 6.9) is included.

Australia AS5100-2:2004 Optional Settings

The Optional Code Settings dialog allows selection of the design values (ULS, SLS, Fatigue) and the load models (components from SM1600) required. Factors used in the calculations may be viewed and modified if required, by accessing the **Advanced** dialog.



The following AS5100-2 options are available:

Design values required

The design values available on the dialog refer to AS5100-2 clauses 6.9 and 6.10 as follows:

- **ULS** Factors are taken from the "Ultimate" column of Table 6.10(A) in clause 6.10. Since Heavy Load Platform Load has a different factor from other traffic loads at the ultimate limit state, it is necessary for the factors in Table 6.10(A) to be considered as part of the optimisation process.
- **SLS** Factors are taken from the "Serviceability" column of Table 6.10(A) in clause 6.10. Since these factors are all unity, they do not affect the optimisation process unless adjusted using the Advanced dialog.
- **Fatigue** As described in clause 6.9, fatigue design verifications require the consideration of only A160 and M1600 truck loads, factored by 0.7. Accordingly

S1600 and HLP loads will not be available if the Fatigue design case is the only case selected for analysis.

The dialog allows selection of one or more design values. The most onerous traffic loading pattern appropriate to each selected design value will be determined, with load factors included as appropriate.

Note

- In AS5100-2 clause 22, traffic loads are always considered together, as a "single block", when combined with loads from other sources. If the ULS, SLS and Fatigue loadcases generated by the traffic load optimiser are combined with other loads, it should be noted that the load factors described above are already included within the optimised traffic loadcases.

SM1600 Loads to Include

According to the design values selected, some or all of the SM1600 load models are available to be included in the analysis. SM1600 load models are deemed mutually exclusive, that is, a traffic load pattern may comprise several A160 loads in various lanes, or several M1600 loads or several S1600 loads, but not a mixture of different load types. As indicated in the section above, S1600 and HLP loads are typically only available when ULS or SLS design values are requested.

- ❑ **A160 axle load** This is a single axle of 2m width positioned centrally within the 3.2m lane width as described in AS5100-2 clause 6.2.2. A dynamic factor applies when considering ULS, SLS or Fatigue (clauses 6.7.1 and 6.9). Consistent with other traffic loads, the wheel loads are treated as discrete point loads rather than being defined with a contact area.
- ❑ **M1600 moving traffic load** This is formed from a uniformly distributed lane load spread over the full width of the lane and placed in adverse areas, and a "M1600 truck" vehicle (four tri-axle groups with one variable axle spacing) which superimposes on the UDL as set out in AS5100-2 clause 6.2.3. The M1600 truck is used without UDL for Fatigue design verifications; additionally an option is provided to switch off UDL for all design values if required. A dynamic factor applies when considering ULS, SLS or Fatigue (clauses 6.7.1 and 6.9). An "M1600 tri-axle group" vehicle is also considered, identifying when this vehicle is more onerous than the full M1600 moving load. A different dynamic factor applies for this configuration of the M1600. (see Table 6.7.2).
- ❑ **S1600 static traffic load** This is formed from a uniformly distributed lane load spread over the full width of lane and placed in adverse areas, and a "S1600 truck" vehicle (four tri-axle groups with one variable axle spacing) which superimposes on the UDL as set out in AS5100-2 clause 6.2.4. The dynamic allowance is zero, in line with Table 6.7.2, unless adjusted manually using the Advanced dialog. An option is provided to switch off UDL for all design values if required.

- **HLP** (Heavy Load Platform) Where specified by an authority a HLP320 or a HLP400 may be required, as per AS5100-2 clause 6.3. These vehicles always straddle two lanes and are placed near the centre of the lanes straddled; to account for errors in the positioning of actual vehicles, the most onerous location within 1.0m of the central position is identified. All other loads are excluded from the straddled lanes but M1600 or S1600, factored at 0.5, is placed in any unobstructed design lanes. See the note below on HLP lane restrictions where necessary. AS5100-7 clause A2.2.4 (vi) allows the central axle spacing to vary from the 1.8m standard, to a gap of between 6m and 15m, for continuous bridges. Within the traffic load optimiser this variable spacing has been conservatively included for single span bridges also. A dynamic factor applies when considering ULS, SLS or Fatigue (clauses 6.7.1 and 6.9); note that the dynamic factor for M1600 or S1600 in the unobstructed lanes has been interpreted by LUSAS to be the same as it would be without the HLP load present.
- **Vehicle(s)** Clicking the "..." button allows selection of one or more HLP vehicles. Where specified by an authority a HLP320 or a HLP400 may be required, as per AS5100-2 clause 6.3
- **Place associated M1600** This check box may be unchecked in order to obtain a result for HLP vehicles alone or HLP vehicles with S1600 only, however, the default is for associated M1600 to be included. In certain cases, it may be required to place associated M1600 loads in the absence of a selected HLP vehicle: see the note below on HLP lane restrictions.
- **Place associated S1600** This check box may be unchecked in order to obtain a result for HLP vehicles alone or HLP vehicles with M1600 only, however, the default is for associated S1600 to be included. In certain cases, it may be required to place associated S1600 loads in the absence of a selected HLP vehicle: see the note below on HLP lane restrictions.

The road carriageway is divided into standard design lanes (3.2m wide) according to AS5100-2 clause 6.5. This will result in a "remaining width" for any carriageways of width not an integer multiple of 3.2m. The lanes are positioned laterally on the bridge to produce the most adverse effects, with the remaining width being placed either side of the lanes or between any of the lanes so as to produce the most onerous arrangement.

Lane factors for A160, M1600 and S1600 load models are according to Table 6.6 or as set in the Advanced dialog, and are allocated so as to produce the most adverse load effect. Where HLP loading is applied according to AS5100-2 clause 6.3, lane factors do not seem to be applicable to the straddling HLP vehicle or to the accompanying M1600 or S1600, which is factored at 0.5. On this understanding, and with reference to AS5100-7 clause A2.2.6, lane factors are not used with HLP loading.

HLP lane restrictions

HLP vehicles in specific lanes. AS5100-2 clause 6.3 indicates that the two lanes which the HLP vehicle straddles may be specified by the authority. By default, the HLP will be placed in the most adverse position, using any possible lane arrangements as determined using clause 6.5. While not a strict implementation of clause 6.3, this is conservative and eliminates the need to identify the HLP lanes.

If it is required to restrict the placement of HLP to two specific lanes, two runs of the traffic load optimiser can be used. The first run would consider a notional carriageway only two lanes wide positioned as required for placement of the HLP vehicle only. The second run would use one or more carriageways for placement of the "accompanying" M1600 or S1600 only. Superposition of the results from the two runs using **combinations** will obtain the combined effects of the HLP and accompanying traffic loads.

Load models excluded from this dialog

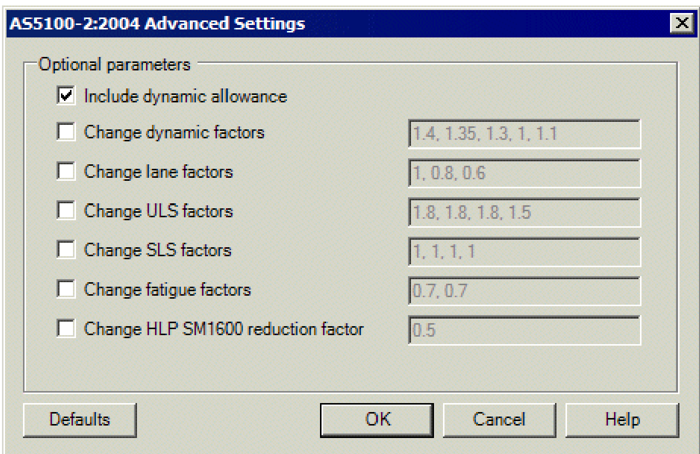
The W80 load of AS5100-2 clause 6.2.1 is a single heavy wheel applied anywhere on the carriageway. It is for local verifications only and therefore is deemed outside the remit of the traffic load optimiser, which is intended for the generation of traffic load patterns applicable to global bridge deck analyses.

Advanced options

- **Edit advanced AS5100-2 options** - See Australia AS5100-2 Advanced Settings

Australia AS5100-2:2004 Advanced Settings

This dialog is used to view and modify dynamic factors, lane factors, load factors, the HLP reduction factors and to request additional output.



Using the checkboxes factors may be entered manually to suit project requirements.

- **Include dynamic allowance** (Selected by default) Dynamic load allowances are required to be included for ULS, SLS and Fatigue design verifications according to AS5100-2 clause 6.7. These are incorporated by default but may be excluded by unticking this check box.
- **Change dynamic factors** Dynamic factors are listed for each load model in turn: A160, M1600 (triaxle group), M1600, S1600, HLP. Default values are derived from AS5100-2 Table 6.7.2 by adding unity to the "allowance" in the table.

- **Check lane factors** Lane factors are listed in descending lane rank: ALF_1, ALF_2, ALF_3 where the subscript numbers correspond to the numbering in the notes under AS5100-2 Table 6.6. ALF_3 is used for the third lane and all subsequent lanes. Default values are from Table 6.6.
- **Change ULS factors** ULS factors are listed for each load model in turn: A160, M1600, S1600, HLP. Default values are from AS5100-2 Table 6.10(A), “ultimate” column.
- **Change SLS factors** SLS factors are listed for each load model in turn: A160, M1600, S1600, HLP. Default values are from AS5100-2 Table 6.10(A), “serviceability” column.
- **Change fatigue factors** Fatigue factors are listed for each applicable load model in turn: A160, M1600. Default values are from AS5100-2 clause 6.9.
- **Change HLP SM1600 reduction factor** This factor is used to reduce accompanying M1600 and accompanying S1600 loads according to AS5100-2 clause 6.3. The default value, as per that clause, is 0.5.
- **Defaults** This button resets all values back to the default according to AS5100-2.

Australia AS5100-7:2004 (Austroads) Loading

This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

When using the country “Australia” and Design code “AS5100-7:2004 (Austroads)” is selected on the main Vehicle Load Optimisation dialog, road traffic loading is generated with reference to Australian Standard **AS5100.7-2004**, Bridge design, Part 7: Rating of existing bridges, Clause A2.2.

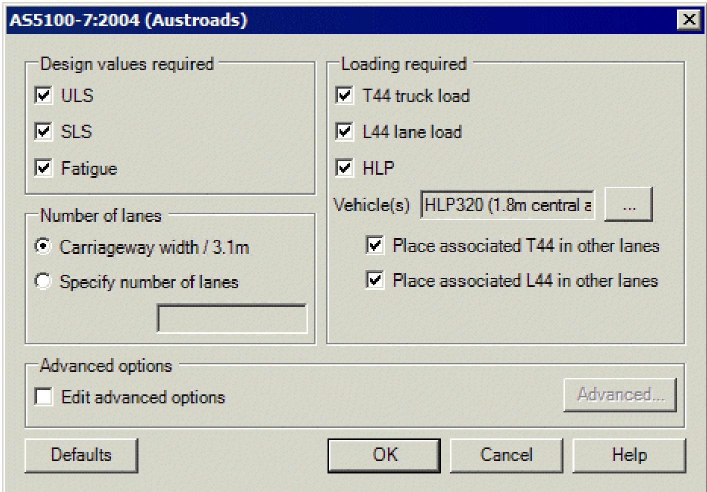
The 1992 Austroads Bridge Design Code is a former design code which is now used for assessment, and is reproduced in AS5100-7 Appendix A Clause A2.2. The loading consists of either L44 loading (a UDL with a concentrated load, or with two concentrated loads on continuous spans) or T44 loading (a Truck load). A heavy vehicle (HLP 320 or HLP400) can replace two lanes of standard loading, with a variable axle spacing for continuous spans.

The road carriageway is divided into standard design lanes (3m wide) according to AS5100-7 clause A2.2.5.1(a), the number of lanes being dependant on the carriageway width divided by 3.1m. Where more than one lane is loaded, multiple lane modification factors to clause A2.2.6 are applied to L44 and T44 loads. Loads are increased by dynamic load allowances to clause A2.2.10, and load factors are applied to all loads as per clause A2.2.9.

Note that the scope of the vehicle load optimisation to AS5100.7 is similar to other implemented codes of practice, that is, restricted to vertical highway traffic loading and global effects. Horizontal components, railway loads and pedestrian loads are not currently included.

Optional code settings dialog

The Optional Code Settings dialog allows selection of the design values (ULS, SLS, Fatigue) and the load models required. Factors used in the calculations may be viewed and modified if required, by accessing the Advanced dialog.



Design values required

The design values available on the dialog refer to AS5100-7 clauses A2.2.8 and A2.2.9 as follows:

- ❑ **ULS** Factors are taken from the Table A4 in clause A2.2.9. Since Heavy Load Platform Load has a different factor from other traffic loads at the ultimate limit state, it is necessary for the factors in Table A4 to be considered as part of the optimisation process.
- ❑ **SLS** Factors are taken from clause A2.2.9. Since these factors are all unity, they do not affect the optimisation process unless adjusted using the Advanced dialog.
- ❑ **Fatigue** As described in clause A2.2.8, fatigue design loads considered are T44 truck and L44 lane load. Accordingly HLP loads will not be available if the Fatigue design case is the only case selected for analysis. No load factors are specified in clauses A2.2.8 or A2.2.9 so unity is used; this may be adjusted using the Advanced dialog.

The dialog allows selection of one or more design values. The most onerous traffic loading pattern appropriate to each selected design value will be determined, with load factors included as appropriate.

Loading required

According to the design values selected, some or all of the load models are available to be included in the analysis. As indicated in the section above, HLP loads are only available when ULS or SLS design values are requested.

- **T44 truck load** This is a five-axled vehicle with a variable spacing between the rear two tandem axle groups, as set out in AS5100-7 Figure A1.
- **L44 lane load** This is formed from a uniformly distributed lane load spread over the 3m lane width and placed in adverse areas, and a concentrated load, also spread over the 3m lane width, which superimposes on the UDL as set out in AS5100-7 clause A2.2.3. The UDL intensity is suitable up to 150m loaded length, as per Figure A2; values for longer loaded lengths can be entered via the Advanced dialog.
- **HLP (Heavy Load Platform)** Where specified by the authority a HLP320 or HLP400 may be required, as per AS5100-7 clause A2.2.4. These vehicles always straddle two lanes and are placed near the centre of the lanes straddled; to account for errors in the positioning of actual vehicles, the most onerous location within 1.0m of the central position is identified. All other loads are excluded from the straddled lanes but T44 or L44, factored at 0.5, is placed in any unobstructed design lanes. See the note below on HLP lane restrictions where necessary.
- **Vehicle(s)** Clicking the "..." button allows selection of one or more HLP vehicles. HLP320 and HLP400 vehicles are described in AS5100-7 clause A2.2.4, formed of 16 axles in two groups of 8 axles and are of 3.6m and 4.5m overall width respectively.
- **Place associated T44 in other lanes / Place associated L44 in other lanes** These checkboxes control which loads are considered in lanes which are not obstructed by HLP loads.

The road carriageway is by default divided into standard design lanes (3m wide) according to AS5100-7 clause A2.2.5.1(a), the number of lanes being dependant on the carriageway width divided by 3.1m. This will result in a "remaining width" which is unloaded. The lanes are positioned laterally on the bridge to produce the most adverse effects, with the remaining width being placed either side of the lanes or between any of the lanes so as to produce the most onerous arrangement.

An alternative lane strategy to clause A2.2.5.1(b) can be achieved using the "Specify number of lanes" option. If this option is used to specify a single lane then that lane is positioned laterally at any location within the carriageway, for most onerous effect. If more than one lane is specified, the carriageway is divided evenly into that many lanes, with the 3m wide loading positioned up to 500mm from the centre of these lanes, so as to produce the most onerous effect.

Multiple lane modification factors for T44 and L44 load models are according to Table A2 or as set in the Advanced dialog, and are allocated so as to produce the most adverse load effect. Where HLP loading is applied to clause A2.2.5.2, the factor for associated loads of 0.5 is used instead of the multiple lane factors.

A dynamic load allowance applies when considering ULS, SLS or Fatigue (clause A2.2.10.2); this can be edited via the Advanced dialog.

HLP lane restrictions

HLP vehicles in specific lanes. AS5100-7 clause A2.2.5.2 indicates that the two lanes which the HLP vehicle straddles may be specified by the authority. By default, the HLP will be placed in the most adverse position, using any possible lane arrangement. While not a strict implementation of clause A2.2.5.2, this is conservative and eliminates the need to identify the HLP lanes altogether.

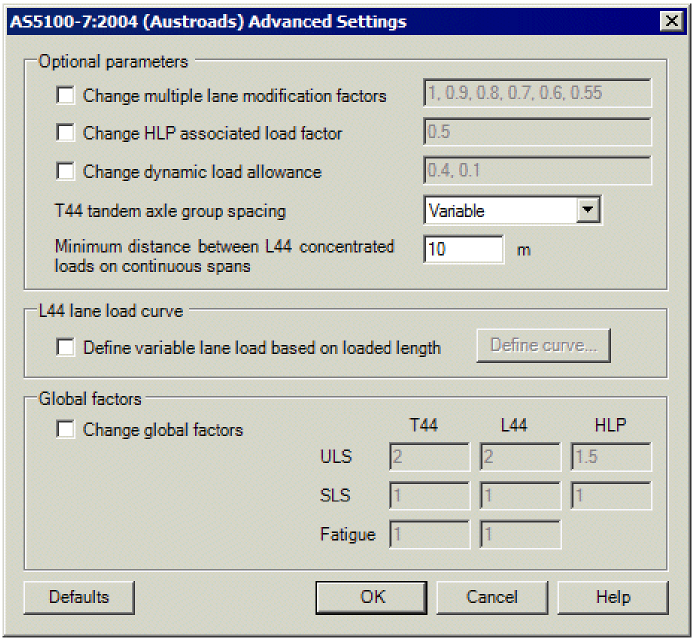
If it is required to restrict the placement of HLP to two specific lanes, two runs of the traffic load optimiser can be used. The first run would consider a notional carriageway only two lanes wide positioned as required for placement of the HLP vehicle only. The second run would use one or more carriageways for placement of the “accompanying” T44 or L44 only, with multiple lane modification factors revised to reflect the accompanying factor instead. Superposition of the results from the two runs may be used to obtain the combined effects of the HLP and accompanying traffic loads.

Load models excluded from this dialog

The W7 wheel load of AS5100-7 clause A2.2.7 is a single heavy wheel applied anywhere on the carriageway. It is for local verifications only and therefore is deemed outside the remit of the traffic load optimiser which is intended for the generation of traffic load patterns applicable to global bridge deck analyses.

Australia AS5100-7:2004 (Austroads) Advanced Settings

This dialog is used to view and modify multiple lane modification factors, a factor for HLP associated loads, dynamic factors, global factors and a number of additional settings.



Optional parameters

- **Change multiple lane modification factors** This option allows the modification factors for multiple lane bridges, from AS5100-7 clause A2.2.6, to be edited. The textbox has a comma separated list of six variables, applicable respectively to: one lane loaded, two lanes loaded, three lanes loaded, four lanes loaded, five lanes loaded, six or more lanes loaded.
- **Change HLP associated load factor** AS5100-7 Clause A2.2.5.2 requires T44 or L44 loads which accompany HLP loads to be half (i.e. 0.5) of their normal value. This option allows the modification of this factor. Note: this factor is considered to replace the multiple lane modification factors when HLP loading is present
- **Change dynamic load allowance** This option allows the modification of the dynamic load allowance from AS5100-7 Clause A2.2.10. The textbox has a comma separated list of two variables, applicable respectively to HLP loads and T44/L44 loads. Note the maximum value from Figure A4, i.e. 0.4, has conservatively been implemented as the default for T44/L44 loads.
- **T44 tandem axle group spacing** The T44 Truck from AS5100-7 Figure A1 has an axle spacing which varies from 3m to 8m; this is implemented by default. This option gives the opportunity to restrict which axle spacings are considered in order to give a faster optimisation process. Axle spacing options should only be restricted when engineering judgement deems this will give a sufficiently accurate result.
- **Minimum distance between L44 concentrated loads on continuous spans** AS5100-7 Clause A2.2.3 allows two concentrated loads which must be in separate spans. This is currently implemented by requiring a minimum distance between the two loads. The required minimum distance to keep them in separate spans, while not forcing them so far apart that they cannot be placed at the local peak adverse influence ordinates, varies depending on span lengths and their ratios to each other. If the defined minimum spacing is too small for a given bridge, Vehicle Load Optimisation may place both loads in the same span. This is incorrect but conservative. If the defined minimum spacing is too large for a given bridge, VLO will not be able to place both loads on the peak influences of their spans. This is incorrect and unconservative. For typical span ratios, a value of 2/3 of the maximum span is usually appropriate.

L44 Lane load curve

- **Define variable lane load based upon loaded length** AS5100-7 Figure A2 defines the intensity of the UDL up to and including a loaded length of 150m. This intensity can be modified, or values for longer loaded lengths defined, using this option.

Global factors

- **Change global factors** The default values provided to AS5100-7 Table A4 may optionally be edited. Note no load factors are specified in clauses A2.2.8 or A2.2.9 for Fatigue, so unity is used.

Defaults button is provided to reset all values back to those specified in AS5100-7.

Australia AS5100-7:2004 (Austroads) - Curve definition

The curve definition dialog allows the definition of a piecewise linear curve. The Traffic Load Optimiser carries out straight line interpolation between the entered values. For more accurate interpolation as many points as possible should be specified.

Canada CAN/CSA-S6-06 (Design) Optional Code Settings

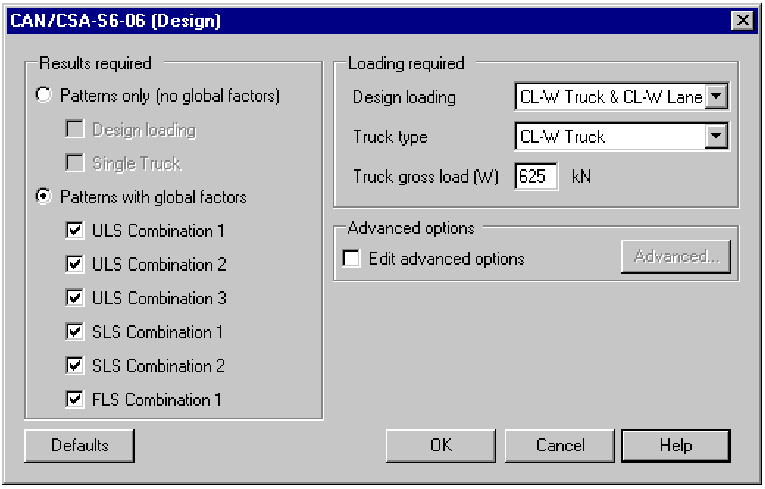
This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

When the country 'Canada' and Design code 'CAN/CSA-S6-06 (Design)', is selected on the main Vehicle Load Optimisation dialog, road traffic loading is generated with reference to CAN/CSA-S6-06 Section 3.

Scope

The scope of the vehicle load optimisation to CAN/CSA-S6-06 is restricted to vertical highway traffic loading and global effects. Horizontal components, railway loads and pedestrian loads are not currently included.

The road carriageway is divided into a number of equal width lanes as per CAN/CSA-S6-06 Table 3.4, which includes a check for the most onerous of 2 or 3 lanes for carriageway widths between 10.0m and 13.5m.



The Optional Code Settings dialog allows selection of the combinations and loads required. Factors used in the calculations may be viewed and modified if required, by accessing the Advanced Settings dialog.

Results required

Loading patterns are available either globally factored (i.e. with a Load Factor for a Limit State to CAN/CSA-S6-06 Table 3.1, or optionally edited in the Advanced Settings dialog) or unfactored as follows:

- ❑ **Patterns only (no global factors)**

  - **Design loading** - Loading to clause 3.8.4.1 (d) - truck load increased by the dynamic load allowance or the lane load, whichever produces the maximum load effect
  - **Single Truck** - Loading to clause 3.8.4.1 (c) - one truck only, placed at the centre of one travelled lane

- ❑ **Patterns with global factors**

  - ULS Combination 1 / ULS Combination 2 / ULS Combination 3 / SLS Combination 1 - as per Design loading pattern but with the relevant global factor from CAN/CSA-S6-06 Table 3.1
  - SLS Combination 2 / FLS Combination - as per Single Truck pattern but with the relevant global factor from CAN/CSA-S6-06 Table 3.1

The dialog allows selection of one or more combinations. The most onerous traffic loading pattern appropriate to each selected combination will be determined, with load factors included as appropriate.

Loading required

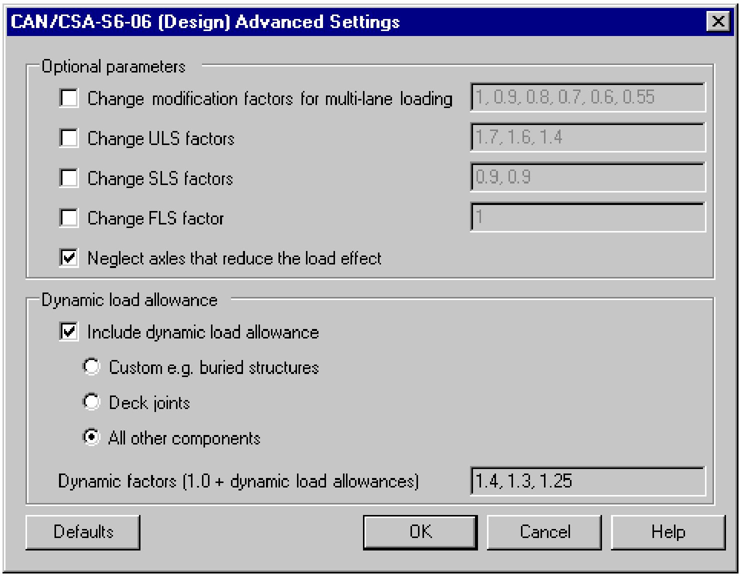
- ❑ **Design loading** - CL-W loads to include. For combinations other than Single Truck patterns, this gives the option to consider only Truck loading, only Lane loading, or both. If both are selected, the most onerous is output for each influence/combination - as per commentary on CAN/CSA-S6-06, clause C3.8.4.2, "Simultaneous loading of different design lanes by truck and lane load need not be considered."
- ❑ **Truck type**. This option allows the selection of the CL-W distribution (to CAN/CSA-S6-06 clause 3.8.3.2), a CL-W-ONT distribution (derived from the CL-625-ONT distribution to CAN/CSA-S6-06 Annex A3.4) or a BCL-W distribution (derived from the BCL-625 distribution to British Columbia Ministry of Transportation and Infrastructure - Bridge Standards and Procedures Manual Volume 1 Figure 3.2a).
- ❑ **Truck gross load (W)**. This setting allows the gross load of the selected Truck to be entered.

Modification factors for multi-lane loading are applied as per CAN/CSA-S6-06 Table 3.5, or optionally edited via the **Advanced Settings** dialog. The optimisation process takes account of these factors when calculating the most onerous load effect, such that every lane may not be loaded.

The dynamic load allowance to CAN/CSA-S6-06 clause 3.8.4.5.1 (or optionally edited via the Advanced Settings dialog) is applied to all Trucks, except for Single Truck loading to clause 3.8.4.1 (c) or when it forms part of lane loading to clause 3.8.3.3. The optimisation process takes account of the dynamic load allowance when calculating the most onerous load effect, such that axles may not be used even when they are adverse.

A **Defaults** button is provided to reset all values back to those specified in CAN/CSA-S6-06.

CAN/CSA-S6-06 (Design) Optional Code Settings (Advanced)



This dialog is used to view and modify modification factors for multi-lane loading, global factors and dynamic factors.

Optional parameters

- **Change modification factors for multi-lane loading** This option allows the modification of the factors from the values in CAN/CSA-S6-06 Table 3.5, which are implemented as default values. The textbox has a comma separated list of six variables, applicable respectively to: one lane loaded, two lanes loaded, three lanes loaded, four lanes loaded, five lanes loaded, six or more lanes loaded.
- **Change ULS factors** The default values provided to CAN/CSA-S6-06 Table 3.1 may optionally be edited. The textbox has a comma separated list of three variables, applicable respectively to: ULS Combination 1, ULS Combination 2, ULS Combination 3.

- **Change SLS factors** The default values provided to CAN/CSA-S6-06 Table 3.1 may optionally be edited. The textbox has a comma separated list of two variables, applicable respectively to: SLS Combination 1, SLS Combination 2.
- **Change FLS factor** The default value provided to CAN/CSA-S6-06 Table 3.1 may optionally be edited.
- **Neglect axles that reduce the load effect** As per CAN/CSA-S6-06 clause 3.8.4.1(a) Truck axles which reduce the load effect are neglected. If this option is deselected then all axles will be included, whether they are adverse or relieving, or even adverse but reducing the load effect(due to dynamic factoring).

Dynamic load allowance

- **Include dynamic load allowance** if deselected, cause the dynamic factors to all be set to 1.0.
- **Custom e.g. buried structures** if selected, causes the textbox to be user-editable.
- **Deck joints** if selected, sets the dynamic factor to 1.5 for one axle and 0.0 for more axles, as per CAN/CSA-S6-06 clause 3.8.4.5.3(a).
- **All other components** if selected,sets the dynamic factors as per CAN/CSA-S6-06 clause 3.8.4.5.3(b) to (d).
- **Dynamic factors** (implemented as 1.0 + dynamic load allowances) are displayed in the textbox and may be optionally edited. The textbox has a comma separated list of three variables, applicable respectively to: one axle used, two axles (or axles nos. 1 to 3) used, three axles (other than axles nos. 1 to 3) or more used.

Notes

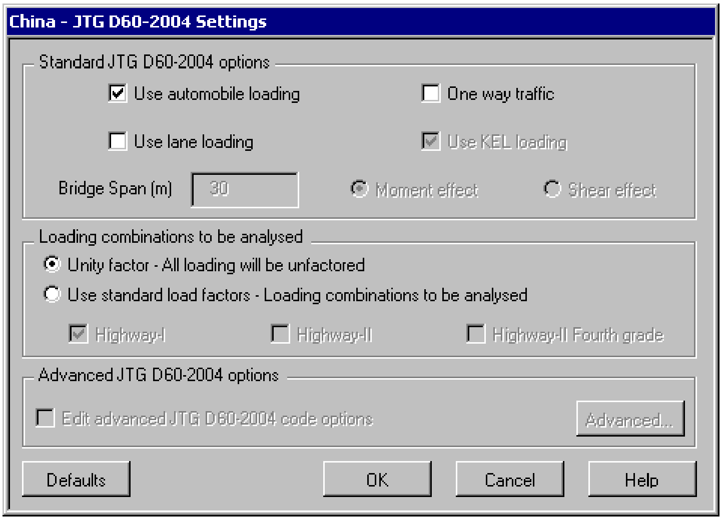
- CAN/CSA-S6-06 clause 3.8.4.1 specifically includes Dynamic Load Allowance for SLS1 and ULS1-3 while not mentioning it for FLS and SLS2. It is therefore interpreted that dynamic load allowance does not apply for FLS or SLS2, despite clause 3.8.4.5.1 stating it shall be applied "unless otherwise specified elsewhere in this code".

Default values

- The **Defaults** button is provided to reset all values back to those specified in CAN/CSA-S6-06.

China - JTG D60-2004 settings

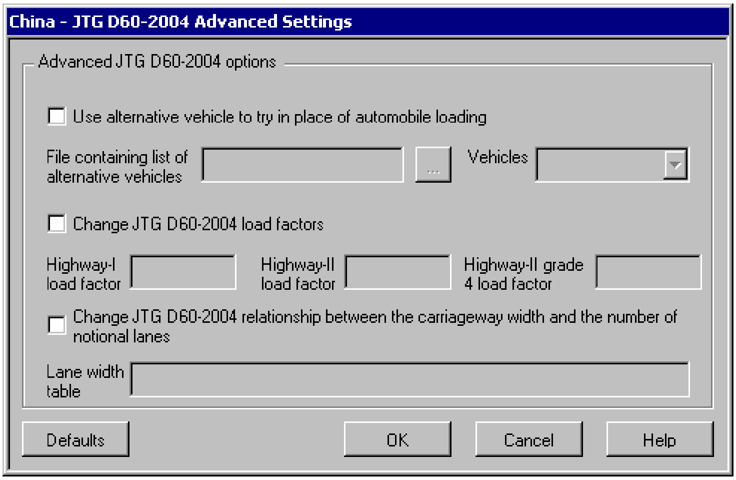
This design code loading is supported by the Autoloader Vehicle Load Optimisation software option.



The following JTG D60-2004 options are available for the selection:

- **Use automobile loading** – Choose this option to load the structure with the standard automobile arrangement as specified in JTG D60-2004 Table 4.3.1-2
- **Use lane loading** – This option will apply lane loading as specified in JTG D60-2004 clause 4 - 1).
- **Knife edge loading (KEL)** - Choose to also apply the knife edge load. Choose the effect required by the KEL as either moment or shear. The KEL will be factored by 1.2 for the shear effect option. The bridge span is required to calculate the intensity of the KEL.
- **Loading combinations to be analysed** can be either set to unity or the standard loadcases can be chosen from the list.
- **Edit advanced JTG D60-2004 code options** - further to the basic code options provided on this dialog, Autoloader allows more advanced changes to the code to be made. To access this functionality ensure that the use standard load factors is chosen and the check box next to the label is selected and click on the Advanced button. However, note that it is unlikely that these advanced options will be required for the majority of work.

China JTG D60-2004 Advanced Settings



The following JTG D60-2004 advanced options are available for selection:

- **Use Alternative vehicle** - defines a list of vehicles to be tried as an alternative to automobile loading. When loading a lane, Autoloader tries to place vehicles from this list within the lane. Each vehicle must be specified in the Chinese Autoloader vehicle library. By default this option is deselected.
- **Change JTG D60-2004 Partial Load factors** - To use this option select the check box next to the label and enter the new factors to be used. By default this option is deselected.
- **Change relationship between carriageway width and the number of notional lanes.** Each 3 values are taken as a lower limit, an upper limit and a number of notional lanes. After calculating the carriageway width, Autoloader works its way down the table, checking the calculated width against the values in the table. By default this option is deselected.

Eurocode Traffic Loading

Traffic loading on bridges to the Eurocodes is specified in two main documents:

- EN1991-2:2003 Eurocode 1: Actions on structures – Part 2: Traffic loads on bridges
- EN1990:2002 +A1:2005 Eurocode: Basis of Structural design

However, the Eurocodes allow a choice of safety related parameters and of certain country-specific data, known collectively as Nationally Determined Parameters (NDPs), which are published in National Annexes that accompany each Eurocode part.

National Annexes supported

Selecting one of the available National Annexes on the main Vehicle Load Optimisation dialog sets default values for NDPs (α_Q, α_q, ψ) and offers traffic load options (e.g. traffic classes, Load Model 3 special vehicles and complementary load models) appropriate to that National Annex. Values may, in any case, be modified to meet specific requirements through the options on the various dialogs.

The following National Annexes are currently supported by LUSAS Traffic Load Optimisation software by first picking the Europe country option:

- **Ireland**
- **Italy**
- **Poland**
- **Recommended Values**
- **Sweden**
- **UK**

The Recommended Values option sets defaults and offers traffic options based only on recommendations in the main Eurocode documents. This option, with or without modification of values, may be of particular use for countries where a National Annex is not yet published or not yet supported.

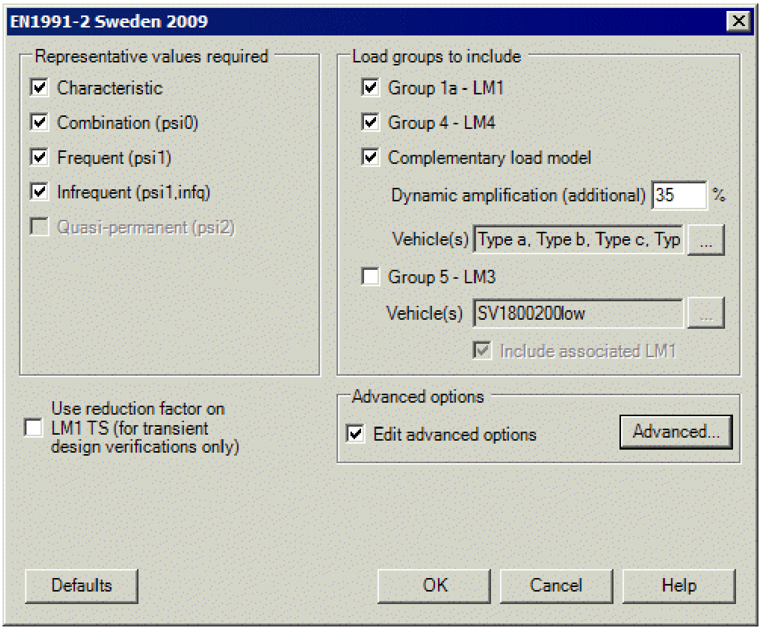
Notes

- The scope of the vehicle load optimisation to Eurocodes is the same as for previously implemented codes of practice, that is, restricted to vertical highway traffic loading and global effects. Railway loads and fatigue loading are not currently included.

EN1991-2 Representative Optional Code Settings

This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

This is a representative dialog for all countries supported. The options available on the dialog depend upon the design code and National Annex supported.



The Optional Code Settings dialog allows defining of the Representative Values that are required and Load Groups that are to be included. Factors used in the calculations may be viewed and modified by accessing the **Advanced Settings** dialog.

Representative values required

The various levels of Representative Values in the Eurocodes are represented on this dialog as follows:

- **Characteristic values** See EN1990:2002 clause 1.5.3.14. Characteristic traffic actions are defined by Table 4.4a in EN1991-2:2003. This includes Group 1a (LM1 with no ψ value used) and Group 5 (generally LM3 with no ψ value together with LM1 with ψ_1 – but depends on National Annex). Characteristic values are used in for the leading variable action ($Q_{k,1}$) in ULS design checks (EN1990 equation 6.9a) and Irreversible SLS checks (equation 6.14a).
- **Combination values** Combination traffic actions are defined by use of ψ_0 from EN1990 table A2.1 (See EN1990 clause 1.5.3.16). It should be noted that ψ_0 for Group 5 is zero, therefore the Optional Code Settings dialog identifies Group 1a (LM1) loading alone as appropriate for the Combination case. Combination values are used for the accompanying variable action ($Q_{k,i}$) in ULS design checks (EN1990 equation 6.9a) and Irreversible SLS checks (equation 6.14a).
- **Frequent values** Frequent traffic actions are defined by Table 4.4b in EN1991-2 or by use of ψ_1 from EN1990 table A2.1 (See EN1990 clause 1.5.3.17) – these two sources are in harmony. It should be noted that ψ_1 for Group 5 is zero, therefore the Optional Code Settings dialog identifies Group 1a (LM1) loading alone as appropriate

for the Frequent case. Frequent values are used for the leading variable action ($Q_{k,1}$) in reversible SLS checks (equation 6.15a).

- **Infrequent values** Infrequent traffic actions are defined by use of $\psi_{1,infq}$ from EN1990 table A2.1 Note 2; which includes Group 1a (LM1) and Group 4 (LM4).
- **Quasi-permanent values** Defined by use of ψ_2 from EN1990 Table A2.1 (See EN1990 clause 1.5.3.18). In all the National Annexes currently implemented and in the Recommended Values settings ψ_2 is zero for all traffic actions, therefore Quasi-permanent cases cannot be defined in the Optional Code Settings dialog.

The dialog allows selection of the Representative Values for which the most onerous effect will be calculated.

Notes

- EN1991-2 does not refer to Combination values specifically; presumably the assumption is that traffic will never be an accompanying action but always a leading action. However the combination values are defined adequately using EN1990 Table A2.1 ψ_0 values and are therefore available should they be required.

Load Groups to include

According to the Representative Values selected, one or more Load groups are available to be included in the analysis. As indicated in the section above, Group 5 loads are typically only available when the Characteristic Values are being sought. The dialog includes only the Load Groups from EN1991-2 Table 4.4a and 4.4b that are relevant:

- **Group 1** comprises Load Model 1 (LM1) tandem system and uniformly distributed loads with the appropriate α, ψ and transient factors where appropriate. Note that for global analysis the tandem system is placed on the centreline of the lane (EN1991-2 clause 4.3.2(1)(a)), however the simplified rules in 4.3.2(6) are not used. Complete tandem systems are used (EN1991-2 clause 4.3.2(1)(a)) and the most onerous length of lane for application of the uniformly distributed load is determined for each lane in turn by integration of the influence surface across the lane width. Footway, cycle track and horizontal loads are excluded from the scope of the optimisation facility.
- **Group 4** comprises Load Model 4, which represents Crowd loading - see EN1991-2 clause 4.3.5. Crowd loading is applied in the carriageway area; footway and cycle track loads are excluded from the scope of the optimisation facility.
- **Complementary load model** Options associated with complementary load models will be available where such a load model is specified in the selected National Annex, according to the Representative Values for which that load model applies.
- **Group 5** is relevant only for Characteristic Values and comprises LM3 (special vehicles) combined with LM1 reduced according to rules given in the National Annex, or – in the case of the Recommended Values option – rules given in the informative Annex A to EN1991-2.

- **Vehicle(s)** Clicking the ... button allows selection of one or more special vehicles as defined in the selected National Annex.
- **Include associated LM1** This check box may be unchecked in order to obtain a result for LM3 special vehicles alone, however, the default is for LM1 to be included.

For all load groups, the lane division is according to EN1991-2 Table 4.1, with the remaining area (EN1991-2 clause 4.2.5(2)) being placed either side of the lanes or between any of the lanes in the carriageway area to produce the most onerous arrangement. Likewise, lane ranking is so as to produce the most onerous effect for the influence under consideration (as EN1991-2 clause 4.2.4(2)).

Reduction factor for transient design verifications

EN1991-2 clause 4.5.3(2) may be invoked with this check box if required.

Traffic Class for LM1

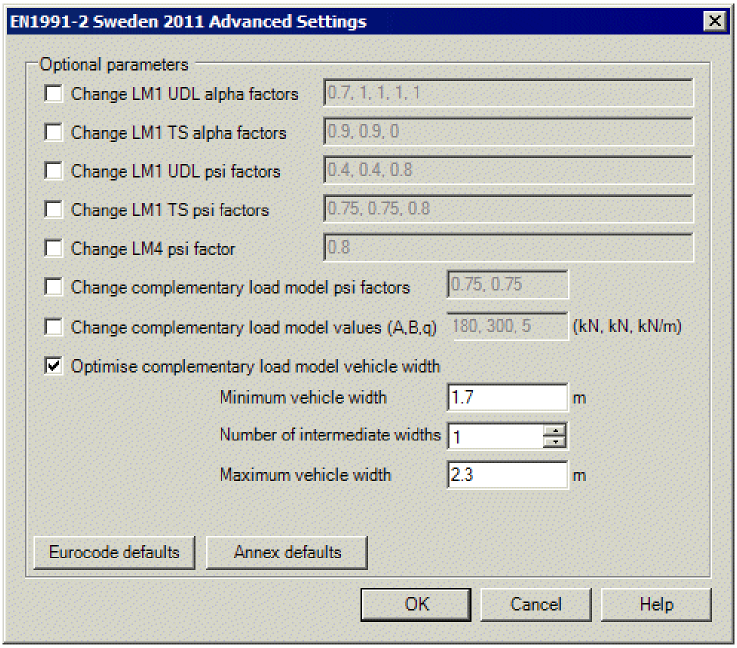
As per EN1991-2 clause 4.3.2 (3) note 2, some National Annexes require a traffic class to be specified for the selection of αQ and αq factors. Where this is the case, the “Traffic Class” box appears on this dialog: for National Annexes where no traffic classes are specific, and for the Recommended Values option, the “Traffic Class” box does not appear on the dialog. See the notes for the relevant National Annex for more information.

Load Groups excluded from appearing on this dialog

- **Group 1b** comprises only Load Model 2 which is for local verifications (“short structural members” to EN1991-2 clause 4.3.1(2)(b)) and so is excluded from the scope of the traffic load optimiser and does not appear on this dialog.
- **Group 2** is relevant only for the Characteristic Values and comprises LM1 (with smaller ψ factors compared to Group 1a) together with horizontal forces. Since horizontal forces are excluded from the scope of the traffic load optimisation software, Group 2 cannot dominate Group 1a and accordingly does not appear on this dialog. Users who wish to combine traffic load patterns with horizontal forces determined outside of the optimisation facility can obtain Characteristic Group 2 results by using Frequent Group 1a, which uses the same factors.
- **Group 3** comprises footway and cycle track loads only, which are excluded from the scope of the optimiser and so Group 3 does not appear on this dialog.

EN1991-2 Optional Code Settings Advanced

A representative dialog for all countries supported is shown. Options available depend upon the design code and National Annex supported.



The Optional Code Settings Advanced dialog is used to view and modify α_Q, α_q and ψ factors or parameters associated with national complementary load models (if applicable) and to request additional output.

Optional parameters

- Adjustment factors (α) for Load Model 1 UDL are in the format α_{q1}, α_{q2}, α_{q3}, α_{qn}, α_{qr}.
- Adjustment factors for Load Model 1 Tandem System are in the format α_{Q1}, α_{Q2}, α_{Q3}.

The numerical subscripts in the above denote lane rank, starting with lane 1 (see EN1991-2 clause 4.2.4(4)).

- Multi-component (ψ) factors for Load Model 1 UDL and Tandem Systems are in the format: ψ_0, ψ_1, $\psi_{1,infq}$
- Multi-component (ψ) factors for Load Model 4 are in the format: $\psi_{1,infq}$

Additional options

For countries where the National Annex specifies a complementary load model under EN1991-2 clause 4.2.1(1) Note 2 (such as Sweden), additional options such as axle loads and multi-component factors may be given in this dialog.

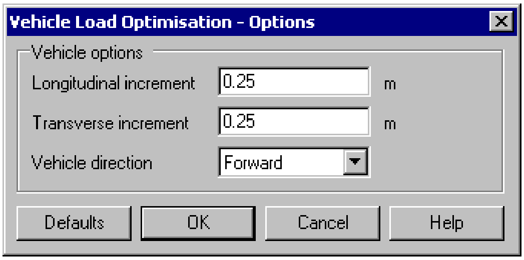
For Sweden, the complementary load model vehicle width can be specified in terms of minimum and maximum width, and the number of intermediate widths to be considered.

The **Eurocode defaults** and the **Annex defaults** buttons permit the viewing of Recommended Values from EN1991-2 or the values stated in the National Annex selected on the main dialog (apart from when the “Recommended Values” option was selected from the main dialog). Using the checkboxes factors may also be entered manually to suit project requirements.In some cases values may be modified to suit national requirements until a particular National Annex is fully implemented and available for those countries not currently supported.

Design Code-based Special Vehicle Selection (All LUSAS TLO Supported countries)

By clicking the ... button on the Optional Code Settings dialog for a VLO design code, a Special Vehicle Selection dialog is displayed. This allows the check-box selection of one or more special vehicles appropriate to the design code or, in the case of the Eurocode, a respective National Annex, for inclusion / exclusion from a VLO assessment.

Optional Loading Parameters (all LUSAS TLO supported countries)



The optional loading parameters dialog allows longitudinal and transverse increments to be set which will determine the accuracy with which the most onerous load locations are calculated. A smaller increment will result in higher accuracy but with an extended processing time.

Vehicle direction (for non-symmetric special vehicles) may be given as **Forward, Reverse** or **Both**. The default is **Both**. Symmetric vehicles are not affected by this setting (and therefore the analysis is run only once for symmetric vehicles).

Notes on implemented National Annexes

Notes relating to the implementation of the National Annexes are provided for the following European EN1991-2 design code options:

- **Ireland**
- **Italy**

- [ ] **Poland**
- [ ] **Recommended Values**
- [ ] **Sweden**
- [ ] **UK**

EN1991-2 Optional Code Settings - Ireland

This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

Selecting **EN1991-2 Ireland 2010** on the main Vehicle Load Optimisation dialog sets default values for NDPs and offers traffic load options according to Irish National Annexes published under the authority of the NSAI. Traffic loading on bridges to the Eurocodes is specified in two main documents:

- **NA to IS EN1991-2:2003 (effective from 9 September 2009)**
- **NA to IS EN 1990:2002+A1:2005 (effective from 29 March 2010)**

Load model 1

Adjustment factors for Load Model 1 (α_Q, α_q) are taken from NA to IS EN1991-2 Table NA.1. According to EN1991-2 clause 4.3.2 (3) Note 2, adjustment factors may correspond to classes of traffic. However, the adjustment factors in Table NA.1 are not dependent upon a selected traffic class and accordingly no traffic class options are offered on the dialog.

Load model 3

- Load Model 3 special vehicles (three SV and four SOV model vehicles) are described in the NA to IS EN1991-2 Fig NA.1 to NA.3 inclusive. One or more vehicles may be selected for use in the calculation of the most onerous load pattern. The selected vehicles are considered one at a time, being placed, together with associated Load Model 1 (switched on by default) according to the rules set out in clause NA.2.16.3.
- Each SV has a central axle spacing that varies (3 possible values). All such axle spacings are considered in calculation of the most onerous load effect. Each SOV has an axle spacing that varies continuously from 1.5 to 40m. By default, this axle spacing is calculated to an accuracy set in the Optional Loading Parameters dialog.
- Dynamic amplification factors (Table NA.2) are included in the calculation of the most onerous Group 5 load pattern and the wheel loads are accordingly factored within the vehicle (discrete point) loading attributes generated when the optimisation process is complete.
- When Group 5 is included, the SV80 is selected as a default. This vehicle is intended to model the effects of typical abnormal vehicles with a maximum gross weight of 80 tonnes and a maximum basic axle load of 12.5 tonnes (NA.2.16.1.1). Project

requirements would dictate if this is the appropriate vehicle and the vehicle selection may need to be adjusted or expanded.

Complementary load model

According to EN1991-2 clause 4.2.1(1) Note 2, a complementary load model may be specified in the National Annex. No such complementary load model is described in the Irish National Annex.

Psi factors

Multi-component (ψ) factors are taken from NA to IS EN1990 Table NA.7. Infrequent values are not required (NA.2.3.3.1) and quasi-static values are not calculated since ψ_2 is given as zero for all traffic loads (Table NA.7). Group 4 loading is not included in the calculation of Frequent values (Note (2) to Table NA.7).

Restrictions on use

The traffic load models in EN1991-2 are applicable for bridges with loaded lengths less than 200m (clause 4.1(1)). In general the use of Load Model 1 is conservative for loaded lengths over 200m (4.1(1) note 1). The Irish National Annex states that Load Model 1 may be used for loaded lengths up to 1500m (NA to IS EN1991-2 clause NA2.6) and no information on load models appropriate beyond that length is given. Calculation of the most onerous load pattern will proceed regardless of the loaded length and therefore patterns generated may be inappropriate for very long loaded lengths.

EN1991-2 Optional Code Settings - Italy

This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

Selecting **EN1991-2 Italy 2007** on the main Vehicle Load Optimisation dialog sets default values for NDPs and offers traffic load options according to Italian National Annexes published by UNI:

- **UNI-EN-1991 – 2 – Eurocodice 1 – Azioni sulle strutture – Parte 2 – Carichi da traffico sui ponti: Appendice nazionale (27 luglio 2007)**
- **UNI-EN-1990 – Criteri generali di progettazione strutturale – Appendice A2 – Applicazioni ai ponti: Appendice nazionale (27 luglio 2007)**

Load model 1

Adjustment factors for Load Model 1 (α_Q, α_q) correspond to traffic class is view of EN1991-2 clause 4.3.2 (3) note 2. Selecting traffic class 1 or class 2 on the dialog results in adjustment factors of 1.0 or 0.8 respectively in accordance with UNI-EN1991-2.

Load model 3

In relation to EN1991-2 clause 4.3.4(1), the Italian National Annex states “When significant, use the special vehicles and rules for application provided in Annex A (informative)”. Accordingly, Load Model 3 special vehicles are taken from EN1991-2 Annex A (informative) Tables A1 and A2. The notes and restrictions for this load model described under “Recommended Values” also apply to EN1991-2 Italy.

Complementary load model

According to EN1991-2 clause 4.2.1(1) Note 2, a complementary load model may be specified in the National Annex. No such complementary load model is described in the Italian National Annex.

Psi factors

Multi-component (ψ) factors are taken from EN1990 Table A2.1 since the Italian National Annex states that the recommended values should be adopted. Infrequent values are not required (A2.2.2(1) note) and quasi-static values are not calculated since ψ2 is given as zero for all traffic loads (EN1990 Table A2). Group 4 loading is not included in the calculation of Frequent values for the reason described under “Recommended Values”.

Restrictions on use

The traffic load models in EN1991-2 are applicable for bridges with loaded lengths less than 200m (clause 4.1(1)). In general the use of Load Model 1 is safe-sided for loaded lengths over 200m (4.1(1) note 1). The Italian National Annex defines a load model which is less conservative than Load Model 1 for structures with loaded lengths > 300m. This alternative Load model is not implemented. Calculation of the most onerous load pattern will proceed using the selected EN1991-2 load models regardless of the loaded length and therefore patterns generated may be over-conservative for very long loaded lengths.

EN1991-2 Optional Code Settings - Poland

This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

Selecting **EN1991-2 Poland 2008** on the main Vehicle Load Optimisation dialog sets default values for NDPs and offers traffic load options according to recommendations in EN1991-2 and EN1990, and offers Load Model 3 Special Vehicles from the informative Annex A to EN1991-2. This is because PN-EN 1991-2:2007 and PN-EN 1990:2004/A1:2008 published by PKN state that European Norms have the status of Polish Norms, with no modification from the English version of the European Norms.

All notes on **Eurocode EN1991-2 Recommended Values** are therefore also applicable to “EN1991-2 Poland 2008”.

EN1991-2 Optional Code Settings - Recommended Values

This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

Selecting **EN1991-2 Recommended Values 2009** on the main Vehicle Load Optimisation dialog sets default values for NDPs according to recommendations in EN1991-2 and EN1990, and offers traffic load options from the informative Annex A to EN1990, as per the CEN documents published by the British Standards Institute.

- **BS EN 1991-2:2003 Eurocode 1: Actions on structures — Part 2: Traffic loads on bridges, Incorporating Corrigendum No. 1 (published 15 December 2004)**
- **BS EN 1990:2002+A1:2005 Eurocode — Basis of structural design, Incorporating corrigendum (published 30 June 2009)**

The default values may be modified as necessary making the "Recommended Values" option useful for countries where a National Annex is not yet published or is not yet supported.

Load model 1

Adjustment factors for Load Model 1 (α_Q, α_q) are taken as 1.0, based on EN1991-2 clause 4.3.2 (3) Note 1. With respect to Note 2; no traffic class options are offered on the dialog but the adjustment factors may be modified in the optional code settings "advanced" dialog.

Load model 3

- Load Model 3 special vehicles are taken from EN1991-2 Annex A (informative) Tables A1 and A2. Each table lists 17 vehicles, however a conflict in the last lines of the two tables means that effectively 18 vehicles are described and accordingly there are 18 vehicles of fixed axle spacing listed in the dialog. One or more vehicles may be selected for use in the calculation of the most onerous load pattern. The selected vehicles are considered one at a time, being placed together with associated Load Model 1 (switched on by default) according to the rules set out in clause A.3 considering low speed transit only (clauses A.3(5) and A.3(7) are not applied).
- When Group 5 is included, the SV1800200 is selected as a default. This vehicle has a gross weight of 180 tonnes and an axle load of 20 tonnes, and is selected only as an example – on the basis that the heaviest vehicles in the Annex are exceptional and the least heavy are covered by the effects of Load model 1 (clause A.2 (2), note 2). Project requirements typically dictate the appropriate vehicle and so the vehicle selection may need to be modified.
- Dynamic amplification (clause A.3(5)) is not used for low speed transit and so is not included in the calculation of the most onerous Group 5 load pattern.

Psi factors

Multi-component (ψ) factors are taken from EN1990 Table A2.1. Infrequent values are not currently implemented (note 2 under table A2.1 is not applied) and quasi-static values are not calculated since ψ_2 is given as zero for all traffic loads (Table A2.1). In EN1990 Table A2.1, the frequent value of Group 4 loading is indicated with ψ_1 factor. However in EN1991-2 Table 4.4a and 4.4b, Group 4 loading is not required in the calculation of Frequent values. This conflict has been resolved by excluding Group 4 from the calculation of frequent values.

Restrictions on use

The traffic load models in EN1991-2 are applicable for bridges with loaded lengths less than 200m (clause 4.1(1)). In general the use of Load Model 1 is conservative for loaded lengths over 200m (4.1(1) Note 1). Calculation of the most onerous load pattern will proceed regardless of the loaded length, and therefore patterns generated may be inappropriate for very long loaded lengths.

EN1991-2 Optional Code Settings - Sweden

EN1991-2 Sweden 2009 and EN1991-2 Sweden 2011 codes are supported by the LUSAS Traffic Load Optimisation software option.

EN1991-2 Sweden 2009

Selecting **EN1991-2 Sweden 2009** on the main Vehicle Load Optimisation dialog sets default values for NDPs and offers traffic load options according to Swedish standards:

- **Specifikation SIS/PAS NA, EN 1991-2:2003 Swedish National Annex NA to Eurocode EN 1991-2:2003 – Traffic loads on bridges, Utgåva 1 (First Edition), Publicerad: maj 2007**
- **VV2009:19 Updated rules for use of Eurocodes on highway projects; Utkom från trycket den 26 juni 2009**
- **VV2009:27 TK Bro; Datum 2009-07-01**

For the purpose of traffic loading optimisation, within the scope described above, the Swedish National Annex to EN1991-2, published by SIS in May 2007, is considered identical to VV2009:19.

Load model 1

Adjustment factors for Load Model 1 (αQ, αq) are taken from SIS/PAS NA to EN 1991-2:2003 clause 4.3.2(3); the same values are given in VVFS 2009:19 Chapter 6, clause 4, Tabell 7.1. According to EN1991-2 clause 4.3.2 (3) note 2, adjustment factors may correspond to classes of traffic. However, the adjustment factors in the Swedish documents are not dependent upon a selected traffic class and accordingly no traffic class options are offered on the dialog.

Load model 3

Load Model 3 special vehicles. TK Bro VV2009:27 clause B.3.4.1.3(e) states that other load models do not apply. However SIS/PAS NA to EN 1991-2 clause 4.3.4(1) allows for the client to specify values for the individual project; VV2009:19 Chapter 6 clause 2 says the same. For flexibility, Load Model 3 Special Vehicles from EN1991-2 Annex A (informative) Tables A1 and A2 are available (although Group 5 is switched off by default). The notes and restrictions for this load model described under **Recommended Values** also apply to EN1991-2 Sweden.

Complementary load model

According to EN1991-2 clause 4.2.1(1) Note 2, a complementary load model may be specified in the National Annex. SIS/PAS NA to EN1991-2 describes a Swedish complementary load model; TK Bro VV2009:27 clause B.3.4.1.3(d) refers to VV2004:43 (superseded by VV2009:19) and VV2009:19 Chapter 6, clause 3 (page 13) repeats the information from the SIS/PAS National Annex.

- The Swedish complementary load model consists of 12 vehicles (type a to type l inclusive). One or more vehicles may be selected for use in the calculation of the most onerous load pattern. The selected vehicle types are considered one type at a time. A single vehicle is placed in the most onerous lane (with lane factor 1.0) with another vehicle of the same type placed in the second most onerous lane (with lane factor 0.8). Uniformly distributed load (q) is placed in adverse areas of lanes 1 and 2, if appropriate, and also in adverse areas of other lanes.
- A number of the complementary load model vehicles have an axle spacing which varies continuously from a stated minimum value, with no set maximum value. By default, this axle spacing is calculated to an accuracy set in the **Optional Loading Parameters** dialog. All complementary load vehicles have axle width that may vary between 1.7 and 2.3m (measured to the centre of action of each wheel load). Minimum and maximum widths can be tested for a specified number of intermediate widths. These widths can be modified in the **Optional Code Settings Advanced** dialog.
- A dynamic amplification factor is applied to the vehicles (not the uniformly distributed load, q) in the calculation of the most onerous Swedish complementary load model pattern and the wheel loads are accordingly factored within the vehicle (discrete point) loading attributes generated when the optimisation process is complete. The dynamic factor entered on the dialog should be calculated from the equation in SIS/PAS NA or VV2009:19 Chapter 6, clause 3. The default value of 35% reflects the maximum allowable value.
- Using the **Optional Code Settings Advanced** dialog, it is possible to view and modify the values used for A, B and q in the complementary load model.
- When the Swedish Complementary Load model is included, all 12 vehicles are selected by default. This means that the most onerous of the 12 will be identified so

that the structural element under consideration can be “designed for the type vehicle that causes the most unfavourable influence” (EN1991-2 clause 4.2.1(1) Note 2). The number of vehicles selected for the optimisation may be reduced using the dialog provided, as appropriate to project requirements.

- It is noted that the Swedish complementary load model vehicles are identical to the “classification loads” of VV2009:61 Clause 2.3.2.2.1 (and Annex 2). However, for classification, VV2009:61 Clause 1.1.5.3 requires the engineer to calculate the maximum values of load magnitudes “A” and “B” that can be carried (also referring to VV2009:62 (MB803) for exceptional loads). Such a calculation is not automated, although values of A and B can be modified as described above.

Psi factors

Multi-component (ψ) factors are generally taken from EN1990 Table A2.1 since the note under VVFS 2009:19 Chapter 7, clause 5, Tabell A2.(S) states that “at least the recommended levels apply”. Infrequent values are not required (SIS/PAS NA, EN 1991-2:2003 clause 2.2(2)) and quasi-static values are not calculated since ψ_2 is given as zero for all traffic loads (EN1990 Table A2). Group 4 loading is not included in the calculation of Frequent values for the reason described under “Recommended Values”.

Multi-component (ψ) factors for the Swedish Complementary load model are taken from TK Bro VV2009:27 Clause B.2.1.2.2. Based on the values given (ψ_0 and ψ_1 but ψ_2 =0), the complementary load model is included in the calculation of characteristic, combination and frequent values, but quasi-static values are not required.

Obtaining most onerous load patterns

The most onerous load patterns returned to the model after the optimisation process are based on a comparison of the Swedish complementary load model (using the selected type vehicles) and any other selected Eurocode load models (Group 1a, Group 4, Group 5). If it is desirable to view the most onerous of each load group, the check box on the Optional Code Settings Advanced dialog should be used.

Restrictions on use

The traffic load models in EN1991-2 are applicable for bridges with loaded lengths less than 200m (clause 4.1(1)). In general the use of Load Model 1 is conservative for loaded lengths over 200m (4.1(1) note 1). In TK Bro VV2009:27 clause B.3.4.1.3.(b) a load model for bridges of span >200m is given, However, this alternative Load model is not implemented. Calculation of the most onerous load pattern will proceed using the EN1991-2 load models and the Swedish Complementary load model, as selected, regardless of the loaded length and therefore patterns generated may be inappropriate for very long loaded lengths.

EN1991-2 Sweden 2011

Selecting **EN1991-2 Sweden 2011** sets default values for NDPs and offers traffic load options according to Swedish standard:

- **TRVFS2011:12 Updated rules for use of Eurocodes on highway projects (Trafikverket, Oct 2011))**

TRVFS2011:12 supersedes VV2009:19 for the implementation of highway traffic loading to the Eurocodes in Sweden. However, in fact the implementation is as described in "EN1991-2 Sweden 2009" above, except for the following modifications:

1. The Swedish complementary load model in TRVFS2011:12 Appendix 3 consists of 14 vehicles (type a to type n inclusive). Vehicle types a to l are as VV2009:19; vehicles type m and n, with fixed axle spacings, have been added.
2. The dynamic amplification factor in TRVFS2011:12 Chapter 6, clause 3 is set at a constant 20%. The default value reflects this change from the maximum of 35% in VV2009:19.
3. Multi-component (ψ) factors for the Swedish Complementary load model are given in TRVFS2011:12 Chapter 7, clause 5 (previously these were only found in TK Bro VV2009:27). However, values are identical in the two documents.

EN1991-2 Optional Code Settings - United Kingdom

This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

Selecting **EN1991-2 UK 2009** on the main Vehicle Load Optimisation dialog sets default values for NDPs and offers traffic load options according to UK National Annexes published by BSI:

- **UK NA to BS EN1991-2:2003 incorporating corrigendum No 1 (May 2008)**
- **UK NA to BS EN 1990:2002+A1:2005 incorporating National Amendment No. 1 (June 2009)**

Load model 1

Adjustment factors for Load Model 1 (αQ, αq) are taken from NA to BS EN1991-2 Table NA.1. According to EN1991-2 clause 4.3.2 (3) note 2, adjustment factors may correspond to classes of traffic. However, the adjustment factors in Table NA.1 are not dependent upon a selected traffic class and accordingly no traffic class options are offered on the dialog.

Load model 3

Load Model 3 special vehicles (three SV and four SOV model vehicles) are described in NA to BS EN1991-2 Fig NA.1 to NA.3 inclusive. One or more vehicles may be selected for use in the calculation of the most onerous load pattern. The selected vehicles are considered one at a time, being placed, together with associated Load Model 1 (switched on by default) according to the rules set out in clause NA.2.16.4.

Each SV has a central axle spacing which varies (3 possible values). All such axle spacings are considered in calculation of the most onerous load effect. Each SOV has an axle spacing which varies continuously from 1.5 to 40m. By default, this axle spacing is calculated to an accuracy set in the Optional Loading Parameters dialog.

Dynamic amplification factors (Table NA.2) are included in the calculation of the most onerous Group 5 load pattern and the wheel loads are accordingly factored within the vehicle (discrete point) loading attributes generated when the optimisation process is complete.

When Group 5 is included, the SV80 is selected as a default. This vehicle is intended to model the effects of STGO Category 2 vehicles with a maximum gross weight of 80 tonnes and a maximum basic axle load of 12.5 tonnes (NA.2.16.1.1). Project requirements would dictate if this is the appropriate vehicle and the vehicle selection may need to be adjusted or expanded.

According to EN1991-2 clause 4.2.1(1) Note 2, a complementary load model may be specified in the National Annex. No such complementary load model is described in the UK National Annex.

Psi factors

Multi-component (ψ) factors are taken from NA to BS EN1990 Table NA.A2.1. Infrequent values are not required (NA.2.3.6.2) and quasi-static values are not calculated since ψ2 is given as zero for all traffic loads (Table NA.A2.1). Group 4 loading is not included in the calculation of Frequent values (note b to Table NA.A2.1).

Restrictions on use

The traffic load models in EN1991-2 are applicable for bridges with loaded lengths less than 200m (clause 4.1(1)). In general the use of Load Model 1 is safe-sided for loaded lengths over 200m (4.1(1) note 1). The UK National Annex states that Load Model 1 may be used for loaded lengths up to 1500m (NA to BS EN1991-2 clause NA2.6) and no information on load models appropriate beyond that length is given. Calculation of the most onerous load pattern will proceed regardless of the loaded length and therefore patterns generated may be inappropriate for very long loaded lengths.

Transit New Zealand Bridge Manual [SP/M/022 2nd Edition (2005), and 3rd Edition (2013)] Optional Settings

This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

When using the country “New Zealand” and Design code “SP/M/022 2nd Edition (2005)” or SP/M/022 3rd Edition (2013), highway traffic loading is generated with reference to:

- **Bridge Manual SP/M/022 2nd Edition (June 2003, with amendments June 2004, September 2004, and July 2005), and 3rd Edition (May 2013)**

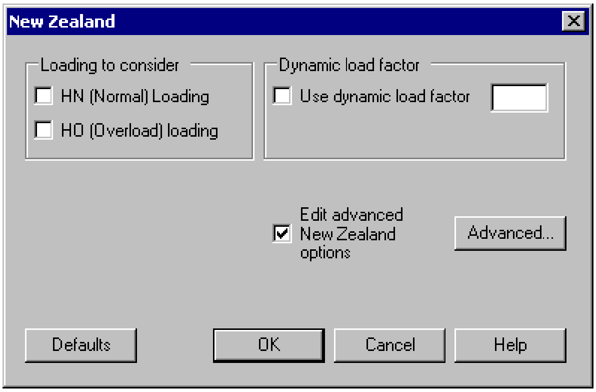
This document was originally published by Transit New Zealand, but since August 2008 it has been the responsibility of the New Zealand Transport Agency.

Note

Vehicle load optimisation to SP/M/022 as implemented in LUSAS TLO is restricted to vertical highway traffic loading and global effects. Fatigue loading, horizontal components, railway loads and pedestrian loads are not currently included. Accidental loads, as described in clause 3.2.3 (d), are considered to be for local effects and are not currently included.

Optional code settings dialog

The Optional Code Settings dialog allows selection of load models (components of HN-HO-72) required, as well as the setting of the dynamic factor. Factors and load intensities used in the calculations may be viewed and modified if required, by accessing the Advanced dialog.



Loading to consider

The traffic loads available on the dialog refer to SP/M/022 clauses 3.2.2 and 3.2.4 as follows:

- **HN (Normal) loading** As described in clause 3.2.2(a) and illustrated in Figure 3.1; this is formed from a uniformly distributed lane load spread over a 3m wide lane and placed in adverse areas, together with a pair of axle loads of 120kN each. The most onerous HN loading may comprise 1 or more loaded lanes, according to the reduction factors in clause 3.2.4.
- **HO (Overload) loading** As described in clause 3.2.2(b) and illustrated in Figure 3.1; this is formed from a uniformly distributed lane load spread over a 3m wide lane and placed in adverse areas, together with a pair of axle loads of 240kN each. The axles are considered as either a knife edge load (alternative (a)) or two wheel loads (alternative (b)). When used, HO loading replaces one element of HN loading in the traffic load pattern. The load pattern may include 1 or more loaded lanes, according to the reduction factors in clause 3.2.4.

The roadway is divided into a number of design lanes of equal width as per clause 3.2.3(b). Where this results in a lane width greater than 3m, the HN and HO loading will be placed in the most adverse transverse position.

The dialog allows selection of HN loading, HO loading or both. The most onerous traffic loading pattern appropriate to each selected load model will be determined, taking into account reduction factors as required. The dynamic factor will also be included if selected (see below).

Note
In SP/M/022 clause 3.5, normal traffic loads are denoted “LL” and overload combinations of traffic loads are denoted “OL”. LL and OL are considered in turn and always factored together in the combinations set out in Table 3.1 (Serviceability Limit State) and Table 3.2 (Ultimate Limit State). The load factors in Table 3.1 and 3.2 are not included in the optimised traffic loadcases, and therefore should normally be included in subsequent combinations.

Dynamic load factor

According to SP/M/022 clause 3.2.5 and Tables 3.1 and 3.2, both normal live load and overload shall be multiplied by a dynamic factor determined from Figure 3.2. The dialog allows the dynamic factor to be switched off if required, but more importantly allows the user to specify the value of that factor.

- The dynamic factor for components above ground and for bearings is 1.3 (default value) for most load effects. However for moments in simple or continuous beams of span the factor reduces for span lengths greater than 12m, to a minimum value of 1.19 at spans of 40m and above.
- The dynamic factor for below ground components is 1.0, to allow for vibration damping by the soil. However, for top slabs of culvert type structures the factor reduces linearly with depth of fill from 1.3 at zero fill to 1.0 at 1m fill.

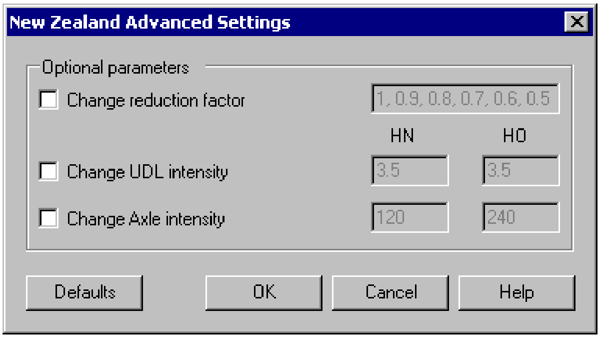
A value (generally 1.3 or less) should be determined and entered into the dialog as required. In the instance where different dynamic factors are required within a bridge analysis, this can be handled by creating more than one Vehicle Load Optimisation analysis and using different settings as appropriate.

Advanced options

- ❑ **Edit advanced New Zealand options** - See Transit New Zealand Bridge Manual (SP/M/022 2nd and 3rd Edition) Advanced Settings

Transit New Zealand Bridge Manual (SP/M/022 2nd Edition (2005), and 3rd Edition (2013)) Advanced Settings

This dialog is used to view and modify reduction factors and load intensities.

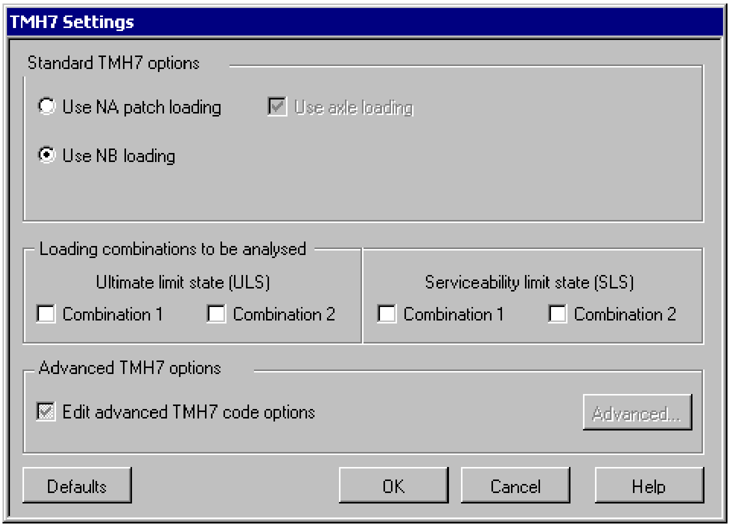


- **Change reduction factor** To allow for the improbability of concurrent loading, where appropriate, the total (traffic) loading is multiplied by a reduction factor, according to clause 3.2.4. Reduction factors are incorporated in the determination of the most onerous load pattern by default, but the values used may be modified by ticking this option and entering values appropriate to the number of elements of HN-HO-72 loading included (1 to 6 comma separated values, or more). To effectively eliminate reduction factors values of 1.0 may be entered for all 6 entries.
- **UDL intensity** A lane load UDL of 3.5kN/m² is used for both HN and HO loading, according to clause 3.2.2 and Figure 3.1.
- **Axle intensity** Total axle loads for HN and HO loading are 120kN and 240kN respectively, according to clause 3.2.2 and Figure 3.1.

Using the checkboxes, values for the UDL and Axle intensities may be entered manually under HN and/or HO as appropriate, to suit project requirements. The Default button resets all values back to the default values according to SP/M/022.

South Africa - TMH7 Settings

This design code loading is supported by the Autoloader Vehicle Load Optimisation software option.



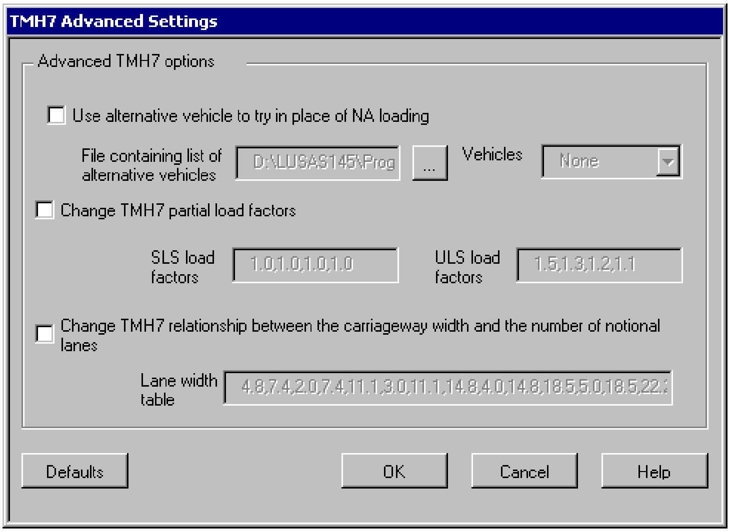
The following South African TMH7 options are available for selection:

- **Use NA loading** - specifies whether to apply the NA loading (which includes axle loading).
- **Use NB loading** - specifies whether to apply a single abnormal vehicle. If selected, the abnormal vehicles are applied to the carriageway. If not selected, then NA loading only is applied.
- **Use axle loading** - specifies whether to apply axle loads. If selected, axle loads are applied to areas with NA loading, according to the standard in use. If not selected, no axle loads are applied. If **Use NA loading** is not selected then **Use axle loading** will not be enabled.
- **Edit advanced TMH7 code options** - further to the basic code options provided on this dialog, Autoloader allows more advanced changes to the code to be made. To access this functionality ensure that the check box next to the label is selected and click on the Advanced button. Note, however, that it is unlikely that these advanced options will be required for the majority of work.

Notes

- Autoloader should be used for global bridge design. Local effects due to accidental wheel loads are not considered and so these effects should be assessed separately.
- The load sequence number is always assumed to be unity.
- The implementation of the South African loading code assumes that the transverse distribution has no significant effect and therefore the NA loading will be distributed over the full width of the notional lane.

TMH7 Advanced Settings



The following South African TMH7 advanced options are available for selection:

- **Use alternative vehicle to try in place of NA loading** - defines a list of vehicles to be tried as an alternative to NA loading. When loading an NA lane, Autoloader tries to place vehicles from this list within the lane, and if the effect is greater than NA loading, uses that vehicle instead. Each vehicle must be specified in the Autoloader vehicle library.
- **Change TMH7 partial load factors** - SLS Load factors allows specification of the partial load factors for the Serviceability Limit State and ULS Load factors allow specification of the partial load factors for the Ultimate Limit State. The list is ordered as follows:
  - NA alone (combination 1)
  - NA alone (combination 2)
  - NB with NA or NB alone (combination 1)
  - NB with NA or NB alone (combination 2)

- **Use TMH7 partial load factors** - SLS Load factors allows specification of the partial load factors for the Serviceability Limit State and ULS Load factors allow specification of the partial load factors for the Ultimate Limit State.
- **Change TMH7 relationship between carriageway width and the number of notional lanes**. Each set of 3 values are taken as a lower limit, an upper limit and a number of notional lanes. After calculating the carriageway width Autoloader works its way down the table, checking the calculated width against the values in the table.

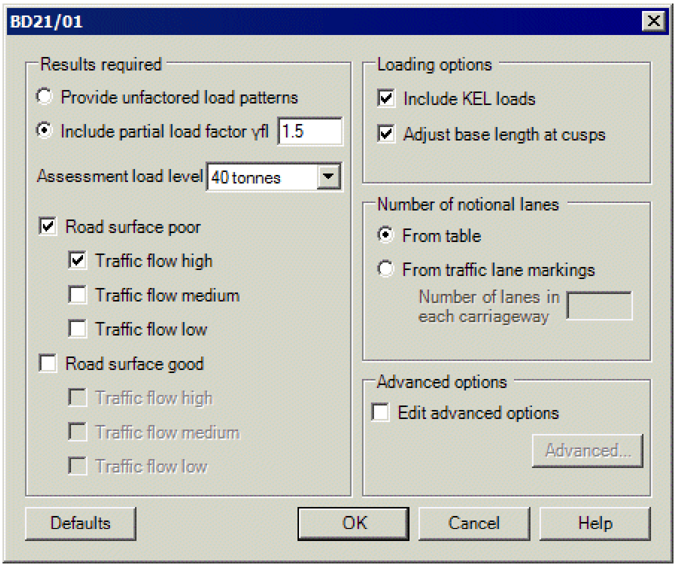
United Kingdom - BD21/01 Optional Code Settings

This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

When country "UK" and Design code "BD21/01" is selected on the main Vehicle Load Optimisation dialog, road traffic loading is generated with reference to BD21/01 Section 5.

The scope of the vehicle load optimisation to BD21/01 is restricted to vertical highway traffic loading and global effects. Horizontal components and pedestrian loads are not currently included.

The Optional Code Settings dialog allows selection of the combinations and loads required. Factors used in the calculations may be viewed and modified if required, by accessing the Advanced Settings dialog.



Results required

- Loading patterns are available either unfactored, or globally factored (i.e. with a partial factor). The default is factored for "Other Structures" to BD21/01 Table 3.1
- **Assessment load level** All Assessment load levels from BD21/01 Figures 5.2 to 5.7 are available.
- **Road surface / Traffic flow** All six categories are available. The dialog allows selection of one or more of these categories. The most onerous traffic loading pattern appropriate to each selected category will be determined.

Note. As per Note to BD21/01 clause 5.27, these categories are not used if the assessment load level selected is Group 1 FE, Group 2 FE or 3 tonnes.

Loading options

- **Include KEL loads** By default, one KEL is included per lane. This may be optionally disabled.
- **Adjust base length at cusps** As required by BD37/01 note to Table 13, "where the influence line has a cusped profile and lies wholly within a triangle joining the extremities of its base to its maximum ordinate, the base length shall be taken as twice the area under the influence line divided by the maximum ordinate". This may be optionally disabled.

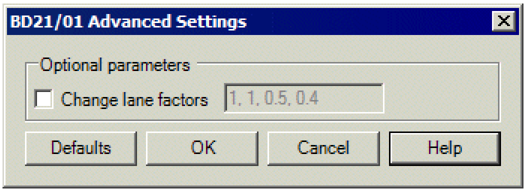
Number of notional lanes

- **From table** The road carriageway is by default divided into a number of equal width lanes as per BD21/01 Table 5.1. This table has a limit of 21.9m, with 6 lanes. This option is hence not compatible with wider carriageways.
- **From traffic lane markings** This option allows the input of the number of lanes as per BD21/01 clause 5.6. If multiple carriageways are selected, this number is used for them all.

It is interpreted in VLO that all lanes will have an equal width, equal to the carriageway width divided by the provided number of lanes. Unequal lane widths (if required) can be achieved by defining each lane as a separate carriageway and using the "From table" option.

A Defaults button is provided to reset all values back to those specified in BD21/01.

United Kingdom - BD21/01 Advanced Settings



The following BD 21/01 advanced options are available for selection:

Optional parameters

- **Change lane factors** This option allows the modification of the lane factors from the values in BD21/01 clause 5.24, which are implemented as default values. The textbox has a comma separated list of four variables, applicable respectively to: Lane 1, Lane 2, Lane 3, Lane 4 and subsequent.

A Defaults button is provided to reset the values back to those specified in BD21/01.

United Kingdom - BD21/01 Implementation Notes

Design Manual for Roads and Bridges Vol 3, Section 4, Part 3: BD21/01 The Assessment of Highway Bridges and Structures, last amended Aug 2001. Available freely from www.standardsforhighways.co.uk

BD21/01 Assessment Live Loading comprises Type HA loading UDL and KEL. The load intensities in clause 5.18 are similar to those in BD37/01, multiplied by a reduction factor (K), which varies according to loaded length and

- Good/poor surfacing
- High/medium/low traffic flow
- Load level, being 40 tonnes, 26 tonnes, 18 tonnes, Group 1 FE, 7.5 tonnes, Group 2 FE or 3 tonnes

The 3 traffic categories, 2 surfacing categories and 7 load levels are represented by the 27 K factor/loaded length curves in BD21/01 Figures 5.2 to 5.7 (Note that as per Clause 5.27 the curves for the 3 tonne and fire engine loading models do not vary between the figures). The load intensities are further multiplied by lane factors (clause 5.24) and divided by an Adjustment Factor (clause 5.23). The loading is applied over a 2.5m width within each notional lane as appropriate. Details of notional lanes are given in clause 5.6. A partial factor for loads may be applied to live loads as per Table 3.1.

The user interface allows the study of several road surface and traffic flow conditions in a single use of the traffic load optimiser. This is to enable the assessing engineer to understand how sensitive the results of the analysis are to such assumptions.

Bridges which are shown to be adequate to support the uppermost (40 tonnes) assessment loading level may then be assessed for their capacity with respect to Special Types General Order (STGO) vehicles (see BD21/01 clause 5.12). This is currently carried out using SV/SOV vehicles to BD86/11; prior to 2001 such ratings were assessed using HB vehicles to BA34/90. Both Codes Of Practice are also available in the LUSAS traffic load optimiser.

The notes below give further clarification on certain clauses of BD21/01 and their implementation within the LUSAS traffic load optimiser, and information on available options.

Positioning of UDL and KEL

BD21/01 clause 5.26 states “The lane loading for any lane determined as in 5.25 above shall be applied to occupy a width of 2.5m, in the most onerous transverse position in that lane.” This is interpreted to mean that the UDL and KEL occupy the same transverse position in each lane. The vehicle load optimiser allows this combined UDL and KEL to occupy the most adverse transverse position within the lane.

BD21/01 clause 5.18 states that the KEL is “uniformly distributed across the lane width”. This is interpreted to mean that the KEL may only be positioned perpendicular to the direction of the lane.

Loaded lengths less than 2m

BD21/01 clause 5.9 requires that for “loaded lengths less than 2m the single axle load and the single wheel load shall be used”. These loads are not implemented in VLO as they are local effects. Loaded lengths less than 2m are therefore treated the same as loaded lengths from 2m to 50m.

Use of BD50/92

BD21/01 clause 5.9 states "For loaded lengths in excess of 50m, the UDL and KEL to be used shall be as described in BD 50 (DMRB 3.4.2)." BD50/92 has been withdrawn from the DMRB, with the Highways Agency website stating "Users seeking BD50/92 should refer to BD101/11". BD101/11, however, does not discuss the use of UDL or KEL for loaded lengths in excess of 50m. The use of the withdrawn BD50/92 is therefore considered to still be valid (for the specific purpose of loaded lengths in excess of 50m only) and is therefore implemented by the vehicle load optimiser.

For loaded lengths below 50m, BD21/01 specifies an Adjustment Factor AF. AF is not specified in BD50/92 for loaded lengths greater than 50m, but given that in BD21/01 it increases with loaded length to a maximum of 1.0 (for 40m to 50m), it has been taken as 1.0 for loaded lengths in excess of 50m within the vehicle load optimiser.

BD50/92 Clause 2.3 defines the K factor for loaded lengths greater than 50m as 0.91 for 40t and 0.4 for 7.5t, without defining the Traffic and Surface they relate to, or K Factors for the other Assessment Load Levels. BD21/01 Figures 5.2 to 5.7 give K factor for 40t for High_poor as 0.91 for all lengths up to 50m. It is therefore considered that the 0.91 factor in BD50/92 is applicable for 40t High_poor, and hence for 40t, 26t, 18t and Group 1 FE (all of which have uniform or near uniform K factors approaching 50m), the K factor for Loaded Lengths in excess of 50m can be taken as that for 50m in Figures 5.2 to 5.7. For 7.5t, Group 2 FE and 3t (all of which have a K factor increasing towards 50m with a value of 0.35 at 50m) it is considered that the K factor can be taken as 0.4 for all loaded lengths in excess of 50m.

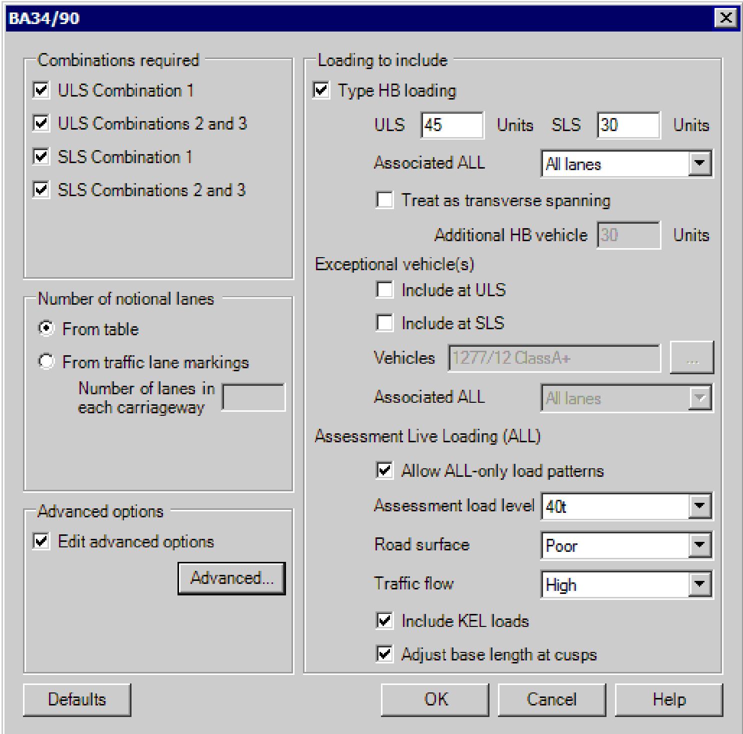
United Kingdom - BA34/90 Settings

This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

When using the country "UK" and Design code "BA34/90", road traffic loading is generated to BA34/90, with reference to BD21/01 and BD37/01.

The scope of the vehicle load optimisation to BA34/90 is restricted to vertical highway traffic loading and global effects. Horizontal components and pedestrian loads are not currently included.

The Optional Code Settings dialog allows selection of the combinations and loads required. Factors used in the calculations may be viewed and modified if required, by accessing the **Advanced Settings** dialog.



Combinations required

The combinations available on the dialog refer to those from BD37/01, as referenced by BA34/90 clause 3.2. As the partial load factors in ULS combinations differ between different loads, it is necessary for the factors to be considered as part of the optimisation process. The limit states and combinations are taken from BD37/01 Clause 4.4 and Table 1.

Number of notional lanes

- **From table** The road carriageway is by default divided into a number of equal width lanes as per BD21/01 Table 5.1. This table has a limit of 21.9m, with 6 lanes. This option is hence not compatible with wider carriageways.
- **From traffic lane marking** This option allows the input of the number of lanes as per BD21/01 clause 5.6. If multiple carriageways are selected, this number is used for them all.

It is interpreted in VLO that all lanes will have an equal width, equal to the carriageway width divided by the provided number of lanes. Unequal lane widths (if required) can be achieved by defining each lane as a separate carriageway and using the “From table” option.

Loading options

- **Type HB loading** HB loading is provided to BD37/01 Clause 6.3. Application with ALL is in accordance with BD37/01 Clause 6.4.2, with ALL taken to replace Type

HA loading. The selection of the option to Treat as transverse spanning generates loading to Clause 6.4.3.1. An option is provided for the Associated ALL, allowing the inclusion of ALL in all lanes, only lanes which do not contain the HB vehicle, or the exclusion of ALL.

- **Exceptional vehicles** Vehicles are provided to the highways Agency's Heavy Load Vehicle Classification Specifications. Application with ALL is in accordance with BD37/01 Clause 6.4.2, with ALL taken to replace Type HA loading and Exceptional vehicles taken to replace Type HB loading. An option is provided for the Associated ALL, allowing the inclusion of ALL in all lanes, only lanes which do not contain the Exceptional vehicle, or the exclusion of ALL.

Assessment Live Loading (ALL)

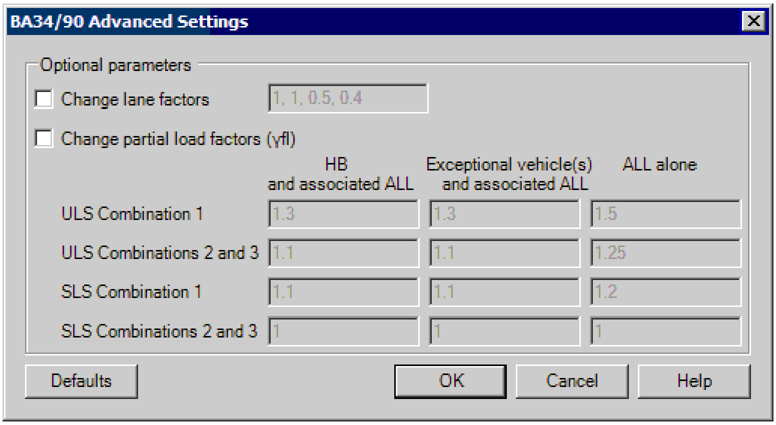
ALL is provided to BD21/01. The reduction factor is provided to Figures 5.2 to 5.7 based on selections made for Assessment load level, Road surface and Traffic flow.

For more detail on interpretations made for ALL, see **BD21/01 Optional Settings.**

- **Allow ALL-only load patterns**. By default an ALL only pattern is included, with ULS load factors to BD37/01 Table 1. This checks against the case where Type HA alone, with a high partial factor, is more onerous than an HB/Exceptional vehicle with ALL at lower partial factors. This loading option may be optionally disabled, but note that this only removes the load pattern with higher factors; if ALL with lower partial factors is still more onerous than the HB/Exceptional vehicle then the optimisation will not place the HB/Exceptional vehicle on the deck.
- **Include KEL loads**. By default, one KEL is included per lane. This may be optionally disabled.
- **Adjust base length at cusps**. As required by BD37/01 note to Table 13, "*where the influence line has a cusped profile and lies wholly within a triangle joining the extremities of its base to its maximum ordinate, the base length shall be taken as twice the area under the influence line divided by the maximum ordinate*". This may be optionally disabled.

A Defaults button is provided to reset all values back to those specified in the design codes.

United Kingdom - BA34/90 Advanced Settings



This dialog is used to view and modify lane factors and partial load factors.

Optional parameters

- ❑ **Change lane factors** This option allows the modification of the lane factors from the values in BD21/01 clause 5.24, which are implemented as default values. The textbox has a comma separated list of four variables, applicable respectively to: Lane 1, Lane 2, Lane 3, Lane 4 and subsequent.
- ❑ **Change partial load factors (γfl)** This option allows the modification of the partial load factors from the values in BD37/01 Table1, which are implemented as default values. Exceptional vehicles (and their associated ALL) are assigned the same factors as for HB vehicles by default.

A Defaults button is provided to reset the values back to those specified in the design codes.

United Kingdom - BA34/90 Implementation Notes

With reference to:

- Design Manual for Roads and Bridges, Volume 3, Section 4, Part 17: BD34/90 Technical Requirements For The Assessment And Strengthening Programme For Highway Structures, last updated Sept 1990
- Design Manual for Roads and Bridges, Volume 3, Section 4, Part 17: BA34/90 Technical Requirements For The Assessment And Strengthening Programme For Highway Structures, last updated Sept 1990
- Heavy Load Vehicle Classification Specifications (Highways Agency, Dec 2006)

Existing UK highway bridges which are shown to be adequate to support 40t Assessment Live Loading may then be assessed for their capacity with respect to Special Types General Order (STGO) vehicles (see BD21/01 clause 5.12). The current Code Of Practice for such assessment is BD86/11, which is also available in the LUSAS Vehicle Load Optimiser.

Prior to 2001, STGO ratings were assessed using HB vehicles to BD34/90. Such ratings are still widely used by highway authorities. BD34/90 makes only the broadest statements about which structures should be assessed using HB vehicles; clause 1.4 refers us to BA34/90 for details, although BD34/90 clause 3.1 (g) (ii) does allow for the assessment of bridges for their capacity to carry specific exceptional vehicles. The LUSAS Traffic Load Optimisation software accordingly includes options for specific vehicles to the Heavy Load Vehicle Classification Specifications.

BA34/90 refers to superseded standards BD21/84 and BD37/88 which have been read as BD21/01 and BD37/01 respectively. In essence, the Assessment Live Loading of BD21/01 is applied together with HB vehicles in patterns and combinations complying with the requirements of BD37/01. The BD21/01 Code is available in the LUSAS Vehicle Load Optimiser; more details about its loading requirements may be found in the appropriate Help entry.

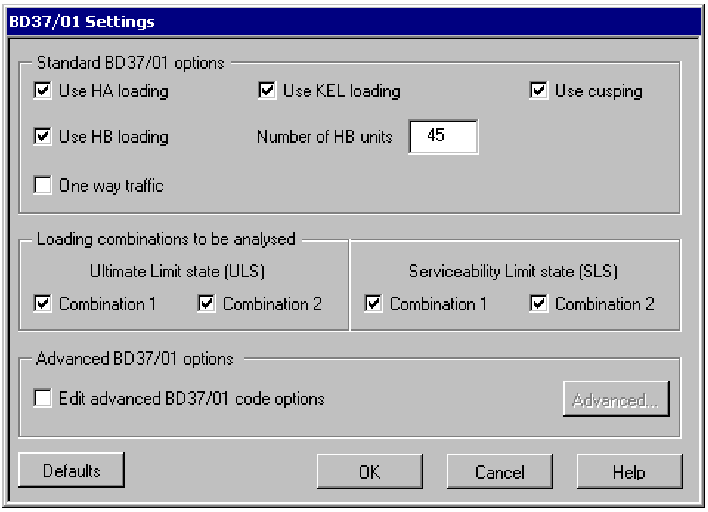
It may be of interest to the highway authority to consider what HB rating or what exceptional vehicles may be safely transported if specific traffic restrictions are in place, for example, if a temporary 3t limit is placed on the bridge during the transit of an exceptional vehicle; if all other vehicular loads are excluded from the bridge; or if all other loads are restricted from the lane carrying the abnormal vehicle; therefore the options for associated live load allow user selection of the loading level required (40t, 26t, 18t, Group 1 FE, 7.5t, Group 2 FE, 3t), road surface category (good, poor) and traffic flow (high, medium, low) as appropriate.

It may also be of interest to the Highway Authority to consider what HB rating or what exceptional vehicles may be safely transported if all other vehicular loads are excluded from the bridge; or if all other loads are excluded from the lane(s) occupied by the abnormal vehicle.

BA34/90 Optional Settings and **BA34/90 Advanced Settings** give further clarification on certain clauses of BA34/90 and their implementation within the LUSAS Traffic Load Optimisation software and information on all available options.

United Kingdom - BD37/01 Settings

This design code loading is supported by the Autoloader Vehicle Load Optimisation software option.

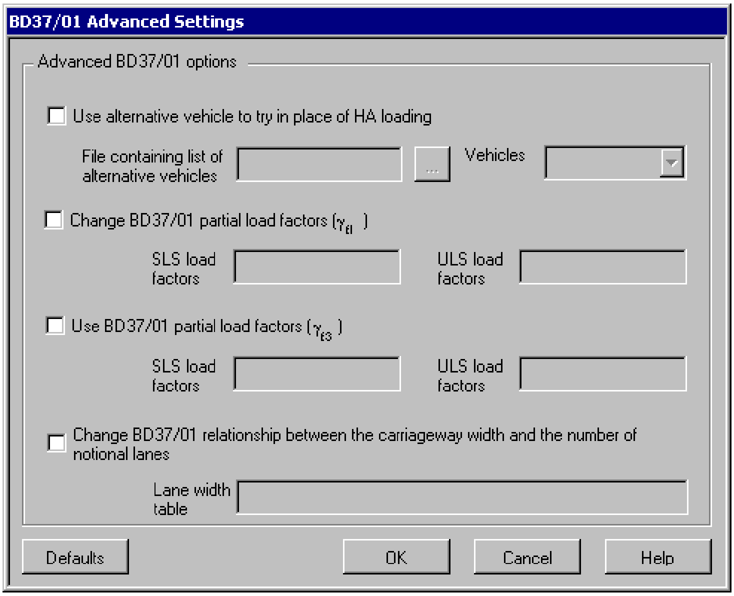


The following BD37/01 options are available for selection:

- **Use HA Loading -** specifies whether to apply the HA loading (which includes KEL loading). To use this option select the check box next to the label, to deselect this option ensure that the check box is not selected. By default this option will be selected.
- **Use HB Loading** - specifies whether to apply an abnormal vehicle. If selected, the abnormal vehicles are applied to the carriageway. If not selected, then HA loading only is applied. To use this option select the check box next to the label, to deselect this option ensure that the check box is not selected. By default this option will be selected.
- **Use KEL Loading** - specifies whether to apply Knife Edge Loads. If selected, Knife Edge Loads are applied to areas with UDL loading, according to the standard in use. If not selected, no KEL'S are applied. To use this option select the check box next to the label, to deselect this option ensure that the check box is not selected. By default this option will be selected, however if Use HA loading is not selected then Use KEL Loading will not be enabled.
- **Use Cusping** - specifies whether to apply cusping when working out loadable areas. If selected, cusping is applied. If not selected, cusping is not applied. To use this option select the check box next to the label, to deselect this option ensure that the check box is not selected. By default this option will be selected.
- **One Way Traffic** - BD37/01 specifies that a bridge carrying traffic in one direction only would result in the doubling of the N value in the calculation of the HA lane factors (ref. BD37/01, Table 14, Note 2). If ONEWAY is selected, then the value of N used in the table is doubled. To use this option select the check box next to the label, to deselect this option ensure that the check box is not selected. By default this option will not be selected

- **Number of HB Units** - specifies the number of HB Units to use. In cases where an abnormal vehicle uses HB Units, the values of the weights of the axles are multiplied by the number of HB Units. By default the number of HB units is set as 45, to change this value enter the new value in the text box.
- **Edit advanced BD37/01 code options** - further to the basic code options provided on this dialog, Autoloader allows more advanced changes to the code to be made. To access this functionality ensure that the check box next to the label is selected and click on the Advanced button. However, note that it is unlikely that these advanced options will be required for the majority of work.

BD37/01 Advanced Settings



The following BD37/01 advanced options are available for selection:

- **Use Alternative vehicle**- defines a list of vehicles to be tried as an alternative to HA loading. When loading an HA lane, Autoloader tries to place vehicles from this list within the lane, and if the effect is greater than HA loading, uses that vehicle instead. Each vehicle must be specified in the Autoloader vehicle library. To use this option select the check box next to the label and enter the vehicle name that you require as an alternative to HA loading. To deselect this option ensure that the check box is not selected. By default this option will be deselected.
- **Change BD37/01 Partial Loadfactors** -SLS Load factors allow specification of the partial load factors for the Serviceability Limit State and ULS Load factors specification of the partial load factors for the Ultimate Limit State. The list is ordered as follows:

- HA alone (combination 1)
- HA alone (combination 2)
- HB with HA or HB alone (combination 1)
- HB with HA or HB alone (combination 2)

To use this option select the check box next to the label and enter the new factors to be used. To deselect this option ensure that the check box is not selected. By default this option will be deselected.

- **Change relationship between carriageway width and the number of notional lanes**. Each 3 values are taken as a lower limit, an upper limit and a number of notional lanes. After calculating the carriageway width Autoloader works its way down the table, checking the calculated width against the values in the table. To use this option select the check box next to the label and enter the values into the text box. To deselect this option ensure that the check box is not selected. By default this option will be deselected.

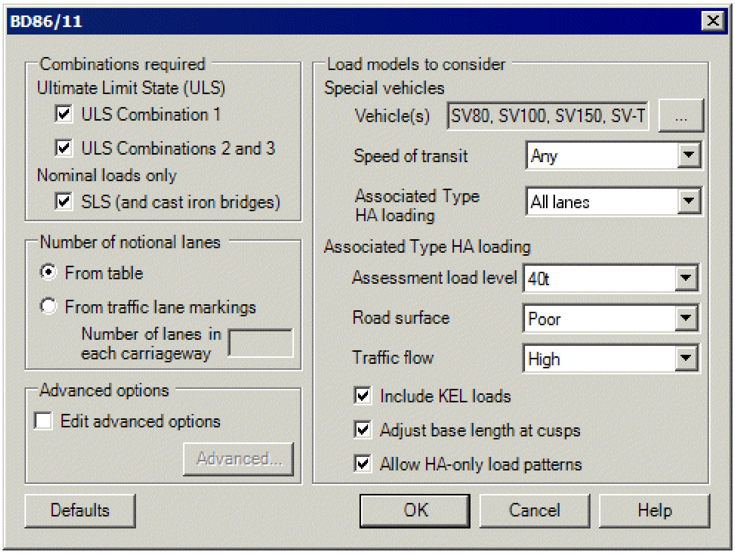
United Kingdom - BD86/11 Optional Code Settings

This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

When using the country “UK” and Design code “BD86/11”, road traffic loading is generated to BD86/11, with reference to BD21/01 and BD37/01.

The scope of the vehicle load optimisation to BD86/11 is restricted to vertical highway traffic loading and global effects. Horizontal components and pedestrian loads are not currently included.

The Optional Code Settings dialog allows selection of the combinations and loads required. Factors used in the calculations may be viewed and modified if required, by accessing the **Advanced Settings** dialog.



Combinations required

The combinations available on the dialog cover all situations from BD86/11 clauses 2.6 to 2.9. As the partial load factors in ULS combinations differ between different loads, it is necessary for the factors to be considered as part of the optimisation process.

Number of notional lanes

- **From table** The road carriageway is by default divided into a number of equal width lanes as per BD21/01 Table 5.1. This table has a limit of 21.9m, with 6 lanes. This option is hence not compatible with wider carriageways.
- **From traffic lane markings** This option allows the input of the number of lanes as per BD21/01 clause 5.6. If multiple carriageways are selected, this number is used for them all.

It is interpreted in VLO that all lanes will have an equal width, equal to the carriageway width divided by the provided number of lanes. Unequal lane widths (if required) can be achieved by defining each lane as a separate carriageway and using the "From table" option.

Loading options

- **Special vehicles** Vehicles are provided to BD86/11 Figures 3.1 to 3.7, and additionally to the highways Agency's Heavy Load Vehicle Classification Specifications. Application with Type HA loading is as per BD86/11 clauses 3.20 to 3.30. An option is provided for speed of transit, allowing any configuration by default, or specifically either Normal speed or Low speed if desired. The Associated Type HA loading can be included in all lanes, only lanes which do not contain the Special vehicle, or can be excluded from all lanes (allowing a vehicle-only optimisation).

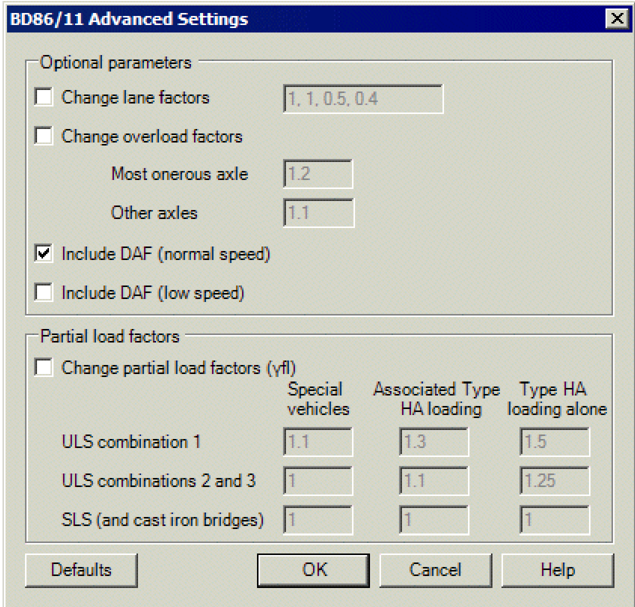
- ❑ **Associated Type HA loading** Type HA loading is provided to BD21/01. The reduction factor is provided to Figures 5.2 to 5.7 based on selections made for Assessment load level, Road surface and Traffic flow.

  For more detail on interpretations made for Type HA loading, see **BD21/01 Optional Settings.**

  - **Include KEL loads** By default, one KEL is included per lane (except special vehicle lanes). This may be optionally disabled.
  - **Adjust base length at cusps** As required by BD37/01 note to Table 13, "*where the influence line has a cusped profile and lies wholly within a triangle joining the extremities of its base to its maximum ordinate, the base length shall be taken as twice the area under the influence line divided by the maximum ordinate*". This may be optionally disabled.
  - **Allow HA-only load patterns** By default a Type HA only pattern is included, with ULS load factors to BD37/01 Table 1. This checks against the case where Type HA alone, with a high partial factor, is more onerous than a special vehicle with Type HA at lower partial factors. This loading option may be optionally disabled, but note that this only removes the load pattern with higher factors; if Type HA with lower partial factors is still more onerous than the special vehicle then the optimisation will not place the special vehicle on the deck.

A Defaults button is provided to reset all values back to those specified in the design codes.

United Kingdom - BD86/11 Advanced Settings



This dialog is used to view and modify lane factors and partial load factors.

Optional parameters

- **Change lane factors** This option allows the modification of the lane factors from the values in BD21/01 clause 5.24, which are implemented as default values. The textbox has a comma separated list of four variables, applicable respectively to: Lane 1, Lane 2, Lane 3, Lane 4 and subsequent.
- **Change overload factors** The overload factors to BD86/11 clause 3.16 are applied to every special vehicle, to the most onerous axle and to all other axles as per the textboxes. These values may be optionally changed from the defaults.
- **Include DAF (normal speed)** By default the Dynamic Amplification Factor to BD86/11 clause 3.17 is applied to all axles of the special vehicles at normal speed. This factor may optionally be removed.
- **Include DAF (low speed)** By default the Dynamic Amplification Factor is not applied to special vehicles at low speed, as per BD86/11 clause 3.24. The factor, to clause 3.17, may optionally be included.

Partial load factors

- **Change partial load factors (γfl)** This option allows the modification of the partial load factors from the default values. These are taken from BD86/11 clauses 2.6 to 2.9; except for Type HA loading alone at ULS, which is taken from BD37/01 Table1.

A Defaults button is provided to reset the values back to those specified in the design codes.

United Kingdom - BD86/11 Implementation Notes

Design Manual for Roads and Bridges Vol 3, Section 4, Part 19: BD86/01 The Assessment of Highway Bridges and Structures for the effects of Special Types General Order (STGO) and Special Order (SO) Vehicles, last amended Nov 2011. Available freely from www.standardsforhighways.co.uk

Existing UK highway bridges which are assessed as adequate to carry the 40t loading of BD21/01 are deemed, in BD86/11 clause 1.3.1, adequate for vehicles complying with The Road Vehicles Construction and Use (C&U) Regulations and Authorised Weight (AW) Regulations. Bridges which have some spare capacity under BD21/01 may therefore be assessed for adequacy to carry one (or both) of the following:

> a) **Vehicles complying with The Road Vehicles (Authorisation of Special Types) General Order (STGO) Regulations**. Five SV load models simulate the effects of real STGO vehicles as per BD86/11 clause 3.10
>
> b) **Special Order (SO) Vehicles**. Four SOV load models simulate the effects of real SO vehicles within limits set out in BD86/11 clause 3.12, Table 3.1.

The SV and SOV load models of BD86/11 are applied together with associated Assessment Live Loading according to BD21/01, which is referred to (within BD86/11) as “Type HA loading”. BD86/11 Clauses 3.20 to 3.29 and Figures 3.9 and 3.10 apply and Type HA loading is excluded over a length preceding and following the SV and SOV load models depending on the speed of transit, being normal speed (25m exclusions) or low speed (5m exclusions). Accordingly options are offered in the user interface.

An overload factor of 1.1 is applied to all axles except for the most onerous axle, for which a value of 1.2 is used as per clause 3.16. A Dynamic Amplification Factor (DAF) is applied to all axles as given in clause 3.17. It is not, however, applied when the vehicle is travelling at low speed with the shorter (5m) exclusion length; see clause 3.24.

It may be of interest to the Highway Authority to consider what SV/SOV vehicle or what exceptional vehicle may be safely transported if specific traffic restrictions are in place, for example, if a temporary 3t limit is placed on the bridge during the transit of an exceptional vehicle; therefore the options for associated live load allow user selection of the loading level required (40t, 26t, 18t, Group 1 FE, 7.5t, Group 2 FE, 3t), road surface category (good, poor) and traffic flow (high, medium, low) as appropriate.

It may also be of interest to the Highway Authority to consider what SV/SOV vehicle or what exceptional vehicle may be safely transported if all other vehicular loads are excluded from the bridge; or if all other loads are excluded from the lane(s) occupied by the abnormal vehicle – as per BD86/11 clause 1.5(ii). Likewise, referring to clause 1.5(ii), the Highway Authority may place special restrictions on load or speed, therefore options to control Overload factor, Dynamic Amplification Factor and/or load pattern are available.

For more information on the Associated Live Loads to BD21/01, see the **BD21/01** help entry.

Prior to 2001, it was mandatory in the UK to carry out STGO assessments using HB vehicles to BA34/90, which is also available within the LUSAS traffic load optimiser.

BD86/11 Optional Settings and **BD86/11 Advanced Settings** notes give further clarification on certain clauses of BD86/11 and their implementation within the LUSAS traffic load optimiser, and information on all available options.

United States of America - AASHTO LRFD (7th Edition) Optional Code Settings

This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

When the country 'United States of America' and the Design code 'AASHTO LRFD 7th Ed', is selected on the main Vehicle Load Optimisation dialog, road traffic loading data and parameters can be specified with reference to AASHTO LRFD Bridge Design Specifications, Seventh Edition.

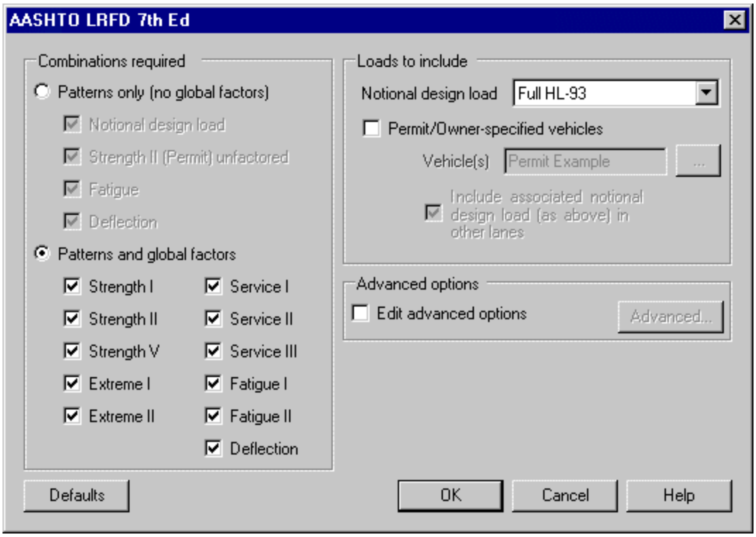
Note that whilst all references in the LUSAS software and documentation are for the AASHTO LRFD 7th Edition, the LUSAS implementation is applicable to the both the AASHTO LRFD 7th Edition and the AASHTO LRFD 6th Edition.

Design codes are also available for many US State Bridge Design Manuals' implementations of AASHTO LRFD. See **State Implementations of AASHTO LRFD (7th Edition)** for more details.

Scope

The scope of the vehicle load optimisation to AASHTO LRFD (7th Edition) is restricted to vertical highway traffic loading and global effects. Horizontal components, railway loads and pedestrian loads are not currently included. The refined design truck for fatigue design of orthotropic decks, to Figure 3.6.1.4.1-1, is not included as it is considered applicable for local effects only.

The road carriageway is divided into standard 12ft design lanes (apart from carriageways from 20ft to 24ft wide which have two equal width lanes). This will result in a “remaining width” for any carriageways of width not an integer multiple of 12ft. The lanes are positioned laterally on the bridge to produce the most adverse effects, with the remaining width being placed either side of the lanes or between any of the lanes so as to produce the most onerous arrangement. The number of design lanes can be modified if required, as per AASHTO LRFD clause 3.6.1.1.1, via the **Advanced Settings** dialog. Loading moves transversely within the design lanes for most onerous effect.



The Optional Code Settings dialog allows selection of the combinations and loads required. Factors used in the calculations and other settings may be viewed and modified if required, by accessing the Advanced Settings dialog.

Combinations required

Loading patterns are available either globally factored (i.e. with a Load Factor for a Limit State to AASHTO LRFD Table 3.4.1-1, or optionally edited in the Advanced Settings dialog) or unfactored as follows:

- ❑ **Patterns only (no global factors)**

  - **Notional design load** - loading to AASHTO LRFD clause 3.6.1.3.1
  - **Strength II (Permit) unfactored** - as Notional design load pattern but with a Permit vehicle replacing one lane of loading, where this is more onerous
  - **Fatigue** - loading to AASHTO LRFD clause 3.6.1.4.1
  - **Deflection** - loading to AASHTO LRFD clause 3.6.1.3.2

- ❑ **Patterns and global factors**

  - Strength I / Strength V / Extreme I / Extreme II / Service I / Service II / Service III - as per Notional design load pattern but with the relevant global factor from AASHTO LRFD Table 3.4.1-1
  - Strength II - as per Strength II (Permit) unfactored pattern but with the relevant global factor from AASHTO LRFD Table 3.4.1-1
  - Fatigue I / Fatigue II - as per Fatigue pattern but with the relevant global factor from AASHTO LRFD Table 3.4.1-1

- Deflection - as per unfactored Deflection pattern but with the Service I global factor from AASHTO LRFD Table 3.4.1-1

The dialog allows selection of one or more combinations. The most onerous traffic loading pattern appropriate to each selected combination will be determined, with load factors included as appropriate.

Loads to include

- ❑ **Notional design load.** If "Full HL-93" is selected, the most onerous of the Design Truck or Design Tandem will be placed in each lane. If "Truck & lane loads only" or "Tandem & lane loads only" is selected, then the Design Truck or Design Tandem respectively will be placed in all lanes. This selection also affects the alternative loading for interior piers, i.e. the two truck loading and two tandem loading (for further details see **Advanced Settings** dialog).
- ❑ **Permit/Owner-specified vehicles**. This option is available when either of the Strength II combinations is selected. If this option is selected then the most onerous of the selected Vehicles will replace one lane of loading for these combinations, provided it is more onerous than the selected Notional design load for that lane.

  - **Vehicle(s)**. Clicking the "..." button allows selection of one or more Permit vehicles. As no Permit vehicles are defined within AASHTO LRFD 7th Edition, an example vehicle is provided. State implementations have relevant vehicles as defined by their State Bridge Design Manuals, as appropriate.
  - **Include associated notional design load (as above) in other lanes**. This check box may be unchecked in order to obtain a result for Permit vehicles alone.

Multiple presence factors are applied to the loading as per AASHTO LRFD clause 3.6.1.1.2, or optionally edited via the Advanced Settings dialog. The optimisation process takes account of the multiple presence factors when calculating the most onerous load effect, such that every lane may not be loaded. As per clause 3.6.1.1.2 they are not applied to the Fatigue loading. By default the 1.20 factor for a single loaded lane is not applied to Permit vehicles alone - the factor of 1.2 for a single lane loaded is due to HL-93 having been originally calibrated for two lanes, and it is assumed that this calibration has not been applied to Permit vehicle definitions. This setting can, however, be modified in the **Advanced Settings** dialog.

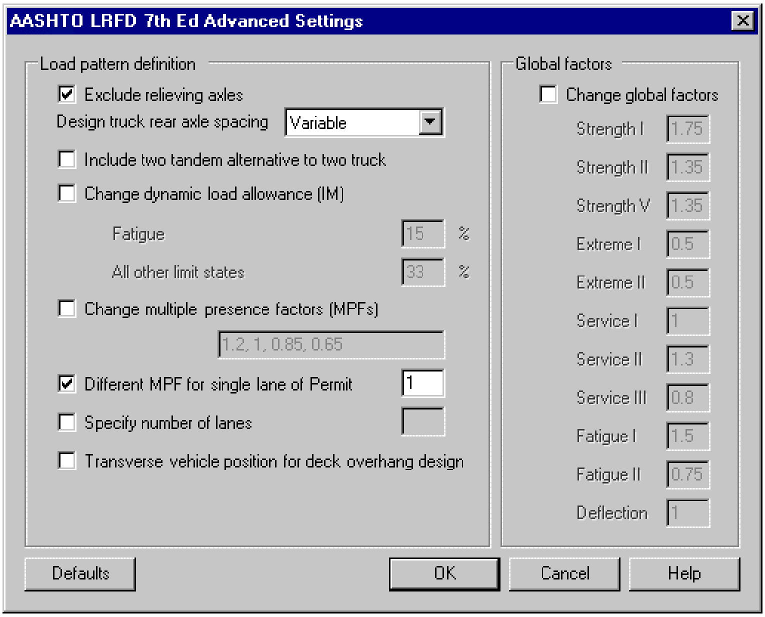
The dynamic load allowance to AASHTO LRFD clause 3.6.2.1 is applied to all vehicles in all combinations, but not to the design lane load. The values for All Other Components to Table 3.6.2.1-1 are applied by default; modifications are possible via the **Advanced Settings** dialog.

Two truck loading and two tandem loading for interior piers is included when the 'Alternative load pattern' option is selected on the Set Influence Surfaces dialog. (See also **Advanced Settings** dialog for further details on the two tandem loading).

A Defaults button is provided to reset all values back to those specified in AASHTO LRFD (or the selected State Bridge Design Manual).

AASHTO LRFD (7th Edition) Optional Code Settings (Advanced)

This dialog is used to view and modify dynamic factors, multiple presence factors, global factors and a number of additional settings.



Load pattern definition

- **Exclude relieving axles**. AASHTO LRFD clause 3.6.1.3.1 states “Axles that do not contribute to the extreme force effect under consideration shall be neglected.” By default relieving axles are therefore excluded from generated loading, but all vehicle axles may be included by unchecking this option.
- **Design truck rear axle spacing**. The Design Truck from AASHTO LRFD clause 3.6.1.2.2 has a rear axle which varies from 14ft to 30ft; this is implemented by default. This option gives the opportunity to restrict which axle spacings are considered in order to give a faster optimisation process. Axle spacing options should only be restricted when engineering judgement deems this will give a sufficiently accurate result. This option does not affect the axle spacing where it is specified elsewhere, i.e. two trucks (for interior piers) will still have a 14ft rear axle to AASHTO LRFD clause 3.6.1.3.1 and for fatigue the truck will still have a 30ft rear axle to AASHTO LRFD clause.
- **Include two tandem alternative to two truck**. This option allows the inclusion, to AASHTO LRFD commentary clause C3.6.1.3.1, of a two tandem alternative to the two truck loading. When selected it does not replace the two truck loading, rather it is an additional load, with the most onerous being selected. This option is unchecked by

default for AASHTO LRFD 7th Ed, but may be checked by default according to a State's Bridge Design Manual.

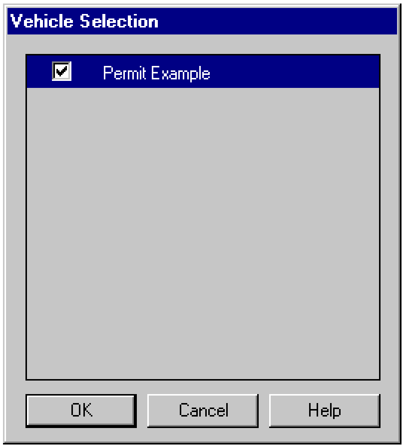
- **Change dynamic load allowance (IM)**. This option allows the modification of the IM from the values in AASHTO LRFD Table 3.6.2.1-1 for All Other Components, which are implemented as default values. Dynamic load allowance is applied to all vehicles in all combinations, but not to the design lane load.
- **Change multiple presence factors (MPFs)**. This option allows the modification of the MPFs from the values in AASHTO LRFD Table 3.6.1.1.2-1, which are implemented as default values. The textbox has a comma separated list of four variables, applicable respectively to: one lane loaded, two lanes loaded, three lanes loaded, more than three lanes loaded.
- **Different MPF for single lane of Permit**. By default the 1.20 factor for a single loaded lane is not applied to Permit vehicles alone - the MPF of 1.2 for a single lane loaded is due to HL-93 having been originally calibrated for two lanes, and it is assumed that this calibration has not been applied to Permit vehicle definitions. This is in accordance with many States' Bridge Design Manuals. However, the default is changed for those States whose manuals specifically use an MPF of 1.2 for Permit vehicles.
- **Specify number of lanes**. By default the carriageway is divided into standard 12ft design lanes, or two equal width lanes for carriageways from 20ft to 24ft wide, as per AASHTO LRFD clause 3.6.1.1.1. This option is provided in accordance with the same clause, allowing the number of design lanes to be specified if their width is less than 12ft. If this option is selected the carriageway will be divided into the specified number of equal width lanes, although the lanes may not be less than 10ft wide due to the width of the notional design loads.
- **Transverse vehicle position for deck overhang design**. As per AASHTO LRFD clause 3.6.1.3.1 the wheel loads should usually be restricted to 2ft from the edge of the design lane / curb, but should be allowed within 1ft of the curb for the design of the deck overhang. By default the wheel loads are therefore restricted to 2ft from the edge of the carriageway, but on selecting this option that distance is reduced to 1ft.

Global factors

- **Change global factors.** The default values provided to AASHTO LRFD Table 3.4.1-1 may optionally be edited. γEQ for Extreme I, which is to be determined on a project specific basis, has been assigned a default of 0.50. NB The default value for the factored Deflection combination is that for Service I, as per AASHTO LRFD clause 3.4.2.2, but may be edited to be different to that for Service I. The value entered is only used for the factored Deflection combination and not for the unfactored combination.
- A **Defaults** button is provided to reset all values back to those specified in AASHTO LRFD (or the selected State Bridge Design Manual).

AASHTO LRFD (7th Edition) Special Vehicle Selection

By clicking the ... button on the Optional Code Settings dialog a Special Vehicle Selection dialog is displayed. This allows the inclusion of a Permit Example.



State Implementations of AASHTO LRFD (7th Edition)

The majority of the Departments of Transportation in the United States publish a Bridge Design Manual or equivalent, which may clarify, add to, or modify the requirements of AASHTO LRFD. The list below contains those States whose manuals have been implemented, along with details of any additions or modifications. If optimisation to a manual not listed below is desired, please contact LUSAS to request its implementation.

Unless otherwise noted for a particular State, all Permit/Owner-specified vehicles below are implemented with a 6-foot transverse wheel spacing and excluding all other loads from their lane, but allowing HL-93 loads in other lanes.

- **Alabama** - ALDOT Structural Design Manual, June 2014 - No changes to AASHTO loading.
- **Delaware** - DelDOT Bridge Design Manual, May 2005 - No changes to AASHTO loading.
- **Florida** - FDOT Structures Design Guidelines, January 2014 - The load factor for Extreme I is set to 0.0 as per FDOT SDG clause 2.1.1
- **Georgia** - Georgia DOT LRFD Bridge and Structure Design Manual, October 2013 - No changes to AASHTO loading.
- **Hawaii** - (Email from HIDOT) - No changes to AASHTO loading.
- **Idaho** - Idaho LRFD Bridge Design Manual, April 2008 - Two tandem loading is included by default as per Idaho LRFD BDM clause 3.6.1.3.1. The load factor for Extreme I is set to 0.0 as per Idaho LRFD BDM clause 3.4.1.

- **Illinois** - Illinois DOT Bridge Manual, January 2012 - No changes to AASHTO loading.
- **Iowa** - Iowa DOT LRFD Bridge Design Manual, July 2014 - No changes to AASHTO loading.
- **Kentucky** - Division of Structural Design Guidance Manual, as modified by Transmittal Memorandum 08-01-2008 - The HL-93 loads are by increased 25% to create KY-HL-93 loading, as per Transmittal Memorandum 08-01 clause 3.6.1.2. This increase is applied to Fatigue and Deflection loading also.
- **Louisiana** - Louisiana LRFD Bridge Design Manual 1st Ed, September 2008 - The Louisiana Special Design Vehicles specified in LADOTD LRFD Bridge Design Manual Figure 3.1 are included in the Strength II limit state.
- **Louisiana** LRFD Bridge Design Manual 1st Ed, September 2008 (inc BDTM to Feb 2009) - The Louisiana Special Design Vehicles specified in LADOTD LRFD Bridge Design Manual Figure 3.1 are included in the Strength II limit state and the Live Load Factor for Service III Limit State to 1.00 as per BDTM.02.
- **Maryland** - (Email from Maryland SHA) - No changes to AASHTO loading.
- **Massachusetts** - Massachusetts LRFD Bridge Manual, 2009 - The load factor for Extreme I is set to 0.0 as per massDOT Bridge Design Manual clause 3.4.3.2.
- **Michigan** - MDOT Bridge Design Manual, August 2009 - The design tandem is replaced with a single 60 kip load and all loads are multiplied by 1.2 to create HL-93 Mod loading as per MDOT Bridge Design Manual clause 7.01.04. The optional dual tandem is also replaced by an optional dual 60 kip load. Fatigue and Deflection loading is not increased from standard AASHTO loading.
- **Mississippi** - (Email from MDOT) - No changes to AASHTO loading.
- **Missouri** - Missouri LRFD Bridge Design Guidelines - No changes to AASHTO loading.
- **Montana** - Montana Structures Manual, August 2002 - No changes to AASHTO loading.
- **Nebraska** - NDOR BOPP, April 2014 - The load factor for Strength I is increased from 1.75 to 2.0 as per NDOR BOPP Clause 2.2.2.
- **New Hampshire** - NHDOT Bridge Design Manual v2.0 (January 2015) - No changes to AASHTO loading.
- **New Mexico** - NMDOT Bridge Procedures and Design Guide, April 2013 - The Permit Vehicle P327-13 is included in the Strength II limit state as per NMDOT Bridge Procedures and Design Guide clause 3.1.13. NB Figure 3.1B plan view shows four wheels of 6.25kip each; transverse spacing between wheel pairs is defined as 6ft but no transverse spacing within the pairs is defined. AASHTO LRFD clause C3.6.1.2.5 gives tyre contact width (in inches) as P(in kip)/0.8 = 6.25/0.8 = 7.8125in. P327-13 has therefore been implemented with 7.8125in transverse spacing between wheels in a pair.
- **North Carolina** - North Carolina Structure Design Manual, February 2014 - No changes to AASHTO loading.

- **North Dakota** - North Dakota DOT Design Manual, July 2011 - No changes to AASHTO loading.
- **Ohio** - Ohio Bridge Design Manual, July 2007 - The load factor for Extreme I is set to 0.0 as per Ohio Bridge Design Manual clause 1003 - S3.10.9.2.
- **Oregon** - Oregon Bridge Design and Drafting Manual, October 2013 - The permit trucks ODOT OR-STP-5BW and ODOT OR-STP-4E are included in the Strength II limit state as per Oregon Bridge Design and Drafting Manual clause 1.3.2.
- **Pennsylvania** - PennDOT Design Manual - Part 4, May 2012, with implementation notes as shown below:

    - Combinations Strength IP, Extreme III, Extreme IV, Service I (with PL), Service IIB, Service III (with PL), Service IIIA and Service IIIB are included as per PennDOT Design Manual Part 4 Part B clause 3.4
    - The design tandem's axles are increased to 31.25kips as per PennDOT Design Manual Part 4 Part B clause 3.6.1.2.3
    - The P-82 permit load as per PennDOT Design Manual Part 4 Part B clause 3.6.1.2.7P is included in combinations Strength II, Extreme III, Extreme IV, Service IIB, Service IIIA and Service IIIB. It is available in one lane with PHL-93 in other lanes if "Include associated notional design load (as above) in other lanes" is checked, or in every lane if the option is unchecked. The P-82 permit load has its own dynamic load allowance setting on the Advanced settings dialog
    - An option is included for "Truck loading on continuous spans" with gives the option of "100% (for moments)" or "90% (for reactions)". This option only affects the pair of design trucks; if a single truck is more onerous it will always be at 100%. The pair of tandems is similarly unaffected. NB Loading for continuous spans is only applicable if the "Alternative load pattern" option is selected, per influence, on the influence selection dialog

- **Rhode Island** - Rhode Island LRFD Bridge Design Manual, January 2007 - The live load for deflection is replaced by the most onerous of 125% of the Design Truck or 33% of the Design Truck with the Design Lane as per LRFD Bridge Design Manual clause 3.4.5. The load factor for Extreme I is set to 0.0 as per LRFD Bridge Design Manual clause 3.2.3.
- **South Carolina** - South Carolina Bridge Design Manual, April 2006 - No changes to AASHTO loading.
- **South Dakota** - (Email from SD DOT) - The SD agency vehicles SD Strength II Long and SD Strength II Short are included in the Strength II limit state.
- **Tennessee** - (Email from Tennessee DOT) - No changes to AASHTO loading.
- **Texas** - Texas DOT Bridge Design Manual, LRFD, March 2013 - No changes to AASHTO loading.
- **Utah** - Design Memorandum 09-2010 - No changes to AASHTO loading.

- **Vermont** - VTrans Structures Design Manual, 2010 - No changes to AASHTO loading.
- **Virginia** - VDOT Modifications to AASHTO LRFD 6th Ed, January 2013 - No changes to AASHTO loading, but note that the Multiple Presence Factors may need manually editing via the Advanced Settings Dialog for specific structural elements and spans, to meet the requirements of VDOT Modifications Section 3.
- **Washington** - WSDOT Bridge Design Manual, August 2012 - Two tandem loading is included by default as per WSDOT Bridge Design Manual clause 3.9.1
- **West Virginia** - WVDOH Bridge Design Manual, March 2004 - No changes to AASHTO loading.
- **Wisconsin** - Wisconsin Bridge Manual, January 2014 - The Wisconsin Standard Permit Vehicle (Wis-SPV) from WisDOT Bridge Manual Figure 45.6-1 is included in the Strength II limit state.
- **Wyoming** - WYDOT Bridge Design Manual, April 2013 - No changes to AASHTO loading.

United States of America - AASHTO Standard Specifications (17th Edition) Optional Code Settings

This design code loading is supported by the LUSAS Traffic Load Optimisation software option.

When using the country “United States of America” and Design code “AASHTO Standard Specifications 17th Ed”, road traffic loading is generated with reference to AASHTO Standard Specifications for Highway Bridges, 17th Edition - 2002.

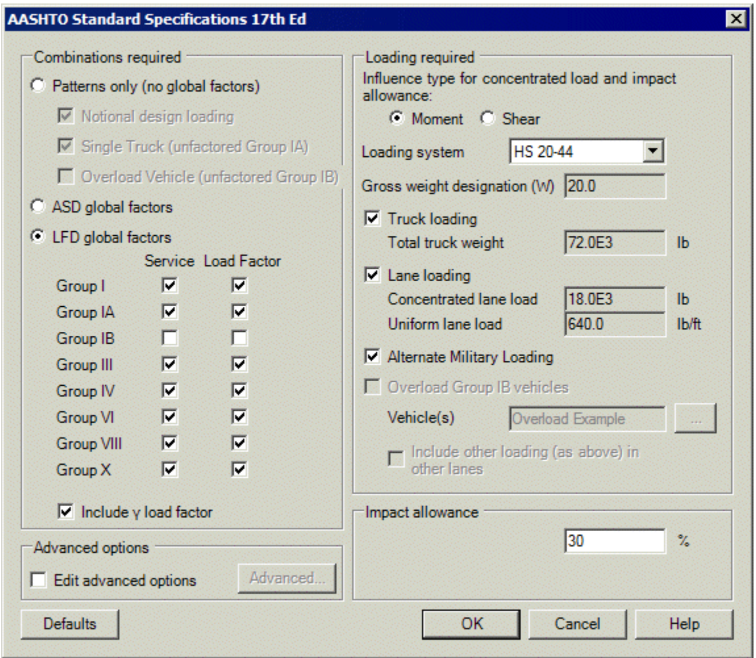
In the Design code droplist you can also select one of many State Design manuals. See **State Implementations of AASHTO Standard Specifications** for more details.

The scope of the vehicle load optimisation to AASHTO Standard Specifications is restricted to vertical highway traffic loading and global effects. Horizontal components, railway loads and pedestrian loads are not currently included.

The roadway width is divided into 12ft design lanes (apart from roadways from 20ft to 24ft wide which have two design lanes of equal width). This will result in a “remaining width” for any roadways of width not an integer multiple of 12ft. The lanes are positioned transversely on the bridge to produce the most adverse effects, with the remaining width being placed either side of the lanes or between any of the lanes so as to produce the most onerous arrangement. The number of design lanes can be modified if required, via the Advanced Settings dialog. Loading moves transversely within the design lanes for most onerous effect.

Attention should be paid to the value set for “Minimum distance between concentrated loads on continuous spans” to ensure that the two concentrated loads are placed for maximum effect. See **AASHTO Standard Specifications 17th Edition Advanced Settings** for more information.

AASHTO Standard Specifications 17th Edition Optional Settings



The Optional Code Settings dialog allows selection of the combinations and loads required. Factors used in the calculations and other settings may be viewed and modified if required, by accessing the **Advanced Settings** dialog.

All clauses referenced below are taken from AASHTO Standard Specifications 17th Edition unless noted otherwise.

Combinations required

Loading patterns are available either globally factored (i.e. with a Load Factor and Coefficient for a Combination to Table 3.22.1A, or optionally edited in the **Advanced Settings** dialog) or unfactored as follows:

- **Patterns only (no global factors)**
  - Notional design load - loading to clause 3.7
  - Single Truck (unfactored Group IA) - loading to clause 3.5.1
  - Overload Vehicle (unfactored Group IB) - loading to clause 3.5.2

- **ASD global factors**

  - Service for: Group I / Group III / Group IV / Group VI / Group VIII / Group X - as per Notional design load pattern but with the relevant β factor (and, if checked below, γ factor) from Table 3.22.1A
  - Service for: Group IA - as per Single Truck pattern but with the relevant β factor (and, if checked below, γ factor) from Table 3.22.1A
  - Service for: Group IB - as per Overload Vehicle pattern but with the relevant β factor (and, if checked below, γ factor) from Table 3.22.1A

- **LFD global factors**

  - As ASD global factors, but β and γ factors are available for both Service and Load Factor combinations

The dialog allows selection of one or more combinations. The most onerous traffic loading pattern appropriate to each selected combination will be determined, with load factors included as appropriate.

Loading required

- **Influence type** The choice of "Moment" or "Shear" affects the required Concentrated lane load.
- **Loading system** The four standard classes of highway loading to clause 3.7.2 are available, as are options for user-defined gross weight designation to either clause 3.7.5 using "H (W)" or clause 3.7.6 using "HS (W)". When a State Implementation specifies "HS 25" loading, this is also made available for selection.
- **Truck loading / Lane loading / Alternate Military Loading** These load types can be removed from the optimisation if desired. Feedback is given on the weight of the truck and lane loads, based on the selections made above.
- **Overload Group IB vehicles** This option is available when any of the Group IB combinations is selected.

  - **Vehicle(s)** Clicking the "..." button allows selection of one or more Overload vehicles. As no Overload vehicles are defined within AASHTO Standard Specs 17th Ed, an example vehicle is provided. State implementations have relevant vehicles as defined by their State Bridge Design Manuals, as appropriate.
  - **Include other loading (as above) in other lanes** Unless otherwise noted, Overload vehicles are assumed to occupy one lane only. If this option is checked, Truck loading, Lane loading and Alternate Military Loading will be allowed in other lanes, provided their options are checked. Note that if the Truck/Lane/Alternate Military Loading is more onerous than the Overload vehicle, the load pattern generated will not include the Overload vehicle. If this

option is unchecked, the load pattern generated will be for the Overload vehicle alone.

Impact allowance

- Impact allowance is applied to all loads as per clause 3.8.1. The same allowance is applied to all loads.

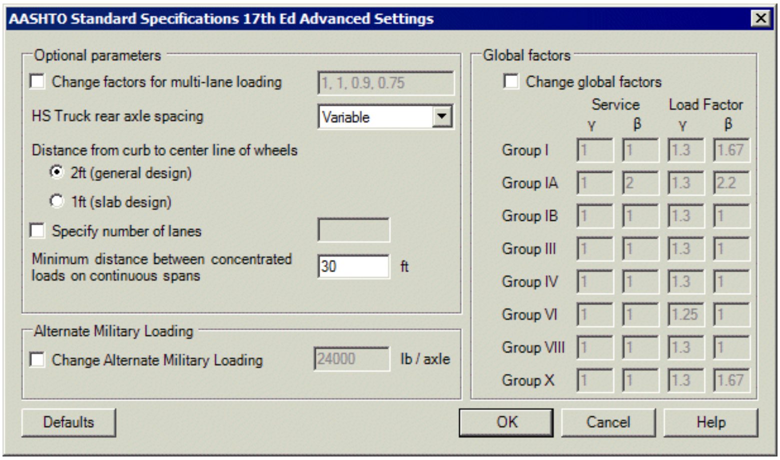
When multiple lanes are loaded, all loads (including any Overload vehicle) are reduced in intensity as specified in clause 3.12.1. The optimisation process takes account of this reduction when calculating the most onerous load effect, such that the most onerous pattern of load may be where the number of lanes loaded is fewer than the maximum available.

Two concentrated loads, for lane loads on continuous spans to clause 3.11.3, are included when the “Include additional load patterns” option is selected on the Influence Surface dialog.

A Defaults button is provided to reset all values back to those specified in AASHTO Standard Specs 17th Edition (or the selected State Bridge Design Manual).

AASHTO Standard Specifications (17th Edition) Advanced Code Settings

This dialog is used to view and modify global factors and a number of other settings.



Optional parameters

- **Change factors for multi-lane loading** This option allows the modification of the factors used to reduce load intensity when multiple lanes are loaded. The default values are based on clause 3.12.1, with the percentage values converted to factors by dividing by 100. The textbox has a comma separated list of four variables, applicable respectively to: one lane loaded, two lanes loaded, three lanes loaded, more than three lanes loaded.

- **HS Truck rear axle spacing** The HS Truck from Figure 3.7.7A has an axle spacing which varies from 14ft to 30ft; this is implemented by default. This option gives the opportunity to limit which axle spacings are considered in order to give a faster optimisation process. Axle spacing options should only be restricted when engineering judgement deems this will give a sufficiently accurate result.
- **Distance from curb to center line of wheels** As per clause 3.24.2 and footnotes to Figures 3.7.6A and 3.7.7A, the wheel loads should usually be restricted to 2ft from the edge of the design lane / curb, but should be allowed within 1ft of the curb for slab design.
- **Specify number of lanes** By default the carriageway is divided into standard 12ft design lanes, or two equal width lanes for carriageways from 20ft to 24ft wide, as per clause 3.6. This option is provided to allow the number of design lanes to be specified if their width is less than 12ft. If this option is selected the carriageway will be divided into the specified number of equal width lanes, although the lanes may not be less than 10ft wide due to the width of the standard truck and lane loads.
- **Minimum distance between concentrated loads on continuous spans** Clause 3.11.3 allows two concentrated loads which must be in separate spans. This is currently implemented by requiring a minimum distance between the two loads. The required minimum distance to keep them in separate spans, while not forcing them so far apart that they cannot be placed at the local peak adverse influence ordinates, varies depending on span lengths and their ratios to each other. If the defined minimum spacing is too small for a given bridge, VLO may place both loads in the same span. This is incorrect but conservative. If the defined minimum spacing is too large for a given bridge, VLO will not be able to place both loads on the peak influences of their spans. This is incorrect and unconservative. For typical span ratios, a value of 2/3 of the maximum span is usually appropriate.

Alternate Military Loading

- **Change Alternate Military Loading** The default for the Alternate Military Loading is 24000 pounds per axle, as per clause 3.7.4. This option allows the value to be edited.

Global factors

- **Change global factors** The default values provided to Table 3.22.1A may optionally be edited. NB Irrespective of what is entered on the Advanced Dialog, γ factors can be removed by unchecking the option of the Code Settings Dialog.

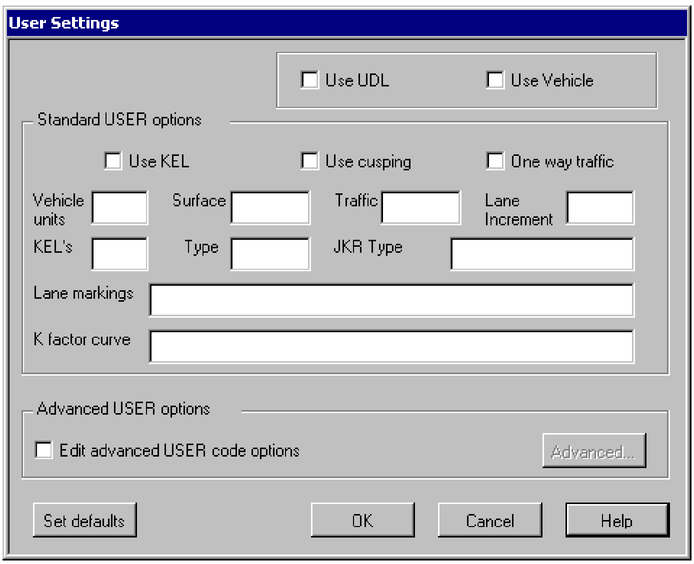
A Defaults button is provided to reset all values back to those specified in AASHTO Standard Specs 17th Ed (or the selected State Bridge Design Manual).

State Implementations of AASHTO Standard Specifications (17th Edition)

Many States' DoTs publish a Bridge Design Manual or equivalent, which may clarify, add to, or modify the requirements of AASHTO Standard Specs 17th Ed. The list below contains those States whose manuals are currently implemented in LUSAS, along with details of any additions or modifications. If optimisation to a manual not listed below is desired, please contact LUSAS Support to request its implementation.

- **Alaska** - Alaska Highway Preconstruction Manual, November 2013 - The design live load is HS-25 as per Alaska Highway Preconstruction Manual clause 1120.3.2.
- **Arizona** - Arizona DOT Bridge Practice Guidelines Section 3, June 2002 - No changes to AASHTO loading
- **Arkansas** - Arkansas Highways Interoffice Memorandum "Live Load for Bridges", October 1984 - No changes to AASHTO loading
- **Colorado** - Colorado DOT Bridge Design Manual Section 3, November 1999 - The Colorado permit Vehicle from Colorado DOT Bridge Design Manual Subsection 3.2 is included. It is available from the Vehicle selection dialog as either “every lane” or “single lane”. The “single lane” vehicle is compatible with the “Include other loading in other lanes” optionsdf
- **District of Columbia** - D.C DoT Design and Engineering Manual, April 2009 - The design live load is HS25 as per D.C DoT Design and Engineering Manual clause 15.1
- **New Hampshire** - New Hampshire DOT Bridge Design Manual, October 2000 - The design live load is HS-25 as per New Hampshire DOT Bridge Design Manual clause 602.1. The Alternate Military Loading is 30000 pounds per axle as per New Hampshire DOT Bridge Design Manual clause 602.1.
- **Puerto Rico** - Puerto Rico Highway Design Manual (El Manual de Diseño de Carreteras), 1979 - No changes to AASHTO loading

User Settings

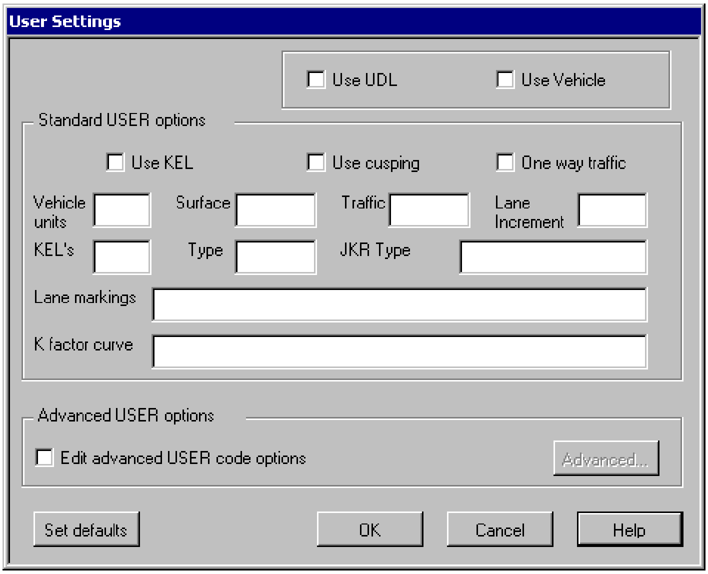


The following options are available for selection:

- **Use UDL -** specifies whether to apply the HA loading (which includes KEL loading). To use this option select the check box next to the label, to deselect this option ensure that the check box is not selected.
- **Use vehicle** - specifies whether to apply an abnormal vehicle. If selected, the abnormal vehicles are applied to the carriageway. If not selected, then HA loading only is applied.
- **Use KEL** - specifies whether to apply Knife Edge Loads. If selected, Knife Edge Loads are applied to areas with UDL loading, according to the standard in use. If not selected, no KEL'S are applied.
- **Use cusping** - specifies whether to apply cusping when working out loadable areas. If selected, cusping is applied. If not selected, cusping is not applied.
- **One way traffic** - specifies that a bridge carrying traffic in one direction only would result in the doubling of the N value in the calculation of the HA lane. If ONEWAY is selected, then the value of N used in the table is doubled.
- **Vehicle units** - specifies the number of HB Units to use. In cases where an abnormal vehicle uses HB Units, the values of the weights of the axles are multiplied by the number of HB Units.
- **Surface** - allows the user to specify the quality of road surface, as defined in the BD 21/97 standard, as **Good** or **Poor.** This option has no effect in any of the other standards. If an invalid value is specified Autoloader gives a warning and uses the default value.
- **Traffic flow** - specify the traffic level as defined in the BD 21/97 standard as **High**, **Medium** or **Low**. This option has no effect in any of the other standards. If an invalid value is specified Autoloader gives a warning and uses the default value.
- **Lane increment** - specifies the increment interval used when moving lanes across the carriageway within the JKR standard. The smaller the increment, the more accurately the position of the vehicle will be calculated on the carriageway, this gives more accurate results. A larger increment gives quicker Autoloader runs. The user must choose through experience an appropriate value. If an invalid value is specified Autoloader gives a warning and uses the default value.
- **KEL's** - specifies the number of KEL's to apply within a specified design lane. This will generally be used when AUSTROADS is applied, but can also be used within the other standards. If an invalid value is specified Autoloader gives a warning and uses the default value.
- **Type** - allows the user to specify either an urban road or a rural road, as defined in the AUSTROADS standard. This option has no effect in any of the other standards. If an invalid value is specified Autoloader gives a warning and the default value is used.
- **JKR Type** - specify either controlled or uncontrolled vehicle movement, as defined in the JKR standard. This option has no effect in any of the other standards. If an invalid value is specified Autoloader gives a warning and uses the default value.

- **Lane markings** - define marked traffic lanes in relation to the carriageway(s). Each set of values consists of a carriageway number and two sets of x and y values, through which the lane edges pass. The traffic lane is assumed to be parallel to the carriageway kerbs.
- **K factor curve** - change the relationship between loaded length and the K factor. Each pair of values consists of a length and a factor for that length. Autoloader performs linear interpolation between these values. If a more accurate value is required as many points as possible should be defined.
- **Edit advanced User code options** - further to the basic code options provide on this dialog Autoloader allows more advanced changes to the code to be made. To access this functionality ensure that the check box next to the label is selected and click on the Advanced button.

User Advanced Settings



The following User advanced options are available for selection:

- **File containing Alternative vehicle** - defines a list of vehicles to be tried as an alternative to HA loading. When loading an HA lane, Autoloader tries to place vehicles from this list within the lane, and if the effect is greater than HA loading, uses that vehicle instead. Each vehicle must be specified in the Autoloader vehicle library.
- **SLS Load factors** - specify the partial load factors for the Serviceability Limit State
- **ULS Load factors** - specify the partial load factors for the Ultimate Limit State.
- **SL** - specify the partial load factors for the Service Load State.

- **LF** - specify the partial load factors for the Load Factor Design.
- **Lane width** - specifies the L44 lane width. If an invalid value is specified Autoloader gives a warning and uses the default value.
- **Pedestrian load** - specify an alternate value for the Pedestrian Load.
- **Lane width table** - Each 3 values are taken as a lower limit, an upper limit, and a number of notional lanes. After calculating the carriageway width, Autoloader works its way down the table, checking the calculated width against the values in the table.

Construction Tables

Overview

The construction table facility can be used to produce Camber, Displacement History, and Incremental Displacement tables of results for selected locations (points) on the model, and for any specified results loadcases. It is available for selected LUSAS Civil & Structural and LUSAS Bridge software products only. It is accessed from the **Civil >** or **Bridge > Construction Table...** menu item.

Prior to selecting the Construction Table menu item, points of interest need to be selected on the model. The construction table facility calculates values for the nodes associated with those selected points, in the order that the points were selected. If the specified loadcases include time dependent effects (e.g. creep), the time at which the values are measured will also be included.

Selecting an analysis

- **Analysis** The analysis containing the loadcases defining the construction stages must be selected. It is not possible to include loadcases from more than one analysis in the construction table.

Table types supported

- **Camber table to target stage** Camber tables are the reverse of Displacement History tables (see below), and are used to set-out a structure towards a target (as-built) profile while accounting for effects during construction, such as dead-load deflection, creep and concrete shrinkage. They can also be used to ensure the deflection of a structure under live-load does not encroach inside a clearance zone (typically referred to as pre-camber). A target (zero deformation) construction stage (loadcase) within the selected analysis must be specified from the drop-down list of those available for the chosen analysis. Where loadcases include nonlinear time-steps it is assumed that the final time-step is representative of the loadcase defining that construction stage as a whole.The values of the Camber table are absolute with reference to an undeformed geometry 'datum'.
- **Displacement history** A Displacement History table reports the absolute displacement of a series of key-points at each stage with reference to an undeformed geometry 'datum'. This information may be used, for example, when constructing reinforced earthwork ramps, to determine the settlement at each rise and the required flexibility of transition slabs, or the total displacement of specified points throughout the construction of a bridge after each stage.

- **Incremental displacements** An Incremental Displacement history table reports the stage-by-stage deformation of a structure. It tabulates the relative displacements between each construction stage, which can then be used in setting-out of structures built using sequential construction methods.

See **Construction Tables Explained** for more details.

Filter options

- **Whole only** reports on deformations for all loadcases in a selected analysis.
- **Specified loadcases** provides the means to specify a subset of sequential, non-repeating loadcases with results from the current analysis to include as stages within the table. Where loadcases include nonlinear time-steps it is assumed that the final time-step is representative of the loadcase defining that construction stage as a whole.
- **Only report vertical axis** reports only on deformations in the **vertical axis** direction for the selected point (and hence node) locations (if checked), or reports on deformations in all three axes (if unchecked).

Notes and limitations

- Loadcase stages selected for inclusion within a construction table must be sequential and cannot repeat.
- The camber table is effectively the inverse of the displacement history. In a simple example, the target geometry of a bridge is cambered upward, so that the final as-built shape is notionally flat: the upward camber is the inverse of a calculated sag from the analysis. However, the approach has limitations and care should be taken when using the Camber Table for structures with nonlinearity. For example, if a beam buckled to the left during the analysis, the Camber Table would suggest a target geometry with that beam heavily cambered to the right. This would not be an appropriate use of the Camber Table function. Similarly, lift-off during construction needs to be considered carefully, since it may present construction risks, and a Camber Table based on an analysis with lift-off would probably not give appropriate target geometries. However, in structures with taut cables or with certain material nonlinearities, for example, the use of the camber table may be acceptable, understanding that it will be an approximation.

Construction Tables Explained

Three different construction tables can be created: Camber, Displacement History, and Incremental Displacement.

Camber table

Camber tables are used to set-out a structure towards a target (as-built) profile while accounting for effects during construction, such as dead-load deflection, creep and concrete shrinkage. Displacement values in a Camber table are absolute values with reference to an undeformed geometry 'datum'. They are effectively the reverse of Displacement History tables (see below).

By definition a Camber table means that the currently undeformed mesh (the model geometry) must be displaced to achieve the same geometry at a later stage. As a result an undeformed mesh 'Datum' is always added to the table.For the Camber table shown below, the target stage was defined as Stage 3. For this, the table was calculated to achieve the undeformed mesh geometry on the 180th day (end of Stage 3).

LUSAS View: Camber

		Point IDs	1	2	4	6	8
		Point IDs	1	2	4	6	8
		Node IDs	1	13	25	37	49
Stage ID	Stage ID	Time					
Datum	Datum		0.0	0.188571			
1:Stage 1: 0 to 60 Days	1:Stage 1: 0 to 60 Days	60.0	0.0	0.124137	0.346919		
2:Stage 2: 60 to 120 Days	2:Stage 2: 60 to 120 Days	120.0	0.0	0.0579217	0.17582	0.350065	
3:Stage 3: 120 to 180 Days	3:Stage 3: 120 to 180 Days	180.0	0.0	0.0	0.0	0.0	
4:Stage 4: 180 to 240 Days	4:Stage 4: 180 to 240 Days	240.0	0.0	-0.0468157	-0.149688	-0.331653	-0.602778
5:Long-term: 240 to 1000 Days	5:Long-term: 240 to 1000 Days	1.0E3	0.0	-0.211046	-0.539432	-1.02585	-1.71095

Model info | Camber

Notes

- Row 4 of the table (the Datum row) specifies the geometry at the start of Stage 1 (0 days)
- Row 5 and 6 specify the target geometry at the start of Stage 2 and Stage 3 respectively
- Rows 7, 8 and 9 specify the resulting geometry at the end of Stage 3, Stage 4 and Long-term respectively.
- Displacements are shown if a node is active in the following stage prior to the target, and the current stage for both the target and those after.

Displacement History table

A Displacement History table reports the absolute displacement of a series of key-points at each stage with reference to an undeformed geometry 'datum'. This information could be used, for example, when constructing reinforced earthwork ramps, to determine the settlement at each rise and the required flexibility of transition slabs, or the total displacement of specified points throughout the construction of a bridge after each construction stage.

The Displacement History table below shows displacement values in all three axes on a model with time-steps. Where a node associated with a point is deactivated during a loadcase no displacement is reported and the respective cell is left blank. If only particular loadcases were specified for inclusion (as opposed to listing results for the whole analysis) the results for each excluded stage will be omitted.

LUSAS View: Displacement History

		Point IDs	1	1	1	2	2	2	4	4	4	6
		Point IDs	1	1	1	2	2	2	4	4	4	6
		Node IDs	1	1	1	13	13	13	25	25	25	37
Stage ID	Stage ID	Time	X	Y	Z	X	Y	Z	X	Y	Z	X
1:Stage 1: 0 to 60 Days	1:Stage 1: 0 to 60 Days	60.0	0.0	0.0	0.0	0.0229145	-0.0644341	0.0				
2:Stage 2: 60 to 120 Days	2:Stage 2: 60 to 120 Days	120.0	0.0	0.0	0.0	0.108715	-0.130649	0.0	0.330909	-0.242645	0.0	
3:Stage 3: 120 to 180 Days	3:Stage 3: 120 to 180 Days	180.0	0.0	0.0	0.0	0.214447	-0.188571	0.0	0.76296	-0.418465	0.0	1.4792
4:Stage 4: 180 to 240 Days	4:Stage 4: 180 to 240 Days	240.0	0.0	0.0	0.0	0.307651	-0.235387	0.0	1.1787	-0.568153	0.0	2.5148
5:Long-term: 240 to 1000 Days	5:Long-term: 240 to 1000 Days	1.0E3	0.0	0.0	0.0	0.352284	-0.399617	0.0	1.39757	-0.957896	0.0	3.0288

Model info | Displacement History

Incremental Displacement table

An Incremental Displacement history table reports the stage-by-stage deformation of a structure. It tabulates the relative displacements between each construction stage, which can then be used in setting-out of structures built using sequential construction methods.

In the Incremental Displacement table example shown below, the stage identified for each row is the 'to' loadcasse, with the previous row defining the 'from' loadcase for the increment. In the case of the first stage row, the displacement increment is 'from' the undeformed mesh. This is true even if particular loadcases are not specified for inclusion, as shown where Row 6 reports the displacement from Stage 2 to Stage 4. Associated nodes are considered active (and therefore valid for inclusion) if they are active in either of the 'from' or 'to' loadcase stages.

LUSAS View: Incremental Displacement

		Point IDs	1	1	1	2	2	2	4	4	4	6
		Point IDs	1	1	1	2	2	2	4	4	4	6
		Node IDs	1	1	1	13	13	13	25	25	25	37
Stage ID	Stage ID	Time	X	Y	Z	X	Y	Z	X	Y	Z	X
1:Stage 1: 0 to 60 Days	1:Stage 1: 0 to 60 Days	60.0	0.0	0.0	0.0	0.0229145	-0.0644341	0.0				
2:Stage 2: 60 to 120 Days	2:Stage 2: 60 to 120 Days	120.0	0.0	0.0	0.0	0.0858001	-0.0662152	0.0	0.272438	-0.171099	0.0	
4:Stage 4: 180 to 240 Days	4:Stage 4: 180 to 240 Days	240.0	0.0	0.0	0.0	0.198937	-0.104737	0.0	0.847795	-0.325509	0.0	1.943
5:Long-term: 240 to 1000 Days	5:Long-term: 240 to 1000 Days	1.0E3	0.0	0.0	0.0	0.044633	-0.16423	0.0	0.218868	-0.389743	0.0	0.514

Model info | Incremental Displacement

Notes applicable to all tables

- The first two rows of each table always show the ID of the selected point and the ID of the implied node that the column refers to.
- The first column of each table contains the Loadcase ID of the loadcase used to form the stage, followed by the name of the loadcase.

- If an analysis has a time component (as it does for the example tables shown) an additional Time column will be prepended to the start of the displacement column data. This will be left blank if an analysis has selected loadcases without a time component.
- Where the table is not limited to displacements in the vertical axis direction an additional header row will be included where axes labels related to the global direction of displacement for that column.
- Table values are reported in model units, with the exception of any time-based values.

Fixed column and row headings (duplicating the information inside the first column and row) are provided for assistance when viewing and scrolling large amounts of data in LUSAS, or when copied to a spreadsheet for third-party graphing purposes.

Cable Tuning Analysis

Overview

A cable tuning analysis calculates load factors for selected lines in a model that represent cables in order to achieve defined target values set for various feature types and results component. Lines representing cables should be assigned a bar or beam element with only one mesh division.

A cable tuning analysis can be set-up by use of the **Analyses > Cable Tuning Analysis** menu item. This is provided in selected Bridge and Civil & Structural software products only. By using the tabbed dialog presented, lines in the model that represent cables are selected for inclusion in the cable tuning analysis, a solution method chosen, and model or results loadcases specified to try and achieve target values that are defined for particular components or features. An 'exact' method, an optimisation facility and two best-fit solution methods are available. The Cable Tuning dialog is functionally similar to the Target Values dialog, with the exception that a Cables page is included to allow the selection and inclusion of lines representing cable stays.

Use of loadcases in cable tuning analysis

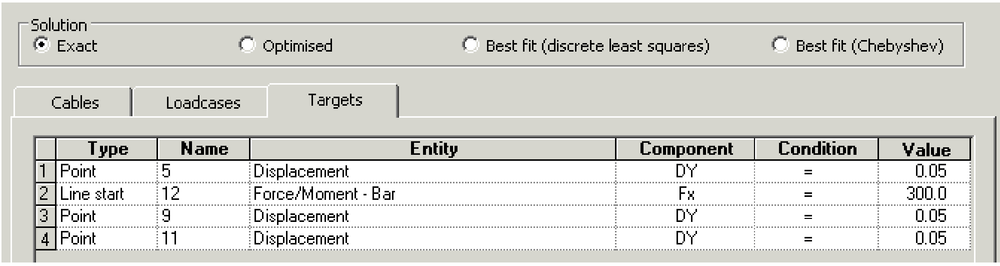
In a cable tuning analysis a unit tensile loadcase is automatically generated for any line selected to represent a cable stay. These loadcases will be factored by the optimisation algorithm to try an achieve a solution. Structural loading can also be applied as part of a cable tuning analysis. The effect of these loads on the cables will also be considered when achieving a target solution.

A cable tuning analysis can be solved independently from any other analyses defined within the model.

When solved, cable tuning results are held in a separate Cable tuning analysis entry in the Analyses Treeview. This contains a folder with the individual solved and generated loadcases for each table, and a results combination representing the achieved target profile. Note that the loadcase naming convention used is automated and cannot be edited, however it indicates the line representing the cable is factored. Double-clicking a Cable tuning analysis entry (or selecting Edit from its context menu) will show the settings used to create it.

Solution types

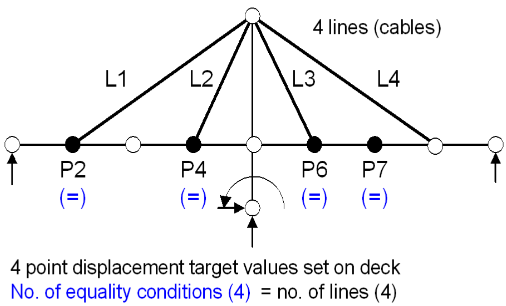
The choice of solution type can be made on any of the cable tuning dialog pages.



	Type	Name	Entity	Component	Condition	Value
1	Point	5	Displacement	DY	=	0.05
2	Line start	12	Force/Moment - Bar	Fx	=	300.0
3	Point	9	Displacement	DY	=	0.05
4	Point	11	Displacement	DY	=	0.05

The type of solution that can be chosen is dependent upon the number of lines selected to represent cables and the number of equality conditions specified (as seen in the Condition column on the Targets grid). An equality condition can be set for either a point or the start or end of a line. The solution type can be set to be:

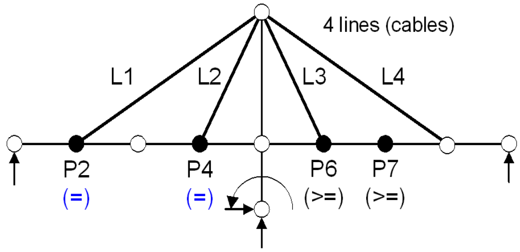
- **Exact**. This requires the number of equality conditions (as seen in the Condition column on the Targets grid) to be the same as the number of lines (representing cables) present in the Included panel on the Cables page. No inequality conditions can be defined for the exact method. Using this option, only one solution is possible.



See **Case study: Cable Stay Analysis** for a basic procedure using the Exact method.

Note that the exact solution for a given set of target values may produce negative factors for some cable and loading arrangements. If this occurs an optimised solution should be investigated where all of the points of interest are permitted to move a small distance to try and achieve a better solution. In the case of Points being used to restrict displacement in a bridge deck (as shown in the accompanying images) this would amount to specifying a distance that would slacken the cables.

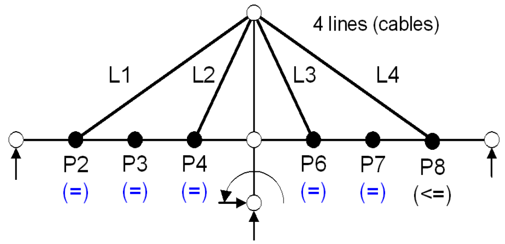
- **Optimised**(an under-determined solution) requires the number of equality conditions to be less than the selected number of lines (representing cables) present in the Included panel on the Cables page that are marked as having a Calculated factor. Any number of targets specifying inequality conditions can be defined. For this option a range of solutions is possible provided that they are not mutually exclusive.



4 point displacement target values set on deck
No. of equality conditions (2) < no. of lines (4)

Options to minimise or maximise the calculated factors for a variety of in-built or specified criteria are provided on the Optimisation criteria page..

- **Best fit (discrete least squares)** and **Best fit (Chebyshev)**(both over-determined solutions) require the number of equality conditions to be greater than or equal to the selected number of lines representing cables in the Included panel on the Cables page that are marked as having a Calculated factor. Any number of targets specifying equality or inequality conditions can be defined. For this approach no unique solution is guaranteed.



6 point displacement target values set on deck
No. of equality conditions (5) > no. of lines (4)

- **Discrete least squares** provides positive and negative load factors. No inequality conditions can be stated for this option.
- **Chebyshev** will always produce positive load factors. Inequality conditions can be stated for this option.

Related pages

The cable tuning dialog pages **Cables**, **Loadcases**, **Targets** and **Optimisation criteria** should be visited to specify all data required for a cable tuning analysis.

Validating cable tuning parameters

- **Validate input** The Validate Input button should be used to check if the number of equality conditions and the number of lines representing cables are valid for the chosen Solution method. Three different outcomes exist.

  - Input is valid - the analysis can proceed.

- An error message regarding an imbalance in the equality conditions and the number of lines representing cables will be displayed requiring correction.
- The message "Exact solution type required" will appear indicating that, because the number of equality conditions and number of lines representing cables is the same, an optimised or best fit solution cannot be obtained. An Exact solution should be used instead.

Identifying a feature in the grid

- ❑ **Identify selection** locates a selected feature (a point or a line) in a model. When the row for a feature is selected in the grid on the dialog, and this checkbox is 'on', the corresponding feature (point/line) is located in the View window by animating concentric shrinking squares.

Saving cable tuning parameters

When the **OK** or **Apply** button is pressed all input is saved, even if it contains errors/inconsistencies. If the input was valid a cable tuning analysis entry will be added to the Analyses Treeview. A Cable tuning Results entry will also be added. If the input is invalid the invalid data sign icon will appear instead, and corrective measures will be needed.

Solving a cable tuning analysis

To solve a cable tuning analysis choose the **Solve Now** menu item on its context menu or press the Solve button on the main toolbar menu. A Stress and Strain loadcase with a unity value is automatically created in the Analyses Treeview and is assigned to the all lines in the model that were selected to represent the cables, this represents a 'unit load' used for optimisation

Viewing results of a cable tuning analysis

- Right-click on the Cable tuning Results entry and select **Set Active** to view results.
- Double-click on the Cable tuning Results entry to display the load factors calculated to achieve (or achieve as near as practicable) the specified target values.

Cable tuning results options

The context menu for a Cable tuning Results entry contains the following menu items:

- ❑ **Calculated Factors** displays the load factors calculated for each loadcase (and hence each line representing a cable) in order to achieve the target values. All entries in the grid are read-only.

- **Create Combination** Creates a combination containing the loadcases used in the Cable tuning results loadcase, with each loadcase factored by the values calculated by the cable tuning analysis. This allows suitable further analysis to be carried out. The menu item is greyed out unless the loadset has been solved.
- **Create Loadcase** Creates a loadcase that could be used in a nonlinear analysis. This loadcase contains the loadings used in the Cable tuning results loadcase, with each loading factored by the values calculated by the cable tuning analysis. The menu item is greyed out unless the loadset has been solved.

Superposition

When a cable stay analysis is carried out, LUSAS cannot verify that superposition holds when combinations or envelopes contain loadcases from different analyses or when nonlinear behaviour is being considered. This may occur if lift-off supports are present, or when geometric or material changes take place between analyses.

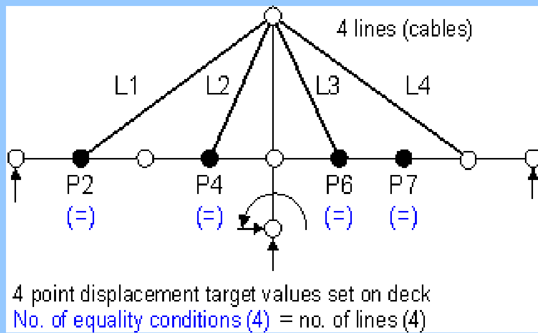
Converting cable tuning analysis data into target values data

The context menu for the cable tuning analysis entry contains the following menu item:

- **Create target values** - this converts the cable tuning loadset data into an equivalent target value loadset. This can then be used for further analysis and investigation using additional criteria. See **Target Values** for more details.

Case Study: Cable Stay Analysis

The basic procedure to carry out a cable stay analysis of a simple 4-stay bridge using the **Exact** method is as follows:



1. Select the same number of points (4) on a bridge deck as lines representing cables of interest (4) and add the lines representing the cables to the point selection.
2. Select **Analyses> Cable Stay Analysis.**
3. On the Cables tab press the **All** button include all the lines from the selection.
4. On the Loadcases tab press the Add to >> button to include all loadcases to be used in the target value analysis.
5. On the Targets tab press the **Add Selected** button to add the 4 points previous selected.
6. Specify target criteria for those points. For example **Displacement**, **DZ**, **0.05**.
7. Check the **Exact** solution is requested and press the **Validate** button to check the input is suitable.
8. Solve the model.
9. Right-click on the Cable tuning Results entry and choose **Calculated Factors** to view the load factors required for each cable to achieve the specfied target values.
10. (Optional) Right-click on the Cable tuning Results entry and choose **Create loadcase** to create a single loadcase with loadings for each cable factored according to the calculated values.

Note that the points (and line start/line ends) selected to define target value criteria can be for any part of a structure.

Worked Example

The worked example 'Cable Stay Bridge' in the *Application Examples Manual (Bridge, Civil and Structural)* shows how linear cable tuning analysis is carried out for a simple structure.

Prestress Loading

Overview

The single and multiple tendon prestress wizards calculate equivalent nodal loading due to post tensioning and assign these forces automatically (and using **search areas)** to selected lines (and hence nodes and elements) of the model for the current active loadcase.

- The single tendon prestress wizard supports beam, tendon, plane stress and solid element modelling of concrete.
- The multiple tendon wizard only supports beam element modelling of concrete.

Design codes supported

For both the single and multiple tendon prestress wizards the computation of tendon forces can be carried out in accordance with AASHTO-LRFD (2nd, 5th, 6th and 7th Editions), BS5400-4:1990, Eurocode EN1992 (Design Draft and 2004 Editions) and JTG D62-2004 codes.

Prestress loading options are accessed from the **Bridge > Prestress Wizard** menu item.

Which to use?

- The **Single tendon prestress wizard** does not take into account any stressing or unstressing of any other tendons. It is for use with beam, tendon, plane stress and solid element modelling of concrete.
- The **Multiple tendon prestress wizard** takes into account elastic shortening due to stressing of other tendons according to the selected design code or user-defined percentage losses. As such it is more suited to staged construction analysis. It is however limited to use with beam element modelling of concrete. If the option to ignore effects due to elastic shortening is chosen the loading computed will be the same as that calculated by the single tendon wizard.

Single Tendon Prestress Wizard

The Single Tendon prestress wizard is accessed from the **Bridge > Prestress Wizard** menu item.

The wizard generates tendon loads as either beam element loads or as discrete loads, depending upon the analysis type chosen. The tendon load attributes created can be seen in the Attributes Treeview. Direct import of tendon forces may also be defined via an Excel spreadsheet. The single tendon wizard does not take into account any individual stressing or unstressing of any other tendons. For sets of tendons it may be used if all tendons are stressed at the same time.

A model may contain many tendons and the single tendon prestress wizard can be used separately on each to derive equivalent nodal loading on the mesh for all the tendons used. Structural concrete surrounding a tendon may either be modelled as a series of beam elements, or by a set of plane stress or solid elements. For examples see the analysis types supported below.

Usage

To use the single tendon prestress wizard:

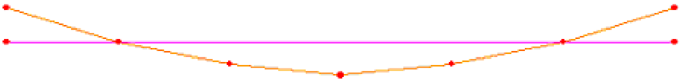
1. Select the spline or combined line defining the tendon.
2. If carrying out a beam analysis, additionally use the **Shift** key to select the beam(s)
3. Select the relevant design code from the **Bridge > Prestress Wizard > Single Tendon** menu item.

The prestress wizard computes the equivalent forces due to the prestress tendon in accordance with a chosen code and assigns them to the current active loadcase of the beam, plane stress or solid elements automatically. After load assignment the forces resulting from the tendon will be shown. Using this simplified force approach, the effects of prestress can be defined in a separate linear loadcase that can be combined with the other loadcases to compute the overall structural behaviour.

Analysis Type

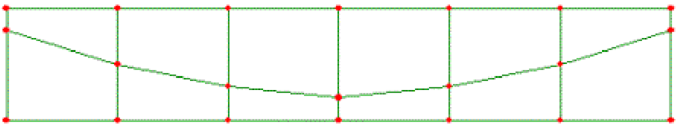
Four analysis types are supported; Beam, Tendon, Plane Stress and Solid.

- **Beam** For the beam analysis method the tendon profile must be defined as a **spline**. No elements need to be assigned to this spline, but thick beam elements should be assigned to the line representing the concrete beam. Beams may be straight or curved and may consist of multiple lines defining each span. Tendon alignment / realignment is independent of the underlying mesh arrangement. With this analysis type the tendon is not included in the analysis model. Lines defining tendon profiles are essentially used solely to derive the forces on the surrounding elements defining the structural concrete and, as such, these tendons are not included in an analysis model. Once used, tendons may be grouped together and made invisible to simplify subsequent model displays.



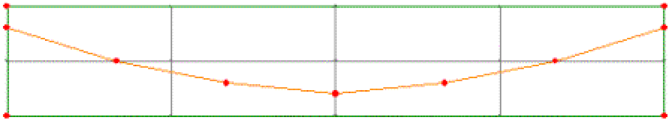
Analysis type: Beam

- **Tendon** For the tendon analysis method the tendon is included in the analysis model. Thick beam elements should be assigned to the lines defining the tendon profile (which must for this analysis type only be defined as a **combined line**) and the concrete beam surrounding the tendon should be modelled with Plane Stress or 3D Solid elements. Note that when using a combined line sufficient points must be used to accurately represent the tendon profile as a series of straight lines. Tendon alignment / realignment is dependent on the underlying mesh arrangement. Note that this analysis method does not allow for tendon realignment as easily as the Plane Stress / Solid analysis option.

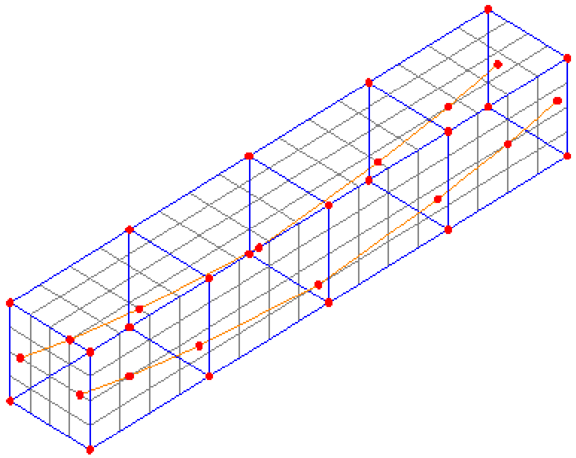


Analysis type: Tendon

- **Plane Stress / Solid** For this analysis method the tendon profile must be defined as either a single line, single arc or a **spline**. No line mesh elements need to be assigned to the spline but plane stress / solid elements must be assigned to the surrounding concrete. The lines representing the tendon are not included in the analysis model. Tendon alignment / realignment is, therefore, independent of the underlying mesh arrangement. Lines defining tendon profiles are essentially used solely to derive the forces on the surrounding elements defining the structural concrete and, as such, are not included in an analysis model. Once used, tendons may be grouped together and made invisible to simplify subsequent model displays. Simple examples of the use of the plane stress and solid methods are shown below.



Analysis type: Plane Stress



Analysis type: Solid

Notes

- An optional report can be generated in HTML file format in your project directory that contains a summary of the tendon properties, tendon geometry and prestress losses for a selected tendon.
- Graph datasets can be optionally created to allow subsequent graphing to be carried out using the Graph Wizard.
- When the prestress definition is computed two graph datasets are created to enable the tendon losses to be visualised against the tendon length. These can be seen in the Utilities Treeview.
- Tendon forces can also be computed directly by the user and imported from a spreadsheet. If this option is selected a filename for the spreadsheet is entered and when the OK button is clicked the tendon geometry is written to the spreadsheet. When this has been done the spreadsheet is displayed and the tendon forces can be added by manually editing the spreadsheet. The forces are then read into Modeller and the procedure follows that described above for the codes.
- If tendon realignment is required the previously assigned discrete loading properties should be removed from the model and the Prestress loading wizard re-run to calculate the new discrete loading properties for the new tendon alignment.
- After creation of the tendon loading attributes the subsequent modification of the tendon profile, or beam lines, will not update the prestress loading until the single tendon wizard is run again.

- Use of the single tendon prestress facility is described in the example 'Linear Analysis of a Post Tensioned Bridge'. See the *Application Examples Manual (Bridge, Civil & Structural)* for details.

Design codes supported

Prestress definition varies slightly according to the design code selected and is explained under separate headings. The following design codes are supported:

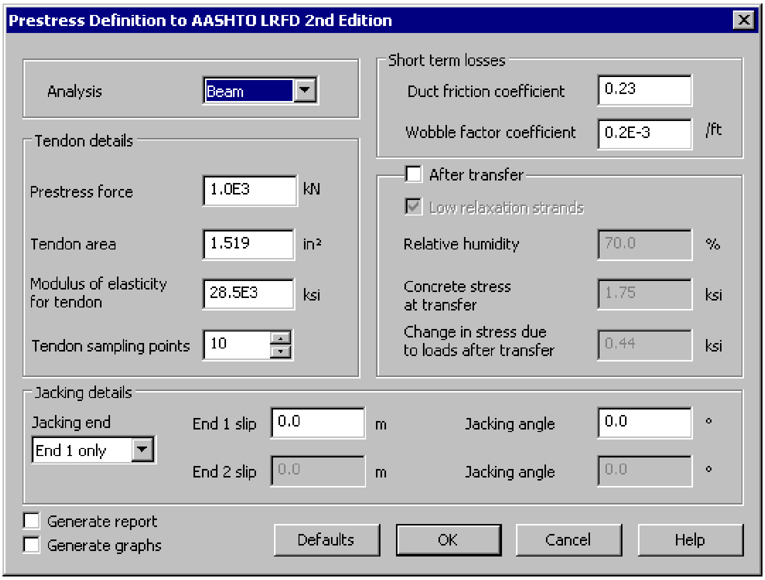
- **AASHTO LRFD 2nd Edition**
- **AASHTO LRFD 5th Edition**
- **AASHTO LRFD 6th Edition**
- **AASHTO LRFD 7th Edition**
- **BS5400-4:1990**
- **DD EN1992-1-1:1992 Eurocode 2**
- **EN 1992-1-1:2004 Eurocode 2**
- **JTG D62-2004**

Prestress definition can also be imported from a spreadsheet.

Single Tendon Prestress to AASHTO LRFD 2nd Edition

The single tendon prestress wizard generates tendon loads as either beam element loads or as discrete loads, depending upon the analysis type chosen. The tendon load attributes created can be seen in the Attributes Treeview.

Prior to using the **single tendon prestress wizard**, first select the spline or combined line defining the tendon and then, but only if carrying out a beam analysis, additionally use the **Shift** key to select the beam(s) prior to selecting the design code from the **Bridge** > **Prestress Wizard** > **Single Tendon** menu items.



Analysis type

Refer to the **single tendon prestress wizard** overview for details on the analysis types supported.

Tendon details

- **Prestress force** The jacking force applied to the tendon or tendon group. Units are displayed on the dialog according to the model units in use.
- **Tendon area** [A_p]. The cross-sectional area of the tendon (or total area of group of strands if a group is being represented by a single load assignment). In general, for internal prestressing, tendons of modest size are advisable to avoid difficulties in housing and anchoring larger tendons with the attendant increase in thickness of members and in the weight of reinforcement. 19 No 0.6" strand (A_p=4.123in²) might be a practical upper limit per tendon. However, the Prestress Wizard will not preclude tendons of larger area.
- **Modulus of Elasticity** [E_p]. Refer to AASHTO 2nd clause 5.4.4.2. The default value is 28,500ksi, representing strand.
- **Tendon sampling points** are the locations along a spline at which calculated equivalent tendon loads will be applied to the model. Note that the original points used to define the spline have no bearing on any calculations that are carried out, they are simply used to ensure a good tendon profile is obtained.

Short term losses

- **Duct Friction coefficient** [μ]. This value is used in the calculation of losses due to friction according to AASHTO 2nd clause 5.9.5.2.2b, and also in determining anchorage losses. Suitable values may be found in Table 5.9.5.2.2b-1; the default value μ = 0.23 is for strand in polyethylene ducts but is also in the suggested range for rigid and semi-rigid galvanised metal sheathing.

- **Wobble factor coefficient** [K (per foot)]. This value is used in the calculation of losses due to friction according to AASHTO 2nd clause 5.9.5.2.2b and also in determining anchorage losses. Example values may be sought in Table 5.9.5.2.2b-1.

After transfer

- **Low relaxation strands** (tick-box) Relaxation losses are calculated according to AASHTO 2nd eqn 5.9.5.4.4c-2, unless this tick-box is checked, causing 30% of the value from eqn 5.9.5.4.4c-2 to be used, in line with the provisions of clause 5.9.5.4.4c.
- **Relative humidity** [H] This value is used in the calculation of losses due to shrinkage, to AASHTO 2nd clause 5.9.5.4.2. H is defined as the “average annual ambient relative humidity (percent)” and guidance can be found in Figure 5.4.2.3.3-1.
- **Concrete stress at transfer** [f_{cgp}]. This value is used for the calculation of elastic shortening losses, according to AASHTO 2nd eqn 5.9.5.2.3b-1 and, when long-term losses are requested, for creep losses using eqn 5.9.5.4.3-1. Creep losses are based on a single value of f_{cgp} for each tendon (combined with Δf_{cdp} defined below as appropriate), and the elastic shortening losses are based on a single f_{cgp} value for each member (calculated from the values given for all the applicable tendons). The value entered for f_{cgp} here should be the stress in the concrete adjacent to the tendon in question, immediately after tensioning and anchoring, due to total prestressing forces and self-weight, at the section of peak moment. Variations in stress arising from permanent actions applied after prestressing (generally, but not exclusively) leading to elastic shortening gains) are not incorporated intp the calculation of initial elastic shortening loss.
- **Change in stress due to loads after transfer** [f_{cdp}]. This value is used for the calculation of the creep losses, according to AASHTO 2nd eqn 5.9.5.4.3-1, and is defined in the code as the “change in concrete stress at centre of gravity of prestressing steel due to permanent loads with the exception of the load acting at the time the prestressing force is applied”. Applied loads and long-term losses generally oppose the initial prestress that dominates f_{cgp} and so creep effects arising from Δ_{fcdp} are subtracted from those arising from f_{cgp} in eqn 5.9.5.4.3-1. Therefore where Δ_{fcdp} represents a reduction in stress from f_{cgp}, a positive value for Δf_{cdp} should be entered into eqn 5.9.5.4.3-1 and this dialog. As in the case of f_{cgp}, a value appropriate to the concrete adjacent to the tendon in question may be used.

Jacking details

Jacking end slip and jacking angle can be defined for end 1 only, end 2 only, or for both ends of the tendon. End 1 is the start of the spline used to define the tendon.

The jacking angle is the angle between the direction of the tendon at the anchorage and the direction that the jack is set to pull in. If the jack pulls the cable axially the jacking angle would be zero.

Generate report

Generates a report as an HTML file in the project directory containing a summary of the tendon properties, tendon geometry and prestress losses for the selected tendon.

Generate graphs

Generates graph datasets in the Utilities Treeview to allow subsequent graphing of prestress losses along tendon using the Graph Wizard.

Defaults

The **Defaults** button sets all previously entered values to those specified when the dialog is first displayed.

Notes

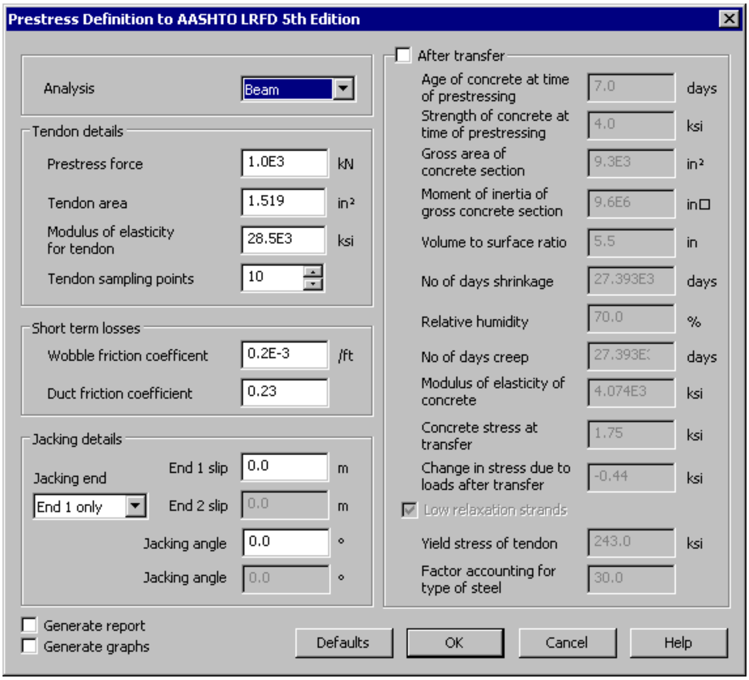
- An approximate check that the specified jacking force is suitable for the tensile strength of commonly available strand is carried out by the wizard but the user should refer to the relevant design code and manufacturers information for the strand strengths and any applicable factors in order to ensure that the tendons are not overstressed.
- If tendon realignment is required the previously assigned discrete loading properties should be removed from the model and the Prestress loading wizard re-run to calculate the new discrete loading properties for the new tendon alignment.
- Use of the single tendon prestress facility is described in the example 'Linear Analysis of a Post Tensioned Bridge'. See the *Application Examples Manual (Bridge, Civil & Structural)* for details.

Single Tendon Prestress to AASHTO LRFD 5th Edition

This implementation is also applicable to AASHTO LRFD 6th and 7th Editions.

The single tendon prestress wizard generates tendon loads as either beam element loads or as discrete loads, depending upon the analysis type chosen. The tendon load attributes created can be seen in the Attributes Treeview.

Prior to using the **single tendon prestress wizard**, first select the spline or combined line defining the tendon and then, but only if carrying out a beam analysis, additionally use the **Shift** key to select the beam(s) prior to selecting the design code from the **Bridge > Prestress Wizard > Single Tendon** menu items.



Analysis type

Refer to the **single tendon prestress wizard** overview for details on the analysis types supported.

Tendon details

- **Prestress force** The jacking force applied to the tendon or tendon group. Units are displayed on the dialog according to the model units in use.
- **Tendon area** [A_p]. The cross-sectional area of the tendon (or total area of group of strands if a group is being represented by a single load assignment). In general, for internal prestressing, tendons of modest size are advisable to avoid difficulties in housing and anchoring larger tendons with the attendant increase in thickness of members and in the weight of reinforcement. 19 No 0.6" strand (A_p=4.123in²) might be a practical upper limit per tendon. However, the Prestress Wizard will not preclude tendons of larger area.
- **Modulus of elasticity for tendon** [E_p]. Refer to AASHTO 5th clause 5.4.4.2. The default value is 28,500ksi, representing strand.
- **Tendon sampling points** are the locations along a spline at which calculated equivalent tendon loads will be applied to the model. Note that the original points used to define the spline have no bearing on any calculations that are carried out, they are simply used to ensure a good tendon profile is obtained.

Short term losses

- **Wobble friction coefficient** [K (per foot)]. This value is used in the calculation of losses due to friction, according to AASHTO 5th clause 5.9.5.2.2b, and also in determining anchorage losses. Example values may be sought in Table 5.9.5.2.2b-1.
- **Duct friction coefficient** [μ]. This value is used in the calculation of losses due to friction according to AASHTO 5th clause 5.9.5.2.2b and also in determining anchorage losses. Suitable values may be found in Table 5.9.5.2.2b-1; the default value $\mu = 0.23$ is for strand in polyethylene ducts, but is also in the suggested range for rigid and semi-rigid galvanised metal sheathing.

After transfer

- **Age of concrete at time of prestressing** [t_i]. This value is used for the calculation of losses due to creep, to AASHTO 5th clause 5.9.5.4.3b.
- **Strength of concrete at time of prestressing** [f'_{ci}]. This value is used in the calculation of losses due to creep, to AASHTO 5th clause 5.9.5.4.3b. f'ci is defined as the "specified compressive strength of concrete at time of initial loading or prestressing; nominal concrete strength at time of application of tendon force (ksi)". According to clause 5.4.2.3.2, f'_{ci} may be taken as $0.8f'_c$ if the concrete age at the time of the initial load is unknown at design stage.
- **Gross area of concrete section** [A_c], and **Moment of inertia of gross concrete section**, [I_c]. These values are used for the calculation of losses due to shrinkage, to AASHTO 5th clause 5.9.5.4.3a, and creep, to clause 5.9.5.4.3b. Where the concrete section varies, suitable intermediate values may be required.
- **Volume to surface ratio** [V/S]. This value is used in the calculation of losses due to shrinkage to AASHTO 5th clause 5.9.5.4.3a and creep to clause 5.9.5.4.3b. Details on how the ratio should be calculated are given in clause 5.4.2.3.2.
- **No of days shrinkage** [t]. This value is used for the calculation of losses due to shrinkage, to AASHTO 5th clause 5.9.5.4.3a. t is defined as the "number of days between end of curing and time being considered for analysis of shrinkage effects". The default value is based on the design life for a bridge, which is 75 years according to clause 1.2.
- **Relative humidity** [H]. This value is used in the calculation of losses due to shrinkage, to AASHTO 5th clause 5.9.5.4.3a, and creep, to clause 5.9.5.4.3b. H is defined as the "average ambient mean relative humidity (percent)" and guidance can be found in Figure 5.4.2.3.3-1.
- **No of days creep** [t]. This value is used for the calculation of losses due to creep, to AASHTO 5th clause 5.9.5.4.3b. t is defined as the "no of days between application of load and time considered for creep calculation". The default value is based on the design life for a bridge, which is 75 years according to clause 1.2.
- **Modulus of elasticity of concrete** [E_c]. This value is used for the calculation of losses due to creep, to AASHTO 5th clause 5.9.5.4.3b. E_c may be calculated for a given concrete grade from the equation in clause 5.4.2.4.

- **Concrete stress at transfer** [f_{cgp}]. This value is used for the calculation of elastic shortening losses, according to AASHTO 5th eqn 5.9.5.2.3b-1, and, when long-term losses are requested, for creep losses, using eqn 5.9.5.4.2b-1. Creep losses are based on a single value of f_{cgp} for each tendon (combined with Δ_{fcd} defined below as appropriate), and the elastic shortening losses are based on a single f_{cgp} value for each member (calculated from the values given for all the applicable tendons). The value entered for f_{cgp} should be the stress in the concrete adjacent to the tendon in question, immediately after tensioning and anchoring, due to total prestressing forces and self-weight at the section of peak moment. Variations in stress arising from permanent actions applied after prestressing (generally, but not exclusively, leading to elastic shortening gains) are not incorporated into the calculation of initial elastic shortening loss.
- **Change in stress due to loads after transfer** [Δf_{cd}]. This value is used for the calculation of the creep losses, according to AASHTO 5th eqn 5.9.5.4.2b-1, and is defined in the code as the "change in concrete stress at centroid of prestressing strands due to long-term losses between transfer and deck placement, combined with deck weight and superimposed loads". For this, a value appropriate to the concrete adjacent to the tendon in question may be used. Δf_{cd} is generally negative since applied loads and long-term losses oppose the initial prestress that dominates f_{cgp}.
- **Low relaxation strands** (tick-box). When invoked, the relaxation loss $\Delta f_{pR2} = \Delta f_{pR1} =$ 1.2ksi is be assumed, according to AASHTO 5th clause 5.9.5.4.2c, and the text boxes for f_{py} and K_L are greyed out. If this assumption is deemed inappropriate, the box should be unticked and the appropriate data entered
- **Yield stress of tendon** [f_{py}]. This value is used for the calculation of relaxation losses according to AASHTO 5th eqn 5.9.5.4.2c-1. f_{py} is defined as "yield stress of prestressing steel.". Suitable values may be based on clause 5.4.4.1.
- **Factor accounting for type of steel** [K_L]. This value is used for the calculation of relaxation losses, according to AASHTO 5th eqn 5.9.5.4.2c-1, when the assumption of $\Delta f_{pR2} = \Delta f_{pR1} =$ 1.2ksi is deemed inappropriate. K_L is defined as "factor accounting for type of steel, taken as 30 for low relaxation strands and 7 for other prestressing steel, unless more accurate manufacturer's data is available.".

Jacking details

Jacking end slip and jacking angle can be defined for end 1 only, end 2 only, or for both ends of the tendon. End 1 is the start of the spline used to define the tendon.

The jacking angle is the angle between the direction of the tendon at the anchorage and the direction that the jack is set to pull in. If the jack pulls the cable axially the jacking angle would be zero.

Generate report

Generates a report as an HTML file in the project directory containing a summary of the tendon properties, tendon geometry and prestress losses for the selected tendon.

Generate graphs

Generates graph datasets in the Utilities Treeview to allow subsequent graphing of prestress losses along tendon using the Graph Wizard.

Defaults

- The **Defaults** button sets all previously entered values to those specified when the dialog was first displayed. Such values are illustrative only and values for use in design calculations should be checked using appropriate source data.

Notes

- The prestress definition dialog expects input units to be the same as the current model unless otherwise stated on the dialogs. When the prestress loads are calculated the prestress forces are converted into the current model units. The current model units can be found on the status bar of LUSAS Modeller.
- An approximate check that the specified jacking force is suitable for the tensile strength of commonly available strand is carried out by the wizard, but the user should refer to the relevant design code and manufacturers information for the strand strengths and any applicable factors in order to ensure that the tendons are not overstressed.
- If tendon realignment is required, the previously assigned discrete loading properties should be removed from the model and the Prestress loading wizard re-run to calculate the new discrete loading properties for the new tendon alignment.
- Use of the single tendon prestress facility is described in the example 'Linear Analysis of a Post Tensioned Bridge'. See the *Application Examples Manual (Bridge, Civil & Structural)* for details.

Single Tendon Prestress to AASHTO LRFD 6th Edition

For details of AASHTO LRFD 6th Edition Tendon Properties please refer to **AASHTO LRFD 5th Edition Tendon Properties (Multi-tendon)**. All relevant clauses in AASHTO LRFD 6th Edition are identical to those in AASHTO LRFD 5th Edition.

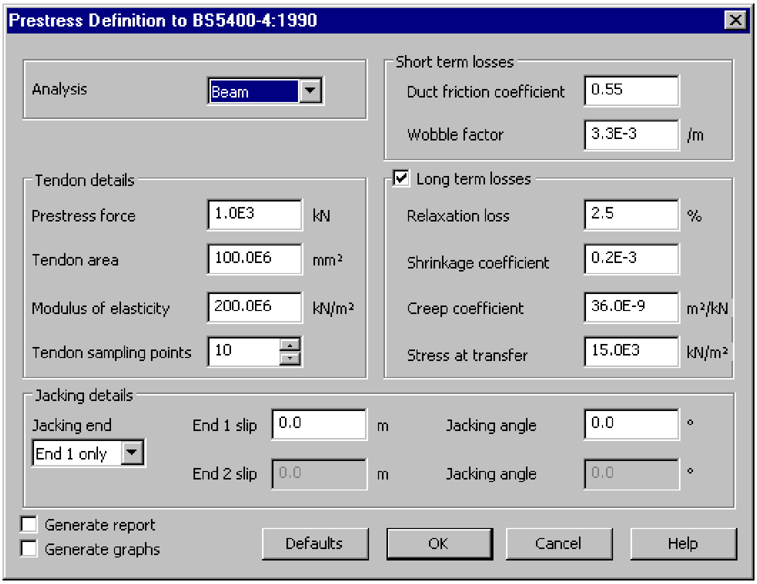
Single Tendon Prestress to AASHTO LRFD 7th Edition

For details of AASHTO LRFD 7th Edition Tendon Properties please refer to **AASHTO LRFD 5th Edition Tendon Properties (Multi-tendon)**. All relevant clauses in AASHTO LRFD 7th Edition are identical to those in AASHTO LRFD 7th Edition.

Single Tendon Prestress to BS5400-4:1990

The single tendon prestress wizard generates tendon loads as either beam element loads or as discrete loads, depending upon the analysis type chosen. The tendon load attributes created can be seen in the Attributes Treeview.

Prior to using the **single tendon prestress wizard**, first select the spline or combined line defining the tendon and then, but only if carrying out a beam analysis, additionally use the **Shift** key to select the beam(s) prior to selecting the design code from the **Bridge > Prestress Wizard > Single Tendon** menu items.



Analysis type

Refer to the **single tendon prestress wizard** overviews for details on the analysis types supported.

Tendon details

- **Prestress force** The jacking force applied to the tendon or tendon group. Units are displayed on the dialog according to the model units in use.
- **Tendon area** for the single tendon or tendon group being defined.
- **Modulus of Elasticity** for the tendon.
- **Tendon sampling points** are the locations along a spline at which calculated equivalent tendon loads will be applied to the model. Note that the original points used to define the spline have no bearing on any calculations that are carried out. They are simply used to ensure a good tendon profile is obtained.

Short term losses

- **Duct Friction coefficient** should be obtained from the design code
- **Wobble factor** is defined per metre length and should be obtained from the design code.

Long term losses

- ❑ **Relaxation loss**
- ❑ **Shrinkage Coefficient**
- ❑ **Creep coefficient**
- ❑ **Stress at transfer**

Jacking details

Jacking end slip and jacking angle can be defined for end 1 only, end 2 only, or for both ends of the tendon. End 1 is the start of the spline used to define the tendon.

The jacking angle is the angle between the direction of the tendon at the anchorage and the direction that the jack is set to pull in. If the jack pulls the cable axially the jacking angle would be zero.

Generate report

Generates a report as an HTML file in the project directory containing a summary of the tendon properties, tendon geometry and prestress losses for the selected tendon.

Generate graphs

Generates graph datasets in the Utilities Treeview to allow subsequent graphing of prestress losses along tendon using the Graph Wizard.

Defaults

The **Defaults** button sets all previously entered values to those specified when the dialog is first displayed.

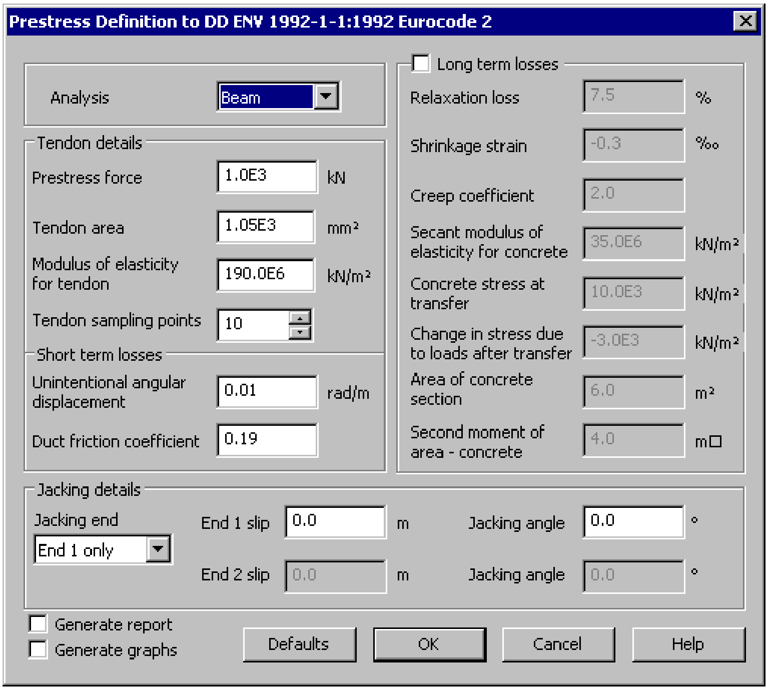
Notes

- An approximate check that the specified jacking force is suitable for the tensile strength of commonly available strand is carried out by the wizard, but the user should refer to the relevant design code and manufacturers information for the strand strengths and any applicable factors in order to ensure that the tendons are not overstressed.
- If tendon realignment is required, the previously assigned discrete loading properties should be removed from the model and the Prestress loading wizard re-run to calculate the new discrete loading properties for the new tendon alignment.
- Use of the single tendon prestress facility is described in the example ‘Linear Analysis of a Post Tensioned Bridge’. See the *Application Examples Manual (Bridge, Civil & Structural)* for details.

Single Tendon Prestress to DD EN1992-1-1:1992 Eurocode 2

The single tendon prestress wizard generates tendon loads as either beam element loads or as discrete loads, depending upon the analysis type chosen. The tendon load attributes created can be seen in the Attributes Treeview.

Prior to using the **single tendon prestress wizard**, first select the spline or combined line defining the tendon and then, but only if carrying out a beam analysis, additionally use the **Shift** key to select the beam(s) prior to selecting the design code from the **Bridge** > **Prestress Wizard** > **Single Tendon** menu items.



Analysis type

Refer to the **single tendon prestress wizard** overviews for details on the analysis types supported.

Tendon details

- **Prestress force** The jacking force applied to the tendon or tendon group. Units are displayed on the dialog according to the model units in use.
- **Tendon area** [A_p]. The cross-sectional area of the tendon (or total area of group of strands if a group is being represented by a single load assignment). In general, for

internal prestressing, tendons of modest size are advisable to avoid difficulties in housing and anchoring larger tendons with the attendant increase in thickness of members and in the weight of reinforcement. 19 No 15mm strands (A_p=2850mm²) or 27 No 13mm strands (A_p=2700mm²) might be a practical upper limit per tendon. However, the Prestress Wizard will not preclude tendons of larger area.

- **Modulus of Elasticity for tendon** [E_s] . Refer to DD ENV 1992-1-1 clause 3.3.4.4 (2). The default value for the modulus of elasticity, E_s, is assumed to be 190kN/mm², representing strand. Values typically fall in the range of 185kN/mm² to 210kN/mm².
- **Tendon sampling points** are the locations along a spline at which calculated equivalent tendon loads will be applied to the model. Note that the original points used to define the spline have no bearing on any calculations that are carried out, they are simply used to ensure a good tendon profile is obtained.

Short term losses

- **Unintentional angular displacement**[k]. This value is used in the calculation of losses due to friction, according to DD ENV1992-1-1 clause 4.2.3.5.5(8), for the component sometimes referred to as "wobble" loss. It is also used in the calculation of anchorage losses. k is defined as "unintentional angular displacement (per unit length) related to the profile of the tendons. Clause 4.2.3.5.5(8) gives the range $0.005 < k < 0.01$ radians/m.
- **Duct Friction coefficient** [μ]. This value is used in the calculation of losses due to friction, according to DD ENV1992-1-1 clause 4.2.3.5.5(8). It is also used in the calculation of anchorage losses. Suitable values may be found in clause 4.2.3.5.5(8); the default value $\mu = 0.19$ is for internal tendons made up of strand filling approximately half of the duct.

Long term losses

- **Relaxation loss** [ρ_t]. This value is used for the calculation of $\Delta_{\sigma pr}$, the absolute value of relaxation loss, which is incorporated into the time dependent loss calculation of DD ENV1992-1-1 eqn 4.10. According to clause 4.2.3.4.1(2), the long term value of the relaxation loss may be assumed to be three times the relaxation losses after 1000h, which may in turn be taken from Figure 4.8, (dependent upon the ratio of the applied prestress to the tensile strength of the tendons and the type of prestressing steel in use). Typical values for class 2 (strand) would therefore be (from Figure 4.8);

  - $\sigma_p/f_{pk} = 0.6$; $\rho 1000 = 1\%$; $\rho\infty = 3\%$
  - $\sigma_p/f_{pk} = 0.7$; $\rho 1000 = 2.5\%$; $\rho\infty = 7.5\%$
  - $\sigma_p/f_{pk} = 0.8$; $\rho 1000 = 4.5\%$; $\rho\infty = 13.5\%$

- **Shrinkage strain** [$\varepsilon_s(t,t0)$]. This value is used in the calculation of losses due to shrinkage, according to DD ENV1992-1-1 clause 4.2.3.5.5(9). $\varepsilon_s(t,t0)$ is "the estimated shrinkage strain, derived from the values in Table 3.4 for final shrinkage (see also 2.5.5 and Appendix 1)" and should be specified "per mil" (‰) in accordance with the code (rather than actual strain).

- **Creep coefficient** [φ(t,t0)]. This value is used in the calculation of losses due to creep, according to DD ENV1992-1-1 clause 4.2.3.5.5(9). Clause 1.7.4 describes φ(t,t0) as the "Creep coefficient, defining creep between times t and t0, related to elastic deformation at 28 days". Clause 4.2.3.5.5(9) refers to clause 2.5.5 which in turn refers to clause 3.1 or (for greater accuracy) Appendix 1. If great accuracy is not required, a value for φ(∞,t0) may be obtained from Table 3.3 – dependent upon notional size, relative humidity and maturity of the concrete when the load is first applied.
- **Secant modulus of elasticity of concrete** [E_{cm}]. This value is used for the calculation of time-dependent losses according to DD ENV1992-1-1 clause 4.2.3.5.5(9). Equation 4.10 requires a modular ratio, $\alpha = E_s/E_{cm}$, where E_{cm} can be obtained from Table 3.2. A likely range of values would be 29-37kN/mm².
- **Concrete stress at transfer** σ_{co}. This value is used for the calculation of elastic shortening losses, to DD ENV1992-1-1 clause 4.2.3.5.5(6) and, when long-term losses are requested, the creep component of time-dependent losses to clause 4.2.3.5.5(9), eqn 4.10. Creep losses are based on a single value of σ_{co} for each tendon (combined with $\Delta\sigma_{cg}$, derived below, as appropriate), and the elastic shortening losses are based on a single σ_{co} value for each member (calculated from the values given for all the applicable tendons). The value entered for σ_{co} here should be the stress in the concrete adjacent to the tendon in question, immediately after tensioning and anchoring, due to total prestressing forces and self-weight averaged along the length of the tendon. Variations in stress arising from permanent actions applied after prestressing (generally but not exclusively leading to elastic shortening gains) are not incorporated in the calculation of initial elastic shortening loss.
- **Change in stress due to loads after transfer** $\Delta\sigma_{cg}$. This is used to adjust the concrete stress at transfer, σ_{co}, to account for stress in the concrete due to permanent loads applied after prestressing. $\sigma_{co} + \Delta\sigma_{cg}$ gives the concrete stress appropriate for the calculation of the creep component of time-dependent losses, according to DD ENV1992-1-1 clause 4.2.3.5.5(9), eqn 4.10 , ($\sigma_{co} + \Delta\sigma_{cg} = [\sigma_{cg} + \sigma_{cpo}]$). $\Delta\sigma_{cg}$ is typically negative since applied loads oppose the prestress that dominates σ_{co}.
- **Area of concrete section** [Ac] and **Second moment of area – concrete** [Ic]. These values are used for the calculation of time-dependent losses, according to DD ENV1992-1-1 clause 4.2.3.5.5(9), eqn 4.10. Where the concrete section varies suitable intermediate values may be required. If the member is a standalone beam element the area of the beam should be used. If the beam member represents part of a larger structure (for example a certain width of slab) then only the area of the portion represented by this particular beam member should be used. Please note that the prestress wizard is designed to work for a single beam member, not for a subdivided portion of slab or wide beam element, so using it in any other way is at an engineer's judgement.

Jacking details

Jacking end slip and jacking angle can be defined for end 1 only, end 2 only, or for both ends of the tendon. End 1 is the start of the spline used to define the tendon.

The jacking angle is the angle between the direction of the tendon at the anchorage and the direction that the jack is set to pull in. If the jack pulls the cable axially the jacking angle would be zero.

Generate report

Generates a report as an HTML file in the project directory containing a summary of the tendon properties, tendon geometry and prestress losses for the selected tendon.

Generate graphs

Generates graph datasets in the Utilities Treeview to allow subsequent graphing of prestress losses along tendon using the Graph Wizard.

Defaults

The **Defaults** button sets all previously entered values to those specified when the dialog was first displayed.

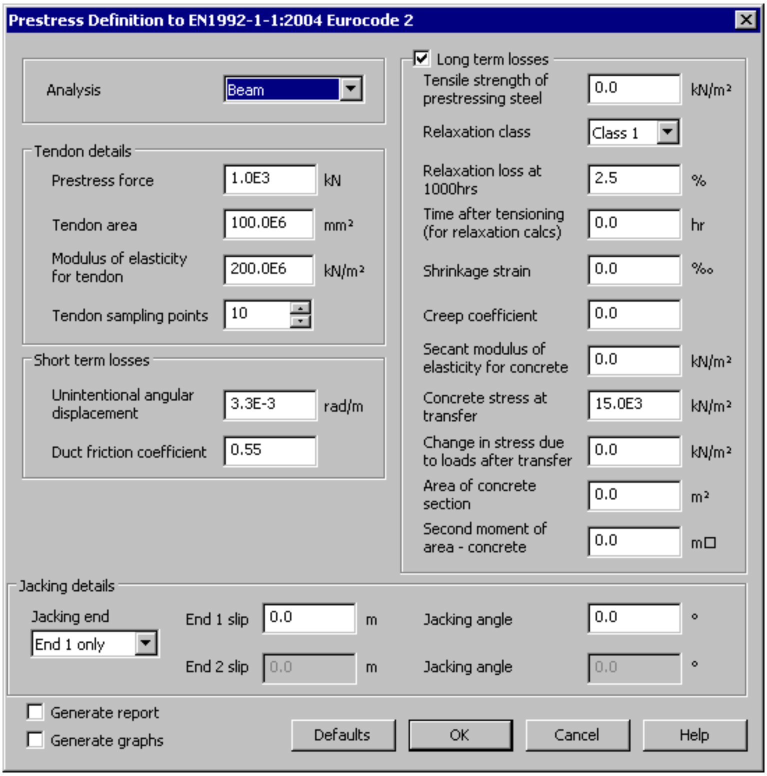
Notes

- The prestress definition dialog expects input units to be the same as the current model unless otherwise stated on the dialogs. When the prestress loads are calculated the prestress forces are converted into the current model units. The current model units can be found on the status bar of LUSAS Modeller.
- An approximate check that the specified jacking force is suitable for the tensile strength of commonly available strand is carried out by the wizard, but the user should refer to EN1992-1-1 clause 5.10.3(2), manufacturers information for the strand strengths and the appropriate National Annex for values of k7 and k8 in order to ensure that the tendons are not overstressed.
- If tendon realignment is required, the previously assigned discrete loading properties should be removed from the model and the Prestress loading wizard re-run to calculate the new discrete loading properties for the new tendon alignment.
- Use of the single tendon prestress facility is described in the example ‘Linear Analysis of a Post Tensioned Bridge’. See the *Application Examples Manual (Bridge, Civil & Structural)* for details.

Single Tendon Prestress to EN 1992-1-1:2004 Eurocode 2

The single tendon prestress wizard generates tendon loads as either beam element loads or as discrete loads, depending upon the analysis type chosen. The tendon load attributes created can be seen in the Attributes Treeview.

Prior to using the **single tendon prestress wizard**, first select the spline or combined line defining the tendon and then, but only if carrying out a beam analysis, additionally use the **Shift** key to select the beam(s) prior to selecting the design code from the **Bridge** > **Prestress Wizard** > **Single Tendon** menu items.



Analysis type

Refer to the **single tendon prestress wizard** overviews for details on the analysis types supported.

Tendon details

- **Prestress force** The jacking force applied to the tendon or tendon group. Units are displayed on the dialog according to the model units in use.

- **Tendon area** [A_p]. The cross-sectional area of the tendon (or total area of group of strands if a group is being represented by a single load assignment). In general, for internal prestressing, tendons of modest size are advisable to avoid difficulties in housing and anchoring larger tendons with the attendant increase in thickness of members and in the weight of reinforcement. 19 No 15mm strands (A_p=2850mm²) or 27 No 13mm strands (A_p=2700mm²) might be a practical upper limit per tendon. However, the Prestress Wizard will not preclude tendons of larger area.
- **Modulus of elasticity for tendon** [E_p]. Refer to EN 1992-1-1:2004 clause 3.3.6. The default value for the modulus of elasticity, E_p is assumed to be 195GPa, representing strand. Values typically fall in the range 185GPa to 210GPa.
- **Tendon sampling points** are the locations along a spline at which calculated equivalent tendon loads will be applied to the model. Note that the original points used to define the spline have no bearing on any calculations that are carried out, they are simply used to ensure a good tendon profile is obtained.

Short term losses

- **Unintentional angular displacement** [k]. This value is used in the calculation of losses due to friction, according to EN1992-1-1:2004 clause 5.10.5.2, for the component sometimes referred to as "wobble" loss, and also in determining anchorage losses. k describes the "unintentional angular displacement" for internal tendons in radians per unit length. Clause 5.10.5.2(3) gives the range $0.005 < k < 0.01$ radians/m.
- **Duct friction coefficient** [μ]. This value is used in the calculation of losses due to friction, according to EN1992-1-1:2004 clause 5.10.5.2(1), and also in determining anchorage losses. Suitable values may be found in Table 5.1; the default value μ = 0.19 is for internal tendons made up of strand filling approximately half of the duct.

Long term losses

- **Tensile strength of prestressing steel** [f_{pk}] This value is used for the calculation of $\Delta\sigma_{pr}$, the absolute value of relaxation losses to EN1992-1-1 clause 3.3.2(7) and incorporated into the time dependent loss calculation of clause 5.10.6(2).
- **Relaxation Class** This is used, along with f_{pk}, to calculate relaxation losses. The Classes 1, 2 and 3 are defined in EN1992-1-1 clause 3.3.2(4).
- **Relaxation loss at 1000 hours**[ρ_{1000}]. This is used, along with fpk and the Relaxation Class, to calculate relaxation losses. Typical values may be found in clause 3.3.2(6) and the default value is based on low relaxation strand (Class 2).
- **Time after tensioning (for relaxation calcs)** [t]. This is used, along with f_{pk}, Relaxation Class and ρ_{1000}, to calculate relaxation losses. According to EN1992-1-1 clause 3.3.2(8), long term (final) values of the relaxation losses may be estimated for a time t equal to 500 000 hours (i.e. around 57 years).
- **Shrinkage strain** [ε_{cs}]. This value is used in the calculation of losses due to shrinkage according to EN1992-1-1:2004 clause 5.10.6(2). The value should be calculated from clause 3.1.4(6), depending on notional size, relative humidity, concrete strength and cement class, and should be specified "per mil" (‰) in accordance with the code (rather than actual strain).

- **Creep coefficient** [φ(t,t0)]. This value is used in the calculation of losses due to creep according to EN1992-1-1:2004 clause 5.10.6(2). The value should be calculated from clause 3.1.4(2), dependent upon the same parameters as the shrinkage strain plus the maturity of the concrete when the load is first applied. Values may be obtained from Figure 3.1.
- **Secant modulus of elasticity for concrete** [E_{cm}]. This value is used for the calculation of time-dependent losses according to EN1992-1-1:2004 clause 5.10.6(2). The clause refers explicitly to obtaining E_{cm} from Table 3.1. A likely range of values would be 27-44MPa.
- **Concrete stress at transfer** [$\Delta\sigma_{c(t)}$]. This value is used for the calculation of elastic shortening losses, according to EN1992-1-1 clause 5.10.5.1(2), and, when long-term losses are requested, for the creep component of time-dependent losses to clause 5.10.6(2). Creep losses are based on a single value of $\Delta\sigma_c(t)$ for each tendon (combined with $\Delta\sigma_{cg}$, defined below, as appropriate), and the elastic shortening losses are based on a single $\Delta\sigma_c(t)$ value for each member (calculated from the values given for all the applicable tendons). The value entered for $\Delta\sigma_c(t)$ here should be the stress in the concrete adjacent to the tendon in question, immediately after tensioning and anchoring, due to total prestressing forces and self-weight averaged along the length of the tendon. Variations in stress arising from permanent actions applied after prestressing (generally but not exclusively leading to elastic shortening gains) are not incorporated into the calculation of initial elastic shortening loss.
- **Change in stress due to loads after transfer** [$\Delta\sigma_{cg}$]. This is used to adjust "Concrete stress at transfer" according to the permanent loads applied after transfer to give the concrete stress appropriate for the calculation of the creep component of time-dependent losses, according to EN1992-1-1 clause 5.10.6(2) ($\sigma_{c,QP} = \Delta\sigma_c(t) + \Delta\sigma_{cg}$). Note that $\Delta\sigma_{cg}$ is typically negative since applied loads oppose the prestress that dominates $\Delta\sigma_c(t)$.
- **Area of concrete section** [A_c], and **Second moment of area – concrete** [Ic]. These values are used for the calculation of time-dependent losses, according to DD ENV1992-1-1 clause 4.2.3.5.5(9), eqn 4.10. Where the concrete section varies suitable intermediate values may be required. If the member is a standalone beam element the area of the beam should be used. If the beam member represents part of a larger structure (for example a certain width of slab) then only the area of the portion represented by this particular beam member should be used. Please note that the prestress wizard is designed to work for a single beam member, not for a subdivided portion of slab or wide beam element, so using it in any other way is at an engineer's judgement.

Jacking details

Jacking end slip and jacking angle can be defined for end 1 only, end 2 only, or for both ends of the tendon. End 1 is the start of the spline used to define the tendon.

The jacking angle is the angle between the direction of the tendon at the anchorage and the direction that the jack is set to pull in. If the jack pulls the cable axially the jacking angle would be zero.

Generate report

Generates a report as an HTML file in the project directory containing a summary of the tendon properties, tendon geometry and prestress losses for the selected tendon.

Generate graphs

Generates graph datasets in the Utilities Treeview to allow subsequent graphing of prestress losses along tendon using the Graph Wizard.

Defaults

The **Defaults** button sets all previously entered values to those specified when the dialog was first displayed. Such values are illustrative only and values for use in design calculations should be checked using appropriate source data.

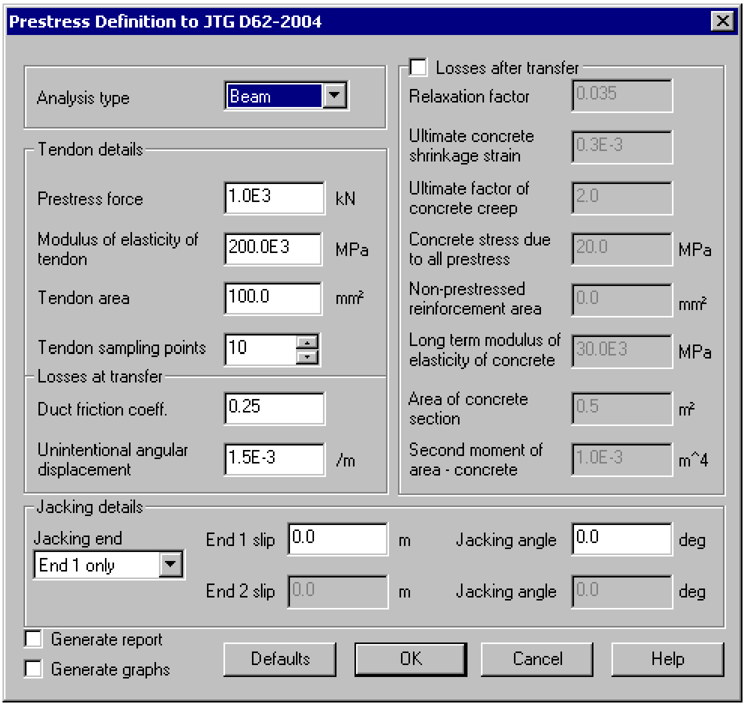
Notes.

- The prestress definition dialog expects input units to be the same as the current model unless otherwise stated on the dialogs. When the prestress loads are calculated the prestress forces are converted into the current model units. The current model units can be found on the status bar of LUSAS Modeller.
- An approximate check that the specified jacking force is suitable for the tensile strength of commonly available strand is carried out by the wizard but the user should refer to EN1992-1-1 clause 5.10.3(2), manufacturers information for the strand strengths and the appropriate National Annex for values of k7 and k8 in order to ensure that the tendons are not overstressed.
- If tendon realignment is required the previously assigned discrete loading properties should be removed from the model and the Prestress loading wizard re-run to calculate the new discrete loading properties for the new tendon alignment.
- Use of the single tendon prestress facility is described in the example ‘Linear Analysis of a Post Tensioned Bridge’. See the *Application Examples Manual (Bridge, Civil & Structural)* for details.

Single Tendon Prestress to JTG D62-2004 code

The single tendon prestress wizard generates tendon loads as either beam element loads or as discrete loads, depending upon the analysis type chosen. The tendon load attributes created can be seen in the Attributes Treeview.

Prior to using the **single tendon prestress wizard**, first select the spline or combined line defining the tendon and then, but only if carrying out a beam analysis, additionally use the **Shift** key to select the beam(s) prior to selecting the design code from the **Bridge** > **Prestress Wizard** > **Single Tendon** menu items.



Analysis type

Refer to the **single tendon prestress wizard** overviews for details on the analysis types supported.

Tendon details

- **Prestress force** The jacking force applied to the tendon or tendon group. Units are displayed on the dialog according to the model units in use.
- **Modulus of elasticity of tendon** Young's modulus of prestress reinforcement.
- **Tendon area** Steel area for the single tendon being defined.
- **Tendon sampling points** Locations along a spline at which calculated equivalent tendon loads will be applied to the model. Note that the original points used to define the spline have no bearing on any calculations that are carried out, they are simply used to ensure a good tendon profile is obtained.

Losses at transfer

- **Duct Friction coefficient** The friction coefficient between the tendon (prestress reinforcement) and the duct (pipe). Refer to JTG D62-2004 Table 6.2.2., for details.
- **Unintentional angular displacement** also known as the influence factor of load deviation, or wobble defined per metre length. See JTG D62-2004 Table 6.2.2.

Losses after transfer

- **Relaxation factor** Refer to JTG D62-2004 Section 6.2.6., for details.
- **Ultimate concrete shrinkage strain** Refer to JTG D62-2004 Table 6.2.7., for details
- **Ultimate factor of concrete creep** Refer to JTG D62-2004 Table 6.2.7., for details
- **Concrete prestress due to all prestress** The stress in concrete due to prestress at the centroid of all tendons. Refer to JTG D62-2004 section 6.1.5., for details
- **Non-prestressed reinforcement area** The sectional area of general (non prestressed) reinforced reinforcement
- **Long-term modulus of elasticity of concrete** The long-term Young's modulus of concrete.
- **Area of concrete section** The net sectional area of concrete after the area of the pipes (ducts) and other weakened parts have been removed.
- **Second moment of area - concrete** Moment of inertia of concrete section

Jacking details

Jacking end slip and jacking angle can be defined for end 1 only, end 2 only, or for both ends of the tendon. End 1 is the start of the spline used to define the tendon.

The jacking angle is the angle between the direction of the tendon at the anchorage and the direction that the jack is set to pull in. If the jack pulls the cable axially the jacking angle would be zero.

Generate report

Generates a report as an HTML file in the project directory containing a summary of the tendon properties, tendon geometry and prestress losses for the selected tendon.

Generate graphs

Generates graph datasets in the Utilities Treeview to allow subsequent graphing of prestress losses along tendon using the Graph Wizard.

Defaults

The **Defaults** button sets all previously entered values to those specified when the dialog was first displayed.

Notes

- The prestress definition dialog expects input units to be the same as the current model unless otherwise stated on the dialogs. When the prestress loads are calculated the prestress forces are converted into the current model units. The current model units can be found on the status bar of LUSAS Modeller.
- An approximate check that the specified jacking force is suitable for the tensile strength of commonly available strand is carried out by the wizard, but the user should

refer to the relevant design code and manufacturers information for the strand strengths and any applicable factors in order to ensure that the tendons are not overstressed.

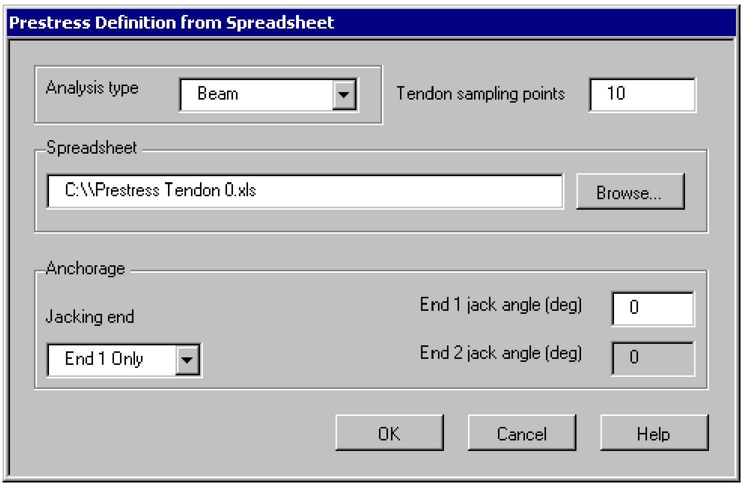
- The loss due to elastic compression of concrete is computed using the simplified formula in JTG D62-2004 Appendix E
- Guidance on the deformation of the anchorage device is provided in JTG D62-2004 Table 6.2.3.
- Use of the single tendon prestress facility is described in the example ‘Linear Analysis of a Post Tensioned Bridge’. See the *Application Examples Manual (Bridge, Civil & Structural)* for details.
- Tendon forces can also be computed directly by the user and imported from a spreadsheet.

Single Tendon Prestress Definition from Spreadsheet

As an alternative to using one of the supplied single tendon prestress design code options, tendon forces can be computed directly by an engineer and imported from a spreadsheet. To aid this, a spreadsheet can be automatically generated by LUSAS containing geometric properties for a particular selected tendon. This means that only the tendon loading data needs to be entered by a user.

Usage

Prior to using this facility the line, arc, spline or combined line defining the tendon must be selected and then, but only if carrying out a beam analysis, the **Shift** key must be used to additionally select the beam(s) prior to selecting the **Bridge > Single Tendon > Spreadsheet Import** menu option.



Analysis type

Refer to the **single tendon prestress wizard** overviews for details on the analysis types supported.

Tendon details

- **Tendon sampling points** are the number of locations along a spline at which calculated equivalent tendon loads will be applied to the model. Note that the original points used to define the spline have no bearing on any calculations that are carried out, they are simply used to ensure a good tendon profile is obtained.

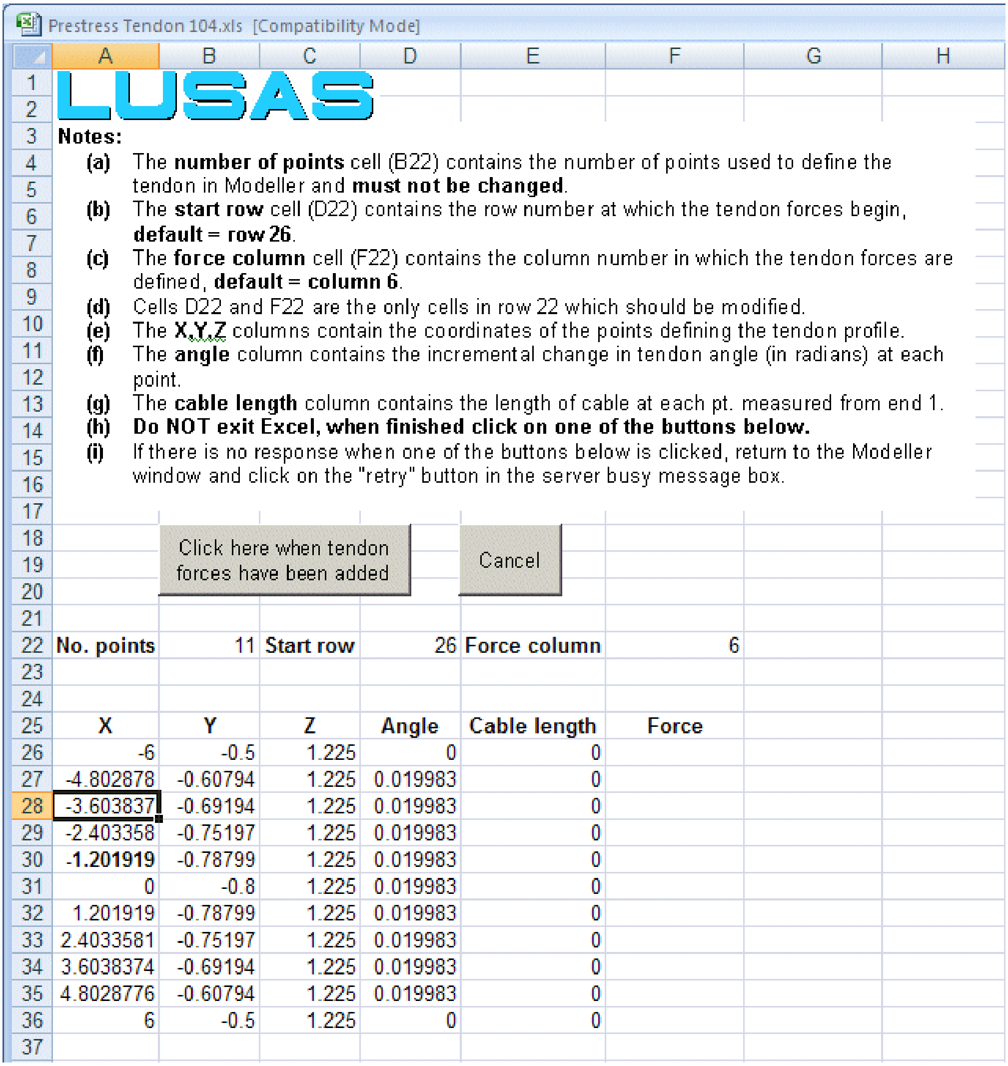
Spreadsheet

A filename for the spreadsheet is automatically entered for creation in the current working directory. When the **OK** button is clicked the previously selected tendon geometry is written to the spreadsheet along with angle and cable length data. The spreadsheet is then automatically opened to allow tendon forces to be manually added. Detailed notes are included on the spreadsheet along with two buttons. One is to be clicked when the tendon forces have been added, the other cancels all input. On pressing the **Click here when tendon forces have been added** button the forces are then read into Modeller and the procedure follows that described for any of the design codes. An example spreadsheet is shown below.

Jacking details

Jacking end slip and jacking angle can be defined for end 1 only, end 2 only, or for both ends of the tendon. End 1 is the start of the spline used to define the tendon.

Example Spreadsheet



Prestress Tendon 104.xls [Compatibility Mode]

LUSAS

Notes:

(a) The **number of points** cell (B22) contains the number of points used to define the tendon in Modeller and **must not be changed**.

(b) The **start row** cell (D22) contains the row number at which the tendon forces begin, **default = row 26**.

(c) The **force column** cell (F22) contains the column number in which the tendon forces are defined, **default = column 6**.

(d) Cells D22 and F22 are the only cells in row 22 which should be modified.

(e) The **X,Y,Z** columns contain the coordinates of the points defining the tendon profile.

(f) The **angle** column contains the incremental change in tendon angle (in radians) at each point.

(g) The **cable length** column contains the length of cable at each pt. measured from end 1.

(h) **Do NOT exit Excel, when finished click on one of the buttons below.**

(i) If there is no response when one of the buttons below is clicked, return to the Modeller window and click on the "retry" button in the server busy message box.

[Click here when tendon forces have been added] [Cancel]

No. points	11	Start row	26	Force column	6

X	Y	Z	Angle	Cable length	Force
-6	-0.5	1.225	0	0	
-4.802878	-0.60794	1.225	0.019983	0	
-3.603837	-0.69194	1.225	0.019983	0	
-2.403358	-0.75197	1.225	0.019983	0	
-1.201919	-0.78799	1.225	0.019983	0	
0	-0.8	1.225	0.019983	0	
1.201919	-0.78799	1.225	0.019983	0	
2.4033581	-0.75197	1.225	0.019983	0	
3.6038374	-0.69194	1.225	0.019983	0	
4.8028776	-0.60794	1.225	0.019983	0	
6	-0.5	1.225	0	0	

Notes

- If tendon realignment is required the previously assigned discrete loading properties should be removed from the model and the Prestress Definition from from Spreadsheet option re-run to calculate the new geometric properties for the new tendon alignment.

Multiple Tendon Prestress Wizard

The Multiple Tendon prestress wizard is accessed from the **Bridge > Prestress Wizard** menu item and is for use with beam elements. It takes account of short and long term losses and can take into account elastic shortening due to stressing of other tendons according to a specified design code or user-defined percentage losses. If the option to ignore effects due to elastic shortening is chosen, the loading computed will be the same as that calculated by the single tendon wizard.

The wizard can be used for a single tendon equivalent nodal loading calculation, but it is primarily for use with multiple tendons and particularly for staged construction where it can calculate and allow for the changing effects of the casting sequence on the tendon properties.

Usage

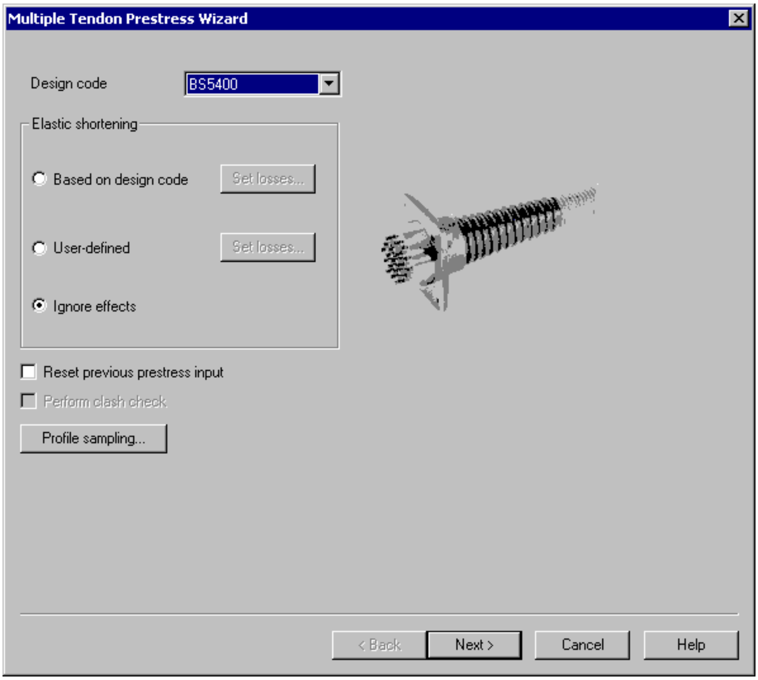
The wizard consists of five pages to assist with tendon definition and assignment and involves the following steps:

1. Selection of design code
2. Definition of tendon profile data
3. Definition of tendon properties according to the design code selected
4. Definition of tendon loading
5. Tendon assignment to loadcases and lines in the model

Notes

- Re-running the Multiple Tendon Prestress Wizard deletes all previously created data before re-calculating for a modified model.
- Use of the multiple tendon prestress facility is described in the example ‘Segmental Construction of a Post Tensioned Bridge’. See the *Application Examples Manual (Bridge, Civil & Structural)* for details.

Selection of Design Code



Design codes supported

Prestress definition varies slightly according to the design code selected. The following design codes are supported:

- **AASHTO LRFD 2nd Edition**
- **AASHTO LRFD 5th Edition**
- **AASHTO LRFD 6th Edition**
- **AASHTO LRFD 7th Edition**
- **BS5400-4:1990**
- **DD EN1992-1-1:1992 Eurocode 2**
- **EN 1992-1-1:2004 Eurocode 2**
- **JTG D62-2004**

Elastic shortening

- Selecting **Based on design code** will allow the elastic shortening to be calculated as described in the selected design code.

- When BS5400-4:1990 is chosen in combination with the Based on design code option the **Set losses…** button is enabled. This allows the average tendon force remaining after attaching subsequent tendons to be specified as a percentage of the total force. This force is applied to all loading that is assigned to the model. Clause 6.7.2.3 of BS5400 Part IV details what is required for elastic shortening.
- When any of **AASHTO LRFD** codes or the **Eurocode** codes are chosen in combination with the Based on design code option the **Set losses…** button will remain disabled and the elastic shortening will be calculated automatically according to clauses specified in each code:

- For **AASHTO LRFD 2nd, 5th, 6th or 7th Edition** the elastic shortening is calculated as described in clause 5.9.5.2.3b.
- For **DD EN1992-1-1:1992 Eurocode 2** elastic shortening is calculated as described in clause 4.2.3.5.5(6).
- For **EN 1992-1-1:2004 Eurocode 2** elastic shortening is calculated as described in clause 5.10.5.1(2).

- Selecting **User-defined** allows the percentage of applied load remaining after subsequent tendons are attached to the same line to be defined. See **Elastic Shortening Average (Multi-tendon)** for more information.
- Selecting **Ignore effects** will not apply any elastic shortening effects to the tendons.
- **Reset previous prestress input** If previous prestress data has been stored in the model, the Reset previous input data option causes the prestress wizard to ignore previous input and allow new data to be entered. If the wizard is cancelled at any time, the previous data saved in the model file will not be lost. Data will only be overwritten when the finish button is chosen on the final dialog.
- **Perform clash check** This is not currently available.

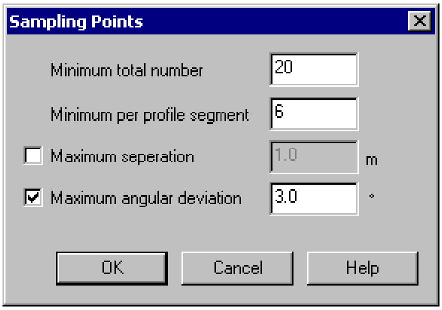
Profile sampling

- Defines the locations along a tendon profile at which calculated equivalent tendon loads will be applied to the model. Note that the original points used to define the line, arc or spline have no bearing on any calculations that are carried out, they are simply used to ensure a good tendon profile is obtained.

Notes

- The wizard expects the input units to be the same as the current model, unless otherwise stated on the dialogs.
- When the prestress loads are calculated the prestress forces are converted into the current model units.
- The current model units can be found on the status bar of LUSAS Modeller.

Tendon Profile Sampling Points



For each line segment (that is for each line, arc or spline) of each tendon profile, tendon sampling points are calculated according to values specified on the Sampling Points dialog. Tendon sampling points are the locations along a tendon profile at which calculated equivalent tendon loads will be applied to the model. Note that the original points used to define the line, arc or spline have no bearing on any calculations that are carried out. They are simply used to ensure a good tendon profile is obtained.

- **Minimum total number** specifies the minimum number of sampling points that will be accepted throughout the tendon profile.
- **Minimum per profile segment** specifies the minimum number of sampling points that will be accepted per line segment of the tendon profile.
- **Maximum separation** specifies the largest distance between sampling points (in modelling units) that will be accepted along each line segment of tendon profile.
- **Maximum angular deviation** specifies the maximum angle of deviation between segments that will be accepted.

From the Minimum total number, Minimum per profile segment and Maximum separation value entries specified a calculated number of sampling points for each line segment will be arrived at. The Maximum angular deviation value will introduce additional sampling points in line segments where the line segment curvature is high.

Notes

- Note that the maximum separation distance is input and displayed in current model units. Consistently with other inputs in Modeller, changing the units after entering a value changes the meaning, not the numerical value that was previously input.

Definition of Tendon Profile (Multiple Tendon Prestress Wizard)

This page of the **multiple tendon prestress wizard** allows tendon profile data to be defined manually, be copied and pasted from a spreadsheet, or be generated by selecting lines, arcs or splines that are defined or imported into Modeller. It supports tendon definition by coordinates in 3D space or by defining coordinates in two 2D planes.

Tendon profiles defined when using this dialog are stored in the Utilities Treeview and are not directly assignable to geometry. Instead, they are read by the multiple tendon prestress wizard and appear on the tendon profile drop-down lists when the Tendon Loading assignment page is in use to allow assignment to lines in the model.

Tendon profile data defined using coordinates and radii is generally referred to in this topic as primary tendon profile data. This is because a smoothing facility may be used to define a minimum radius around or through which two intersecting tendon definition lines will be shaped, resulting in a modified profile. Note that the calculated tendon profile shape (after all smoothing has been applied) can be included in a tendon summary report in a format that is suitable for setting-out the tendon on-site.

Creating a tendon profile

The Tendon Profile dialog can be used to create any number of tendon profiles.

- To create a new tendon profile enter a tendon name and select the **Create** button. Coordinate data can then be entered for this new profile in the blank grid. Use **Rename** and **Delete** to change and remove tendon profile names from the Utilities Treeview.
- Over-typing a new tendon profile name with existing tendon profile data displayed will duplicate that data into a new definition.

Previously defined tendon profile data can be accessed using the drop-down list and the up/down control alongside the tendon profile name.

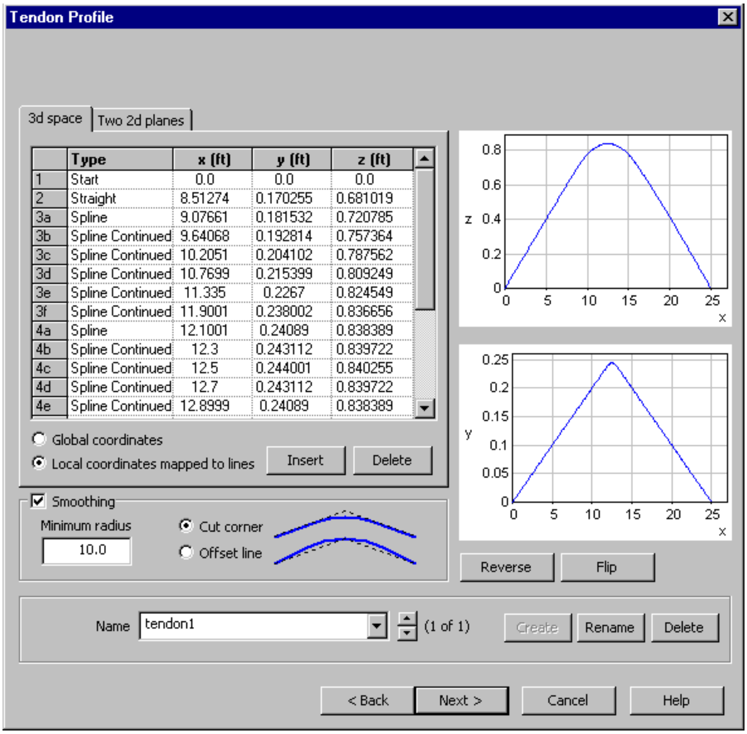
Tendon profiles can be defined by:

- ❑ **Coordinate input**
- ❑ **Spreadsheet import**
- ❑ **Drawing a tendon profile in Modeller**

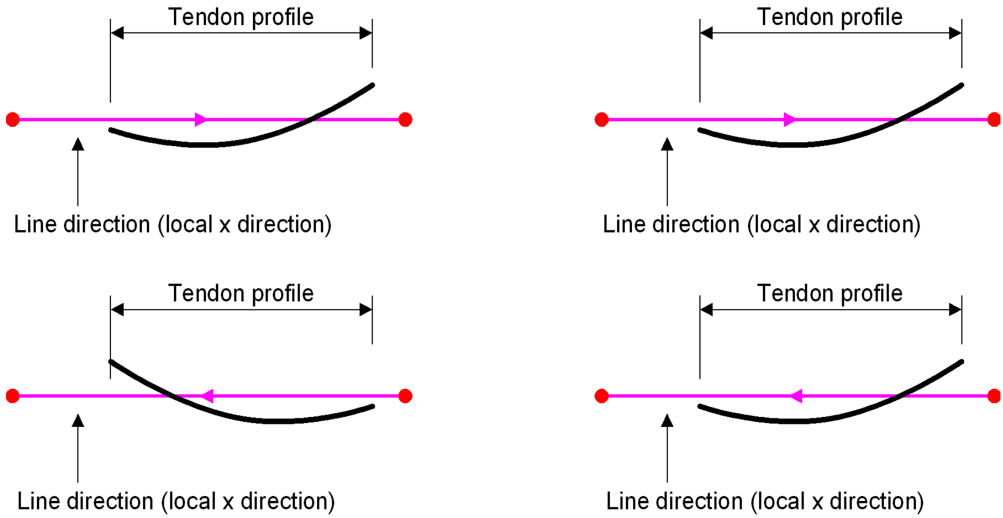
Defining a tendon profile by coordinate input

Defining a tendon profile in 3D space

This requires the selection of the 3D space tab and the input of X,Y and Z coordinates to define a tendon. These can be defined as global or local coordinates.



Primary tendon profile data is defined by adding rows to the grid and specifying the coordinates of each defining point along the tendon. The profile coordinates can be entered in either **Global coordinates** or **Local coordinates mapped to lines**. Local coordinates are relative to the lines the profile is assigned to using the multiple tendon prestress wizard.



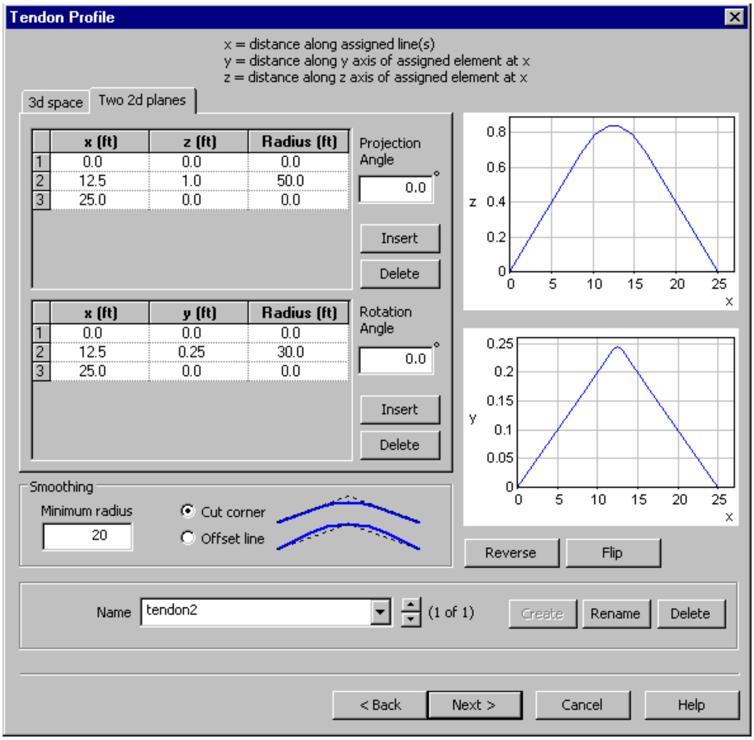
Local coordinates mapped to lines **Global coordinates**

As the coordinates are entered, the tendon shape that they define is visualised in the adjacent graphs. Use the tab key to create a new row beneath the last row of entered data. Note that the use of different scales for the vertical and horizontal axes of the graphs can lead to some apparently visually impossible radii fitting at line intersections.

- **Type** Clicking on the drop-list button in this cell of the table allows the line segment type to be specified. Options are **Straight**, **Arc Bulge**, **Arc Centre**, **Arc End**, **Spline**, **Spline Continued**, **Parabola Bulge** and **Parabola End**
- The **Insert** and **Delete** buttons provide the means to create a new row above a selected row, or to delete rows.
- The **Reverse** button reverses the order of all of the rows in the table whilst maintaining the tendon shape.
- The **Flip** button mirrors the shape of the tendon profile about its mid-x coordinate.

Defining a tendon profile in two 2D planes

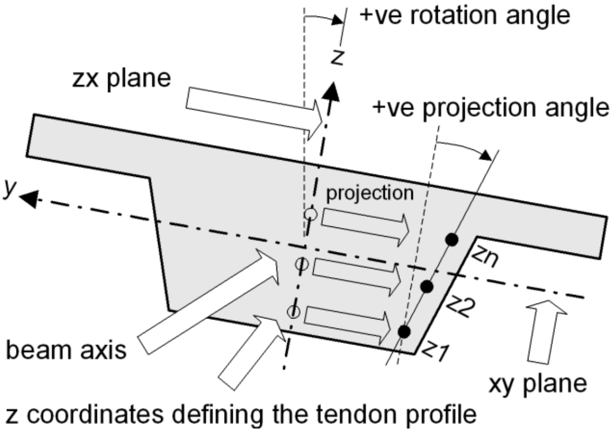
Defining primary tendon profile data in two 2D planes, as opposed to the more common definition in 3D space, is a preferred method in certain countries. This requires the selection of the Two 2D planes tab and the input of X,Y and radius values for both ZX and XY planes. Tendon profile data can be different in each plane. Radius values need only be defined for line intersections. Projection and rotation angles can additionally be defined to cope with inclined webs and rotated beam sections.



Primary tendon profile data is defined for each plane by adding rows to the grid and specifying the in-plane coordinates of each defining point along the tendon. Use the tab key to create a new row beneath the last row of entered data. As the values are entered, the tendon shape that they define is visualised in the adjacent graphs. Note that the use of different scales for the vertical and horizontal axes of the graphs can lead to some apparently visually impossible radii fitting at intersections. Use of a zero radius generates point intersections.

- The **Insert** and **Delete** buttons provide the means to create a new row above a selected row, or to delete rows. These manipulations can be carried out independently for each of the two planes.
- The **Reverse** button reverses the order of all of the rows in the table whilst maintaining the shape.
- **Flip** mirrors the shape of the tendon profile about its mid-x coordinate.
- **Projection angle** Use of a projection angle overcomes the difficulties in calculating z distances of an inclined elliptically shaped tendon in a 2D plane. Using a projection angle causes the z coordinates defining the shape of the tendon to be projected to the plane on which they would lie. For a vertical z axis the projection angle is that between the zx plane and the inclined plane onto which the zx curve will be projected. For a box section this would be the angle between vertical and the inclined web (and would be zero if the web was vertical)

- **Rotation angle** This rotates a tendon definition about the beam axis to which it has been assigned. For a beam with a vertical z axis this would be the angle between the zx plane and vertical. For a box section this might represent camber of the whole section (zero if the section is level).



Smoothing

Smoothing allows the definition of a minimum radius around or through which two intersecting tendon definition lines will be shaped.

There are two options:

- **Cut corner** adds a radius transition between two lines inside of their defined intersection point (as per the diagram on the dialog.
- **Offset line** adds a radius transition between two lines through their defined intersection point

For primary tendon profile data defined in 3D space, if a minimum smoothing radius is defined it will be applied at all intersections along the profile. Use of a zero radius generates point intersections.

For primary tendon profile data defined in two 2D planes a minimum smoothing radius will only be applied at intersections if the radii otherwise defined for those intersections are smaller.

Defining a tendon profile by spreadsheet import

Tendon geometry and line segment information can be imported from a spreadsheet using standard copy and paste facilities. Both four column (type and coordinate data) and three column (coordinate data only) widths are supported. Where the copied data includes the line segment 'Type' the text is expected to match the drop-down list item names. Three column data is expected to be numeric and will paste into the coordinate cells wherever it is pasted.

Defining a tendon profile in Modeller

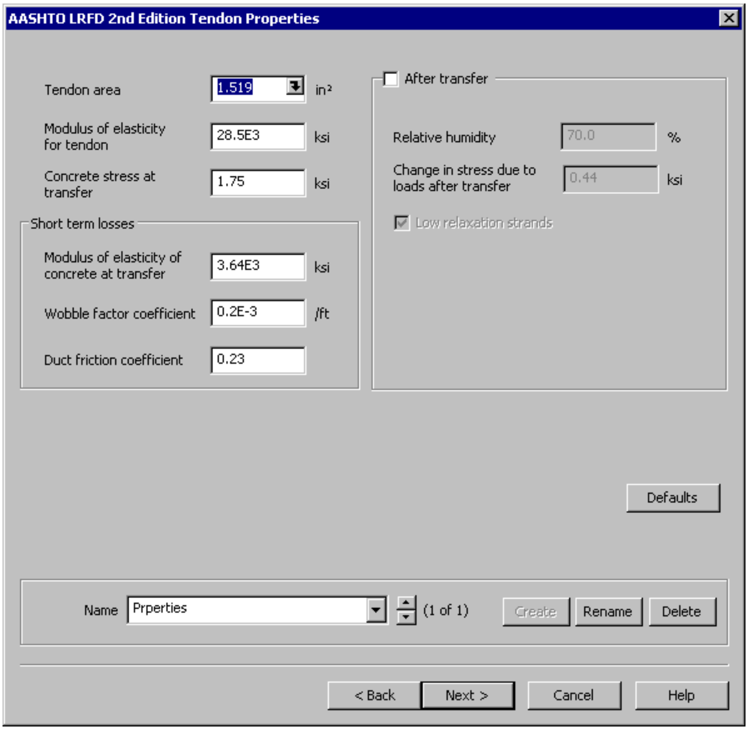
Lines, arcs and splines that represent a tendon can, alternatively, be defined in LUSAS Modeller and then be selected prior to accessing this dialog. Line segments will be listed in the table with their setting-out information.

Notes

- If the line to which the tendon is assigned is an arc or spline, and the tendon profile is generated with a straight local x-axis, this will be “wrapped” to follow the path of the assigned curve, and therefore there is no need to curve the tendon profile. In this situation it may be better to specify the profile as a curve in global coordinates.
- Tendon profile data defined within the Prestress Wizard is also stored in the Utilities Treeview. Editing of tendon profile data in the Utilities Treeview (that is, outside of the prestress wizard) is permitted but all attributes in the Attributes Treeview that were previously created by a Prestress Wizard will be marked with the warning symbol to show that the Prestress Wizard must be re-run to recalculate new values.
- It is not possible to convert 3D space data tendon profile definition into a Two 2D Plane tendon profile definition.
- Tendon profiles can also be defined in isolation and outside of the multiple tendon prestress wizard by using the **Bridge > Prestress Wizard > Tendon Profile** menu item.
- For tendon profiles defined in two 2D planes there is no requirement for the number of inputs to be the same for the XY and YZ planes or for the X coordinates to be coincident, although the first and last sets of values in both grids must span the same X range.
- Tendon profiles defined in two 2D planes can be viewed in 3D tendon coordinate format (by pressing the 3D Space tab). Changes can be made to 3D Space data viewed in this way but if made, the two 2D planes definition will be lost.

Defining AASHTO LRFD 2nd Edition Tendon Properties (Multi-tendon)

This page of the **multiple tendon prestress wizard** allows tendon properties including short and long and term losses to be entered.



- To create a tendon property enter an attribute name and select the **Create** button. Then enter tendon property data as described below:

Tendon details

- **Tendon area** [A_{ps}]. In general, for internal prestressing, tendons of modest size are advisable, to avoid difficulties in housing and anchoring larger tendons with the attendant increase in thickness of members and in the weight of reinforcement. 19 No 0.6" strand (Aps=4.123in²) is considered to be a practical upper limit per tendon. However, the Prestress Wizard will not preclude tendons of larger area.
- **Modulus of elasticity for tendon** [E_p]. Refer to AASHTO 2nd clause 5.4.4.2. The default value is 28,500ksi, representing strand.
- **Concrete stress at transfer** [f_{cgp}]. This value is used for the calculation of elastic shortening losses according to AASHTO 2nd eqn 5.9.5.2.3b-1 and, when long-term losses are requested, for creep losses using eqn 5.9.5.4.3-1. Creep losses are based on a single value of f_{cgp} for each tendon (combined with Δf_{cdp} below as appropriate), and the elastic shortening losses are based on a single f_{cgp} value for each member (calculated from the values given for all the applicable tendons). The value entered for f_{cgp} here should be the stress in the concrete adjacent to the tendon in question, immediately after tensioning and anchoring, due to total prestressing forces and self-weight, at the section of peak moment. Variations in stress arising from permanent

actions applied after prestressing (generally but not exclusively leading to elastic shortening gains) are not incorporated in the calculation of initial elastic shortening loss.

Short term losses

- **Modulus of elasticity for concrete at transfer** [E_{ci}]. This value is used for the calculation of elastic shortening losses according to AASHTO 2nd eqn 5.9.5.2.3b-1. It may be derived from the formula given in clause 5.4.2.4 (substituting f'_{ci} in place of f'_c). In the apparent absence of guidance on values for f'_{ci} in AASHTO 2nd, AASHTO 5th clause 5.4.2.3.2, states f'_{ci} may be taken as $0.8f'_c$ if the concrete age at the time of initial load is unknown during design. A likely range of values is considered to be Eci = 2522 - 5148ksi.
- **Wobble factor** [K] (per foot). This value is used in the calculation of losses due to friction according to AASHTO 2nd clause 5.9.5.2.2b, and also in determining anchorage losses. Values may be sought in Table 5.9.5.2.2b-1.
- **Duct friction coefficient** [μ]. This value is used in the calculation of losses due to friction, according to AASHTO 2nd clause 5.9.5.2.2b, and also in determining anchorage losses. Values may be found in Table 5.9.5.2.2b-1; the default value μ = 0.23 is for strand in polyethylene ducts, but is also in the suggested range for rigid and semi-rigid galvanised metal sheathing.

After transfer

- **Relative humidity** [H]. This value is used in the calculation of losses due to shrinkage to AASHTO 2nd clause 5.9.5.4.2. H is defined as the "average annual ambient relative humidity (percent)" and guidance can be found in Figure 5.4.2.3.3-1.
- **Change in stress due to loads after transfer** Δf_{cdp}. This value is used for the calculation of the creep losses according to AASHTO 2nd eqn 5.9.5.4.3-1 and is defined in the code as the "change in concrete stress at centre of gravity of prestressing steel due to permanent loads with the exception of the load acting at the time the prestressing force is applied". Applied loads and long-term losses generally oppose the initial prestress which dominates f_{cgp} and so creep effects arising from Δf_{cdp} are subtracted from those arising from f_{cgp} in eqn 5.9.5.4.3-1. Therefore where Δf_{cdp} represents a reduction in stress from f_{cgp}, a positive value for Δf_{cdp} should be entered into eqn 5.9.5.4.3-1 and this dialog. As in the case of f_{cgp}, a value appropriate to the concrete adjacent to the tendon in question may be used.
- **Low relaxation strand** (tick-box). Relaxation losses are calculated according to AASHTO 2nd eqn 5.9.5.4.4c-2, unless this tick-box is invoked, whereon 30% of the value from eqn 5.9.5.4.4c-2 is used in line with the provisions of clause 5.9.5.4.4c.

Defaults

The **Defaults** button sets all previously entered values to those specified when the dialog was first displayed. Such values are illustrative only and values for use in design calculations should be checked using appropriate source data.

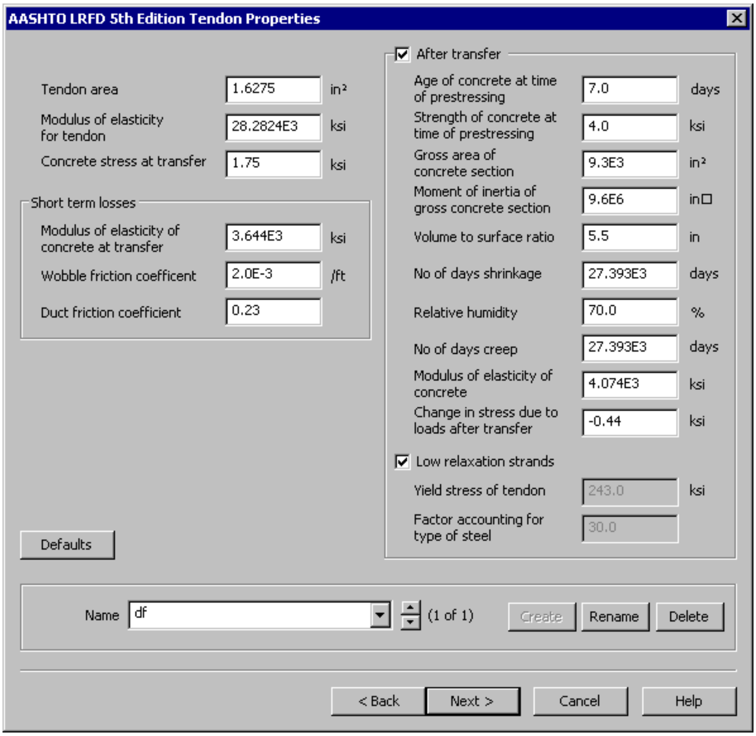
Notes

- The prestress definition dialog expects input units to be the same as the current model unless otherwise stated on the dialogs. When the prestress loads are calculated the prestress forces are converted into the current model units. The current model units can be found on the status bar of LUSAS Modeller.
- An approximate check that the specified jacking force is suitable for the tensile strength of commonly available strand is carried out by the wizard but the user should refer to the relevant design code and manufacturers information for the strand strengths and any applicable factors in order to ensure that the tendons are not overstressed.

Defining AASHTO LRFD 5th Edition Tendon Properties (Multi-tendon)

This page of the **multiple tendon prestress wizard** allows tendon properties including short and long and term losses to be entered.

This page is also applicable to AASHTO LRFD 6th and 7th Editions.



- To create tendon property enter an attribute name and select the **Create** button. Then enter tendon property data as described below:

Tendon details

- ❑ **Tendon area** [A_p]. The cross-sectional area of the tendon (or total area if a group is being represented by a single load assignment). In general, for internal prestressing, tendons of modest size are advisable to avoid difficulties in housing and anchoring larger tendons with the attendant increase in thickness of members and in the weight of reinforcement. 19 No 0.6” strand (A_p=4.123in²) is considered to be a practical upper limit per tendon. However, the Prestress Wizard will not preclude tendons of larger area.
- ❑ **Modulus of elasticity for tendon** [E_p]. Refer to AASHTO 5th clause 5.4.4.2. The default value is 28,500ksi, representing strand.
- ❑ **Concrete stress at transfer** [f_{cgp}]. This value is used for the calculation of elastic shortening losses according to AASHTO 5th eqn 5.9.5.2.3b-1 and, when long-term losses are requested, for creep losses using eqn 5.9.5.4.2b-1. Creep losses are based on a single value of f_{cgp} for each tendon (combined with Δf_{cd}, below, as appropriate), and elastic shortening losses based on a single f_{cgp} value for each member (calculated from the values given for all the applicable tendons). The value entered for f_{cgp} here should be the stress in the concrete adjacent to the tendon in question, immediately after tensioning and anchoring, due to total prestressing forces and self-weight at the section of peak moment. Variations in stress arising from permanent actions applied after prestressing (generally, but not exclusively, leading to elastic shortening gains) are not incorporated into the calculation of initial elastic shortening loss.

Short term losses

- ❑ **Modulus of elasticity for concrete at transfer** [E_{ci}]. This value is used for the calculation of elastic shortening losses according to AASHTO 5th eqn 5.9.5.2.3b-1. It may be derived from the formula given in clause 5.4.2.4 (substituting f'_{ci} in place of f'_c). According to clause 5.4.2.3.2, f'_{ci} may be taken as $0.8f'_c$ if the concrete age at the time of the initial load is unknown at the design stage. A likely range of values would be E_{ci} = 2525 - 5422ksi.
- ❑ **Wobble friction coefficient** [K (per foot)]. This value is used in the calculation of losses due to friction according to AASHTO 5th clause 5.9.5.2.2b and also in determining anchorage losses. Values may be sought in Table 5.9.5.2.2b-1
- ❑ **Duct friction coefficient** [μ]. This value is used in the calculation of losses due to friction, according to AASHTO 5th clause 5.9.5.2.2b, and also in determining anchorage losses. Suitable values may be found in Table 5.9.5.2.2b-1; the default value μ = 0.23 is for strand in polyethylene ducts, but is also in the suggested range for rigid and semi-rigid galvanised metal sheathing.

After transfer

- ❑ **Age of concrete at time of prestressing** [t_i]. This value is used for the calculation of losses due to creep to AASHTO 5th clause 5.9.5.4.3b.

- **Strength of concrete at time of initial loading** [f'_{ci}]. This value is used in the calculation of losses due to creep to AASHTO 5th clause 5.9.5.4.3b. f'_{ci} is defined as the "specified compressive strength of the concrete at the time of initial loading or prestressing; nominal concrete strength at time of application of tendon force (ksi)". According to clause 5.4.2.3.2, f'ci may be taken as 0.8f'c if concrete age at time of initial load is unknown at design stage.
- **Gross area of concrete section** [A_c], and **Moment of inertia of gross concrete section**, [I_c]. These values are used for the calculation of losses due to shrinkage, to AASHTO 5th clause 5.9.5.4.3a, and creep, to clause 5.9.5.4.3b. Where the concrete section varies, suitable intermediate values may be required.
- **Volume to surface ratio** [V/S]. This value is used in the calculation of losses due to shrinkage to AASHTO 5th clause 5.9.5.4.3a, and creep, to clause 5.9.5.4.3b. Details on how the ratio should be calculated are given in clause 5.4.2.3.2.
- **No of days shrinkage** [t]. This value is used for the calculation of losses due to shrinkage to AASHTO 5th clause 5.9.5.4.3a. t is defined as the "number of days between end of curing and time being considered for analysis of shrinkage effects". The default value is based on the design life for a bridge, which is 75 years according to clause 1.2.
- **Relative humidity** [H]. This value is used in the calculation of losses due to shrinkage, to AASHTO 5th clause 5.9.5.4.3a, and creep, to clause 5.9.5.4.3b. H is defined as the "average ambient mean relative humidity (percent)" and guidance can be found in Figure 5.4.2.3.3-1.
- **No of days creep** [t]. This value is used for the calculation of losses due to creep to AASHTO 5th clause 5.9.5.4.3b. t is defined as the "no of days between application of load and time considered for creep calculation". The default value is based on the design life for a bridge, which is 75 years, according to clause 1.2.
- **Modulus of elasticity of concrete** [E_c]. This value is used for the calculation of losses due to creep to AASHTO 5th clause 5.9.5.4.3b. E_c may be calculated for a given concrete grade from the equation in clause 5.4.2.4.
- **Change in stress due to loads after transfer** [Δ_{fcd}]. This value is used for the calculation of the creep losses according to AASHTO 5th eqn 5.9.5.4.2b-1, and is defined in the code as the "change in concrete stress at centroid of prestressing strands due to long-term losses between transfer and deck placement, combined with deck weight and superimposed loads". In fact, a value appropriate to the concrete adjacent to the tendon in question may be used. Δ_{fcd} is generally negative since applied loads and long-term losses oppose the initial prestress that dominates f_{cgp}.
- **Low relaxation strands** (tick-box). When invoked, the relaxation loss $\Delta f_{pR2} = \Delta f_{pR1} =$ 1.2ksi is be assumed according to AASHTO 5th clause 5.9.5.4.2c, and the text boxes for f_{pt} and K_L are greyed out. If this assumption is deemed inappropriate, the box should be unticked and the appropriate data entered
- **Yield stress of tendon** [f_{py}]. This value is used for the calculation of relaxation losses according to AASHTO 5th eqn 5.9.5.4.2c-1. f_{py} is defined as "yield stress of prestressing steel.". Suitable values may be based on clause 5.4.4.1.

- **Factor accounting for type of steel** [K_L]. This value is used for the calculation of relaxation losses according to AASHTO 5th eqn 5.9.5.4.2c-1, when the assumption of $\Delta f_{pR2} = \Delta f_{pR1} = 1.2$ksi is deemed inappropriate. K_L is defined as "factor accounting for type of steel, taken as 30 for low relaxation strands and 7 for other prestressing steel, unless more accurate manufacturer's data is available.".

Defaults

- The **Defaults** button sets all previously entered values to those specified when the dialog was first displayed. Such values are illustrative only and values for use in design calculations should be checked using appropriate source data.

Notes

- The prestress definition dialog expects input units to be the same as the current model unless otherwise stated on the dialogs. When the prestress loads are calculated the prestress forces are converted into the current model units. The current model units can be found on the status bar of LUSAS Modeller.

Defining AASHTO LRFD 6th Edition Tendon Properties (Multi-tendon)

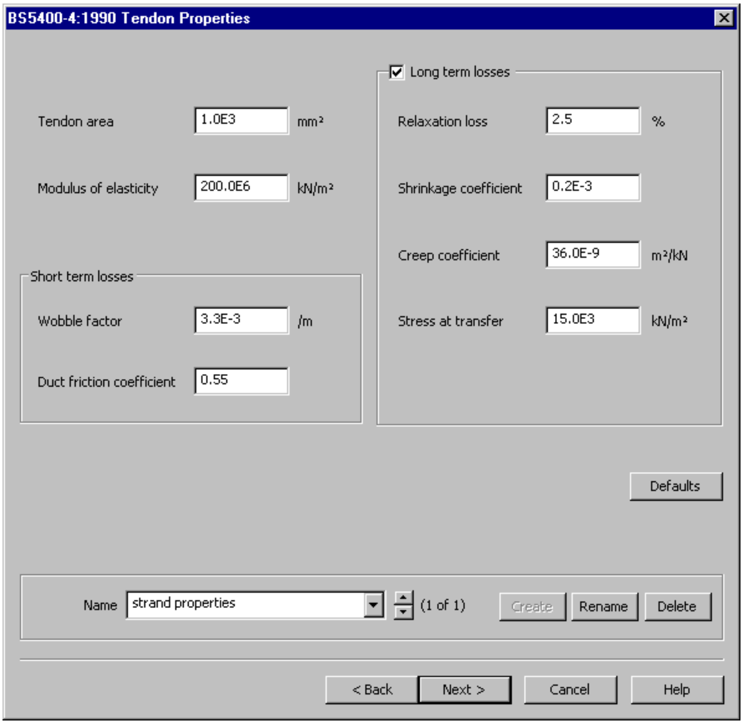
For details of AASHTO LRFD 6th Edition Tendon Properties please refer to **AASHTO LRFD 5th Edition Tendon Properties (Multi-tendon)**. All relevant clauses in AASHTO LRFD 6th Edition are identical to those in AASHTO LRFD 5th Edition.

Defining AASHTO LRFD 7th Edition Tendon Properties (Multi-tendon)

For details of AASHTO LRFD 7th Edition Tendon Properties please refer to **AASHTO LRFD 5th Edition Tendon Properties (Multi-tendon)**. All relevant clauses in AASHTO LRFD 7th Edition are identical to those in AASHTO LRFD 5th Edition.

Defining BS5400-4:1990 Tendon Properties (Multi-tendon)

This page of the **multiple tendon prestress wizard** allows tendon properties including short and long and term losses to be entered.



- To create a tendon property enter an attribute name and select the **Create**button. Then enter tendon property data as described below:

Tendon details

- ❑ **Tendon area** The cross-sectional area of the tendon (or total area of group of strands if a group is being represented by a single load assignment)
- ❑ **Modulus of elasticity**

Short term losses

- ❑ **Wobble factor** is defined per metre length and should be obtained from design code.
- ❑ **Duct friction coefficient** should be obtained from design code.

Long term losses

- ❑ **Relaxation loss** is a percentage of the prestress force.
- ❑ **Shrinkage Coefficient**
- ❑ **Creep coefficient**
- ❑ **Stress at transfer** Compressive stress in the concrete adjacent to the tendon due to prestress loading only, averaged along the length of the tendon.

Defaults

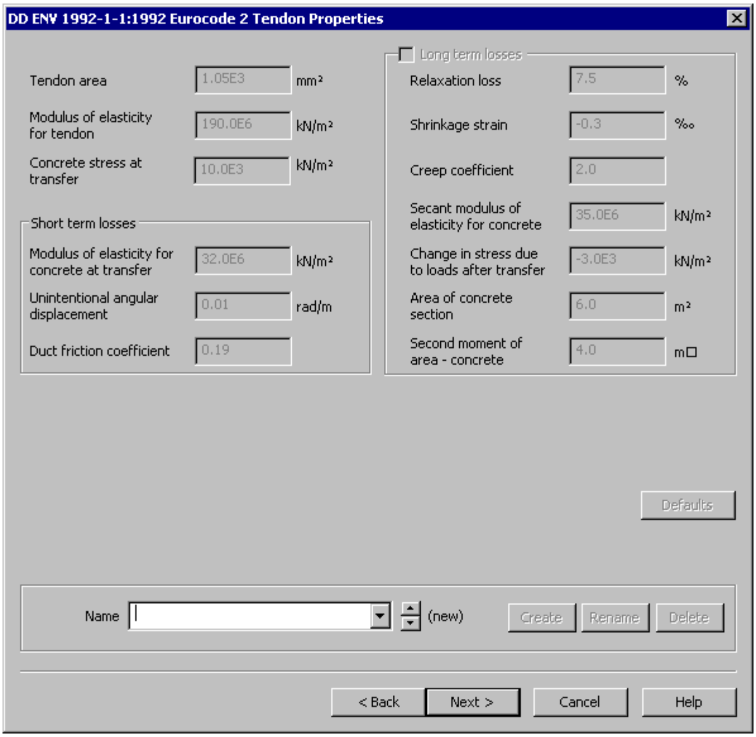
The **Defaults** button sets all previously entered values to those specified when the dialog was first displayed.

Notes

- The prestress definition dialog expects input units to be the same as the current model unless otherwise stated on the dialogs. When the prestress loads are calculated the prestress forces are converted into the current model units. The current model units can be found on the status bar of LUSAS Modeller.

Defining DD EN1992-1-1:1992 Eurocode 2 Tendon Properties

This page of the **multiple tendon prestress wizard** allows tendon properties including short and long and term losses to be entered.



- To create tendon property enter an attribute name and select the **Create** button. Then enter tendon property data as described below:

Tendon details

- **Tendon area** [A_p]. The cross-sectional area of the tendon (or total area of group of strands if a group is being represented by a single load assignment). In general, for internal prestressing, tendons of modest size are advisable to avoid difficulties in housing and anchoring larger tendons with the attendant increase in thickness of members and in the weight of reinforcement. 19 No 15mm strands (A_p=2850mm²) or 27 No 13mm strands (A_p=2700mm²) might be a practical upper limit per tendon. However, the Prestress Wizard will not preclude tendons of larger area.
- **Modulus of elasticity of tendon** [E_s] . Refer to DD ENV 1992-1-1 clause 3.3.4.4 (2). The default value for the modulus of elasticity, E_s, is assumed to be 190kN/mm², representing strand. Values typically fall in the range of 185kN/mm² to 210kN/mm².
- **Concrete stress at transfer** [σ_{co}]. This value is used for the calculation of elastic shortening losses to DD ENV1992-1-1 clause 4.2.3.5.5(6) and, when long-term losses are requested, the creep component of time-dependent losses to clause 4.2.3.5.5(9) eqn 4.10. Creep losses are based on a single value of σ_{co} for each tendon (combined with $\Delta\sigma_{cg}$ below, defined as appropriate), and the elastic shortening losses are based on a single σ_{co} value for each member (calculated from the values given for all the applicable tendons). The value entered for σ_{co} here should be the stress in the concrete adjacent to the tendon in question, immediately after tensioning and anchoring, due to total prestressing forces and self-weight averaged along the length of the tendon. Variations in stress arising from permanent actions applied after prestressing (generally but not exclusively leading to elastic shortening gains) are not incorporated in the calculation of initial elastic shortening loss.

Short term losses

- **Modulus of elasticity of concrete at transfer** [$E_{cm(t)}$]. This value is used for the calculation of elastic shortening losses to DD ENV1992-1-1 clause 4.2.3.5.5(6). This is the secant modulus of elasticity for concrete, adjusted for concrete age at the time of application of the load, under the provisions of DD ENV 1992-1-1 clause 3.1.2.5.2(4). There are no formulae for the strength of concrete developing with time in DD ENV 1992-1-1, therefore reference may be made to the CEB-FIP Model Code 1990, clauses 2.1.6.1 and 2.1.3.2, or EN1992-1-1:2004 clause 3.1.2. A likely range of values would be 24-37kN/mm².
- **Unintentional angular displacement** [k]. This value is used in the calculation of losses due to friction, according to DD ENV1992-1-1 clause 4.2.3.5.5(8), for the component sometimes referred to as “wobble” loss. It is also used in the calculation of anchorage losses. k is defined as “unintentional angular displacement (per unit length) related to the profile of the tendons". Clause 4.2.3.5.5(8) gives a range of $0.005 < k < 0.01$ /m.
- **Duct friction coefficient** , μ. This value is used in the calculation of losses due to friction, according to DD ENV 1992-1-1 clause 4.2.3.5.5(8). It is also used in the

calculation of anchorage losses. Suitable values may be found in clause 4.2.3.5.5(8); the default value $\mu = 0.19$ is for internal tendons made up of strand filling roughly half of the duct.

Long term losses

- **Relaxation loss** [ρ_t]. This value is used for the calculation of $\Delta_{\sigma pr}$, the absolute value of relaxation loss, which is incorporated into the time dependent loss calculation of DD ENV 1992-1-1 eqn 4.10. According to clause 4.2.3.4.1(2), the long term value of the relaxation loss may be assumed to be three times the relaxation losses after 1000h, which may in turn be taken from Figure 4.8, dependent upon the ratio of the applied prestress to the tensile strength of the tendons and the type of prestressing steel in use. Typical values for class 2 (strand) would therefore be (from Figure 4.8);

  - $\sigma_p/f_{pk} = 0.6$; ρ1000 = 1%; ρ∞ = 3%
  - $\sigma_p/f_{pk} = 0.7$; ρ1000 = 2.5%; ρ∞ = 7.5%
  - $\sigma_p/f_{pk} = 0.8$; ρ1000 = 4.5%; ρ∞ = 13.5%

- **Shrinkage strain** [ε_s(t,t0)]. This value is used in the calculation of losses due to shrinkage according to DD ENV 1992-1-1 clause 4.2.3.5.5(9). ε_s(t,t0) is "the estimated shrinkage strain, derived from the values in Table 3.4 for final shrinkage (see also 2.5.5 and Appendix 1)" and should be specified "per mil" (‰) in accordance with the code (rather than actual strain).
- **Creep coefficient** [φ(t,t0)]. This value is used in the calculation of losses due to creep, according to DD ENV 1992-1-1 clause 4.2.3.5.5(9). Clause 1.7.4 describes φ(t,t0) as the "Creep coefficient, defining creep between times t and t0, related to elastic deformation at 28 days". Clause 4.2.3.5.5(9) refers to clause 2.5.5 that in turn refers to clause 3.1 or (for greater accuracy) Appendix 1. If greater accuracy is not required, a value for φ(∞,t0) may be obtained from Table 3.3, dependent upon notional size, relative humidity and maturity of the concrete when the load is first applied.
- **Secant modulus of elasticity of concrete** [E_{cm}]. This value is used for the calculation of time-dependent losses according to DD ENV 1992-1-1 clause 4.2.3.5.5(9). Equation 4.10 requires a modular ratio, $\alpha = E_s/E_{cm}$, where E_{cm} can be obtained from Table 3.2. A likely range of values would be 29-37kN/mm².
- **Change in stress due to loads after transfer** $\Delta\sigma_{cg}$. This is used to adjust the concrete stress at transfer, σ_{co}, to account for stress in the concrete due to permanent loads applied after prestressing. $\sigma_{co} + \Delta\sigma_{cg}$ gives the concrete stress appropriate for the calculation of the creep component of time-dependent losses according to DD ENV 1992-1-1 clause 4.2.3.5.5(9), eqn 4.10 that is $\sigma_{co} + \Delta\sigma_{cg} = (\sigma_{cg} + \sigma_{cpo})$. $\Delta\sigma_{cg}$ is typically negative, since applied loads oppose the prestress that dominates σ_{co}.
- **Area of concrete section** [Ac] and **Second moment of area – concrete** [Ic]. These values are used for the calculation of time-dependent losses, according to DD ENV 1992-1-1 clause 4.2.3.5.5(9) eqn 4.10. Where the concrete section varies, suitable

intermediate values may be required. If the member is a standalone beam element the area of the beam should be used. If the beam member represents part of a larger structure (for example a certain width of slab) then only the area of the portion represented by this particular beam member should be used. Please note that the prestress wizard is designed to work for a single beam member, not for a subdivided portion of slab or wide beam element, so using it in any other way is at the engineer's judgement.

Defaults

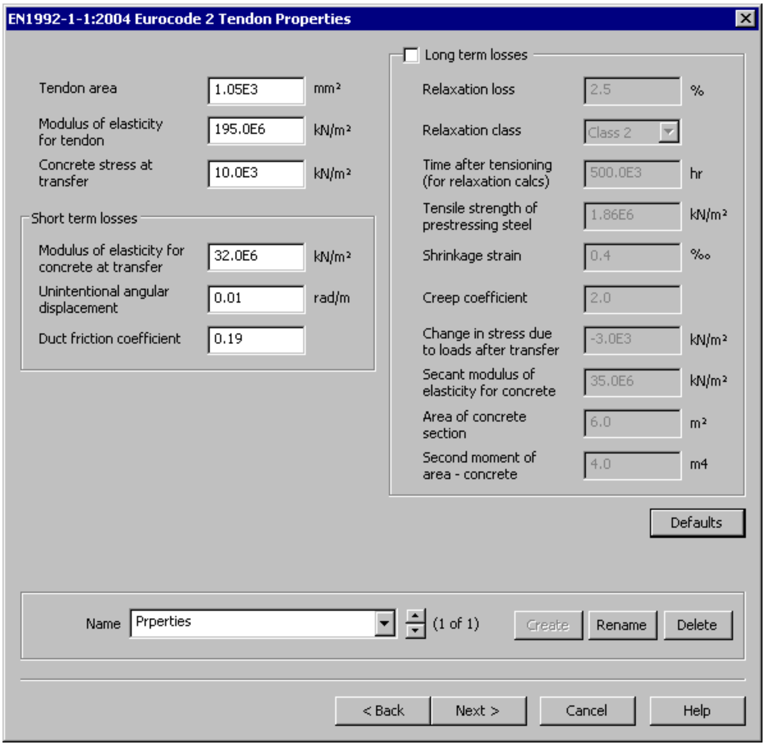
The **Defaults** button sets all previously entered values to those specified when the dialog was first displayed. Such values are illustrative only and values for use in design calculations should be checked using appropriate source data.

Notes.

- The prestress definition dialog expects input units to be the same as the current model unless otherwise stated on the dialogs. When the prestress loads are calculated the prestress forces are converted into the current model units. The current model units can be found on the status bar of LUSAS Modeller.

Defining EN 1992-1-1:2004 Eurocode 2 Tendon Properties (Multi-tendon)

This page of the **multiple tendon prestress wizard** allows tendon properties including short and long and term losses to be entered.



- To create a tendon property enter an attribute name and select the **Create** button. Then enter tendon property data as described below:

Tendon details

- ❑ **Tendon area** [A_p]. The cross-sectional area of the tendon (or total area of group of strands if a group is being represented by a single load assignment). In general, for internal prestressing, tendons of modest size are advisable to avoid difficulties in housing and anchoring larger tendons with the attendant increase in thickness of members and in the weight of reinforcement. 19 No 15mm strands (A_p=2850mm²) or 27 No 13mm strands (A_p=2700mm²) might be a practical upper limit per tendon. However, the Prestress Wizard will not preclude tendons of larger area.
- ❑ **Modulus of elasticity for tendon** [E_p]. Refer to EN 1992-1-1:2004 clause 3.3.6. The default value for the modulus of elasticity, E_p is assumed to be 195GPa, representing strand. Values typically fall in the range of 185GPa to 210GPa.
- ❑ **Concrete stress at transfer** [$\Delta\sigma_c(t)$]. This value is used for the calculation of elastic shortening losses, according to EN1992-1-1 clause 5.10.5.1(2), and, when long-term losses are requested, for the creep component of time-dependent losses, to clause 5.10.6(2). Creep losses are based on a single value of $\Delta\sigma_c(t)$ for each tendon (combined with $\Delta\sigma_{cg}$ below as appropriate), and the elastic shortening losses are based

on a single $\Delta\sigma_c(t)$ value for each member (calculated from the values given for all the applicable tendons). The value entered for $\Delta\sigma_c(t)$ here should be the stress in the concrete adjacent to the tendon in question, immediately after tensioning and anchoring, due to total prestressing forces and self-weight, averaged along the length of the tendon. Variations in stress arising from permanent actions applied after prestressing (generally, but not exclusively, leading to elastic shortening gains) are not incorporated into the calculation of initial elastic shortening loss.

Short term losses

- **Modulus of elasticity for concrete at transfer** [$E_{cm}(t)$]. This value is used for the calculation of elastic shortening losses, according to EN1992-1-1:2004 clause 5.10.5.1(2). $E_{cm}(t)$ is defined as "the secant modulus of elasticity of concrete at time t". From clause 5.10.3, time t=t0 (immediately after tensioning and anchoring) should be used for this calculation. Thus E_{cm} from Table 3.1 may need to be adjusted according to clause 3.1.3(3). A likely range of values would be 26-39MPa.
- **Unintentional angular displacement** [k]. This value is used in the calculation of losses due to friction, according to EN1992-1-1:2004 clause 5.10.5.2, for the component sometimes referred to as "wobble" loss and also in determining anchorage losses. k describes the "unintentional angular displacement" for internal tendons in radians per unit length. Clause 5.10.5.2(3) gives the range $0.005 < k < 0.01$ rad/m.
- **Duct friction coefficient** [μ]. This value is used in the calculation of losses due to friction according to EN1992-1-1:2004 clause 5.10.5.2(1), and also in determining anchorage losses. Suitable values may be found in Table 5.1; the default value $\mu = 0.19$ is for internal tendons made up of strand filling roughly half of the duct.

Long term losses

- **Tensile strength of prestressing steel** [f_{pk}] This value is used for the calculation of $\Delta_{\sigma pr}$, the absolute value of relaxation losses to EN1992-1-1 clause 3.3.2(7) and incorporated into the time dependent loss calculation of clause 5.10.6(2).
- **Relaxation Class** This is used along with f_{pk} to calculate relaxation losses. Classes 1, 2 and 3 are defined in EN1992-1-1 clause 3.3.2(4).
- **Relaxation loss at 1000 hours** [ρ_{1000}]. This is used, along with f_{pk} and the Relaxation Class, to calculate relaxation losses. Typical values may be found in clause 3.3.2(6) and the default value is based on low relaxation strand (Class 2).
- **Time after tensioning (for relaxation calcs)** [t]. This is used, along with f_{pk}, Relaxation Class and ρ_{1000} to calculate relaxation losses. According to EN1992-1-1 clause 3.3.2(8), long term (final) values of the relaxation losses may be estimated for a time (t) equal to 500 000 hours (i.e. around 57 years).
- **Shrinkage strain** [ε_{cs}]. This value is used in the calculation of losses due to shrinkage, according to EN1992-1-1:2004 clause 5.10.6(2). The value should be calculated from clause 3.1.4(6), depending on notional size, relative humidity, concrete strength and cement class, and should be specified "per mil" (‰) in accordance with the code (rather than actual strain).

- **Creep coefficient** [φ(t,t0)]. This value is used in the calculation of losses due to creep, according to EN1992-1-1:2004 clause 5.10.6(2). The value should be calculated from clause 3.1.4(2), dependent upon the same parameters as the shrinkage strain plus the maturity of the concrete when the load is first applied. Values may be taken from Figure 3.1.
- **Secant modulus of elasticity for concrete** [E_{cm}]. This value is used for the calculation of time-dependent losses according to EN1992-1-1:2004 clause 5.10.6(2). The clause refers explicitly to obtaining E_{cm} from Table 3.1. A likely range of values would be 27-44MPa.
- **Change in stress due to loads after transfer** [$\Delta\sigma_{cg}$]. This is used to adjust "Concrete stress at transfer" according to the permanent loads applied after transfer, to give the concrete stress appropriate for the calculation of the creep component of time-dependent losses, according to EN1992-1-1 clause 5.10.6(2), that is, $\sigma_{c,QP} = \Delta\sigma_c(t) + \Delta\sigma_{cg}$. Note that $\Delta\sigma_{cg}$ is typically negative since applied loads oppose the prestress that dominates $\Delta\sigma_c(t)$.
- **Area of concrete section** [Ac], **Second moment of area – concrete** [I_c].The area and second moment of area of the concrete member to which the prestress forces will be applied. These values are used for the calculation of time-dependent losses, according to EN1992-1-1:2004 clause 5.10.6(2). Where the concrete section varies, suitable intermediate values may be required. If the member is a standalone beam element the area of the beam should be used. If the beam member represents part of a larger structure (for example a certain width of slab) then only the area of the portion represented by this particular beam member should be used. Please note that the prestress wizard is designed to work for a single beam member, not for a subdivided portion of slab or wide beam element, so using it in any other way is at an engineer's judgement.

Defaults

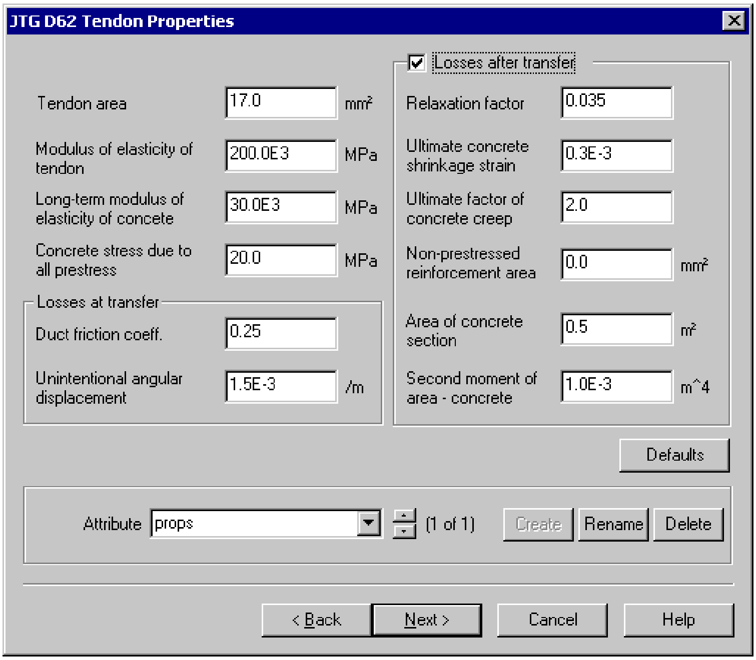
The **Defaults** button sets all previously entered values to those specified when the dialog was first displayed. Such values are illustrative only and values for use in design calculations should be checked using appropriate source data.

Notes.

- The prestress definition dialog expects input units to be the same as the current model unless otherwise stated on the dialogs. When the prestress loads are calculated the prestress forces are converted into the current model units. The current model units can be found on the status bar of LUSAS Modeller.
- An approximate check is carried out by the wizard that the specified jacking force is suitable for the tensile strength of commonly available strand, but the user should refer to EN1992-1-1 clause 5.10.3(2), manufacturer's information for the strand strengths and the appropriate National Annex for values of k7 and k8 in order to ensure that the tendons are not overstressed.

Defining JTG D62-2004 Tendon Properties (Multi-tendon)

This page of the **multiple tendon prestress wizard** allows tendon properties including short and long and term losses to be entered.



- To create tendon property enter an attribute name and select the **Create** button. Then enter tendon property data as described below:

Tendon details

- ❑ **Tendon area** Steel area for the tendon being defined.
- ❑ **Modulus of elasticity of tendon** Young's modulus of prestress reinforcement.
- ❑ **Long-term modulus of elasticity of concrete The long-term Young's modulus of concrete.**
- ❑ **Concrete stress due to all prestress** The stress in concrete due to prestress at the centroid of all tendons. See JTG D62-2004 section 6.1.5.

Losses at transfer

- ❑ **Duct friction coefficient** The friction coefficient between the tendon (prestress reinforcement) and the duct (pipe). See JTG D62-2004 Table 6.2.2.
- ❑ **Unintentional angular displacement** also known as the influence factor of load deviation or wobble. This is defined per metre length. See JTG D62-2004 Table 6.2.2.

Losses after transfer

- **Relaxation factor** See JTG D62-2004 Section 6.2.6.
- **Ultimate concrete shrinkage strain** See JTG D62-2004 Table 6.2.7.
- **Ultimate factor of concrete creep** See JTG D62-2004 Table 6.2.7
- **Concrete prestress due to all prestress** The stress in concrete due to prestress at the centroid of all tendons. See JTG D62-2004 section 6.1.5.
- **Non-prestressed reinforcement area** The sectional area of general (non prestressed) reinforced reinforcement
- **Area of concrete section** The net sectional area of concrete after the area of the pipes (ducts) and other weakened parts have been removed.
- **Second moment of area- - concrete** Moment of inertia of concrete section

Defaults

The **Defaults** button sets all previously entered values to those specified when the dialog was first displayed.

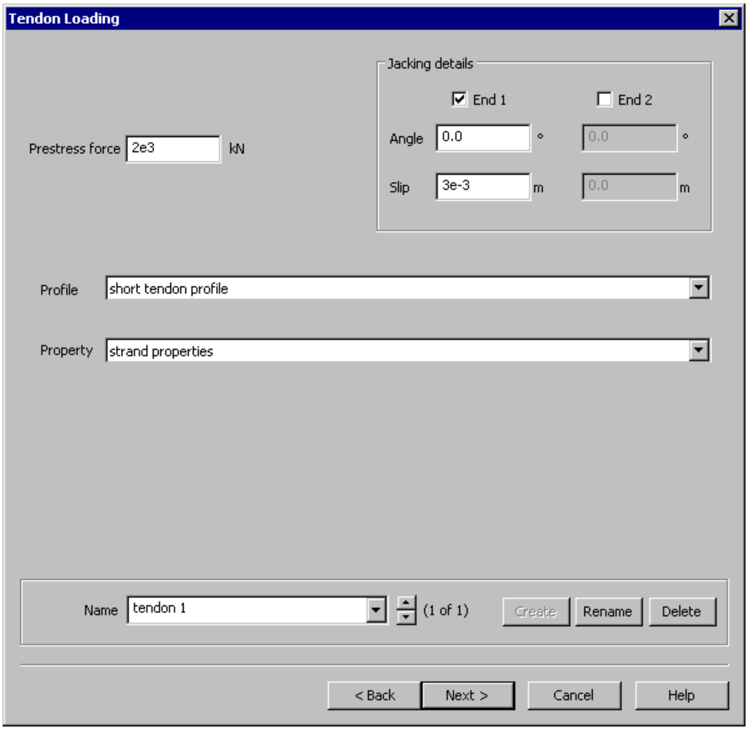
Notes

- The prestress definition dialog expects input units to be the same as the current model unless otherwise stated on the dialogs. When the prestress loads are calculated the prestress forces are converted into the current model units. The current model units can be found on the status bar of LUSAS Modeller.
- The loss due to elastic compression of concrete is computed using the simplified formula in JTG D62-2004 Appendix E.
- Guidance on the deformation of the anchorage device is provided in JTG D62-2004 Table 6.2.3.

Definition of Tendon Loading

This page of the **multiple tendon prestress wizard** allows a tendon load to be associated with previously entered profiles and properties.

The tendon profile and property drop-down lists are populated with entries present in the Utilities Treeview.



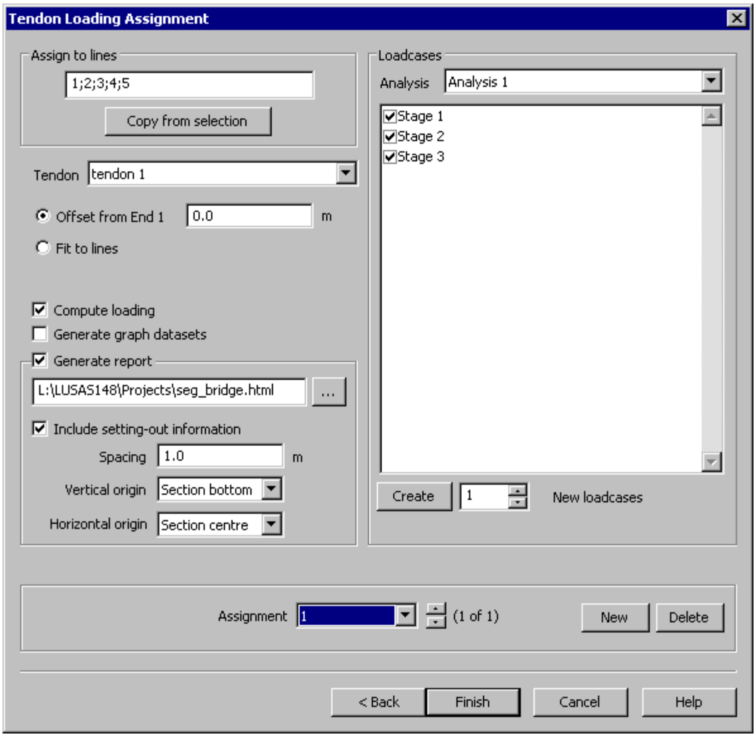
- To create a tendon load enter a dataset title and select the **Create** button. Tendon load data can then be entered.
- The tendon force must be defined in addition to jacking details. Anchorage losses are defined as slippage from either, or both ends.

Notes

- An approximate check that the specified jacking force is suitable for the tensile strength of commonly available strand is carried out by the wizard but the user should refer to the relevant design code and manufacturers information for the strand strengths and any applicable factors in order to ensure that the tendons are not overstressed.

Tendon Loading Assignment

This last page of the **multiple tendon prestress wizard** allows the assignment of the tendon loads that have been defined in the previous dialog.



Assigning tendon loads to loadcases

- Tendon loads can be assigned to single or multiple loadcases within a chosen Analysis using the checklist boxes and drop-down list in the Loadcases panel. Selecting a loadcase will automatically select all subsequent loadcases. To switch the subsequent loadcases off, select the loadcase following the chosen loadcase and this will toggle the activation of the subsequent loadcases.
- New loadcases for tendon assignment can be created by entering a quantity and selecting the **Create** button. These loadcases will not be generated in the model until the tendon calculation is complete.

Assigning tendon loads to lines

- If the **Tendon analysis type** of modelling is being used, before the tendon assignment can be undertaken any lines in the model representing the tendon must be assigned tendon properties must be meshed with beam elements.
- Tendon loads can be assigned to lines entered in the line numbers box directly, or by selecting the lines on the model prior to starting the wizard, and clicking **Copy from**

selection. These lines may be straight or curved and may consist of multiple lines defining each span.

- Checks are made, on saving, that the lines selected for assignment form a single continuous open-ended path. Unconnected lines, multiple paths, branching and /or a closed loop are reported as an error.
- Line numbering and line direction determines the tendon profile direction for the superimposed tendon. Line direction must be consistent along the selected lines representing the beam assignments. Any inconsistencies in line direction will cause an error.
- **Offset from End 1** allows the selected tendon profile to be displaced with reference to the starting point of the line or lines stated in the Assign to Lines cell.
- **Fit to Lines** is used if a tendon profile needs to be contracted or expanded to match the length of a line or series of lines. This process is part of the mapping of the tendon to the lines and does not change the tendon profile definition. The mapping does not provide a uniform scaling so arcs in the profile will become ellipses, and the y and z maxima and minima will not change.

Computing loading

- ❑ **Compute loading** when checked causes the loading calculation and assignments to be carried out on finishing the wizard. When not checked no calculations will take place but the data entered, as long as it is valid, will be stored to the model. This is useful if it is required to exit the wizard during the process of entering data. The **Next>** button on the preceding dialogs can be clicked to get to this final page to enter a single assignment and then exit the wizard assured that the current entered data will be stored to the model.

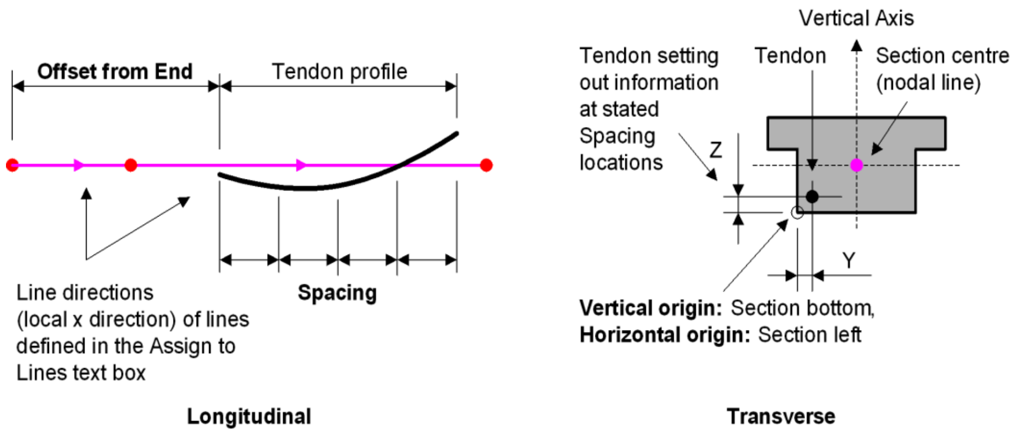
Generating graphs of tendon losses

- ❑ **Generate graph datasets** should be selected if a graphical visualisation of the tendons is required. When the prestress definition is computed graph datasets will be created, which can be graphed using the graph wizard to enable the tendon losses to be visualised against the tendon length.

Generating reports of tendon assignments

- ❑ **Generate report** creates an HTML document useful for quality assurance purposes in the current working folder (or the specified directory). This document is automatically displayed in the default web browser at the end of the calculation. The report comprises a table of contents of all the tendon calculations; summaries for the tendon properties and tendon profile (original input and sampling points used) ; followed by the prestress losses and the prestress load components as used in the LUSAS loading attributes. Tendon setting-out information can be optionally added to this report.
- ❑ **Include setting-out information** The calculated tendon profile shape (after all smoothing has been applied) can be included in a tendon summary report in a format that is suitable for setting-out the tendon on-site.

- Tendon profile setting out data can be tabulated for a defined **Spacing** along a beam with reference to **Vertical origin** and **Horizontal origin**. The origin settings can be set to be an Assigned line or the top or bottom, left or right, or centre of a geometrically defined cross-section. Note that 'Section centre' may differ from 'Assigned line' by any eccentricity that may be applied.
- Each tendon assignment writes a separate setting-out chapter where the setting-out tendon profile information is stated as relative distances between setting-out points. In some cases, several assignments may generate duplicate tables of data, for example where the same profile has been used on many lines, but the details of the assigned lines may be different so a separate report is provided as a safety measure.
- At any given point on the structure the interpretation of 'Section top', 'Section bottom', 'Section left' or 'Section right' depends upon the specified vertical direction (set using Utilities > Vertical Axis), the assigned line, and the direction of increasing x distance along the tendon profile. 'Section top' is always interpreted as vertical, which (for non-level decks) is not necessarily perpendicular to the section at that point. A 'forwards' direction is always along the assigned line, but may be opposite to the line's x-axis, as it must follow the tendon's increasing x value. 'Section left' and 'Section right' are calculated from the cross-product between the 'up' and 'forwards' directions at each point.



Selected tendon profile setting-out terms explained

Completing the Wizard

- Clicking the **Finish** button will save all of the entered data to the model and, if selected, compute the tendon forces and assign them to the relevant loadcases.

Prestress Wizard-created data

In using the Prestress Wizard, prestress loading is added to the analysis model as equivalent discrete point loading. In doing so, search areas are created and used automatically by the prestress wizard to define the target geometry to be loaded. By right-clicking on a tendon loading assignment in the Attributes Treeview and choosing Visualise from the context menu the search area used for that assignment may be viewed.

- Load factors are used to represent elastic shortening. These can be seen for each stage in the Analyses Treeview.
- Graphing datasets (which allow of graphing of prestress losses etc) are added to the Utilities Treeview.
- Tendon profiles defined by the Prestress Wizard are added to the Utilities Treeview.

Editing Prestress Wizard-created data

If any tendon assignments need modifying, the Multiple tendon prestress wizard may be re-run (previously entered values will be retained) and the Next button can be clicked to get to the tendon loading assignment page. Incorrect assignments can be deleted or simply corrected as required before clicking the Finish button to recalculate the equivalent tendon loading on the model.

- Editing of tendon profile data in the Utilities Treeview. (that is, outside of the wizard) is permitted but all attributes that require re-calculating by re-running the prestress wizard will be marked with the warning symbol . Re-running the prestress wizard will update all values accordingly.
- Editing of any attribute data calculated and generated by the prestress wizard is not permitted.

Deletion of prestress attributes

- Deletion of individual prestress attributes is not permitted since it will invalidate any calculations (e.g. elastic shortening) that have been made. Deletion of all prestress attributes (i.e profiles, properties) is possible and is achieved by selecting the **Delete all** menu item on the context menu for each item.
- Re-running the Multiple Tendon Prestress Wizard deletes all previously created data before re-calculating for a modified model.

Note that if some other aspect of the model has been changed (for example the geometry or mesh) then the attributes will be marked with a warning ⚠ symbol to show that the wizard must be re-run to recalculate new values.

Rail Track-Structure Interaction Analysis

Overview

The LUSAS Rail Track Analysis software option permits track/bridge interaction analysis to the International Union of Railways Code UIC774-3. Dialogs that enable model building, definition of loading and post-processing of results are accessed from the **Bridge** (or Civil) **> Rail Track Analysis UIC774-3** menu item.

Track and bridge interaction models are built automatically in LUSAS from geometric, material property, and loading data defined in a MS Excel spreadsheet. Both thermal loading to the track and train loading due to acceleration and braking forces can be defined. In accordance with the UIC774-3 code of practice, a user-specified element length is used to define the longitudinal embankment and bridge features. Rail clips, ballast movement, bearings and pier stiffness are all included in the analysis model. The model building dialogs allow for either one train crossing one or more structures, or for multiple trains crossing the same structure

When running an analysis, deck temperature loading can be considered in isolation for subsequent analysis of multiple rail configurations, or a full analysis can be carried out considering the combined temperature in the deck and rail loading. Because the response of the ballast and/or clips is nonlinear a nonlinear analysis always needs to be carried out.

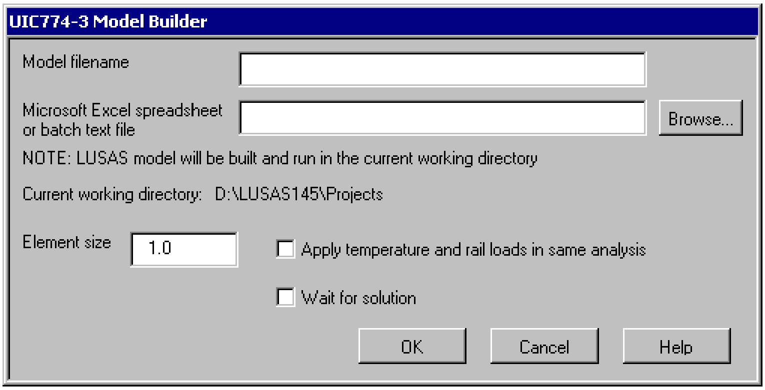
Results can be produced in either Excel spreadsheet or standard LUSAS results file format. User-defined load combinations can be specified. Spreadsheet results include deformations, forces/moments and axial stresses in the rails of the tracks and deformations and forces / moments in the deck structure.

The use of individual dialogs are explained in the following topics. For more detailed information refer to the *Rail Track Analysis Manual.*

Worked example

A worked example, **Track-Structure Interaction to UIC774-3**, examines the track-structure interaction between a braking train and a single span bridge to replicate (as far as the original test data allows) testcase E1-3 which can be found in Appendix D.1 of the UIC774-3 Code of Practice. This example can be found in the *Rail Track User Manual.*

UIC774-3 Model Builder



To use the UIC774-3 analysis option a spreadsheet describing the model geometry, properties and loading should have been defined. To do this use the template located in the <Lusas Installation Folder>\Programs\Scripts\User directory and enter the required information into the appropriate locations. All units for the analysis are metric and are listed for the sections within this spreadsheet.

Note: Each 'rail' in the Geometric Properties definition represents a single track on the structure and therefore should have geometric properties equal to double that of a single rail. Up to two tracks can be present on the structure.

Analysing a Single Structure

If a single analysis is being carried out then a model filename should be specified with or without the *.mdl extension. A directory should not be specified with this model filename as the current working directory is used for saving the model.

The Microsoft Excel filename should be entered or located using the **Browse…** button

Batch Processing Multiple Structures

If multiple models are to be built and analysed the batch processing facilities can be used. For this a batch text file (with a *.txt extension) should be created that contains the name of one valid Excel spreadsheet per line that defines the models. If the spreadsheet exists and contains valid data then the model will be built and solved in the same directory as the Excel spreadsheet. The number of Excel spreadsheets is unlimited.

The batch text filename should be entered into the dialog or located using the Browse… button and choosing Batch text file (*.txt) as the file type. No model filename should be entered as this will be defined by the basename of the Excel spreadsheet in the batch text file.

Common Input for Single or Multiple Structures

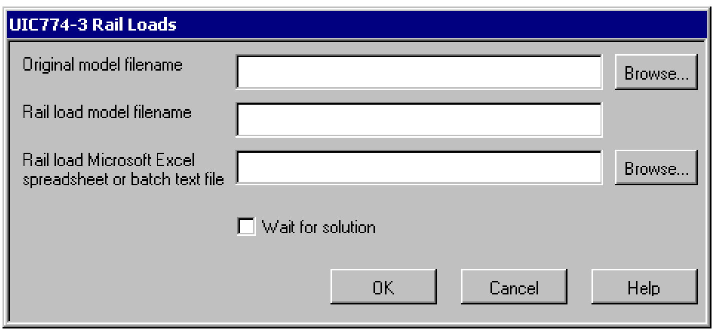
The element size to use in the Finite Element mesh should be specified in this box. According to the UIC774-3 Code of Practice, the maximum element size that is permitted in an analysis is 2.0m (Clause 1.7.3). The dialog therefore generally allows element sizes of 0 < Element Size ≤ 2.0m for the building of the models. Larger element sizes can be used (up to the length of the smallest bridge deck span) but a warning will be issued about non-compliance with the UIC774-3 Code of Practice. Note that for large bridges and/or embankments the use of small element sizes can generate excessively large models which take significant time to manipulate / solve. Use of element sizes below 1.0m should be used with caution.

By default, only the temperature loading is applied to the decks of the structure which allows multiple train load configurations to be applied to the same structure. If, however, the combined deck temperature and rail loading is to be solved in a single analysis then the Apply temperature and rail loads in same analysis option should be selected.

If the option to wait for the solution is selected then all of the analyses will be run from Modeller and nothing can be carried out in the current Modeller window until the solution has finished. For relatively small structures or analyses with a limited set of parametric trainset loading locations this is may be fine. If a large number of parametric trainset loading locations are included in an analysis and/or a large number of models are being built using the batch processing then waiting for the solution can take a considerable amount of time.

On clicking the OK button the model(s) will be built and solved automatically by the software. A log file called UIC774-3_BuildModel.log will be created in the current working directory if batch processing is used to report any errors encountered during the batch process.

UIC774-3 Rail Loads



If only the temperature effects have been applied to the model using the UIC774-3 Model Builder dialog then single or multiple rail configurations can be applied using this dialog.

Note: Attempting to use this dialog on a model that has not been created using the model builder dialog or that has loading other than temperature loads will generate an error message.

Applying train loads to the current model

If the current model loaded was generated from the Build Model... dialog with the Apply temperature and rail loads in same analysis option turned off then this option can be selected. If this option is not selected then the Original model filename entry is available for manual selection of the original model containing only temperature loads.

Analysing a Single Structure with One Rail Load Configuration

If a single rail load configuration is to be analysed for a single structure either enter the filename of the existing LUSAS temperature model created using the model builder or select it using the **Browse…** button. Enter the filename for the new model (omitting any path description as the model will be saved in the current working directory). This filename can be the same as the original model but it is recommended that an alternative filename is used so the original model remains unchanged. Finally enter the filename for the Excel spreadsheet that contains the rail load configuration description or select it using the **Browse…** button.

Analysing a Single Structure or Multiple Structures with Multiple Rail Load Configurations

If multiple models and / or multiple rail load configurations are to be analysed then only the batch text file should be entered or selected using the **Browse…** button and choosing Batch text file (*.txt) as the file type. The batch text file should be TAB delimited and contain the filename of the original model file, the filename of the new model to be created and the filename of the Excel spreadsheet containing the rail loading, e.g.

Viaduct 1.mdl Viaduct 1_RailConfig1.mdl Viaduct 1_RailConfig1.xls

Viaduct 1.mdl Viaduct 1_RailConfig2.mdl Viaduct 1_RailConfig2.xls

Viaduct 1.mdl Viaduct 1_RailConfig3.mdl Viaduct 1_RailConfig3.xls

Viaduct 2.mdl Viaduct 2_RailConfig1.mdl Viaduct 2_RailConfig1.xls

Viaduct 2.mdl Viaduct 2_RailConfig2.mdl Viaduct 2_RailConfig2.xls

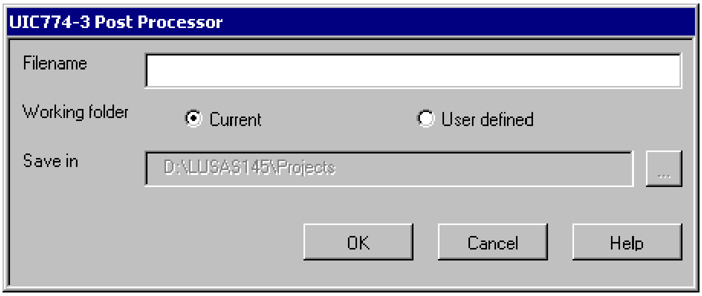
In the above example, two different viaduct temperature models have been selected and three rail load configurations chosen for the first and two for the second models. The number of entries in the batch text file is unlimited.

Common Input for Single or Multiple Structures / Rail Loads

If the option to wait for the solution is selected then all of the analyses will be run from Modeller and nothing can be carried out in the current Modeller window until the solution has finished. For relatively small structures or analyses with a limited set of parametric trainset loading locations this is may be fine. If a large number of parametric trainset loading locations are included in an analysis and/or a large number of models are being built using the batch processing then waiting for the solution can take a considerable amount of time.

On clicking the **OK** button the combined temperature and rail load model(s) will be built from the original temperature model(s) and solved automatically by the software. A log file called UIC774-3_RailLoads.log will be created in the current working directory and will report any errors encountered during the batch process.

UIC774-3 Post-Processor



The models created by the UIC774-3 model building software can be post-processed in Microsoft Excel with the post-processing dialog. This allows the extraction of the results along with automatic generation of graphs and enveloping in Microsoft Excel. In addition, if enveloping is performed in Microsoft Excel then additional summary tables are generated for key quantities that need to be checked for compliance with UIC774-3 under clause 1.7.2 of the Code of Practice.

On startup of the dialog, if valid UIC774-3 model groups are found, namely "Track 1", "Track 2" and "Decks" then the results can be extracted using these groups or for individual selected rail nodes. If, however, these groups are not found then the current selection will be used if it contains lines with thick 3D engineering beams assigned. If no valid groups or selection are found then the post-processor will report an error.

To use the post-processor, enter the filename for the Excel file that will be created. The directory in which to place the Microsoft Excel file can be selected but is, by default, the current working directory.

On clicking OK the post-processor will extract the results from all of the results loadcases. If envelopes or basic combinations are defined in the model then the option to process envelopes and generate the additional summary tables will not be available and all envelopes (without association) and basic combinations defined in the model file will be extracted. If the groups are being processed and no envelopes or basic combinations are defined in the model then the option to process the envelopes in Microsoft Excel will become available. If multiple results files are loaded, for example if multiple rail load configurations have been analysed and the results loaded into Modeller, then the results for all of these results files will be extracted into the Microsoft Excel spreadsheet.

On opening the spreadsheet, if the model originally contained the UIC774-3 groups ("Track 1", "Track 2" and "Decks") then the results for each item will be placed into a separate worksheet within the spreadsheet. If the selection was used for post-processing a single worksheet will be placed into the Microsoft Excel spreadsheet. Additional worksheets will also be generated if enveloping is carried out in Microsoft Excel. If individual rail nodes were selected and post-processed the spreadsheet will contain a separate worksheet for each node that lists the rail stresses seen at the location of the node for all results processed.

RC Slab Designer

Overview

The RC Slab Designer is accessed using the **Bridge> RC Slab Design** menu item. It enables plotting of contours and values that indicate flexural reinforcement requirements at Ultimate Limit State (ULS) or design crack width at Serviceability Limit State (SLS) for those design codes that support this. The calculations carried out are for reinforced concrete slabs (without prestressing) that are modelled using plate or shell elements.

Codes of Practice supported

The Codes of Practice available using the Country and Design code drop-lists are:

- **AASHTO LRFD 5th edition (2010)** and **AASHTO LRFD 6th edition (2012)** – listed under United States of America.
- **AS5100.5-2004** and **AS3600-2009** - both listed under Australia
- British Standards: **BS5400-4** (bridges), **BS8007** (Structures retaining aqueous liquids) and **BS8110** (buildings) – all listed under United Kingdom.
- **CAN/CSA S6-06** – listed under Canada.
- **Eurocode EN1992-1-1** (buildings): implemented for Austria, Belgium, Bulgaria, Cyprus, Czech, Denmark, Finland, France, Germany, Greenland, Ireland, Italy, Netherlands, Poland, Romania, Singapore, Slovakia, Spain and United Kingdom. Default settings are listed under Europe (Eurocode Default Values).
- **Eurocode EN1992-2** (bridges): implemented for Bulgaria, Cyprus, Denmark, Finland, Ireland, Italy, Spain, Sweden and United Kingdom. Default settings are listed under Europe (Eurocode Default Values).
- **IRC: 112:2011** and **IRS:CBC-1997** - listed under India.
- **SS CP65-1999** – listed under Singapore.

Using the RC Slab Designer

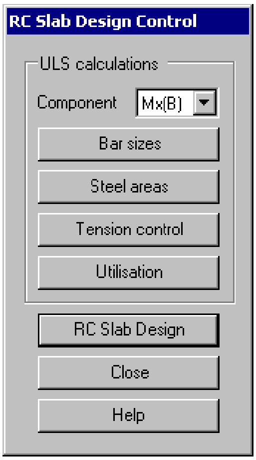
Prior to obtaining results from the RC Slab Designer, a slab model comprising plate or shell elements must be created, loaded and solved. Loadcases considered and load combinations created should include those appropriate to ULS and/or SLS depending on the calculations to be carried out.

If the elements in a model are orientated such that their local axes vary from one another, or if an alternative coordinate system is required, the results may need to be transformed to a consistent direction. This can be done by using the Results transformation button on the Contours and Values properties dialogs. See **Results Transformation** for details. The most recently set results transformation is used by the RC Slab Designer when calculating and displaying results.

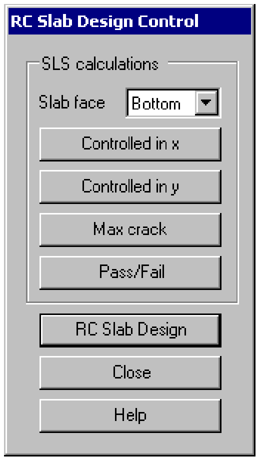
Running the RC Slab Designer (using the Bridge> RC Slab Design menu item) provides access to up to three pages:

1. The main options page, including selection of Design Code and calculation type
2. Reinforcement Details page
3. Crack Width Settings page (only required for SLS calculations)

When the RC Slab Designer is in use and when the Finish button is pressed on either the **Reinforcement Details** page (for ULS Calculations) or on the **Crack Width Calculation** settings page (for SLS calculations) an RC Slab Design Control dialog appropriate to the settings made will be displayed. These dialogs provide buttons to plot chosen results contours for the previously stated Code of Practice. The contours displayed are based on the active loadcase (or combination), using the most recently set results transformation, and are re-computed if a different loadcase is set active. A button is also provided to re-access the Slab designer and change settings based upon results previously viewed. The results components set by the RC Slab Design Control dialogs are explained in detail below.



RC Slab Design Control dialog for ULS design



RC Slab Design Control dialog for SLS design

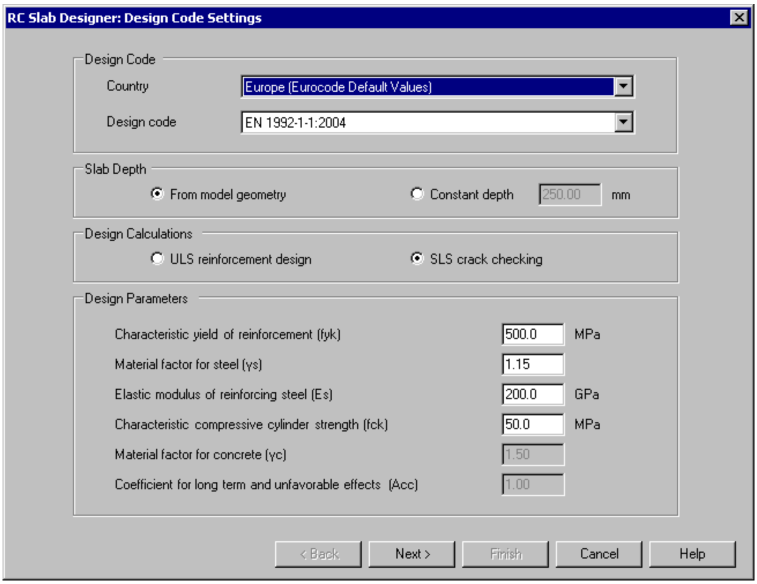
See **RC Slab Designer : Viewing Results** for more details.

Notes

- All RC Slab Designer calculations are based on a moment field (Mx, My, Mxy), and in-plane effects from shell elements are neglected. This may be unconservative and users should ensure that in-plane effects are not significant if using the RC Slab Designer with shell element results. If the RC Slab Designer is deemed appropriate for use, it is helpful to ensure that all shell elements in the slab are orientated with the surface normals aligned, pointing upwards (towards the top of the slab).
- See **Background to Calculations Carried out by the Slab Designer** for additional information.

General Design Code Settings and Parameters

On the main Design Code Settings page of the Slab Designer the design code, the slab depth, the design calculation method and the design parameters are defined.



Design code

- **Country** The list contains countries for which design codes are supported. Europe is included in the list, offering calculations to EN1992 with defaults based on Recommended values from the main Eurocode documents. This option, with or without modification of values, may be of particular use for countries where a National Annex is not yet published or not yet supported.
- **Design Code** A list of Codes of Practice appropriate to the country previously selected. Calculations will be carried out in accordance with the selected Code of Practice.

Slab depth

This sets the slab depth which will be used for all calculations. Two options are provided:

- **From model geometry** The thickness specified in the geometric attribute assigned to each surface in the model is used in the slab depth calculations carried out. This is the default option.
- **Constant depth** By specifying a constant slab thickness the likely implications of changing the slab depth can be considered in advance of adjusting the geometric properties in the model and re-solving for those revised properties. It is recommended that final calculations use the From model geometry option, with the model re-analysed with updated slab thicknesses where appropriate since slab depth has implications in terms of stiffness and, in most cases, self weight.

For both options the slab designer checks that the slab depth is compatible with other inputs of cover, bar size etc.

Design calculations.

ULS reinforcement design or SLS crack checking may be selected, although for some Codes of Practice the latter is not available. The two options are mutually exclusive since, different load factors would typically be applicable depending on the calculation being undertaken.

- **ULS reinforcement design** This option provides checks on flexural resistance based on proposed reinforcement arrangements. The calculations are based on Wood-Armer moments and resistances determined ignoring reinforcement in the compression zone. Each of the four layers of reinforcement may be checked in turn.
- **SLS crack checking** This option provides calculation of design crack widths in top and bottom faces for proposed reinforcement arrangements. The calculation is based on principal moments and makes allowance for the non-alignment of principal moments and reinforcement. Crack widths can be viewed according to the reinforcement direction controlling the crack, such that each of the four layers of reinforcement can be checked in turn.

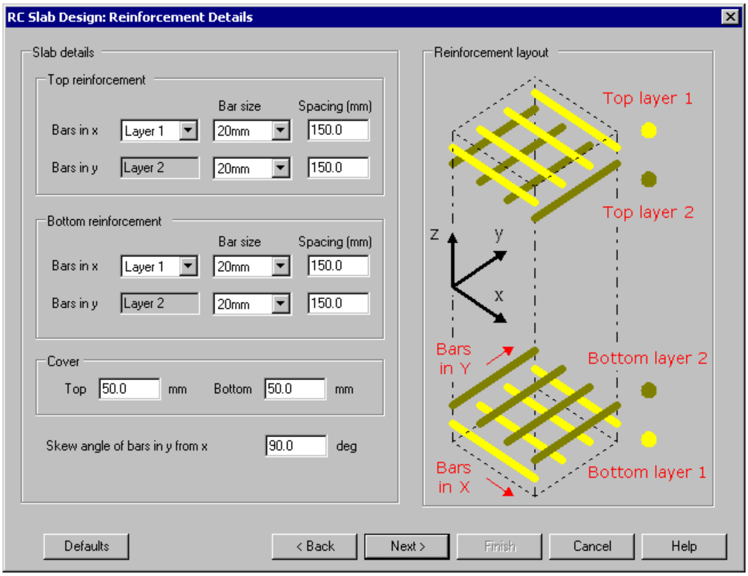
Design Code Parameters.

Design Code specific parameters are explained separately for:

- **AASHTO LRFD**
- **AS5100 and AS3600**
- **British Standards (various)**
- **CAN/CSA S6-06**
- **EN1992**
- **IRC:112-2011**
- **IRS:CBC-1997**
- **SS CP65**

Reinforcement Details

The Reinforcement Details page of the RC Slab Designer is essentially the same for all Codes of Practice and requires top and bottom reinforcement layout, size, spacing, cover and skew angle of bars in y from x to be defined.



- ❑ **Bar sizes** The sizes and designations that are shown are appropriate to the region for the selected Code of Practice. These are documented separately for each supported design code.
- ❑ **Bar spacing** Bar spacing is measured centre-to-centre, perpendicular to the bar direction.
- ❑ **Cover** Cover is measured from outermost part of the bars laid in the outermost layer of reinforcement to the nearest concrete face. It is intended that the value entered here is the actual cover, which may include allowance for fixing tolerance. A different nominal cover can be entered on the crack width calculation page of the wizard (page 3) where necessary.
- ❑ **Skew angle of bars in y from x** Used to arrange bars in a non-orthogonal arrangement where the skew angle can lie between 0 and 180 degrees (90 degrees in perpendicular). The angle entered defines the orientation of the y reinforcement with respect to the x-direction, measured in an anti-clockwise direction. The angle specified is used to determine Wood Armer results in the x- and skew-directions (for comparison to flexural resistance in the two corresponding directions) and in the determination of crack widths when chosen.

Design Code specific parameters are explained separately for:

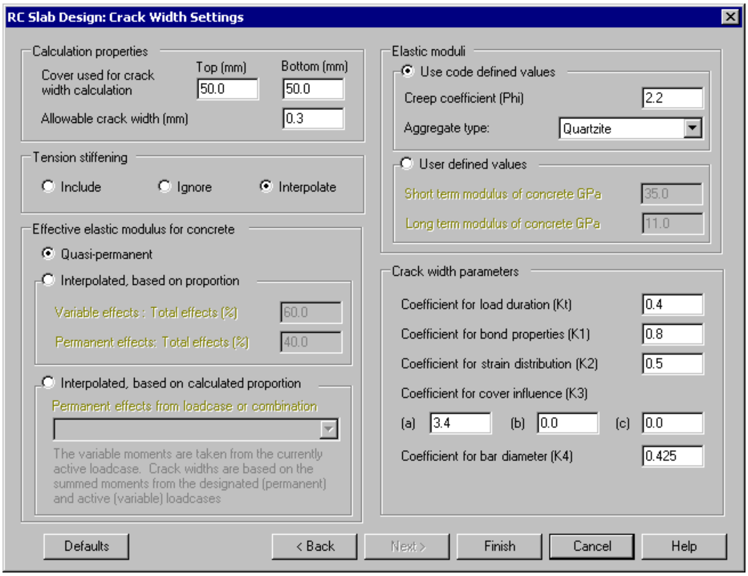
- ❑ **AASHTO LRFD**

- AS5100 and AS3600
- British Standards (various)
- CAN/CSA S6-06
- EN1992
- IRC:112-2011
- IRS:CBC-1997
- SS CP65

Please refer to the relevant link/page for more details.

Crack Width Calculation Settings

The crack width calculation page of the RC Slab Designer will appear only if the SLS crack checking option was selected on the main Design Code Settings page.



Design Code specific parameters are explained separately for:

- AASHTO LRFD
- AS5100 and AS3600
- British Standards (various)
- CAN/CSA S6-06
- EN1992
- IRC:112-2011
- IRS:CBC-1997
- SS CP65

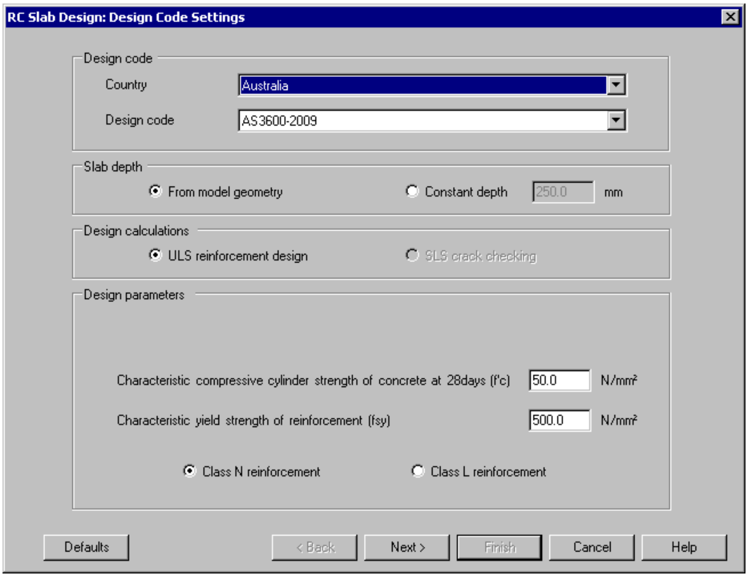
Note that not all design codes supported by the RC Slab Designer provide methodology for explicit crack width calculation.

Design Code Settings and Parameters for Australia (AS3600 / AS5100)

Selecting **Australia** and one of the design code options on the main Design Code Settings page of the RC Slab Designer will make it perform calculations, and offer defaults, based on the appropriate Australia publication below:

- **AS 3600-2009** Australian Standard - Concrete Structures
- **AS 5100.5-2004** Australian Standard - Bridge Design - Part 5

Note that regardless of model units, the Slab Designer uses units in keeping with the design code, i.e. N/mm², millimetres etc., as displayed on the dialog.



General Settings

See RC Slab Designer : General Design Code Settings and Parameters for details.

Design Calculations.

ULS reinforcement design is catered for but SLS crack checking is not available. This is because AS3600 and AS5100.5 code requirements do not include a crack width calculation.

Design Parameters

For AS 3600-2009 and AS 5100.5-2004 the following parameters are required:

- **Characteristic compressive cylinder strength of concrete, f'c** When ULS reinforcement design is selected, f'_c is used in the calculation of the lever arm, z, and in determining if the section is tension-controlled. When SLS crack checking is selected, the elastic modulus for concrete is determined using this value in the absence of specific user input for that parameter. Typical values lie in the range 20 to 100N/mm². The default is 50N/mm²
- **Characteristic yield strength of reinforcement, fsy** When ULS reinforcement design is selected f_{sy} is used in the calculation of the area of reinforcing steel required (and bar diameters) and in determining if the section is tension-controlled. Typical values would be f_y=250, 460 or 500N/mm² and the default is 500N/mm².

Reinforcement Details

See **RC Slab Designer : Reinforcement Details** for general information.

- **Bar sizes** Nominal diameters (and cross-sectional areas) correspond to those readily available in Australia
- **Spacing** [s] Typical values lie in the range 40mm to 300mm. The default is 150mm.
- **Cover** [c_{act}] This is the cover used to calculate effective depths, as distinct from nominal cover (c_{nom}), used in the calculation of crack widths (see below). For various reasons, including durability considerations or fixing tolerances, the specified c_{act} may be greater than c_{nom}. Typical values for c_{act} would, however, lie in the range 25 to 65mm. The default is 50mm.

ULS Reinforcement Design to AS3600 and AS5100.5

See **Background to Calculations Carried out by the Slab Designer** for general information.

Area of steel required

The formulae used are summarised below:

AS 3600	AS5100.5
$A_{s,req} = \dfrac{M_d}{\emptyset f_{yd} z}$	$A_{s,req} = \dfrac{M_d}{\emptyset f_{yd} z}$
$z = d - \dfrac{\gamma k_u d}{2}$	$z = d - \dfrac{\gamma k_u d}{2}$

$$k_u d = \frac{f_{yd} A_{s,prov}}{\alpha_2 f_c' b \gamma}$$

$$Where\ \alpha_2 = 1.0 - 0.003 f_c'$$
$$and\quad \gamma = 1.05 - 0.007 f_c'$$

$$k_u d = \frac{f_{yd} A_{s,prov}}{0.85 f_c' b \gamma}$$

$$Where\ \gamma = [0.85 - 0.007(f_c' - 28)]$$

The nomenclature used above is as in the relevant codes excepting that:

$A_{s,req}$ = Area of tension reinforcement required

Tension control ratio

The slab is ductile when the appropriate inequality below is met:

AS 3600	AS5100.5
$k_u \le 0.36$	$k_u \le 0.40$

Viewing ULS results

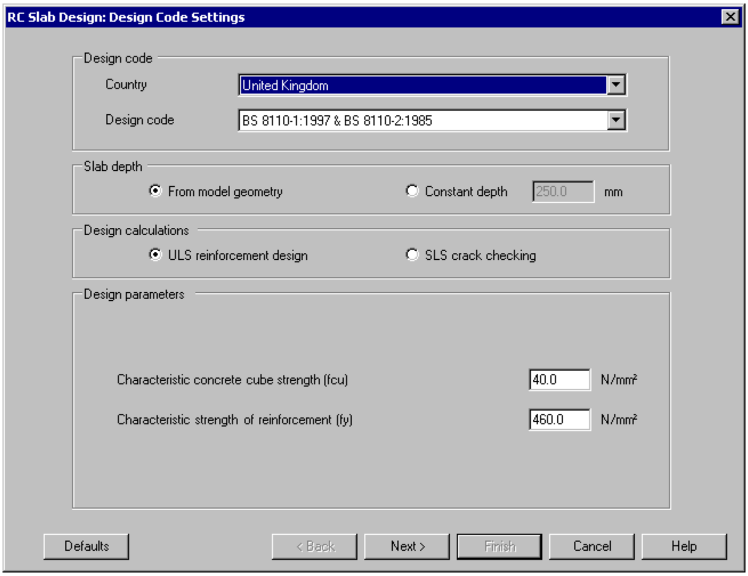
See **RC Slab Designer : Viewing Results** for details of plotting ULS reinforcement design contours.

Design Code Settings and Parameters for United Kingdom (Eurocode / British Standards)

Selecting **United Kingdom** and one of Eurocode or British Standard design code options on the main Design Code Settings page of the RC Slab Designer will make it perform calculations, and offer defaults, based on the appropriate BS publication below:

- **BS EN1992-1-1:2004/NA:2005,** Design of Concrete Structures - general rules and rules for buildings
- **BS EN1992-2:2005/NA:2007,** Design of Concrete Structures - concrete bridges - Design and detailing rules
- **BS 8110–1:1997 & BS8110-2:1985** Code of practice for design and construction, and for special circumstances
- **BS 8007:1987**, Code of practice for design of concrete structures for retaining aqueous liquids
- **BS 5400-4: 1990**, Steel, concrete and composite bridges – Part 4: Code of practice for design of concrete bridges

Note that regardless of model units, the Slab Designer uses units in keeping with the design code, i.e. N/mm², millimetres etc., as shown on the dialog.



General Settings

See **RC Slab Designer : General Design Code Settings and Parameters** for details.

Design Parameters

For BS EN1992-1-1:2004/NA:2005 and BS EN1992-2:2005/NA:2007 see **RC Slab Designer : Design Code Settings and Parameters for Eurocode**

For BS8110–1:1997 & BS8110–2:1985, BS8007:1987, and BS5400-4: 1990 the following parameters are required:

- **Characteristic concrete cube strength, fcu** When ULS reinforcement design is selected, f_{cu} is used in the calculation of the lever arm, z, and in determining if the section is tension-controlled. When SLS crack checking is selected, the elastic modulus for concrete is determined using this value in the absence of specific user input for that parameter. Typical values lie in the range 20 to 60N/mm². The default is 40N/mm²
- **Characteristic strength of reinforcement, fy** When ULS reinforcement design is selected f_y is used in the calculation of the area of reinforcing steel required (and bar diameters) and in determining if the section is tension-controlled. It is not used for SLS crack checking calculations. Typical values would be f_y=250, 460 or 485N/mm² (see BS8110-1 Table 3.1 and BS5400-4 Table 6) and the default is 460N/mm².

Reinforcement Details

See **RC Slab Designer : Reinforcement Details** for general information.

- **Bar sizes** Nominal diameters (and cross-sectional areas) correspond to those in BS4449 Table 7.
- **Spacing** [s] Typical values lie in the range 25mm to 300mm (BS8110-1 clause 3.12.11 and BS5400-4 clause 5.8.8, with a typical aggregate size of 20mm). The default is 150mm.
- **Cover** [c_{act}] This is the cover used to calculate effective depths, as distinct from nominal cover (c_{nom}), used in the calculation of crack widths (see below). For various reasons, including durability considerations or fixing tolerances, the specified cact may be greater than c_{nom}. Typical values for c_{act} would, however, lie in the range 20 to 75mm (BS8110-1 clause 3.3, BS5400-4 Table 13 and BD57/01 clause 3.1). The default is 50mm.

ULS Reinforcement Design to British Standards

See **Background to Calculations Carried out by the Slab Designer** for general information.

British Standard-specific assumptions for nominal resistance calculations are:

The formulae are according to BS8110-1 clause 3.4.4.4 (as referenced from clause 3.5.1 regarding slabs). BS8007 clauses 2.2.1 and 3.2.1 indicate that ULS moment checks are determined according to BS8110-1 and BS8007 does not amend the method of BS8110-1 in this regard. For bridges, BS5400-4 clause 5.3.2.3 (as referenced from clause 5.4.2 regarding slabs) is used.

Area of steel required

The formulae used are summarised below.

BS8110	BS5400-4
$A_s = \dfrac{M}{0.95 f_y z}$	$A_{s,req} = \dfrac{M_u}{0.87 f_y z}$
$z = d\left\{0.5 + \sqrt{0.25 - \dfrac{K}{0.9}}\right\} \ngtr 0.95d$	$z = \left(1 - \dfrac{1.1 f_y A_{s,prov}}{f_{cu} b d}\right) d \ngtr 0.95d$
$K = \dfrac{M}{b d^2 f_{cu}}$	

The nomenclature used above is as in the relevant codes excepting that:

$A_{s,req}$ = Area of tension reinforcement required

$A_{s,prov}$ = Area of tension reinforcement provided

Tension control ratio

The slab is ductile when the appropriate inequality below is met:

BS8110	BS5400-4
$\frac{K}{K'} = \frac{K}{0.156} \leq 1$	$\frac{M_u/bd^2 f_{cu}}{0.15} \leq 1$

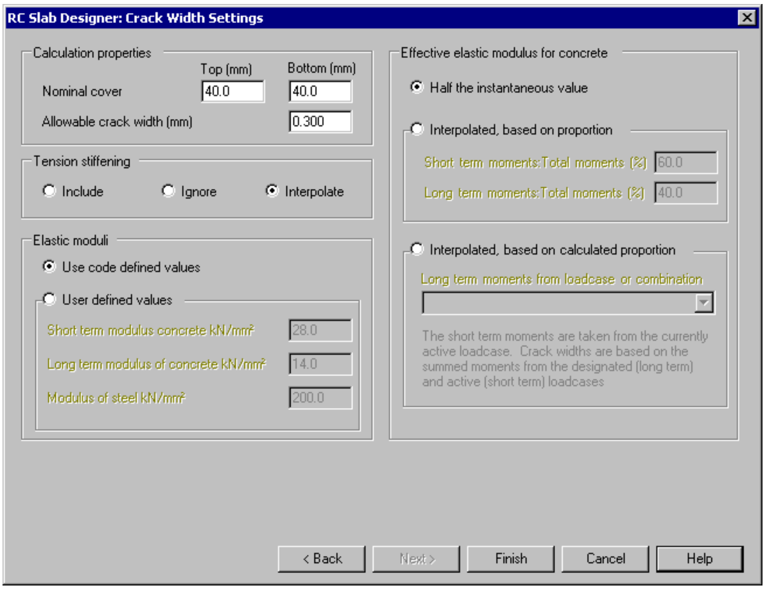
Viewing ULS results

See **RC Slab Designer : Viewing Results** for details of plotting ULS reinforcement design contours.

Crack Width parameters

The Crack Width Settings page of the RC Slab Designer will appear only if the SLS crack checking option was selected on the main Design Code Settings page.

Settings for the British Standards options (non-Eurocodes) are described here:



Calculation properties

- **Nominal Cover** [c_{nom}] This value affects the design crack width calculation by changing the location at which the design crack width is assessed. In essence, the crack width is calculated at a plane lying a distance, c_{nom}, from the layer of bars controlling the crack. When using BS8110-2, it may be assumed that c_{nom} is equal to the actual cover, cact, making the crack width plane described coincident with the tension face of the concrete. When using BS5400-4 in conjunction with BD57/01 clause 3.1, however, $c_{act} = c_{nom} + 10$mm, and therefore the crack width plane is internal to the concrete, also see BA57/01 clause 5.2. The value entered here should therefore be the nominal cover, c_{nom} to be used in crack width calculations – whether this is identical to, or less than, c_{act}. Typical values lie in the range 20 to 70mm and should be determined from BS8110-1 clause 3.3, Tables 3.3 and 3.4, BS8007 clause 2.7.6 or BS5400-4 Table 13 and clause 5.8.2 as appropriate. The default is 40mm.

- Allowable crack width (mm) When using BS8007, this value will determine which expression from BS8007 Appendix B is used for the calculation of design crack width. It will also be used to scale the crack width contours. When using either BS8110 or BS5400, the value entered is only used to the scale the crack width contours. Additionally, BS8007 clause 2.2.3.3(a) requires the allowable crack width to be set as either 0.1 or 0.2mm. BS8110-1 clause 3.12.11.2 indicates a limitation of 0.3mm, while BS5400-4 Table 1 relates design crack width requirements to exposure conditions, with values from 0.1mm to 0.25mm. The default value is taken as 0.2mm.

Tension stiffening

BS8110-2 cl 3.8.3, eqn 13, BS8007 Appendix B clause B.4 and BS5400-4 cl 5.8.8.2 eqn 25 include a reduction in the calculated crack width due to the stiffening effect of tension carried by concrete which remains bonded to the reinforcement in the tensile zone at small strains. Where reinforcement does not lie perpendicular to cracks, whether the “tension stiffening” reduction should be included is a matter of debate (see SLS Crack Checking in **Background to Slab Designer Calculations).** The Slab Designer offers three options for crack check calculations:

- **Include** Considers tension stiffening for all angles
- **Ignore** Tension stiffening is completely ignored for conservative design.
- **Interpolate** (Default option) Scales linearly from 0% tension stiffening at α=25°, to 100% at α=0°.

Elastic Moduli

The elastic moduli for concrete and reinforcing steel are used in calculation of flexural strains in the slab, leading to the calculation of a design crack width.

- **Use code defined values** When this option is invoked, the moduli used in calculations (as required), will be as stated in the table below.
- **User defined values** When this option is invoked, users may enter values for elastic moduli instead of using those determined automatically. The default values are based on the values and expressions in the codes of practice described previously.

Design code	Short term modulus for concrete	Long term modulus for concrete	Elastic modulus for steel
BS8110	$E_{c,short} = 0.2 \times f_{cu} + 20$	$E_{c,long} = E_{c,short/2}$	200kN/mm²
	BS8110-2 eqn 17, clause 7.2	BS5400-4 clause 4.3.2.1 (see note 1)	BS8110-1 clause 2.5.4
BS8007		As for BS8110	
		BS8007 clause 2.2.1	
BS5400-4	$E_{c,short} = 0.27 \times f_{cu} + 20$	$E_{c,long} = E_{c,short/2}$	200kN/mm²
	BD44/95 (see note 2)	BS5400-4	BS5400-4

Notes

1. The reference to BS5400-4 is for an approximation taking into account the effect of creep under long term loading, since reference to BS8110-2 clause 3.6(a)(3) requires determination of a creep coefficient. This approximation is broadly validated with reference to CIRIA Report 110.
2. This equation (rounded to 2 significant figures) closely approximates BS5400-4 Table 3 although the value for fcu=25N/mm² is calculated as Ec=27kN/mm² instead of 26kN/mm² as given in Table 3. Use of the equation is felt preferable as it is more flexible for values of fcu other than those given in Table 3.

Calculate Effective Elastic Modulus for Concrete

In the calculation of crack widths, strains are assessed using an effective elastic modulus for concrete. This effective modulus may be calculated in a number of ways to suit the user and code of practice as follows:

- **Half the instantaneous value** The effective elastic modulus for concrete is taken as half the instantaneous value, as per BS8110-2 clause 3.8.3. This assumption is also conservative in consideration of BS5400-4 clause 4.3.2.1. This is the default option.
- **Interpolated, based on proportion** The effective elastic modulus for concrete is calculated using a ratio, R, of moments due to live loading (M_q) to total moments,

(M_g+M_q). R is entered by the user as a percentage and the following expression (based on strain compatibility) is used to determine the effective modulus:

$$E_{c,eff} = \frac{E_{c,short} \times E_{c,long}}{E_{c,long} R + E_{c,short}(1 - R)}$$

Assuming a suitable single value for R can be determined by the user, this option suits the approach of BS5400-4 clause 4.3.2.1(b) when $E_{c,long}$ = $E_{c,short}$/2, i.e. when "use code defined values" is selected.

- **Interpolated, based on calculated proportion** The effective elastic modulus for concrete is calculated using the ratio, R, as described above. In this case, however, R is evaluated at each node. The moments due to permanent loads, M_g, are taken from the loadcase selected in the droplist. The moments due to live loads, M_q, are taken from the active loadcase. The ratio at each node is:

$$R = \frac{M_q}{M_g + M_q}$$

Consistent with the above expression, total moments used for the calculation of crack width are based on M_g + M_q. This option suits the approach of BS5400-4 clause 4.3.2.1(b) when $E_{c,long}$ = $E_{c,short}$/2 i.e. when "use code defined values" is selected. The need for a suitable single value for R to be determined by the user is avoided.

SLS crack width calculation to British Standards

See **Background to Calculations Carried out by the Slab Designer** for general information.

Crack width calculations for the British Standards are carried out by reference to BS8110-2 clause 3.8.3 and BS5400-4 clause 5.8.8.2(a) and may be summarised (with differences shown) as follows:

BS8110	BS5400-1

$$\epsilon_s = \frac{M}{E_s A_s \left(d - \frac{x}{3}\right)}$$

$$\epsilon_1 = \frac{\epsilon_s (a' - x)}{(d - x)}$$

Reduction due to strain stiffening

$$\epsilon_2 = \frac{b_t(h-x)(a'-x)}{3E_sA_s(d-x)} \quad \epsilon_2 = \left[\frac{3.8b_th(a'-d_c)}{\epsilon_sA_s(h-d_c)}\right]\left[\left(1-\frac{M_q}{M_g}\right)10^{-9}\right]$$

Strain at cracking level

$$\epsilon_m = \epsilon_1 - \epsilon_2$$

Distance to controlling bar

$$a_{cr} = \sqrt{(h-d)^2 + \left({}^{s}/_{2}\right)^2} - {}^{\emptyset}/_{2}$$

Crack widths

$$w = \frac{3a_{cr}\epsilon_m}{1+2\left(\frac{a_{cr}-c_{min}}{h-x}\right)} \qquad w = \frac{3a_{cr}\epsilon_m}{1+2\left(\frac{a_{cr}-c_{nom}}{h-d_c}\right)}$$

The spacing and bar diameter of the bars identified as "controlling" the crack are used to determine the distance from the crack to the nearest bar, a_{cr}.

From BS8007 clauses B.2 to B.4, flexural cracks are calculated as for BS8110-2, except that ε_2 is given as:

$$\epsilon_2 = \frac{b_t(h-x)(a'-x)}{3E_sA_s(d-x)}$$ for crack width limit of 0.2mm

$$\epsilon_2 = \frac{1.5b_t(h-x)(a'-x)}{3E_sA_s(d-x)}$$ for crack width limit of 0.1mm

Viewing SLS results

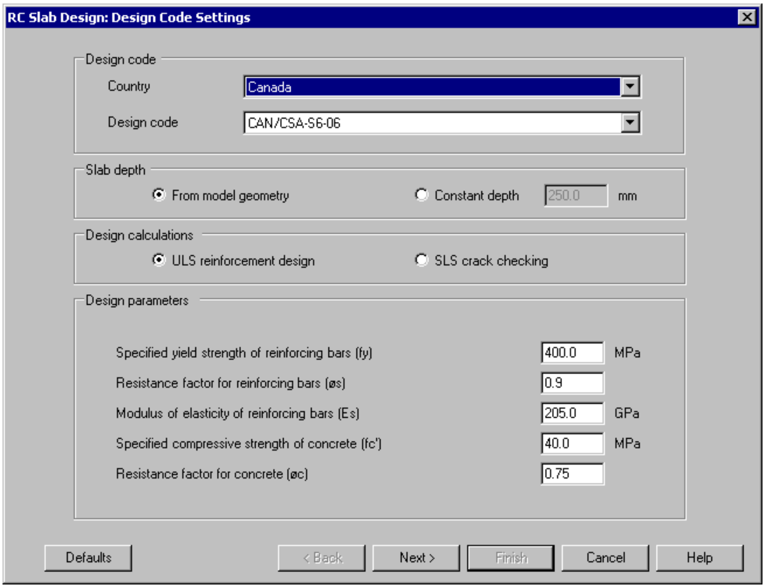
See **RC Slab Designer : Viewing Results** for details of plotting SLS reinforcement design contours.

Design Code Settings and Parameters for CAN/CSA S6-06

Selecting **Canada** on the main Design Code Settings page of the RC Slab Designer performs calculations, and offers defaults, based on the following publications:

- **CAN/CSA-S6-06** Canadian Highway Bridge Design Code. Canadian Standards Association, Ontario, Nov 2006
- **S6.1-06** Commentary on CAN/CSA-S6-06, Canadian Highway Bridge Design Code. Canadian Standards Association, Ontario, Nov 2006

Regardless of model units, the Slab Designer uses units in keeping with the design code, i.e. MPa, millimetres etc.



General Settings

See **RC Slab Designer : General Design Code Settings and Parameters** for details.

Design Parameters

- **Specified compressive strength of concrete (fc')**: When ULS reinforcement design is selected, $f_{c'}$ is used in the calculation of the lever arm, and in determining if the section is tension-controlled. When SLS crack checking is selected, the elastic modulus for concrete is determined using this value in the absence of specific user input. Values acceptable in clause 8.4.1.2 lie in the range 30 to 85MPa. The default is $f_{c'}$=40MPa.
- **Specified yield strength of reinforcement (fy)**: When ULS reinforcement design is selected, f_y is used in the calculation of the area of reinforcing steel required (and bar

diameters) and in determining if the section is tension-controlled. It is not used for SLS crack checking calculations. Values acceptable in clause 8.4.2.1.3 lie in the range f_y=300 to 500MPa. The default is f_y=400MPa.

- **Modulus of elasticity of reinforcing bars (Es)**: When ULS reinforcement design is selected, E_s is used in determining if the section is tension-controlled. When SLS crack checking is selected, the value is used in the calculation of the depth in compression. From clause 8.4.3.3(b) the typical value, taken as the default, is E_s=205GPa
- **Resistance factor for concrete (ϕc)**: When ULS reinforcement design is selected, ϕ_c is used in determining the distance between the neutral axis and the compressive face, hence affecting the lever arm calculated, and the classification of the section with regard to tension-control. It is not used for SLS crack checking calculations. From clause 8.4.6 & Table 8.1 the expected value, taken as the default, is $\phi_c = 0.75$.
- **Resistance factor for reinforcing bars (ϕs)**: When ULS reinforcement design is selected, ϕ_s is used as the partial factor on material for the factored flexural resistance as seen in the commentary C8.8.4.1. It is also used in determining the distance between the neutral axis and the compressive face, hence affecting the lever arm calculated, and the classification of the section with regard to tension-control. It is not used for SLS crack checking calculations. From clause 8.4.6 & Table 8.1 the expected value, taken as the default, is $\phi_s = 0.90$.

Reinforcement Details

See **RC Slab Designer : Reinforcement Details** for general information.

Code-specific advice is as follows:

- **Bar sizes** Standard metric bar designations are used, representing the nominal bar diameter in millimetres rounded to the nearest 5mm.
- **Spacing** [s] Typical values lie in the range 50mm to 300mm. The default is 150mm.
- **Cover** [c_{actual}] This is the cover used to calculate effective depths, as distinct from nominal cover (c_{nom}), used in the calculation of crack widths (see below). For various reasons, including durability considerations or fixing tolerances, the specified actual cover (c_{act}) may be greater than the nominal cover. Typical values for actual cover would, however, lie in the range 30 to 80mm (Table 8.5). The default is 50mm.

Bar Designations

Bar Size	Bar Size Nominal Diameter (mm)
10M	11.3
15M	16
20M	19.5
25M	25.2
30M	29.9
35M	35.7
45M	43.7
55M	56.4

ULS Reinforcement Design to CAN/CSA S6-06

See **Background to Calculations Carried out by the Slab Designer** for general information.

From clause 8.8.4.1, the factored flexural resistance is calculated in accordance with clause 8.8.3, assuming the relationship between concrete strain and the concrete compressive stress to be rectangular. The basic formula used can be derived directly from moment equilibrium or found with reference to the commentary C8.8.4.1, by eliminating irrelevant terms for prestress and flange width etc.

CAN/CSA-S6-06

Area of steel:

$$A_{s,req} = \frac{M_d}{\phi_s f_y \left(d_s - \frac{a}{2}\right)}$$

$$a = \frac{\phi_s f_y A_{s,prov}}{b.\alpha_1 \phi_c f_c'}$$

$$\alpha_1 = \max(0.85 - 0.0015 f_c', 0.67)$$

The nomenclature used above is as in the code excepting that:

M_d = factored moment at the section

$A_{s,req}$ = Area of tension reinforcement required

$A_{s,prov}$ = Area of tension reinforcement provided

For tension control the slab is ductile when the appropriate inequality below is met:

$$\frac{c}{d_s} \leq \left[\frac{\phi_s f_y}{E_s \varepsilon_{cu3}} + 1\right]^{-1}$$

$$c = \frac{a}{\beta_1}$$

$$\beta_1 = \max(0.97 - 0.0025 f_c{}', 0.67)$$

Where ε_{cu3} is the maximum usable strain at the extreme concrete compression fibre, taken as 0.0035 according to clause 8.8.3(c)

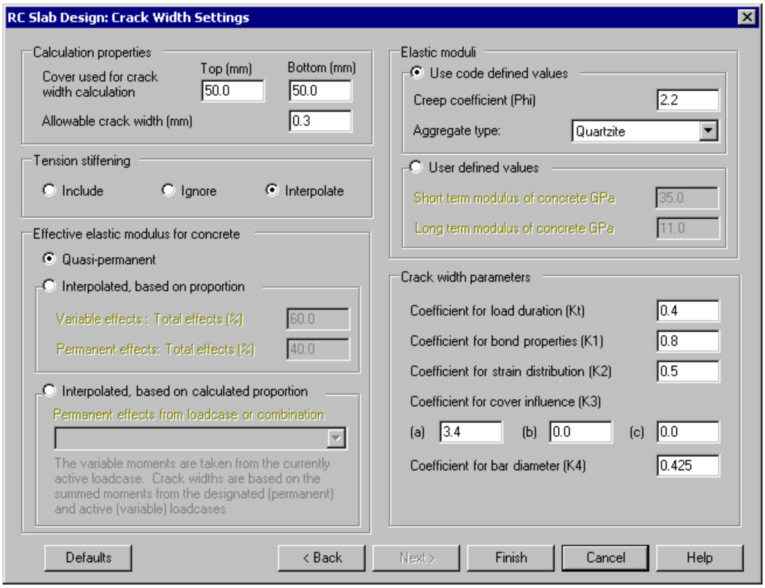
Viewing ULS results

See **RC Slab Designer : Viewing Results** for details of plotting ULS reinforcement design contours.

Crack Width Settings for CAN/CSA S6-06

See **Background to Calculations Carried out by the Slab Designer** for general information.

The Crack Width Settings page is only available if the SLS crack checking option is selected on the Design Code Settings page. Settings for CAN/CSA-S6-06 are described here:



Calculation properties

- **Cover used for crack width calculation** [c_{nom}] This value affects the design crack width calculation by changing the location at which the design crack width is assessed. In essence, the crack width is calculated at a plane lying a distance, c_{nom}, from the layer of bars controlling the crack. It is typically assumed that c_{nom} is equal to the actual cover, c_{act}, making the crack width plane described coincident with the tension face of the concrete. However when the actual cover is significantly greater than the cover required in Table 8.5, setting c_{nom} to the lower value will calculate the crack width on a plane internal to the concrete, resulting in a smaller calculated crack width, an approach that may be deemed appropriate in certain circumstances. Values for c_{nom} would therefore typically be by reference to Table 8.5 and the default is 50mm..
- **Allowable crack width (mm)** [w_{max}]. This value is used only to scale the crack width contours. A suitable value from clause 8.12.3.1, Table 8.6, would generally be either 0.25 or 0.35mm. The default is w_{max}=0.25mm

Tension Stiffening

Tension Stiffening (fw/fs). The expression for the average strain in the reinforcement, ε_{sm} includes a reduction due to the stiffening effect of tension carried by concrete. Where reinforcement does not lie perpendicular to cracks, whether the "tension stiffening" reduction should be included is a matter of debate (see SLS Crack Checking in **Background to Slab Designer Calculations).** The Slab Designer offers crack check calculations with the tension stiffening component included, ignored (conservative) or interpolated. The "interpolated" option scales linearly from 0% tension stiffening at α=25°, to 100% at α=0°, and this is the default option.

- **Include** Considers tension stiffening for all angles
- **Ignore** Tension stiffening is completely ignored for conservative design.
- **Interpolate** (Default option) Provides a linear scale from 0% tension stiffening at α=25°, to 100% at α=0°.

Calculation of effective elastic modulus for concrete

In the calculation of crack widths, strains are assessed using an effective elastic modulus for concrete. This effective modulus may be calculated in a number of ways to suit the user and code of practice as follows:

- **Quasi-permanent** For this the effective elastic modulus for concrete is taken as the long-term value (from user input or calculated as described below). This is generally regarded as conservative in respect of the elastic and creep deformations that might be expected as a result of the mix of permanent and transient loads that are likely to affect the structure. Accordingly, this is the default option.
- **Interpolated, based on proportion** For this the effective elastic modulus for concrete is calculated using a ratio, R, of moments due to live loading (M_q) to total moments, (M_g+M_q). R is a single suitable value, determined by the user, that is entered as a percentage. The following expression (based on strain compatibility) is used to determine the effective modulus:

$$E_{c,eff} = \frac{E_{c,short} \times E_{c,long}}{E_{c,long} R + E_{c,short}(1 - R)}$$

- **Interpolated, based on calculated proportion** For this the effective elastic modulus for concrete is calculated using the ratio, R, as described above. In this case, however, R is evaluated at each node. The moments due to permanent loads, M_g, are taken from the loadcase selected in the droplist. The moments due to live loads, M_q, are taken from the active loadcase. The ratio at each node is:

$$R = \frac{M_q}{M_g + M_q}$$

  Consistent with the above expression, total moments used for the calculation of crack width are based on M_g + M_q. The need for a suitable single value for R to be determined by the user is avoided.

Elastic Moduli

The elastic moduli for concrete and reinforcing steel are used in calculation of flexural strains in the slab, leading to the calculation of a design crack width.

- **Use code defined values** When this option is invoked, the moduli used in calculations (as required), are as stated in the table below.

Short term modulus for concrete	Long term modulus for concrete

$$E_{c,short} = E_{c,28} = \left(3000\sqrt{f'_c} + 6900\right)\Big(E_{c,long} = E_c(\infty, 28) = E_{c,28}[1 + \phi(\infty$$

Based on clause 8.4.1.7 **Based on clause 8.4.1.6**

The use of code defined values requires the entry of two further parameters:

- **Creep coefficient φ** (∞, 28) : This should be determined from clause 8.4.1.6.3. The default is taken as $\phi(\infty, 28) = 2.2$..
- **Mass density of concrete γc** : From clause 8.2 and clause 3.6, Table 3.3. The default is taken as γ_c=2450kg/m³.
- **User defined values** When this option is invoked, values for elastic moduli may be entered instead of using those determined automatically. The default values are based on the expressions described previously.

Canada crack width factors

- **Coefficient for aggregate type (kag)** In clause 8.4.1.8.1, the cracking strength of concrete, fcr, is seen to be dependent upon a constant, here named k_{ag}, and the root of the specified compressive strength. Clause 8.4.1.8.1 takes a value of k_{ag} in the range 0.3 to 0.4 according to the density of the concrete. The default is taken as k_{ag}=0.4, appropriate to normal-density concrete.
- **Coefficient for bond properties (kb)** In clause 8.12.3.2, k_b is taken as 1.2 for components with epoxy-coated reinforcing steel and 1.0 for all other components. The default is k_b = 1.0.
- **Coefficient for strain distribution (kc)** In clause 8.12.3.2, k_c is a coefficient which takes account of the form of the strain distribution, with the values 0.5 for bending and 1.0 for pure tension. The commentary C8.12.3.2 refers to DD ENV 1992-1-1, in which clause 4.4.2.4(3) gives guidance for cases of eccentric tension. However since the scope of the slab designer is restricted to moment fields, the appropriate value would in most cases be 0.5 and the default is accordingly k_c=0.5.
- **Coefficient for the cause of cracking (βc)** In clause 8.12.3.2, β_c is seen to be determined by the cause of the cracking and the section depth. When cracking is caused by load, β_c=1.7. When cracking is caused by superimposed deformations, β_c is taken as 1.7 where the section has a minimum dimension greater than 800mm, and taken as 1.3 where the minimum dimesion is less than or equal to 300mm. The default is β_c=1.7.
- **Ratio of max to mean crack width κc.** In the commentary in clause 8.12.3.2, "Maximum crack width, which is the quantity required for design, is obtained by multiplying average crack width by the parameter k_c.". It appears that the coefficient, k_c, here is not the same as k_c in clause 8.12.3.2 (described above), since that has a value of 0.5 for sections in bending, which would not be appropriate for obtaining a maximum value from an average value. The ratio is therefore here named κ_c. No suggested value for κ_c is given in either the code or the commentary but it is thought

to generally lie in the range $1.0 < \kappa_c \leq 1.7$. In the absence of data, the default is taken as $\kappa_c=1.0$

SLS crack width calculation to CAN/CSA-S6-06

Crack width calculations for CAN/CSA-S6-06 are carried out by reference to clauses 8.12.3.2 and 8.4.1.8.1 and 8.8.4.4:

CAN/CSA-S6-06

Effective area of concrete in tension:

$$A_{ct} = b.\min\left\{2.5(h-d_s), \frac{1}{3}(h-x)\right\}$$

$$\rho_c = \frac{A_s}{A_{ct}}$$

Crack spacing:

$$s_{rm} = 50 + 0.25k_c\frac{d_b}{\rho_c}$$

Cracking stress and cracking moment:

$$f_{cr} = k_{cr}\sqrt{f_c'}$$

$$M_{cr} = \frac{bh^3}{12}\frac{f_{cr}}{(d_s + d_b + c_{nom} - x)}$$

Stresses in the tension reinforcement (cracked section):

$$f_s = \frac{M_s}{A_{s,\text{prov}}(d_s - x/3)} \qquad f_w = \frac{M_{cr}}{A_{s,\text{prov}}(d_s - x/3)}$$

Average strains in the reinforcement:

$$\varepsilon_{sm} = \frac{f_s}{E_s}\left[1 - \left(\frac{f_w}{f_s}\right)^2\right]$$

Average crack widths:

$$w = k_b \beta_c s_{rm} \varepsilon_{sm}$$

Maximum crack width:

$$w_k = \kappa_c w$$

Viewing SLS results

See **RC Slab Designer : Viewing Results** for details of plotting SLS reinforcement design contours.

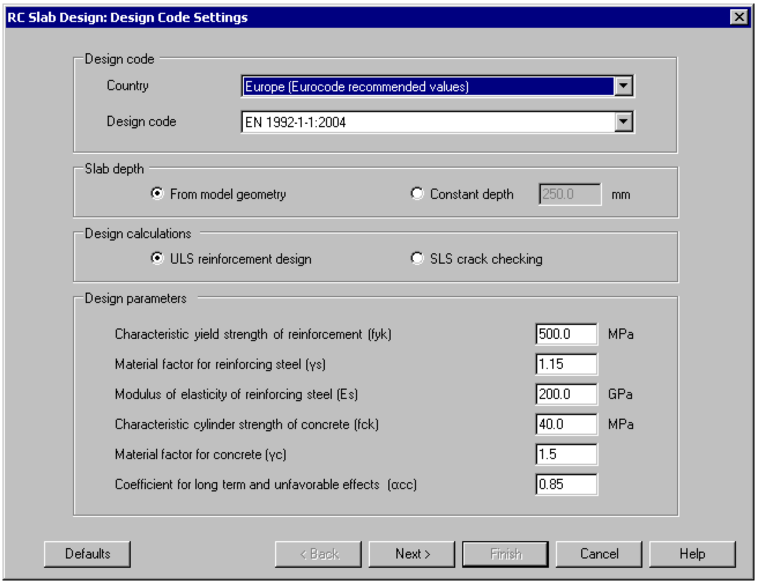
Design Code Settings and Parameters for Europe (Eurocodes)

Selecting **Europe (Eurocode Default Values)** and one of the EN design code options on the main Design Code Settings page of the RC Slab Designer will make it perform calculations, and offer defaults, based on either:

- **EN1992-1-1:2004 Eurocode 2**: Design of concrete structures - Part 1-1: General rules and rules for buildings. English version published within BS EN1992-1-1:2004, British Standards Institute, 23 December 2004
- **EN1992-2:2005 Eurocode 2**: Design of concrete structures - Part 2: Concrete bridges – Design and detailing rules. English version published within BS EN1992-2:2005, British Standards Institute, 2 December 2005

Selecting a supported **Country** and a corresponding **Eurocode** will set the Nationally Determined Parameters (NDPs) for that country and code. See **Eurocode supported countries and their respective National Annexes** for more details. The NDPs are read from an external file and can be set manually extended with additional NDPs, which will then be made available in the country selection on this page.

Note that regardless of model units, the Slab Designer uses units in keeping with the design code, i.e. N/mm², millimetres etc.



General Settings

See **RC Slab Designer : General Design Code Settings and Parameters** for details.

Design Parameters

- **Characteristic yield of reinforcement (fyk)** : When ULS reinforcement design is selected, f_{yk} is used in the calculation of the area of reinforcing steel required (and bar diameters) and in determining if the section is tension-controlled. It is not used for SLS crack checking calculations. Values acceptable in clause 3.2.2(3) lie in the range f_{yk}=400 to 600MPa. The default is f_{yk}=500MPa.
- **Material factor for steel (γs)** : When ULS reinforcement design is selected, γ_s is used in determining the design strength for concrete, hence affecting calculation of the depth in compression, the lever arm, classification of the section with regard to tension-control and the area of steel required directly. It is not used for SLS crack checking calculations. From clause 2.4.2.4 & Table 2.1N γ_s is an NDP and therefore suitable defaults are taken from the appropriate National Annex. Since different values apply for "accidental" design situations as compared to "persistent and transient" design situations, the defaults are for the latter. Broadly speaking γ_s=1.15 is adopted by most countries.

- **Elastic modulus of reinforcing steel (Es)** : When ULS reinforcement design is selected, Es is used in determining if the section is tension-controlled. When SLS crack checking is selected, the value is used in the calculation of the depth in compression. From clause 3.2.7(4) the typical value, taken as the default, is Es=200GPa
- **Characteristic compressive cylinder strength of concrete (fck)** : When ULS reinforcement design is selected, f_{ck} is used in the calculation of the lever arm, and in determining if the section is tension-controlled. When SLS crack checking is selected, the elastic modulus for concrete is determined using this value in the absence of specific user input. Values in EN1992-1-1 Table 3.1 lie in the range 12 to 90MPa. The default is f_{ck}=40MPa.
- **Material factor for concrete (γc)** : When ULS reinforcement design is selected, γ_c is used in determining the design strength for concrete, hence affecting calculation of the depth in compression, the lever arm, and the classification of the section with regard to tension-control. It is not used for SLS crack checking calculations. From clause 2.4.2.4 & Table 2.1N γ_c is an NDP and therefore suitable defaults are taken from the appropriate National Annex. Since different values apply for "accidental" design situations as compared to "persistent and transient" design situations, the defaults are for the latter. Broadly speaking γ_c=1.15 is adopted by most countries.
- **Coefficient for long term and unfavourable effects (Acc)** : When ULS reinforcement design is selected, α_{cc} is used in determining the design strength for concrete, hence affecting calculation of the depth in compression, the lever arm, and the classification of the section with regard to tension-control. It is not used for SLS crack checking calculations. From clause 2.4.2.4 and Table 2.1N α_{cc} is an NDP and therefore suitable defaults are taken from the appropriate National Annex. Broadly speaking values of α_{cc} =1.0 (buildings) and α_{cc} =0.85 (bridges) are adopted by most countries.

Reinforcement Details

See **RC Slab Designer : Reinforcement Details** for general information.

- **Bar sizes** Standard metric bar designations are used.
- **Spacing** [s] Typical values lie in the range 25mm to 300mm. The default is 150mm.
- **Cover** [c_{act}] This is the cover used to calculate effective depths, the "nominal cover specified on the drawings" as in clause 4.4.1.1(2). This may sometimes be distinct the cover distance used in the calculation of crack widths (see below). Typical values for cover would lie in the range 15 to 65mm. The default is 45mm.

ULS Reinforcement Design Output

See **Background to Calculations Carried out by the Slab Designer** for general information.

The flexural resistance is calculated in accordance with clauses 3.1.6 and 3.1.7 and Figure 3.5 – assuming a rectangular stress distribution in the compression zone. The basic formula used can be derived directly from moment equilibrium and strain compatibility.

Area of steel required

The formulae used are summarized below:

EN1992

$$z = d - \frac{\lambda x}{2} \quad ; \quad f_{cd} = \frac{\alpha_{cc} f_{ck}}{\gamma_c}$$

$$\lambda = \text{if}\left\{f_{ck} \leq 50\text{MPa}, 0.8, 0.8 - \frac{f_{ck} - 50}{400}\right\}$$

$$\eta = \text{if}\left\{f_{ck} \leq 50\text{MPa}, 1.0, 1.0 - \frac{f_{ck} - 50}{200}\right\}$$

$$\rho_l = \frac{A_{s,prov}}{bd} \qquad x = d\frac{f_{yd}}{\eta f_{cd} \lambda}\rho_l \qquad z = d - \frac{\lambda x}{2}$$

$$A_{s,req} = \frac{M_d}{f_{yd} z}$$

The nomenclature used above is as in the code excepting that:

M_d = factored moment at the section

$A_{s,req}$ = Area of tension reinforcement required

$A_{s,prov}$ = Area of tension reinforcement provided

Tension control ratio

The slab is ductile when the appropriate inequality below is met:

$$\varepsilon_{cu3} = \text{if}\left\{f_{ck} < 50\text{MPa}, 3.5\text{‰}, 2.6 + 35\left[\frac{(90 - f_{ck})}{100}\right]^4 \text{‰}\right\}$$

$$\frac{x}{d} \leq \left[\frac{f_{yd}}{E_s \varepsilon_{cu3}} + 1\right]^{-1}$$

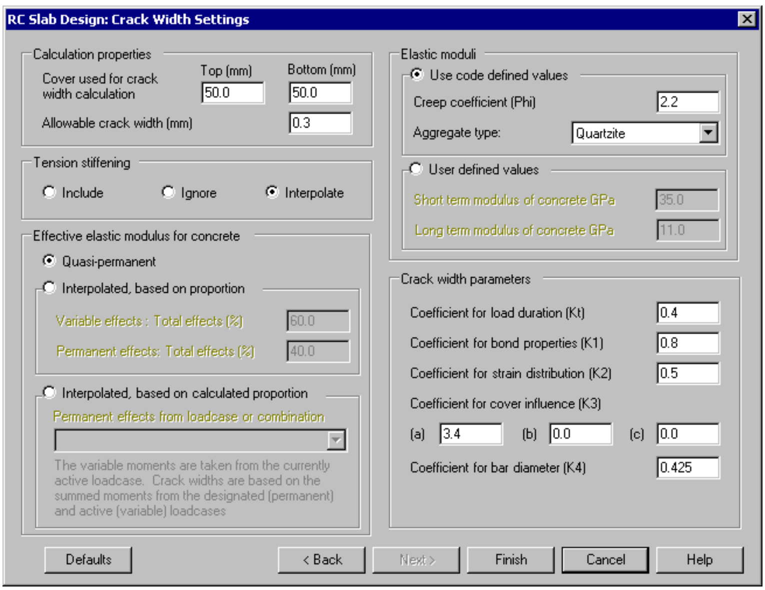
Viewing ULS results

See **RC Slab Designer : Viewing Results** for details of plotting ULS reinforcement design contours.

Crack Width Settings

See **Background to Calculations Carried out by the Slab Designer** for general information.

The Crack Width Settings page is only available if the SLS crack checking option is selected on the Design Code Settings page. Settings for Eurocodes are described here:



Calculation properties

- **Cover used for crack width calculation** The cover used for crack width calculation, c_{cr}. This value affects the design crack width calculation by changing the location at which the design crack width is assessed. In essence, the crack width is calculated at a plane lying a distance, c_{cr}, from the layer of bars controlling the crack. It is typically assumed that c_{cr} is equal to the actual cover (typically based on c_{nom}), making the crack width plane described coincident with the tension face of the concrete. However when the actual cover is significantly greater than the cover required in clause 4.4.1, setting c_{cr} to the lower value will calculate the crack width on a plane internal to the concrete, resulting in a smaller calculated crack width, an approach which may be deemed appropriate in certain circumstances. Values for c_{cr} would therefore typically be by reference to clause 4.4.1 and the default is 30mm.

- **Allowable crack width** [w_{max}] This value is used only to scale the crack width contours. From EN1992-1-1 clause 7.3.1(5), Table 7.1N, w_{max} is a NDP and therefore suitable defaults are taken from the appropriate National Annex. Broadly speaking a value of w_{max}=0.30 is adopted by most countries.

Tension Stiffening

The expression for the average strain in the reinforcement, ε_{sm} includes a reduction in consideration of the stiffening effect of tension carried by concrete. Where reinforcement does not lie perpendicular to cracks, whether the "tension stiffening" reduction should be included is a matter of debate (see SLS Crack Checking in **Background to Slab Designer Calculations).** The Slab Designer offers crack check calculations with the tension stiffening component included, ignored (conservative) or interpolated.

- **Include** Considers tension stiffening for all angles
- **Ignore** Tension stiffening is completely ignored for conservative design.
- **Interpolate** (Default option) Scales linearly from 0% tension stiffening at α=25°, to 100% at α=0°.

Calculation of effective elastic modulus for concrete

In the calculation of crack widths, strains are assessed using an effective elastic modulus for concrete. This effective modulus may be calculated in a number of ways to suit the user and code of practice as follows:

- **Quasi-permanent, based upon creep coefficient** For this the effective elastic modulus for concrete is taken as the long-term value (from user input or calculated as described below). This is generally regarded as conservative in respect of the elastic and creep deformations that might be expected as a result of the mix of permanent and transient loads that are likely to affect the structure. Accordingly, this is the default option.
- **Interpolated, based on proportion** For this the effective elastic modulus for concrete is calculated using a ratio, R, of moments due to live loading (M_q) to total moments, (M_g+M_q). R is a single value, determined by the user, which is entered as a percentage. The following expression (based on strain compatibility) is used to determine the effective modulus:

$$E_{c,eff} = \frac{E_{c,short} \times E_{c,long}}{E_{c,long} R + E_{c,short}(1 - R)}$$

- **Interpolated, based on calculated proportion** For this the effective elastic modulus for concrete is calculated using the ratio, R, as described above. In this case, however, R is evaluated at each node. The moments due to permanent loads, M_g, are taken from the loadcase selected in the droplist. The moments due to live loads, M_q, are taken from the active loadcase. The ratio at each node is:

$$R = \frac{M_q}{M_g + M_q}$$

Consistent with the above expression, total moments used for the calculation of crack width are based on $M_g + M_q$. The need for a suitable single value for R to be determined by the user is avoided.

Elastic Moduli

The modulus of elasticity of concrete, Ec, is used in calculation of flexural strains in the slab, leading to the calculation of a design crack width.

- **Use code defined values** When this option is invoked, the moduli used in calculations (as required), will be as follows:

  - Short term modulus based upon Table 3.1

    $$E_{c,short} = E_{cm} = 22\left(\frac{f_{cm}}{10}\right)^{0.3} \quad \text{where } f_{cm} = f_{ck} + 8$$

    This value must be adjusted for the aggregate in use according to clause 3.1.3(2).

  - Long term modulus for concrete assuming linear creep as in clause 7.2(2), can be based on clause 3.1.4.

    $$E_{c,long} = E_c(\infty, t_0) = E_{cm}\left[1 + \frac{\varphi(\infty, t_0)}{1.05}\right]^{-1}$$

  Use of the code defined values option requires the entry of two further parameters:

- **Creep coefficient φ** : This should be determined from clause 3.1.4 (Fig 3.1). The default is taken as ψ = 2.2.
- **Aggregate type** : From clause 3.1.3(2) four types are offered which modify the Modulus of Elasticity as appropriate.
- **User defined values** When this option is invoked, values for elastic moduli may be entered instead of using those determined automatically. The default values are based on the expressions described previously.

Eurocode crack width factors

- **Coefficient for load duration (Kt)** : Coefficient for duration of loading, k_t. In clause 7.3.4(2), k_t is given as 0.6 for short term loading or 0.4 for long term loading. For most nations, crack width calculations are for the quasi-permanent design situation in the Eurocode (see EN1992-1-1 Table 7.1N and EN1992-2 Table 7.101N), therefore the default is taken as k_t=0.4.
- **Coefficient for bond properties (K1)** : In clause 7.3.4(3), k_1 is a coefficient which takes account of the bond properties of the bonded reinforcement, taken as 0.8 for high bond bars or 1.6 for bars with an effectively plain surface. The default is k_1 = 0.8.
- **Coefficient for strain distribution (K2)** : In clause 7.3.4(3), k_2 is a coefficient which takes account of the form of the strain distribution, with the values 0.5 for bending and 1.0 for pure tension. Since the scope of the slab designer is restricted to moment fields, the appropriate value would in most cases be 0.5 and the default is accordingly k_2=0.5.
- **Coefficient for cover influence (K3)**Coefficient controlling the influence of cover on crack spacing. In clause 7.3.4(3), k_3 is an NDP. It requires special treatment for some nations and is taken to be of the form:

  $$k_3 = k_{3A}.\varnothing^{k_{3B}}.c_{cr}^{\ k_{3C}}$$

  where generally $k_{3B} \geq 0$ and $k_{3C} \leq 0$. For most nations, k_{3A}=3.4, k_{3B}=0 and k_{3C}=0, but defaults are set as appropriate to the National Annex.

- **Coefficient for bar diameter (K4)** : Coefficient controlling the influence of bar diameter on crack spacing. In clause 7.3.4(3), k_4 is an NDP and therefore suitable defaults are taken from the appropriate National Annex. Broadly speaking a value of k_4 =0.425 is adopted by most countries.

SLS crack width calculation to Eurocode

Crack width calculations for EN1992 are carried out by reference to clause 7.3.4 and 7.3.2(3):

EN1992

Effective area of concrete in tension:

$$A_{c,eff} = b.\min\left\{2.5(h-d), \frac{h-x}{3}, \frac{h}{2}\right\}$$

$$\rho_{p,eff} = A_{s,prov}/A_{c,eff}$$

Crack spacing:

$$s_{r,max} = if\left\{s \le 5\left(c + \frac{\phi}{2}\right), k_3 c + \frac{k_1 k_2 k_4 \phi}{\rho_{p,eff}}, 1.3(h - x)\right\}$$

Cracking strain showing tension stiffening component in square brackets:

$$\varepsilon_{sm} - \varepsilon_{cm} = \frac{\sigma_s - k_t f_{ct,eff}\alpha_e}{E_s} - \left[\frac{k_t \frac{f_{ct,eff}}{\rho_{p,eff}}}{E_s}\right]$$

The mean value of the tensile strength of the concrete effective at the time when cracks may be first expected to occur:

$$f_{ct,eff} = f_{ctm} = \text{if}\left\{f_{ck} \le 50, 0.3 f_{ck}{}^{2/3}, 2.12\ln\left(1 + \frac{f_{ck} + 8}{10}\right)\right\}$$

Crack widths:

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

Viewing SLS results

See **RC Slab Designer : Viewing Results** for details of plotting SLS reinforcement design contours.

Eurocode supported countries and their respective National Annexes

Selecting a Eurocode country and “EN1992-1-1” or “EN1992-2” design code makes the Slab Designer perform calculations, and offer defaults, based on the following publications:

Country	National Annex reference document
Austria	ONORM EN 1992-1-1/NA:2007
Belgium	prNBN EN 1992-1-1/NA:2007
Bulgaria	BDS EN 1992-1-1:2005/NA:2011 and BDS EN 1992-2:2006/NA:2012
Cyprus	CYS EN 1992-1-1:2004/NA:2009 and CYS EN 1992-2:2005/NA:2009
Czech Republic	CSN EN 1992-1-1/NA:2007
Denmark	DS/EN 1992-1-1 DK NA:2011 and VD/EN 1992-2 DK NA:2009
Finland	SFS-EN 1992-1-1/NA:2009 and SFS-EN 1992-2/NA:2010
France	NF EN 1992-1-1/NA:2007
Germany	prDIN EN 1992-1-1/NA:2009
Greenland	DS/EN 1992-1-1 GL NA:2009
Ireland	I.S. EN 1992-1-1/NA: 2010 and I.S. EN 1992-2/NA: 2010
Italy	UNI-EN 1992-1-1/ NA:2007 and UNI-EN 1992-2/NA:2007
Netherlands	NEN EN 1992-1-1/NA:2007
Poland	PN EN 1992-1-1/NA:2008
Romania	SR EN 1992-1-1/NB 2008 and SR EN 1992-2/NB 2009
Singapore	SS EN 1992-1-1/NA 2008 inc Amd 1
Slovakia	STN EN 1992-1-1/NA:2007
Spain	AN/UNE-EN 1992-1-1: 2012 and AN/UNE-EN 1992-2: 2012
Sweden	TRVFS 2001:12 (EN 1992)
United Kingdom	BS EN 1992-1-1:2004/ NA: 2005 and BS EN 1992-2:2005/ NA: 2007

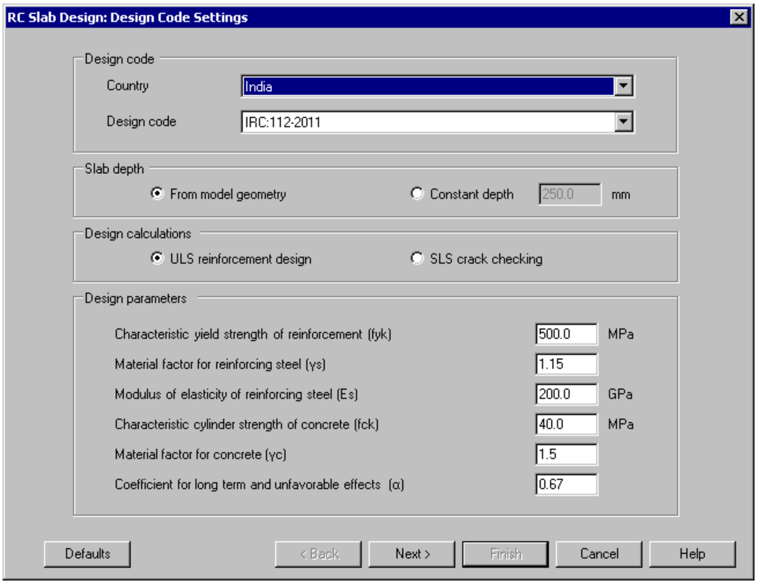
Nationally determined parameters (NDP) for these countries are stored in the **EurocodeParameters.xml** file in the **<LUSAS Installation Folder>\Programs\scripts\User** folder. When a national annex of the Eurocode is selected the NDP's are read from the xml file and displayed in the text fields on the dialog for use in the calculations. Additional national annexes added to the file will be displayed in the list of available design codes. Note the design calculations carried out for all Eurocodes are the same, only the values of the NDPs change when selecting national annexes.

Design Code Settings and Parameters for India IRC:112-2011

Selecting **India** and the IRC:112-2011 design code option on the main Design Code Settings page of the RC Slab Designer will make it perform calculations, and offer defaults, based on:

- **IRC: 112:2011**: Code of Practice for Concrete Road Bridges, Indian Roads Congress. (Indian Highway Bridge Design Code)

Note that regardless of model units, the Slab Designer uses units in keeping with the design code, i.e. N/mm², millimetres etc., as shown on the dialog.



General Settings

See **RC Slab Designer : General Design Code Settings and Parameters** for details.

Design Parameters

- **Characteristic yield of reinforcement (fyk)** : When ULS reinforcement design is selected, f_{yk} is used in the calculation of the area of reinforcing steel required (and bar diameters) and in determining if the section is tension-controlled. It is not used for SLS crack checking calculations. Values acceptable in clause 6.2.2, Table 18.1 lie in the range f_{yk}=240 to 600MPa. The default is f_{yk}=500MPa.
- **Material factor for steel (γs)** : When ULS reinforcement design is selected, γ_s is used in determining the design strength for concrete, hence affecting calculation of the depth in compression, the lever arm, classification of the section with regard to tension-control and the area of steel required dierctly. It is not used for SLS crack checking calculations. Broadly speaking γ_s=1.15 is adopted by most countries.
- **Elastic modulus of reinforcing steel (Es)** : When ULS reinforcement design is selected, E_s is used in determining if the section is tension-controlled. When SLS crack checking is selected, the value is used in the calculation of the depth in compression. From clause 6.2.2 the typical value, taken as the default, is E_s=200GPa
- **Characteristic compressive cylinder strength of concrete (fck)** : When ULS reinforcement design is selected, f_{ck} is used in the calculation of the lever arm, and in determining if the section is tension-controlled. When SLS crack checking is selected,

the elastic modulus for concrete is determined using this value in the absence of specific user input for that parameter. Refer to IRC:112-2011 Table 6.5 for a range of values. The default is f_{ck}=40MPa.

- **Material factor for concrete (γc)** : When ULS reinforcement design is selected, γ_c is used in determining the design strength for concrete, hence affecting calculation of the depth in compression, the lever arm, and the classification of the section with regard to tension-control. It is not used for SLS crack checking calculations. Broadly speaking γ_c=1.15 is adopted by most countries.
- **Coefficient for long term and unfavourable effects (Acc)** : When ULS reinforcement design is selected, α_{cc} is used in determining the design strength for concrete, hence affecting calculation of the depth in compression, the lever arm, and the classification of the section with regard to tension-control. It is not used for SLS crack checking calculations. For IRC:112-2011 a value of α_{cc} =0.67 is used.

Reinforcement Details

See **RC Slab Designer : Reinforcement Details** for general information.

- **Bar sizes** Standard metric bar designations are used.
- **Spacing** [s] Typical values lie in the range 25mm to 300mm. The default is 150mm.
- **Cover** [c_{act}] This is the cover used to calculate effective depths, the "nominal cover specified on the drawings". This may sometimes be distinct from the cover distance used in the calculation of crack widths (see below). The default is 45mm.

ULS Reinforcement Design Output

See **Background to Calculations Carried out by the Slab Designer** for general information.

The flexural resistance is calculated in accordance with clauses 6.4.2.8(1)(a) and 6.4.2.8(1)(b) and Figure A2-4 – assuming a rectangular stress distribution in the compression zone. The basic formula used can be derived directly from moment equilibrium and strain compatibility.

IRC:112-2011

Area of steel:

$$z = d - \frac{\lambda x}{2} \quad ; \quad f_{cd} = \frac{\alpha_{cc} f_{ck}}{\gamma_c}$$

$$\lambda = \text{if}\left\{f_{ck} \le 60\text{MPa}, 0.8, 0.8 - \frac{f_{ck} - 60}{500}\right\}$$

$$\eta = \text{if}\left\{f_{ck} \leq 60\text{MPa}, 1.0, 1.0 - \frac{f_{ck} - 60}{250}\right\}$$

$$\rho_l = \frac{A_{s,prov}}{bd} \qquad x = d\frac{f_{yd}}{\eta f_{cd}\lambda}\rho_l \qquad z = d - \frac{\lambda x}{2}$$

$$A_{s,req} = \frac{M_d}{f_{yd} z}$$

The nomenclature used above is as in the code excepting that:

M_d = factored moment at the section

$A_{s,req}$ = Area of tension reinforcement required

$A_{s,prov}$ = Area of tension reinforcement provided

For tension control the slab is ductile when the appropriate inequality below is met:

$$\varepsilon_{cu3} = \text{if}\left\{f_{ck} > 60\text{MPa}, 2.6 + 35\left[\frac{(90 - 0.8f_{ck})}{100}\right]^4, 3.5\right\}‰$$

$$\frac{x}{d} \leq \left[\frac{f_{yd}}{E_s \varepsilon_{cu3}} + 1\right]^{-1}$$

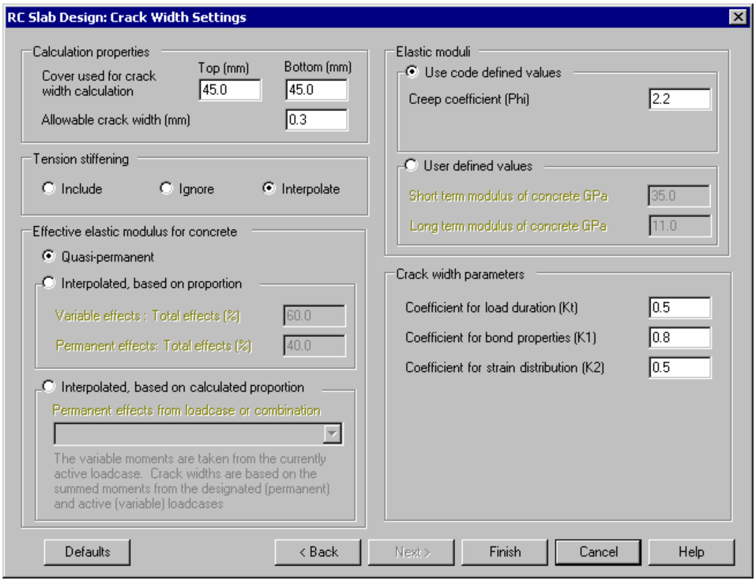
Viewing ULS results

See **RC Slab Designer : Viewing Results** for details of plotting ULS reinforcement design contours.

Crack Width Settings

See **Background to Calculations Carried out by the Slab Designer** for general information.

The Crack Width Settings page is only available if the SLS crack checking option is selected on the Design Code Settings page.



Calculation properties

- **Cover used for crack width calculation** The cover used for crack width calculation, c_{cr}. This value affects the design crack width calculation by changing the location at which the design crack width is assessed. In essence, the crack width is calculated at a plane lying a distance, c_{cr}, from the layer of bars controlling the crack. It is typically assumed that c_{cr} is equal to the actual cover (typically based on c_{nom}), making the crack width plane described coincident with the tension face of the concrete. However when the actual cover is significantly greater than the cover required in clause 14.3.2.1 Table 14.2, setting c_{cr} to the lower value will calculate the crack width on a plane internal to the concrete, resulting in a smaller calculated crack width, an approach which may be deemed appropriate in certain circumstances. Values for c_{cr} would therefore typically be by reference to clause 14.3.2.1 and the default is 45mm.
- **Allowable crack width** [w_{max}] This value is used only to scale the crack width contours. Refer to IRC:112-2011 Table 12.1. A default of 0.3mm is used.

Tension Stiffening

The expression for the average strain in the reinforcement, εsm includes a reduction in consideration of the stiffening effect of tension carried by concrete. As described in SLS crack checking where reinforcement does not lie perpendicular to cracks, whether the "tension stiffening" reduction should be included is a matter of debate. The Slab Designer offers crack check calculations with the tension stiffening component included, ignored (conservative) or interpolated.

- **Include** Considers tension stiffening for all angles
- **Ignore** Tension stiffening is completely ignored for conservative design.
- **Interpolate** (Default option) Scales linearly from 0% tension stiffening at α=25°, to 100% at α=0°.

Calculation of effective elastic modulus for concrete

In the calculation of crack widths, strains are assessed using an effective elastic modulus for concrete. This effective modulus may be calculated in a number of ways to suit the user and code of practice as follows:

- **Quasi-permanent, based upon creep coefficient** For this the effective elastic modulus for concrete is taken as the long-term value (from user input or calculated as described below). This is generally regarded as conservative in respect of the elastic and creep deformations that might be expected as a result of the mix of permanent and transient loads that are likely to affect the structure. Accordingly, this is the default option.
- **Interpolated, based on proportion** For this the effective elastic modulus for concrete is calculated using a ratio, R, of moments due to live loading (M_q) to total moments, (M_g+M_q). R is a single value, determined by the user, that is entered as a percentage. The following expression (based on strain compatibility) is used to determine the effective modulus:

$$E_{c,eff} = \frac{E_{c,short} \times E_{c,long}}{E_{c,long}R + E_{c,short}(1 - R)}$$

- **Interpolated, based on calculated proportion** For this the effective elastic modulus for concrete is calculated using the ratio, R, as described above. In this case, however, R is evaluated at each node. The moments due to permanent loads, M_g, are taken from the loadcase selected in the droplist. The moments due to live loads, M_q, are taken from the active loadcase. The ratio at each node is:

$$R = \frac{M_q}{M_g + M_q}$$

  Consistent with the above expression, total moments used for the calculation of crack width are based on $M_g + M_q$. The need for a suitable single value for R to be determined by the user is avoided.

Elastic Moduli

The modulus of elasticity of concrete, E_c, is used in calculation of flexural strains in the slab, leading to the calculation of a design crack width.

- **Use code defined values** When this option is invoked, the moduli used in calculations (as required), will be as follows:

- **Short term modulus of concrete** based upon Table 3.1

$$E_{c,short} = E_{cm} = 22\left(\frac{f_{cm}}{10}\right)^{0.3} \quad \text{where} \quad f_{cm} = f_{ck} + 10$$

- **Long term modulus of concrete** assuming linear creep as in clause 7.2(2), can be based on clause 3.1.4.

$$E_{c,long} = E_c(\infty, t_0) = E_{cm}[1 + \varphi(\infty, t_0)]^{-1}$$

Use of code defined values requires the entry of a further parameter:

- **Creep coefficient φ** : This should be determined from clause 6.4.2.7, Table 6.9. The default is taken as ψ = 2.2.
- **User defined values** When this option is invoked, values for elastic moduli may be entered instead of using those determined automatically. The default values are based on the expressions described previously.

Crack width parameters

- **Coefficient for load duration (Kt)** : Coefficient for duration of loading, k_t. In clause 12.3.4(2), k_t is given as 0.6 for short term loading or 0.4 for long term loading. The default is taken as k_t=0.4.
- **Coefficient for bond properties (K1)** : In clause 12.3.4(3), k_1 is a coefficient which takes account of the bond properties of the bonded reinforcement, taken as 0.8 for high bond bars or 1.6 for bars with an effectively plain surface. The default is k_1 = 0.8.
- **Coefficient for strain distribution (K2)** : In clause 12.3.4(3), k_2 is a coefficient which takes account of the form of the strain distribution, with the values 0.5 for bending and 1.0 for pure tension. Since the scope of the slab designer is restricted to moment fields, the appropriate value would in most cases be 0.5 and the default is accordingly k_2=0.5.

SLS crack width calculation

Crack width calculations for IRC:112-2011 are carried out by reference to clauses 12.3.4 and 12.3.3:

IRC:112-2011

Effective area of concrete in tension:

$$A_{c,eff} = b.\min\left\{2.5(h-d), \frac{h-x}{3}, \frac{h}{2}\right\}$$

$$\rho_{p,eff} = A_{s,prov}/A_{c,eff}$$

Crack spacing:

$$s_{r,max} = if\left\{s \le 5\left(c + \frac{\phi}{2}\right), k_3 c + \frac{k_1 k_2 k_4 \phi}{\rho_{p,eff}}, 1.3(h - x)\right\}$$

Cracking strain showing tension stiffening component in square brackets:

$$\varepsilon_{sm} - \varepsilon_{cm} = \frac{\sigma_s - k_t f_{ct,eff} \alpha_e}{E_s} - \left[\frac{k_t \frac{f_{ct,eff}}{\rho_{p,eff}}}{E_s}\right]$$

The mean value of the tensile strength of the concrete effective at the time when cracks may be first expected to occur:

$$f_{ct,eff} = f_{ctm} = \text{if}\left\{f_{ck} \le 60, 0.259 f_{ck}{}^{2/3}, 2.27\ln\left(1 + \frac{f_{cm}}{12.5}\right)\right\}$$

Crack widths:

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

Viewing SLS results

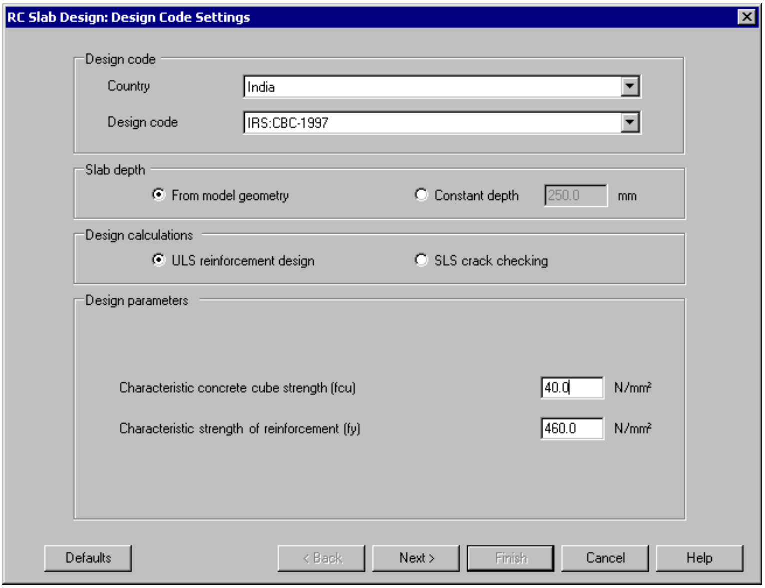
See **RC Slab Designer : Viewing Results** for details of plotting SLS reinforcement design contours.

Design Code Settings and Parameters for India (IRS:CBC-1997)

Selecting **India** and the IRS:CBC-1997 design code options on the main Design Code Settings page of the RC Slab Designer will make it perform calculations, and offer defaults, based upon:

- **IRS:CBC-1997**, IRS Concrete Bridge Code 1997, Incorporating A & C slip no 7, 2003. Indian Railway Standard, Code of Practice for Plain, Reinforced and Prestressed Concrete for General Bridge Construction Research Designs and Standards Organisation

Note that regardless of model units, the Slab Designer uses units in keeping with the design code, i.e. N/mm², millimetres etc., as shown on the dialog



General Settings

See **RC Slab Designer : General Design Code Settings and Parameters** for details.

Design Parameters

The following parameters are required:

- **Characteristic concrete cube strength, fcu** When ULS reinforcement design is selected, f_{cu} is used in the calculation of the lever arm, z, and in determining if the section is tension-controlled. When SLS crack checking is selected, the elastic modulus for concrete is determined using this value in the absence of specific user input for that parameter. Typical values lie in the range 20 to 60N/mm². The default is 40N/mm²
- **Characteristic strength of reinforcement, fy** When ULS reinforcement design is selected f_y is used in the calculation of the area of reinforcing steel required (and bar diameters) and in determining if the section is tension-controlled. It is not used for SLS crack checking calculations. Typical values would be f_y=250, 460 or 485N/mm². The default is 460N/mm².

Reinforcement Details

See **RC Slab Designer : Reinforcement Details** for general information.

- **Bar sizes** Nominal diameters (and cross-sectional areas).
- **Spacing** [s] Typical values lie in the range 25mm to 300mm with a typical aggregate size of 20mm). The default is 150mm.
- **Cover** [c_{act}] This is the cover used to calculate effective depths, as distinct from nominal cover (c_{nom}), used in the calculation of crack widths (see below). For various reasons, including durability considerations or fixing tolerances, the specified cact may be greater than c_{nom}. Typical values for c_{act} would, however, lie in the range 20 to 75mm. The default is 50mm.

ULS Reinforcement Design

See **Background to Calculations Carried out by the Slab Designer** for general information.

For nominal resistance calculations IRS:CBC-1997 clause 15.4.2.2.1 is used.

Area of steel required

The formulae can be summarized as follows:

$$A_{s,req} = \frac{M_u}{0.87 f_y z}$$

$$z = \left(1 - \frac{1.1 f_y A_{s,prov}}{f_{cu} b d}\right) d \ngtr 0.95d$$

The nomenclature used above is as in the relevant codes excepting that:

$A_{s,req}$ = Area of tension reinforcement required

$A_{s,prov}$ = Area of tension reinforcement provided

Tension control ratio

The slab is ductile when the inequality below is met:

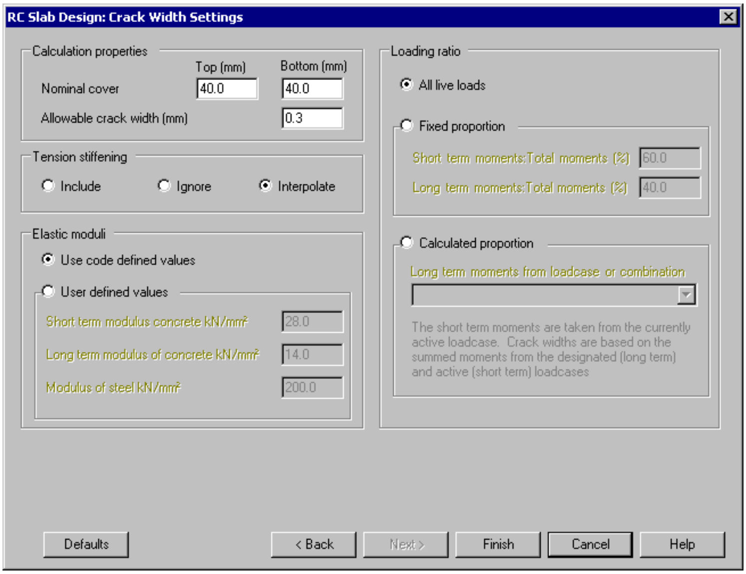
$$\frac{M_u / b d^2 f_{cu}}{0.15} \leq 1$$

Viewing ULS results

See **RC Slab Designer : Viewing Results** for details of plotting ULS reinforcement design contours.

Crack Width parameters

The Crack Width Settings page of the RC Slab Designer will appear only if the SLS crack checking option was selected on the main Design Code Settings page.



Calculation properties

- **Nominal Cover** [c_{nom}] This value affects the design crack width calculation by changing the location at which the design crack width is assessed. In essence, the crack width is calculated at a plane lying a distance, c_{nom}, from the layer of bars controlling the crack. Typical values lie in the range 25 to 75mm and should be determined from IRS:CBC-1997 clause 15.9.2.2. The default is 50mm.
- **Allowable crack width (mm)** The value entered is only used to the scale the crack width contours. IRS:CBC-1997 Table 10 relates design crack width requirements to exposure conditions, with values from 0.1mm to 0.3mm. The default value is taken as 0.2mm.

Tension stiffening

IRS:CBC-1997 cl 15.9.8.2.1 shows an equation which includes a reduction in the calculated crack width in consideration of the stiffening effect of tension carried by concrete which remains bonded to the reinforcement in the tensile zone at small strains. As described in SLS crack checking where reinforcement does not lie perpendicular to cracks, whether the "tension stiffening" reduction should be included is a matter of debate. The Slab Designer offers three options for crack check calculations:

- **Include** Considers tension stiffening for all angles
- **Ignore** Tension stiffening is completely ignored for conservative design.
- **Interpolate** (Default option) Scales linearly from 0% tension stiffening at α=25°, to 100% at α=0°.

Elastic Moduli

The elastic moduli for concrete and reinforcing steel are used in calculation of flexural strains in the slab, leading to the calculation of a design crack width.

- **Use code defined values** When this option is invoked, the moduli used in calculations (as required), will be as stated in the table below.
- **User defined values** When this option is invoked, users may enter values for elastic moduli instead of using those determined automatically. The default values are based on the values and expressions in the code of practice described previously.

Short term modulus for concrete	Long term modulus for concrete	Elastic modulus for steel
E_c,short = 0.27×f_{cu} + 20 See note 1.	m = 280/f_{ck}	200kN/mm²

Notes

1. This equation (rounded to 2 significant figures) closely approximates Cl 5.5.2.1 Table 3 although the value for f_{ck}=25N/mm² is calculated as Ec=27kN/mm² instead of 26kN/mm² as given in Table 3. Use of the equation is felt preferable as it is more flexible for values of f_{ck} other than those given in Table 3.

Loading Ratio

In the calculation of crack widths, strains are assessed using an effective elastic modulus for concrete. This effective modulus may be calculated in a number of ways to suit the user and code of practice as follows:

- **All live loads** This assumption is conservative in consideration of Clause 5.2.6. This is the default option.
- **Fixed proportion** The effective elastic modulus for concrete is calculated using a ratio, R, of moments due to live loading (M_q) to total moments, (M_g+M_q). R is entered by the user as a percentage and the following expression (based on strain compatibility) is used to determine the effective modulus:

$$E_{c,eff} = \frac{E_{c,short} \times E_{c,long}}{E_{c,long} R + E_{c,short}(1 - R)}$$

Assuming a suitable single value for R can be determined by the user, this option suits the approach of clause 5.2.6 when $E_{c,long} = E_{c,short}/2$, i.e. when “use code defined values” is selected.

- **Calculated proportion** The effective elastic modulus for concrete is calculated using the ratio, R, as described above. In this case, however, R is evaluated at each node. The moments due to permanent loads, M_g, are taken from the loadcase selected in the droplist. The moments due to live loads, M_q, are taken from the active loadcase. The ratio at each node is:

  $$R = \frac{M_q}{M_g + M_q}$$

  Consistent with the above expression, total moments used for the calculation of crack width are based on $M_g + M_q$. This option suits the approach of clause 5.2.6 when $E_{c,long} = E_{c,short}/2$ i.e. when “use code defined values” is selected. The need for a suitable single value for R to be determined by the user is avoided.

SLS crack width calculation

See **Background to Calculations Carried out by the Slab Designer** for general information.

Crack width calculations are carried out by reference to IRS:CBC-1991 clause 15.9.8.2.1 as follows:

IRS:CBC-1997

$$\epsilon_s = \frac{M}{E_s A_s \left(d - \frac{x}{3}\right)}$$

$$\epsilon_1 = \frac{\epsilon_s (a' - x)}{(d - x)}$$

Reduction due to strain stiffening

$$\epsilon_2 = \left[\frac{3.8 b_t h(a' - d_c)}{\epsilon_s A_s (h - d_c)}\right]\left[\left(1 - \frac{M_q}{M_g}\right)10^{-9}\right]$$

Strain at cracking level

$$\epsilon_m = \epsilon_1 - \epsilon_2$$

Distance to controlling bar

$$a_{cr} = \sqrt{(h - d)^2 + \left({}^{s}/_{2}\right)^2} - {}^{\emptyset}/_{2}$$

Crack widths

$$w = \frac{3 a_{cr} \epsilon_m}{1 + 2\left(\frac{a_{cr} - c_{nom}}{h - d_c}\right)}$$

The spacing and bar diameter of the bars identified as "controlling" the crack are used to determine the distance from the crack to the nearest bar, a_{cr}.

Viewing SLS results

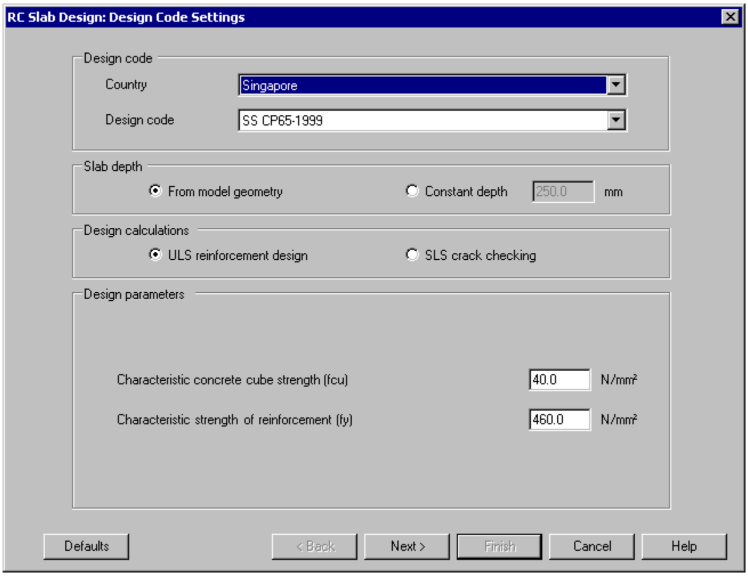
See **RC Slab Designer : Viewing Results** for details of plotting SLS reinforcement design contours.

Design Code Settings and Parameters for Singapore (Eurocode / CP65-1999)

Selecting **Singapore** and one of the design code options on the main Design Code Settings page of the RC Slab Designer will make it perform calculations, and offer defaults, based on the appropriate Singapore publication below:

- **SS EN1992-1-1/NA 2008 inc Amd 1** Design of Concrete Structures (buildings)
- **SS CP65-1999** Design of Concrete Structures - general rules for buildings

Note that regardless of model units, the Slab Designer uses units in keeping with the design code, i.e. N/mm², millimetres etc., a shown on the dialog.



General Settings

See **RC Slab Designer : General Design Code Settings and Parameters** for details.

Design Parameters

For SS EN1992-1-1/NA 2008 inc Amd 1 see **RC Slab Designer : Design Code Settings and Parameters for Eurocode**

For SS CP65-1999 the following parameters are required:

- **Characteristic concrete cube strength, fcu** When ULS reinforcement design is selected, f_{cu} is used in the calculation of the lever arm, z, and in determining if the section is tension-controlled. When SLS crack checking is selected, the elastic modulus for concrete is determined using this value in the absence of specific user input for that parameter. Typical values lie in the range 20 to 60N/mm². The default is 40N/mm²
- **Characteristic strength of reinforcement, fy** When ULS reinforcement design is selected f_y is used in the calculation of the area of reinforcing steel required (and bar diameters) and in determining if the section is tension-controlled. It is not used for SLS crack checking calculations. Typical values would be f_y=250, 460 or 485N/mm². The default is 460N/mm².

Reinforcement Details

See **RC Slab Designer : Reinforcement Details** for general information.

- **Bar sizes** Nominal diameters (and cross-sectional areas) .
- **Spacing** [s] Typical values lie in the range 25mm to 300mm, (with a typical aggregate size of 20mm). The default is 150mm.
- **Cover** [c_{act}] This is the cover used to calculate effective depths, as distinct from nominal cover (c_{nom}), used in the calculation of crack widths (see below). For various reasons, including durability considerations or fixing tolerances, the specified cact may be greater than c_{nom}. Typical values for c_{act} would, however, lie in the range 20 to 75mm (CP65 clause 3.3). The default is 50mm.

ULS Reinforcement Design to CP65-1999

See **Background to Calculations Carried out by the Slab Designer** for general information.

The formulae are according to CP65 clause 3.4.4.4 (as referenced from clause 3.5.1 regarding slabs).

Area of steel required

The formulae used are summarized below:

CP65

$$A_{s,req} = \frac{M_d}{0.87 f_y z}$$

$$z = d\left\{0.5 + \sqrt{0.25 - \frac{K}{0.9}}\right\} \ngtr 0.95d$$

$$K = \frac{M}{bd^2 f_{cu}}$$

Where:

$A_{s,req}$ = Area of tension reinforcement required

Tension control ratio

The slab is ductile when the appropriate inequality below is met:

CP65

$$\frac{K}{K'} = \frac{K}{0.156} \leq 1$$

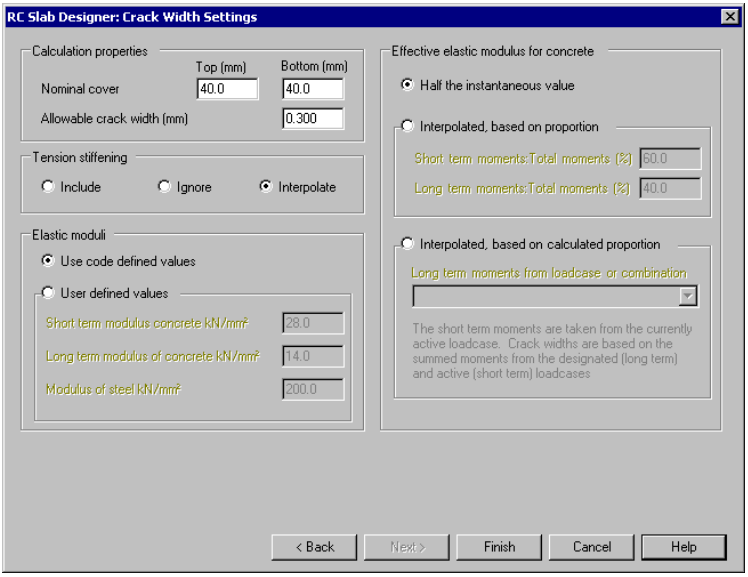
Viewing ULS results

See **RC Slab Designer : Viewing Results** for details of plotting ULS reinforcement design contours.

Crack Width parameters for CP65-1999

The Crack Width Settings page of the RC Slab Designer will appear only if the SLS crack checking option was selected on the main Design Code Settings page.

Settings for the CP65-1999 code are described here:



Calculation properties

- **Nominal Cover** [c_{nom}] This value affects the design crack width calculation by changing the location at which the design crack width is assessed. In essence, the crack width is calculated at a plane lying a distance, c_{nom}, from the layer of bars controlling the crack. For CP65-1999 it may be assumed that c_{nom} is equal to the actual cover, $c_{act,}$ making the crack width plane described coincident with the tension face of the concrete. Typical values lie in the range 20 to 70mm and should be determined from BS8110-1 clause 3.4, Tables 3.4 and 3.5. The default is 40mm.

- **Allowable crack width (mm)** For CP65-1999 the value is used to the scale the crack width contours. Clause 3.12.11.2 indicates a limitation of 0.3mm. The default value is taken as 0.2mm

Tension stiffening

CP65-1999 cl 3.8.3, eqn 13, includes a reduction in the calculated crack width in consideration of the stiffening effect of tension carried by concrete which remains bonded to the reinforcement in the tensile zone at small strains. As described in SLS crack checking where reinforcement does not lie perpendicular to cracks, whether the "tension stiffening" reduction should be included is a matter of debate. The Slab Designer offers three options for crack check calculations:

- **Include** Considers tension stiffening for all angles
- **Ignore** Tension stiffening is completely ignored for conservative design.
- **Interpolate** (Default option) Scales linearly from 0% tension stiffening at α=25°, to 100% at α=0°.

Elastic Moduli

The elastic moduli for concrete and reinforcing steel are used in calculation of flexural strains in the slab, leading to the calculation of a design crack width.

- **Use code defined values** When this option is invoked, the moduli used in calculations (as required), will be as stated in the table below.
- **User defined values** When this option is invoked, users may enter values for elastic moduli instead of using those determined automatically. The default values are based on the values and expressions described previously.

Short term modulus for concrete	Long term modulus for concrete	Elastic modulus for steel
$E_{c,short}$ = 0.2×fcu + 20	$E_{c,long} = E_{c,short/2}$	200kN/mm²
CP65-Part 2 1999 eqn 17, clause 7.2	BS5400-4 clause 4.3.2.1 (see note 1)	CP65-1999 clause 2.5.4

Notes

1. The reference to BS5400-4 is for an approximation taking into account the effect of creep under long term loading, since reference to CP65-Part 2 1999 clause 3.6(a)(3) requires determination of a creep coefficient. This approximation is broadly validated with reference to CIRIA Report 110.

Calculate Effective Elastic Modulus for Concrete

In the calculation of crack widths, strains are assessed using an effective elastic modulus for concrete. This effective modulus may be calculated in a number of ways to suit the user and code of practice as follows:

- **Half the instantaneous value** The effective elastic modulus for concrete is taken as half the instantaneous value, as per BS8110-2 clause 3.8.3. This is the default option.
- **Interpolated, based on proportion** The effective elastic modulus for concrete is calculated using a ratio, R, of moments due to live loading (M_q) to total moments, (M_g+M_q). R is entered by the user as a percentage and the following expression (based on strain compatibility) is used to determine the effective modulus:

$$E_{c,eff} = \frac{E_{c,short} \times E_{c,long}}{E_{c,long}R + E_{c,short}(1-R)}$$

- **Interpolated, based on calculated proportion** The effective elastic modulus for concrete is calculated using the ratio, R, as described above. In this case, however, R is evaluated at each node. The moments due to permanent loads, M_g, are taken from the loadcase selected in the droplist. The moments due to live loads, M_q, are taken from the active loadcase. The ratio at each node is:

$$R = \frac{M_q}{M_g + M_q}$$

  Consistent with the above expression, total moments used for the calculation of crack width are based on $M_g + M_q$.

SLS crack width calculation to CP65-1999

See **Background to Calculations Carried out by the Slab Designer** for general information.

Crack width calculations are carried out by reference to CP65 clause 3.8.3 and may be summarised as follows:

CP65

$$\epsilon_s = \frac{M}{E_s A_s \left(d - \frac{x}{3}\right)}$$

$$\epsilon_1 = \frac{\epsilon_s (a' - x)}{(d - x)}$$

Reduction due to strain stiffening

$$\epsilon_2 = \frac{b_t(h-x)(a'-x)}{3E_sA_s(d-x)}$$

Strain at cracking level

$$\epsilon_m = \epsilon_1 - \epsilon_2$$

Distance to controlling bar

$$a_{cr} = \sqrt{(h-d)^2 + \left({}^{s}/_{2}\right)^2} - {}^{\emptyset}/_{2}$$

Crack widths

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

The spacing and bar diameter of the bars identified as "controlling" the crack are used to determine the distance from the crack to the nearest bar, a_{cr}.

Viewing SLS results

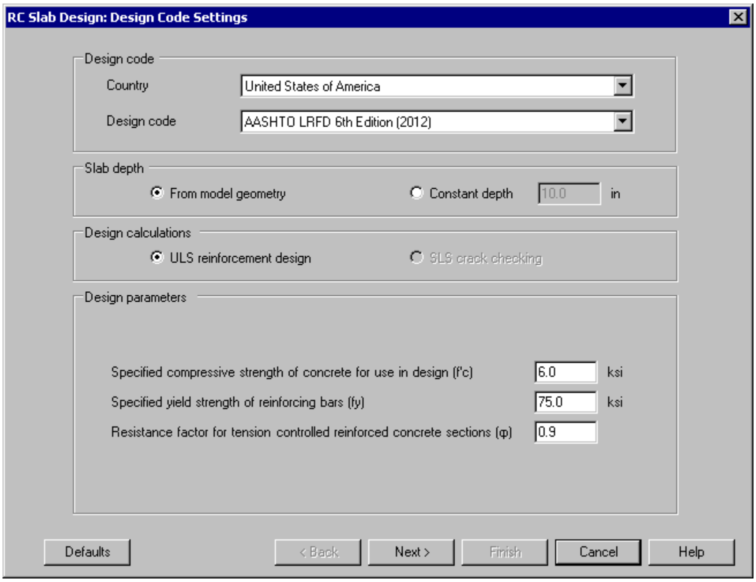
See **RC Slab Designer : Viewing Results** for details of plotting SLS reinforcement design contours.

Design Code Settings and Parameters for AASHTO LRFD

Selecting **United States of America** and one of the design code options on the main Design Code Settings page of the RC Slab Designer makes it perform calculations, and offer defaults, based on the appropriate publication below:

- ❑ **AASHTO LRFD Bridge Design Specifications, 6th Edition**, American Association of State Highway and Transportation Officials, 2012
- ❑ **AASHTO LRFD Bridge Design Specifications, 5th Edition**, American Association of State Highway and Transportation Officials, 2010

Note that regardless of model units, the Slab Designer uses units in keeping with the design code, i.e. ksi, inches etc., as shown on the dialog.



General Settings

See **RC Slab Designer : General Design Code Settings and Parameters** for details.

Design Calculations.

ULS reinforcement design is catered for but SLS crack checking is not available. This is because AASHTO LRFD requirements do not include a crack width calculation, instead relying on minimum reinforcement requirements (clause 5.7.3.3.2) and minimum spacing of reinforcing bars (clause 5.7.3.4).

Design Parameters

- ❑ **Specified compressive strength of concrete for use in design, f'c.** Typical values lie in the range 2.4 to 10.0ksi (clause 5.4.2.1 and Table C5.4.2.1-1). The default is 6.0ksi (41MPa).
- ❑ **Specified yield strength of reinforcing bars, fy.** Typical values lie in the range 60 to 75ksi (clauses 5.4.3.1 and 9.7.2.5). The default is 75ksi (520MPa).

- **Resistance factor for tension controlled reinforced concrete sections ϕ.** According to clause 5.5.4.2.1, ϕ=0.9 for tension controlled reinforced concrete sections, and this is the default value.

Reinforcement Details

See **RC Slab Designer : Reinforcement Details** for general information. AASHTO-specific advice is provided here:

- **Bar sizes** Imperial bar designations are used as shown in the table. The default size is #6.
- **Spacing** [s] Typical values lie in the range 2" to 12" (from practical considerations and clause 5.7.3.4, although clause 9.7.2.5 suggests 18" as a maximum). The default is 6" (approximately 150mm)
- **Cover** [d_c] Typical values lie in the range 0.8" to 4" (from Table 5.12.3-1). The default is 2" (approximately 50mm)

Bar Designations

Bar size	Bar Size Nominal Diameter (inches)
#3	0.375 = ⅜
#4	0.500 = ½
#5	0.625 = ⅝
#6	0.750 = ¾
#7	0.875 = ⅞
#8	1
#9	1.128
#10	1.27
#11	1.41
#14	1.693
#18	2.257
#18J	2.337

ULS Reinforcement Design to AASHTO

See **RC Slab Designer : Viewing Results** for details of plotting ULS reinforcement design contours.

See **Background to Calculations Carried out by the Slab Designer** for general information.

AASHTO-specific assumptions for nominal resistance calculations are:

- Tensile strength of concrete is neglected, as per clause 5.7.2.1
- A rectangular stress block of $0.85f'_c$ is assumed (see clause 5.7.2.2 and eqn 5.7.3.1.1-4 in the absence of prestressing steel)
- f_y replaces f_s on the assumption that $c/d_s \leq 0.6$, as per clause 5.7.2.1

A check is carried out for $c/d_s \leq 0.6$ and a warning occurs if this inequality does not hold at any location on the slab.

Area of steel required

On the above basis, the formulae used can be summarized as:

AASHTO LRFD

$$A_{s,req} = \frac{M_u}{\phi f_y z}$$

$$z = d_s - \frac{\beta_1 c}{2}$$

$$c = \frac{f_y A_{s,prov}}{0.85 f'_c \beta_1 b}$$

$$\beta_1 = \text{if}\left\{f'_c < 4, 0.85, \max\left(0.85 - \frac{f'_c - 4}{20}, 0.65\right)\right\}$$

The nomenclature used above is as in AASHTO LRFD, plus:

$A_{s,req}$ = Area of reinforcing steel required

$A_{s,prov}$ = Area of reinforcing steel provided

Tension control ratio

Inequality is met when the slab is ductile.

AASHTO LRFD

$$\frac{0.005c}{0.003\,(d_s - c)} \leq 1$$

Viewing results

See **RC Slab Designer : Viewing Results** for details of plotting ULS reinforcement design contours.

Background to the RC Slab Designer Calculations

ULS reinforcement design

Where a reinforced concrete slab is subject to a general set of bending and twisting moments (Mx, My, Mxy), these must be rationalised into components which can be used to design the 4 layers of reinforcement – top and bottom, X and Y directions. The method used here is that attributed to Wood & Armer (1969), which optimises the reinforcement requirements based on total weight of reinforcement.

Codes of Practice generally set out principles or rules in terms of the flexural resistance of beams. In accordance with industry practice, the Slab Designer uses these rules together with Wood-Armer moments to determine ULS reinforcement requirements. To prevent excessive compression in the concrete, due to the combined effect of moments in orthogonal directions, Cope & Clark (1984) recommend that slabs be "tension controlled" (sometimes termed "under-reinforced"), i.e. the resistance of the section should be controlled by yielding of the tensile reinforcement rather than crushing of the concrete. Adding compression steel is advised against, and accordingly the RC Slab Designer assumes that all reinforcement specified is tensile.

The RC Slab Designer enables users to check that the slab is tension-controlled, by use of the appropriate contour option. The plot is based on a ratio, the specifics of which are detailed according to the Code of Practice, calculated at each node. If the slab is not tension-controlled, reinforcement calculations may be unconservatively inaccurate.

SLS crack checking

Where a reinforced concrete slab is subject to a general set of bending and twisting moments, it may be reasonably assumed that tensile cracks will form perpendicular to the directions of principal moments.

Codes of Practice generally set out principles or rules in terms of cracking in beams, where, by nature, the principal moments are aligned with the direction of the reinforcement. Such methods can, however, be applied to slabs (where the principal moments may not coincide with the reinforcement directions), with the crack assumed perpendicular to the principal moment direction, using an equivalent area of reinforcement. The equivalent area of reinforcement used by the RC Slab Designer is that given by the expression attributed to Cope & Clark (1984):

$$A_n \approx \sum_{i=1}^{N} A_i cos^4 \alpha_i$$

Where:

A_n = equivalent area of steel per unit width in the n-direction

A_i = area of steel per unit width in the i-direction (of total N layers of steel)

α_i = angle between the direction of ith layer of reinforcement and the direction perpendicular to the crack

The equivalent area calculated using this expression is described by Cope & Clark as being conservative for synclastic bending but unconservative for anticlastic bending. However the alternative equation offered by the same authors gives singularities where $\alpha = \pm 45°$, and furthermore the equation above appears to give conservative answers compared to the method of Jofriet & McNeice. It therefore is adopted for all crack width calculations in the RC Slab Designer.

The crack widths calculated are therefore notionally measured in the direction of the principal moment, based on an effective area of reinforcing steel which includes contributions from the two reinforcement directions in the tension zone of the slab as appropriate. That said, the reinforcement most nearly orthogonal to the crack (or widest crack, if cracking occurs in two directions) must be identified as the "controlling" direction. The "Controlled in x" and "Controlled in y" contours allow identifying which bars to modify in order to most efficiently control cracking in the slab.

Code of Practice rules for crack widths in beams generally include a reduction in consideration of the stiffening effect of tension carried by concrete that remains bonded to the reinforcement in the tensile zone at small strains. However, where reinforcement does not lie perpendicular to cracks, as has been described to be the case in slabs, whether the "tension stiffening" reduction should be included is a matter of debate. Cope & Clarke suggest that tension stiffening should be ignored completely when principal moments act at α>25° to the reinforcement. The RC Slab Designer offers crack check calculations with the tension stiffening component included, ignored (conservative) or interpolated. The "interpolated" option scales linearly from 0% tension stiffening at α=25°, to 100% at α=0°, and this is the default option.

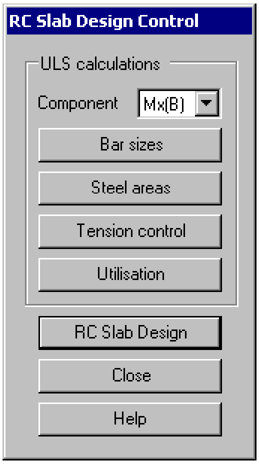
In the case of crack-width calculations, it may be necessary to nominate a long-term loadcase on the **RC Slab Designer : Crack Width Calculation Setttings** page, in which case the active loadcase is used for additional short-term loading.

Viewing Results with the RC Slab Design Control Dialog

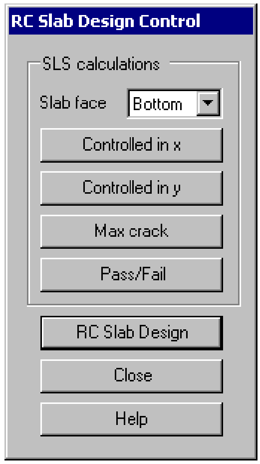
When the RC Slab Designer is in use and when the Finish button is pressed on either the Reinforcement Details page (for ULS Calculations) or on the Crack Width Calculation settings page (for SLS calculations) an RC Slab Design Control dialog appropriate to the settings made will be displayed. These dialogs provide buttons to plot chosen results contours for a previously chosen Code of Practice.

The contours or values calculated and displayed are based on the active loadcase (or combination), using the most recently set results transformation, and are re-computed if a different loadcase is set active. Appropriate contour key values are automatically set by each RC Slab Design Control dialog option used. A button is also provided to re-access the RC Slab Designer and change settings based upon results previously viewed.

The results components set by the RC Slab Design Control dialogs are explained in detail below



RC Slab Design Control dialog
for ULS design



RC Slab Design Control dialog
for SLS design

ULS Reinforcement Design Contours

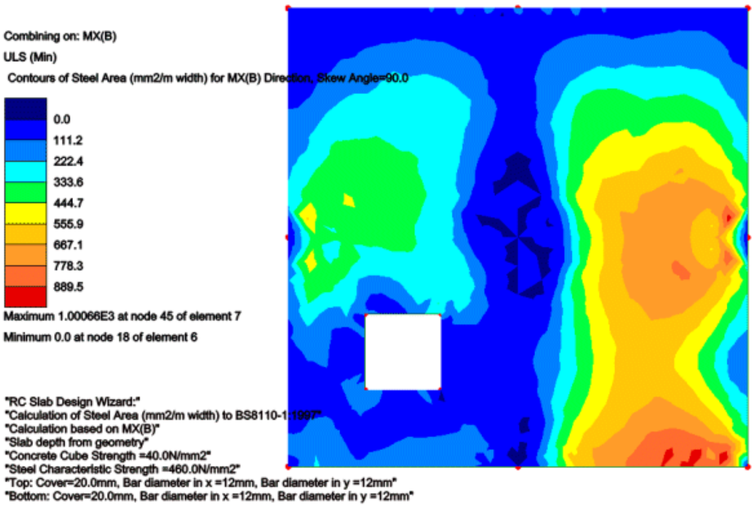
For Ultimate Limit State, and for each of the 4 layers of reinforcement, Mx(T), My(T), Mx(B) and My(B), (where 'T' signifies Top of slab and 'B' signifies Bottom) the following Slab Designer results components can be contoured:

- ❑ **Bar sizes** Assuming the spacing provided in page 2 of the wizard is to be used, this contour indicates the minimum bar size required to obtain the steel area described above.
- ❑ **Steel areas** Using the input provided through the wizard, this contour indicates the minimum area of reinforcing steel per unit width required to obtain a flexural resistance equal to the design Wood-Armer moment from the active loadcase. The minimum area for both faces and both directions must be provided for a safe design.

The effective depth and lever arm used in the calculations is, in each case, based on the bar size and other parameters specified in page 2 of the wizard.

- **Tension Control** This contour indicates whether the slab may be regarded as tension-controlled in respect of the current layer of reinforcement. This is expressed using a ratio (the specifics of which are detailed according to the Code of Practice) calculated from the reinforcement specified in page 2 of the wizard. Values <1 indicate that the slab is tension-controlled and thus the slab may be regarded as ductile. Values >1 indicate that reinforcement calculations may be unconservative; use of compression steel in slabs is not recommended and so slab depth may need to be increased.
- **Utilisation** Defined as the ratio of the design Wood-Armer moment from the active loadcase to the calculated flexural resistance e.g. Mx(B)/Mrx(B). The flexural resistance is calculated using the reinforcement specified on page 2 of the wizard and is appropriate to the face and direction of the Wood-Armer moment. Values <1 indicate that the reinforcement is adequate.
- **RC Slab Design** re-opens the Design Code Settings page of the slab designer.

Note: The RC Slab Designer can provide checks on flexural resistance based on proposed reinforcement arrangements. In-plane effects are not considered. Where overall compression or tension exists in a slab (modelled using shell elements), the reinforcement requirements indicated by use of the Slab Designer will be inaccurate and unconservative. Calculations for ULS shear resistance are not included.



Typical ULS reinforcement contour plot

SLS Reinforcement Design Contours

For Serviceability Limit State the following Slab Designer results components are available for the top face and the bottom face:

- **Controlled in X** Whilst the crack widths are determined from principal moments and do not necessarily correspond to a particular reinforcement direction, the crack is considered to be controlled by the reinforcement to which it is most nearly orthogonal, or, when the principal direction lies at an angle no more aligned to either reinforcement direction, by the outermost reinforcement layer. This contour reports the maximum crack widths controlled by the x-direction reinforcement, enabling the user to identify if the maximum cracks observed are best controlled by modifying x-direction steel.
- **Controlled in Y** This contour reports the maximum crack widths controlled by the y-direction reinforcement, enabling the user to identify if the maximum cracks observed are best controlled by modifying y-direction steel.
- **Max Crack** This contour indicates the maximum crack width calculated for the face of the slab in question. The crack width reported is the width perpendicular to the crack, which may not be orthogonal to the reinforcement provided.
- **Pass/ fail** This contour indicates how the maximum calculated crack width compares to the allowable crack width specified on page 3 of the wizard.
- **RC Slab Design** re-runs the slab design wizard

Code of Practice crack width formulae are generally invalidated when certain stress limits are exceeded. Therefore, although stresses are not output in contour form, they are calculated by the Slab Designer and if the calculated values exceed codified limits a message box will report the maximum value.

Note: The RC Slab Designer can provide calculation of design crack widths in top and bottom faces for proposed reinforcement arrangements. As for ULS reinforcement design, in-plane effects are not considered. The crack widths indicated by use of the RC Slab Designer will be inaccurate where overall compression or tension exists in a slab, and unconservative. SLS deflections and curvatures are not calculated.

General Results Processing

When the RC Slab Designer is in use, most RC Slab Designer results components can also be accessed via the contour layer's Contour Properties dialog. The results components are accessed from the RC Slab Design Entity but note that the Contour Range values are pre-set by the RC Slab Designer and may require manual adjustment.

When the RC Slab Designer is not in use, general results processing (such as plotting displacements, forces/moments, stresses etc) can be carried out, however no access to the slab designer-specific results components is provided.

Crack Width Calculation to EN 1992-1-1

Overview

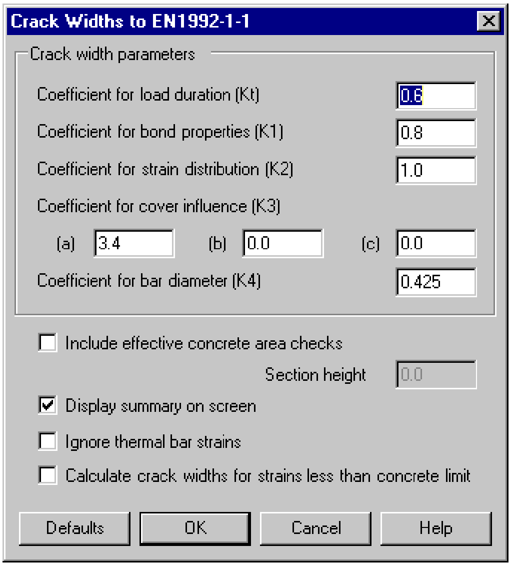
The Crack Width calculation facility is accessed using the **Bridge / Civil> Crack Widths to EN 1992-1-1** menu item. This is provided in selected Bridge and Civil & Structural software products only. It enables plotting of contours of design crack widths in accordance with the EN 1992-1-1 design code. The calculations carried out are for reinforced concrete structures that are modelled using the Smoothed Multi Crack Concrete Model (Model 102), that have steel reinforcement modelled with reinforcement attributes, using a linear steel material model.

See **Crack Width Calculation Methods Available** for other crack width calculation options.

Using the Crack Width Calculator

Prior to obtaining results from the Crack Width Calculator, a 2D or 3D model containing lines assigned with bar **reinforcement attributes** and meshed with bar elements must have been created, loaded and solved. Loadcases considered and load combinations created should include those appropriate to the calculations to be carried out.

Running the Crack Width Calculator provides access to the Crack Width parameters dialog.



Crack Width Control dialog for EN 1992-1-1

Crack width parameters : Coefficients to be stated

- **Coefficient for load duration (kt)** a factor dependent on the load duration, (EN 1991-1-1:2004 equation 7.9)
- **Coefficient for bond properties (k1)** takes account of the bond properties of the bonded reinforcement, (EN 1991-1-1:2004 equation 7.11)
- **Coefficient for strain distribution (k2)** takes account of the distribution of strain, (EN 1991-1-1:2004 equation 7.11)
- **Coefficient for cover influence (k3)**Coefficient controlling the influence of cover on crack spacing. (EN 1991-1-1:2004 equation 7.11). In clause 7.3.4(3), k_3 is an NDP. It requires special treatment for some nations and is taken to be of the form:

  $$k_3 = k_{3A}.\varnothing^{k_{3B}}.c_{cr}{}^{k_{3C}}$$

  where generally $k_{3B}\geq 0$ and $k_{3C}\leq 0$. For most nations, $k_{3A}=3.4$, $k_{3B}=0$ and $k_{3C}=0$, but defaults are set as appropriate to the National Annex. This equation accommodates the requirements of the two national annexes that account for cover and bar diameter.

- **Coefficient for bar diameter (k4)** controls the influence of bar diameter on crack spacing, (EN 1991-1-1:2004 equation 7.11)

Note that the distribution of strain is considered to be accounted for in the nonlinear concrete model and therefore it is recommended that k2=1.0

Additional options

- **Include effective concrete area checks** The effective depth of concrete is limited by EN 1991-1-1 to less than or equal to half the beam depth or to one third of the depth minus the neutral axis. Due to the nature of the analysis the location of the neutral axis is unknown and therefore this check cannot be carried out. The option to limit the effective concrete height to half the section height is provided and for this the section height must be entered manually. The default approach to neglect the check is considered conservative.
- **Display summary on screen** The option to display a summary of crack width information is provided.
- **Ignore thermal bar strains** In coupled (Structural and Thermal) analyses steel strains include both temperature and mechanical strains. In calculating the crack widths it is generally only the mechanical strains that are of interest. The option to ignore thermal strains should therefore be used in coupled analyses when crack widths are to be calculated based on the mechanical steel strains only.
- **Calculate crack widths for strains less than the concrete limit** It is assumed that cracking only occurs when the tensile strain limit of the concrete is exceeded. This assumption can be overridden and theoretical crack widths calculated for concrete strains less that the theoretical cracking strain by selecting this option

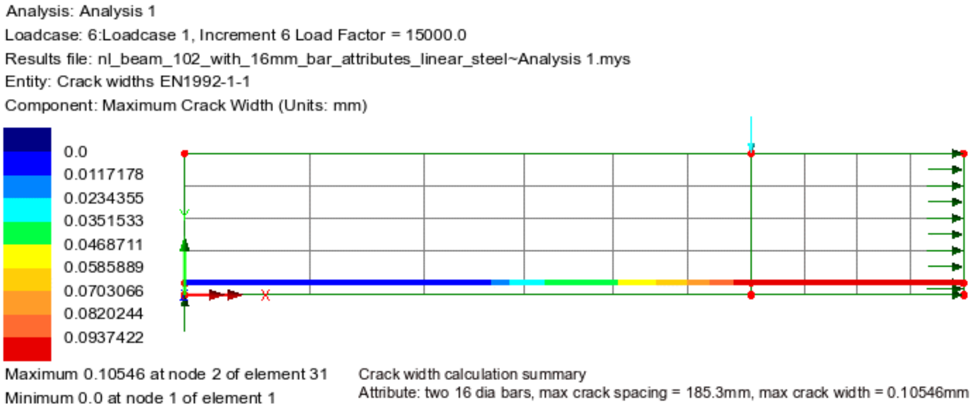
Results displayed

Crack width contours are plotted along the bar elements (actually on the surface of the fleshed bar section) corresponding to the steel strains used in the calculation, and not on any concrete face or surface in the model. They are plotted for an active loadcase (or combination) and are re-computed if a different loadcase is set active. This visualisation method is used because the approach to calculating crack widths in EN 1991-1-1:2009 is generally unclear as to where the crack width calculation applies.

Crack width information can be plotted for the Contour and Values layers as components of Maximum Crack Width under the Crack Widths EN1991-1 entity. These settings are automatically made when the Crack Width calculation facility is used.

A Crack Widths Control dialog provides an easy means of returning to and changing the crack width calculation parameters used.

A typical crack width contour plot for a 2D plane strain model is shown below.



Notes

- All calculations are based upon EN1992-1-1 clause 7.3.4.
- Crack width calculation to EN1991-1-1 can only be carried out using this utility when a nonlinear concrete material, that has a concrete tensile strength, such as Smoothed Multi Crack Concrete Model (Model 102), is used, bar reinforcement is modelled with **reinforcement attributes**, and a linear steel material model is used to represent the steel reinforcement. This is because of the steel strains used in the calculations.
- Crack width calculation relies on attributes being assigned to geometry and available during calculation so the menu item is disabled and not available when only a results file is loaded.
- If reinforcement attribute values are changed the model must be re-solved prior to running the Crack Width calculation facility.

Load Combination Wizards

Overview

In addition to Basic or Smart load combinations that can be defined for any design code, loading combinations can be defined using load combination wizards for the BD37/88, BRO or the Korean Highway codes.

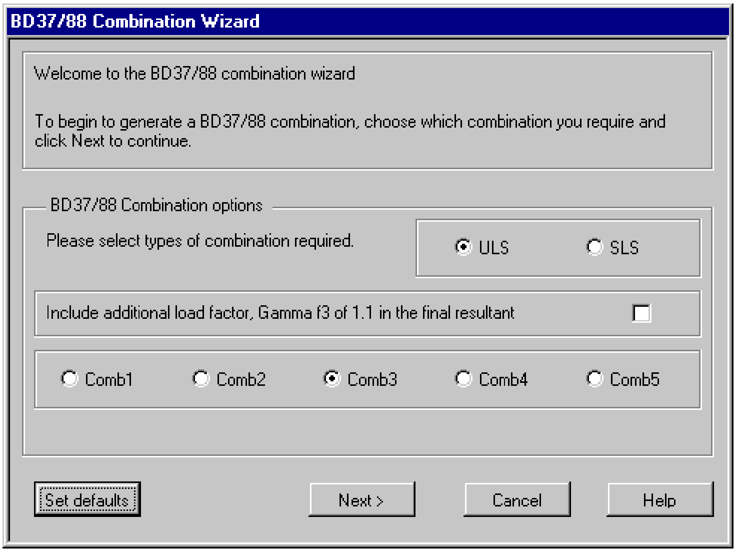
Load combination wizards should only be used in conjunction with a corresponding predefined load template. Load templates are selected from the New Model dialog and predefine the possible characteristic loadcases. Loads are assigned to these characteristic loadcases and the combination generator then combines these loadcases and creates load combinations to give the resultant maximum and minimum ULS or SLS loadcases in accordance with the chosen design code.

Notes:

- If a model exists which has not been created from the appropriate startup template all loading should be removed and the loadcases should be defined using the vbs file **<Code>Loadcases.vbs** which is located in the **\<Lusas Installation Folder>\Programs\Scripts\Treeview** directory.
- The loads should be applied to the model as un-factored loads as the load factors are placed into the combination automatically by the combination wizard in accordance with the code. The factors taken into account within the combinations are those stated within the appropriate code.
- The use of bridge load combinations is described in the BRO Slab Analysis example. See the *Application Examples Manual* for LUSAS Bridge.

BD37/88 Combinations

The BD37/88 combination wizard allows combinations to be generated in accordance with the UK Highways Agency Department Standard BD37/88. In order to use the combination wizard a model must have been created using the BD37/88 startup template, with loads assigned to the required loadcases as named in the template. The factors taken into account within the combinations are those stated within Table 1 of BD37/88.

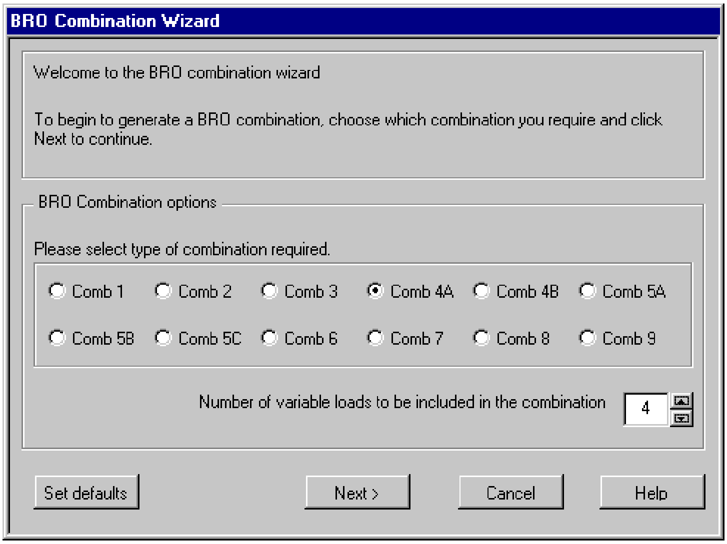


To use the combination wizard:

- Specify which limit state combination is required, **ULS** (Ultimate Limit State) or **SLS** (Serviceability Limit State)
- Specify if the additional factor Gamma f3 is to be included in the final combination. This should only be used for concrete structures designed to BS5400 part4.
- Specify the combination required.

BRO Combinations

The BRO combination wizard allows combinations to be generated in accordance with the Swedish Highways Agency Department Standard BRO. In order to use the combination wizard a model must have been created using the BRO startup template, with loads assigned to the required loadcases as named in the template. The factors taken into account within the combinations are those stated within BRO.



To use the combination wizard:

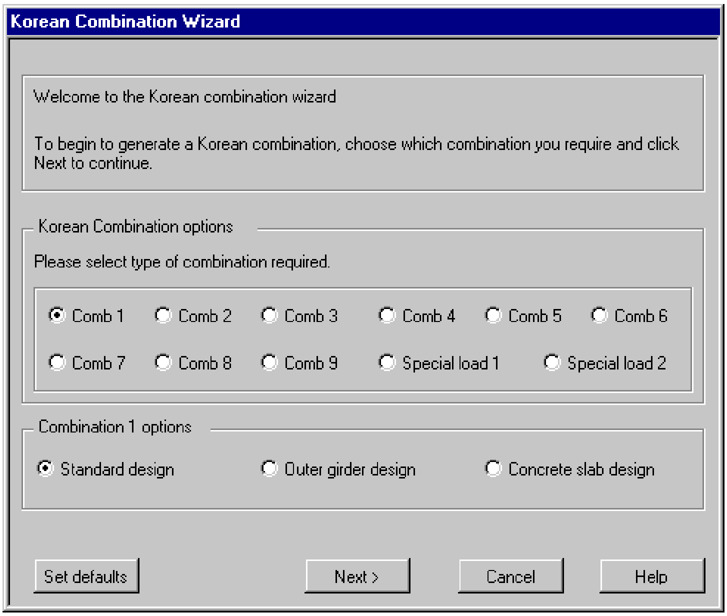
- Specify the combination required.
- Specify the number of variable loadcases to be considered in the variable loadcases combination.

Korean Highway Code Combinations

The Korean combination wizard allows combinations to be generated in accordance with the Korean Highways code. In order to use the combination wizard a model must have been created using the Korean (Bridge) startup template, with loads assigned to the required loadcases as named in the template.

The factors taken into account within the combinations are those stated within the following Table:

Combination Name	Loading Combination and factors
1	U=1.3D+2.15(L+I)+1.3CF+1.7H+1.3Q
2	U=1.3D+1.7H+1.3Q+1.3W
3	U=1.3D+1.3(L+I)+1.3CF+1.7H+1.3Q+1.3(0.5W+WL+BK)
4	U=1.3D+1.3(L+I)+1.3CF+1.7H+1.3Q+1.3G
5	U=1.25D+1.65H+1.25Q+1.25W+1.25G
6	U=1.25D+1.25(L+I)+1.25CF+1.65H+1.25Q+1.25(0.5W+WL+BK)+1.25G
7	U=1.0(D+H+Q+E)
8	U=1.3D+1.3(L+I)+1.3CF+1.7H+1.3Q+1.3CO
9	U=1.2D+.55H+1.2Q+1.2CO
Special load 1	U=1.3D+2.85(L+I)
Special load 2	U=1.3D+1.3(L+I)+1.3CF+1.7H+1.3Q



To use the combination wizard:

- Specify the combinations required.
- If Combination 1 is being considered specify which of the three options is required; Standard design, Outer girder design or Concrete slab design.

Exporting data for use with the Steel and Composite Deck Designer

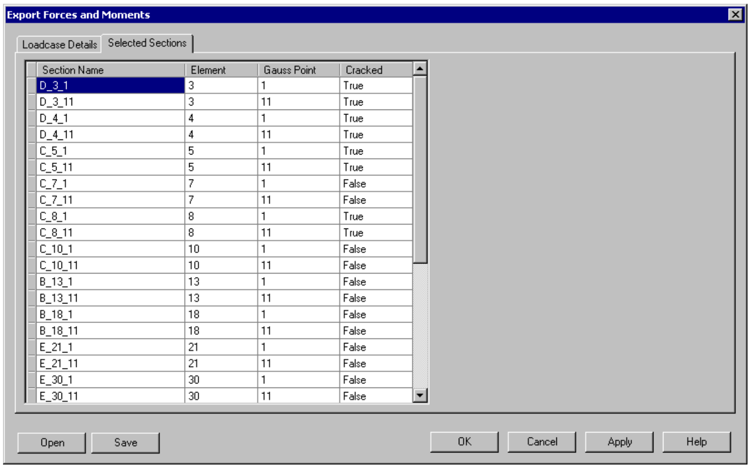
Overview

By using the **Bridge> Composite Deck Designer (PontiEC4) ...** menu item relevant LUSAS model file and results data for a selected analysis and for pre-selected elements (see above) can be exported for use in the Steel and Composite Deck Designer software option. In exporting the forces and moments LUSAS will use the Dataset name of the assigned geometric attribute to determine if the section is cracked (see below). The Analysis from which the geometric features are to be used must be specified in the Analysis selection dialog before the main export dialog is launched.

Prior to exporting data from LUSAS a model must be solved and the beam elements for which forces and moments are to be exported should be selected. The selected elements should coincide with the sections (or changes of section) at which design checks will be undertaken in the Steel and Composite Deck Designer. The forces and moments will be exported for the first and last Gauss point of each element.

Selected Sections

The sections listed on the Selected Sections page of the dialog are based upon the elements selected prior to selecting the **Bridge> Composite Deck Designer...** menu item.



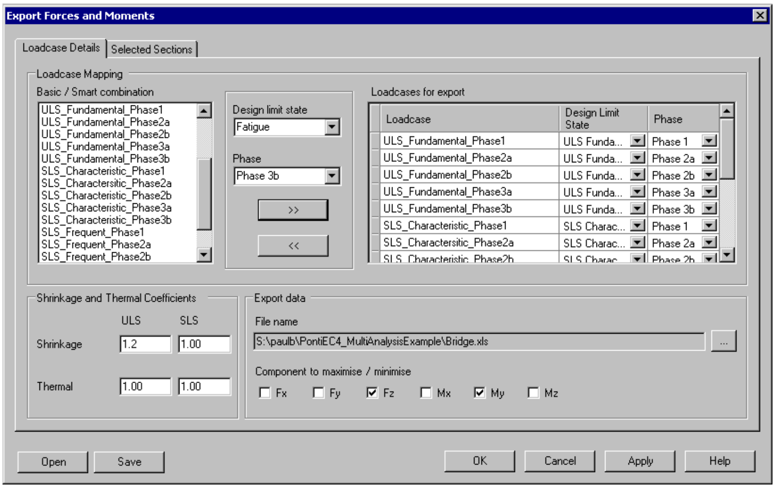
Notes

- For each selected element in the model , the table defines two sections, representing the first and last Gauss points. Each section has the following information: section name, element name (number), gauss point number, section name, and cracked/uncracked section status.
- Section names are created from: "[Segment Name]+"_"+ "[Element number]+"_"+[Gauss Point Number]
- The segment and the phase name are derived from the assigned geometric attribute, by splitting the attribute name on the double underscore separator "__". For this reason the cross-section geometric properties in the LUSAS model should have a name that follows this standard: [Segment Name] +"__"+ [Phase name]; furthermore the phase name should be: {F1, F2a, F2b, F3a, F3b, cracked}.
- The cracked status of the section is a Boolean value taken as true when the phase name is "cracked" and false otherwise.

Forces and moments will be exported for each of the sections listed and for each design combination and construction phase selected on the Loadcase details page.

Loadcase Details

On the Loadcase Details page of the dialog the loadcase combinations defined within LUSAS are associated with design limit states and phases defined in the Steel and Composite Deck Designer. For each mapped Smart Combination, maximum and minimum values will be exported for each of the selected force components. This allows the maximum design values to be computed accounting for all relevant load factors. The shrinkage/thermal coefficients will be exported for all non-cracked sections (see below).



Notes

- To map a Basic or Smart combination select it in the left hand panel, choose a Design limit state and a Phase from the drop-down lists provided, and click the >> button to add it to the Loadcase for Export grid on the right.
- **Shrinkage and Thermal Coefficients** can be defined for the Ultimate Limit State and Serviceability Limit State. The thermal and shrinkage coefficients are used by the Steel and Composite Deck designer in calculating the primary effects due to shrinkage and thermal actions in uncracked sections only (identified by the word 'False' in the cracked column of the Selected Sections Table). The hyperstatic (secondary) effects are determined from the LUSAS analysis. These same factors should therefore be specified separately in the relevant LUSAS smart combination
- **Component to maximise / minimise** are the components that will be considered in the smart combination

Saving and re-using defined data

- The **Save** button saves all Defined Loadcase Details and Selected Sections data to a input file with a .inp extension.
- The **Open** button will allow a previously saved input file to be chosen and re-populate the dialog pages with the data in that file.

Exporting data

- Assembled data can be exported into a spreadsheet for use by the Steel and Composite Deck Designer by pressing the **OK** button. The ellipsis button can be used to change folder and filename. Typical output is shown below.

	1	2	3	4	5	6	7	8	9	10	11	12
1	*Ponti EC4*					*N*		*V*	*T*	*M*		
2	**Section**	**Element**	**GaussPt**	**Component**	**Phase**	**Fx**	**Fy**	**Fz**	**Mx**	**My**	**Mz**	**γ ψ**
3	D_3_1	3	1	Fz (Max)	Phase 1	4.2940E+04	1.7073E+02	2.4724E+06	-1.0750E+02	2.4513E+07	6.1363E+02	
4	D_3_11	3	11	Fz (Max)	Phase 1	4.2940E+04	1.7073E+02	2.4878E+06	-1.0750E+02	2.6684E+07	7.6307E+02	
5	D_4_1	4	1	Fz (Max)	Phase 1	4.2938E+04	2.2986E+01	2.2245E+06	-1.0540E+02	1.7175E+07	2.7736E+02	
6	D_4_11	4	11	Fz (Max)	Phase 1	4.2938E+04	2.2986E+01	2.2496E+06	-1.0540E+02	2.0361E+07	3.1009E+02	
7	C_5_1	5	1	Fz (Max)	Phase 1	4.2938E+04	2.2986E+01	2.1242E+06	-1.0164E+02	1.4620E+07	2.4973E+02	
8	C_5_11	5	11	Fz (Max)	Phase 1	4.2938E+04	2.2986E+01	2.1392E+06	-1.0164E+02	1.7182E+07	2.7736E+02	
9	C_7_1	7	1	Fz (Max)	Phase 1	4.2933E+04	5.1090E+00	1.8640E+06	-1.1520E+02	8.4254E+06	2.1987E+02	
10	C_7_11	7	11	Fz (Max)	Phase 1	4.2933E+04	5.1090E+00	1.8745E+06	-1.1520E+02	9.9922E+06	2.2416E+02	
11	C_8_1	8	1	Fz (Max)	Phase 1	4.2933E+04	5.1090E+00	1.9395E+06	-1.1520E+02	9.9922E+06	2.2416E+02	
12	C_8_11	8	11	Fz (Max)	Phase 1	4.2933E+04	5.1090E+00	1.9539E+06	-1.1520E+02	1.2254E+07	2.3009E+02	

ULS Fundamental | SLS Characteristic | SLS Frequent | Fatigue | Model Info

The output file can be read directly into the Steel and Composite Deck Designer to carry out section design checks.

The Composite Deck Designer

The Composite Deck Designer is a software option that carries out comprehensive calculations for multiple sections on steel/composite bridge decks to the Eurocodes, allowing otherwise time-consuming and error-prone manual design calculations to be carried out efficiently. Force and moment results for selected bridge deck elements are provided by LUSAS and loadcase combinations defined within LUSAS are associated with design limit states and phases defined in the Composite Deck Designer.

Design calculations covering ULS bending, stress, shear and interaction; SLS stress, web breathing and cracking, and fatigue checks for main members and connectors are supported in the Composite Deck Designer. Multiple sections with different properties (haunches, stiffeners, etc) can be considered. Results, output in tabbed dialogs, visually show values that pass or fail. Graphs and a report containing all input data and output with references to the Eurocode clauses can be easily created.

Design checks for a number of construction phases are supported:

- Phase 1 – Self weight
- Phase 2a – Permanent loads
- Phase 2b – Concrete Shrinkage
- Phase 2c – Imposed deformations/prestressing
- Phase 3a – Thermal actions
- Phase 3 b – Traffic loads

For each construction phase design checks are made for each of the following combinations:

- ULS Fundamental – EN 1990 6.4.3.2 (3)
- SLS Characteristic – EN 1990 6.5.3 (2a)
- SLS Frequent – EN 1990 6.5.3 (2b)
- Fatigue – EN 1992-1-1 6.8.3

See **Exporting data for use with the Composite Deck Designer**.

Additional information

For more details on the Composite Deck Designer visit the LUSAS website.

Frame Results

Overview

The Frame Results processor (currently for restricted use only) allows the automatic tabulation of results for 2D and 3D frames to Microsoft Excel. Results from all loadcases, envelopes, basic combinations and smart combinations are output into a formatted spreadsheet for the parts selected.

To use

In order to use this facility the model must consist of 2D or 3D engineering thick beams (BEAM, BMS3 and BTS3 elements). In addition, only straight lines or combined lines containing only straight lines with these elements can be post-processed at present. Models can contain arcs and splines but results for these parts cannot be extracted using this facility.

Multiple straight lines can be selected for processing but should not contain any branching or closed loops. The lines will be grouped into line sets governed by their connectivity and output to the spreadsheet. For each line set a separate table and page is generated in the spreadsheet.

In addition to the results, Quality Assurance information about the LUSAS analysis is included at the top of each page along with a logo. The logo is defined by the **report_logo.bmp** bitmap image in the **\<Lusas Installation Folder>\Programs\Scripts\User** directory. This logo can be replaced to allow your own company logo to be automatically included on the spreadsheet.

The results can be output as individual line fractions (the default) or as chainage distances from the start of the first line in the selection to the end of the last line.

Limitations:

- Ensure that only post-processing beam elements is done when using this facility
- Only 2D or 3D thick engineering beam (BEAM, BMS3 and BTS3) elements are supported
- Output is only generated at the end nodes of the beam elements and may omit peak results and / or discontinuities of forces / moments that occur within elements due to internal beam loads.

- Output is reported in the Microsoft Excel Spreadsheet as values at each mesh node and these are stated at fractional distances along the parent line feature.
- Only straight lines or combined lines containing only straight lines are supported
- Only a single results file is supported
- Only Microsoft Excel is supported
- For outputting by line fractions the order of the output is moments followed by forces instead of the normal forces then moments based upon user requests.