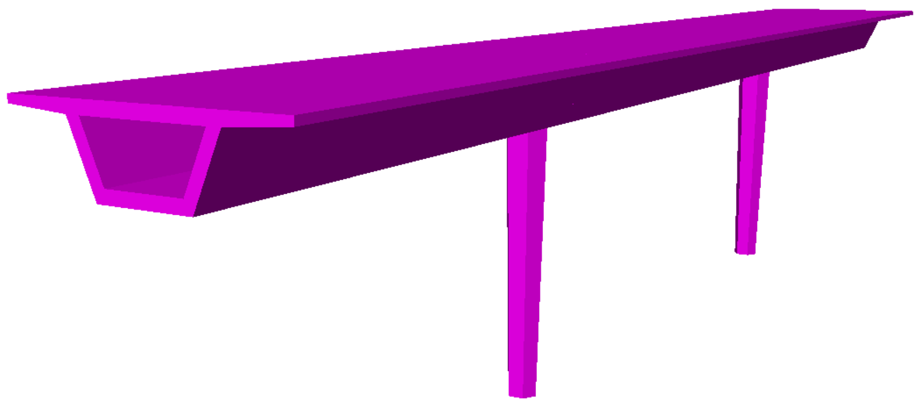
RC Frame Design to EN 1992-2

For LUSAS version:	19.0
For software product(s):	LUSAS *Civil & Structural* or LUSAS *Bridge*.
With product option(s):	RC Frame Design
Note: This example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

This example concentrates on the design of the reinforced concrete members of a pedestrian bridge in accordance with EN 1992-2:2005. Values for Nationally Determined Parameters (NDPs) are taken from the UK National Annex.



The bridge consists of a 3-span reinforced concrete deck of box section. The central span is 25m and the end spans are 14m. A pier is provided at each end of the central span. The piers are 6m high, hexagonal in shape and tapered. The concrete is grade C40/50 and the reinforcement is grade 500B throughout. Vertical support is provided at each end of the bridge. To concentrate on the definition of beam reinforcement and frame design no diaphragms in the deck are modelled.

In addition to the self-weight of the concrete deck and piers, the following loads are included:

- Superimposed dead load of 6kN/m to the deck
- Live uniform load of 20kN/m to the deck
- Service vehicle load consisting of two axles 3.0m apart with loads of 115kN and 65kN to the axles
- Horizontal load of 108kN (applied with either the live uniform load or the service vehicle load)

The live loads are based on Groups of loads gr1 and gr2 for footbridges to the UK National annex to EN 1991-2:2003.

Units used are kN, m, t, s, C throughout.

Objective

- ❑ To carry out design checks to a selected design code and confirm suitability of concrete member sizes and reinforcement details.

Keywords

3D, Frame, Concrete, Steel – Reinforcing bar, Reinforcement, Design code checks, Eurocode 2, EN 1992-2, Design Member, Design Attribute, Load Combinations, Enveloping, Member Utilisation Ratio, Reporting

Associated Files

- ❑ **rc_frame_bridge_preliminary.mdl** Model of the bridge excluding any reinforcement details. Open and use this model if you do not wish to follow all the modelling steps described in this example. See below for details.
- ❑ **rc_frame_bridge_modelling.vbs** carries out the entire modelling of the example, including defining reinforcement details. This file is for use where stated in case of user errors in preparing the model for a design check.

Modelling

Running LUSAS Modeller

For details of how to run LUSAS Modeller, see the heading ***Running LUSAS Modeller*** in the Examples Manual Introduction.

Choose your modelling method

This worked example provides you with the choice to:

- **Use a supplied model -** where all the modelling steps described in this example have been completed, with the exception of defining reinforcement details. Use this option if you only wish to see how to define reinforcement and carry out a design check, or:
- **Create a new model –** where all modelling steps as described in this example should be followed to build and save the model in preparation for defining reinforcement details.

Using a supplied model

File
Open…

Open the read-only file **rc_frame_bridge_preliminary.mdl** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Depending upon your access rights it may be necessary to copy this file to another folder before it can be opened, and remove the read-only protection before it can be saved to the new folder. Once opened:

The bridge model that is ready for defining member reinforcement will be displayed.

File
Save As…

- Save the file as \<LUSAS Installation Folder>\Projects\rc_frame_bridge
- Save the model into this new folder as **rc_frame_bridge**
- Continue from the section in this example titled "***Defining member reinforcement***"

Creating a new model

Note. This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command **File>New** to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.

- Enter the file name as **rc_frame_bridge**
- Use the **Default** working folder.
- Ensure an Analysis type of **Structural** is set.
- Set the Analysis category as **3D**
- Change the model units to **kN,m,t,s,C**
- Ensure that timescale units are **Seconds**
- Ensure the Layout grid is set as **None**
- Enter the title as **RC Pedestrian Bridge Design**

- Click **OK**

Feature Geometry

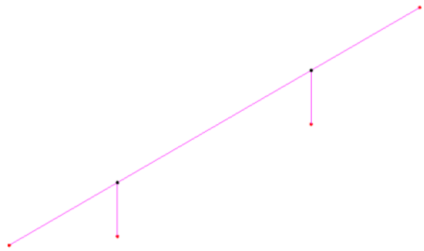
Geometry
Line >
By Coords...

- Enter coordinates of (**0,0,6**), and using the **Tab** key to move to the next entry field on the dialog enter additional points of (**14,0,6**), (**39,0,6**) and (**53,0,6**) to define the deck. With all the coordinates entered click the **OK** button.

- Select the two internal points on the deck.

Geometry
Line >
By Sweeping...

- Enter a distance of **-6** in **Z** direction and click **OK** to create the piers.



This completes the geometry definition for the bridge.

Select the isometric button to view the frame in 3D.

Meshing

For the deck members a standard thick beam line mesh is used.

Attributes
Mesh >
Line...

- Select **Thick beam**, **Linear** elements.

- Enter the attribute name as **Thick Beam** and click **OK.**

- Select the three lines defining the deck, and drag and drop the mesh attribute **Thick Beam** from the treeview onto the selected features and click **OK**.

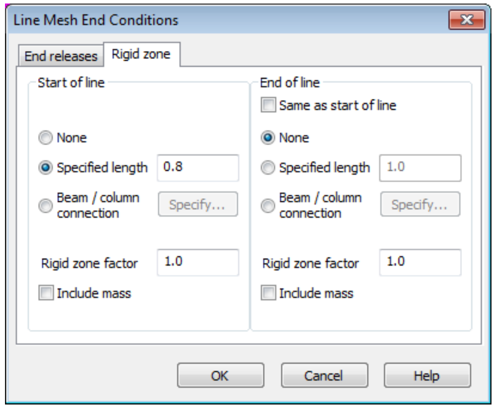
One way to model the distance between the top of the piers and the centroid of the deck is to define a rigid zone for the start of the line mesh, of a length equal to the distance (0.8m – as shown later in the example) from the centroid of the box section to its connection with the pier.

Attributes
Mesh >
Line...

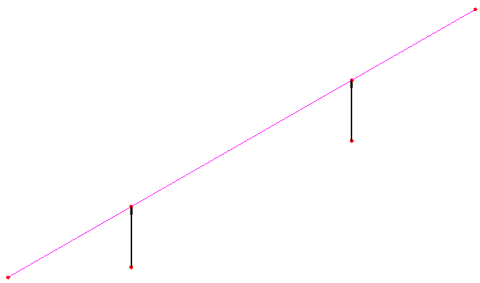
- Select **Thick beam**, **Linear** elements, press the **End Conditions** button and select the **Rigid Zone** tab,

- Specify a rigid zone length of **0.8** and for the End of line **un-tick** the **Same as start of line** checkbox. Ensure **None** is selected and click **OK**.

- On the main line mesh dialog enter the name as **Thick Beam Rigid End** and click **OK.**



- Select the two lines defining the piers, and drag and drop the mesh attribute **Thick Beam Rigid End** from the treeview onto the selected features and click **OK**.

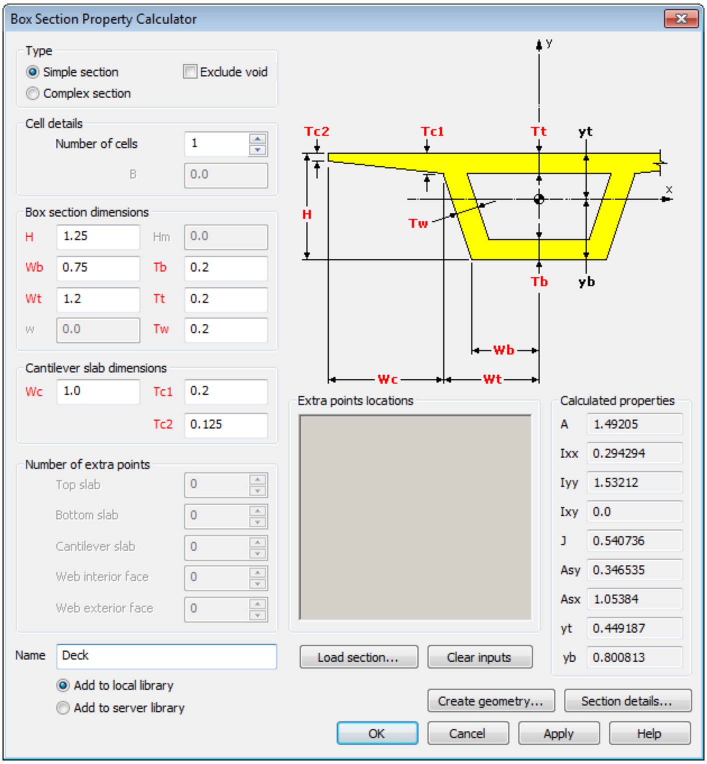


Defining cross-sectional properties

To define the deck box section

Tools
Section Property Calculator >
Box Section

- Ensure the **Simple section** type is selected and enter values as shown on the following dialog.



Note the value of yt equals 0.45. This is used to calculate the distance at which horizontal vehicle loading should be applied later in the example.

- Enter the name as **Deck**
- Click the **Section details…** button to see the section that will be created and the additional properties calculated by the values entered.
- Click **Close** to return to the main dialog.
- Ensure **Add to local library** is selected and click **OK**.

To define the hexagonal pier sections

The piers vary in section size from top to bottom, so two section sizes must be defined.

Tools
Section Property Calculator >
More sections
Hexagonal Solid…

- Enter **0.8** for distance D.
- Click in the name and the name **HSS D=0.8** will be automatically created.
- Ensure **Add to local library** is selected and click **Apply**.

- Enter **0.55** for distance D, click in the Name field to update the name and click **OK** to add **HSS D=0.55** to the local library.

Add sections to the Attributes treeview

All required section properties have now been defined and saved in the local user library. The sections must now be added to the treeview before they can be assigned to the model.

Adding deck section to the Treeview

Attributes
Geometric >
Section Library...

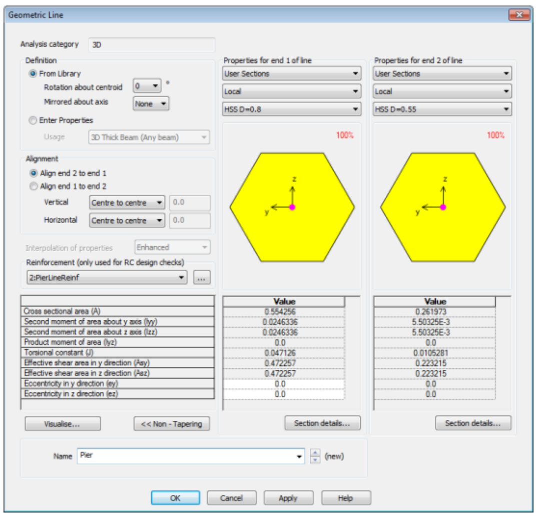
The standard sections dialog will appear.

- Click on the drop-down list button to change the selection from UK Sections to **User Sections**.
- Ensure a **Local** library is being accessed.
- Click on the drop-down list button and select **Deck** from the list of local library items.
- Enter an attribute name of **DeckNominal** and click **OK**.

Adding pier section to the Treeview

The pier section will be used to define a tapering member as follows:

Attributes
Geometric >
Tapered Section...

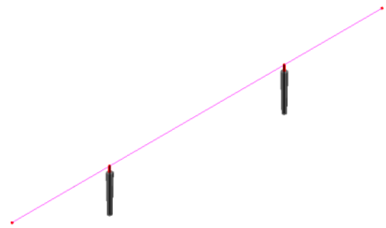


- In the ‘Properties for end 1 of line’ panel, click on the drop-down list button to change the selection from UK Sections to **User Sections**
- Ensure a **Local** library is being accessed.
- Click on the drop-down list button and select **HSS D=0.8** from the list of local library items.
- In the ‘Properties for end 2 of line’ panel, click on the drop-down list button to change the selection from UK Sections to **User Sections** for Properties for end 2 of line.
- Ensure a **Local** library is being accessed.

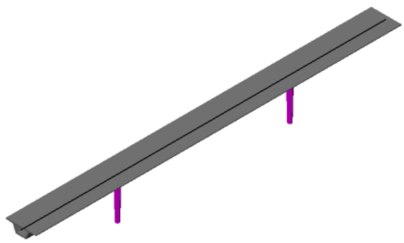
- Click on the drop-down list button and select **HSS D=0.55** from the list of local library items.
- For the Vertical alignment ensure that **Centre to centre** is selected.
- For the Horizontal alignment ensure that **Centre to centre** is selected.
- Enter an attribute name of **Pier** and click **OK**.

Assigning the geometric line properties

- Select the two lines representing the piers and drag and drop the **Pier** geometric line attribute from the treeview onto them.
- If necessary, press the button turn on the fleshing.



- Select the three lines representing the deck, then drag and drop the **DeckNominal** geometric line attribute from the treeview onto them.



Material Properties

Attributes
Material >
Material Library...

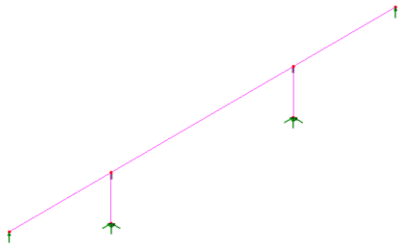
- Select material type **Concrete**, region **Europe**, standard **EN1992-2:2005** and grade **C40/50**. Leave the name set as **Iso1** and click **OK** to add the material attribute to the Treeview.
- Select the whole model and drag and drop the material attribute **Iso1 (C40/50 Concrete EN1992-2:2005)** from the Treeview onto the selected Surfaces.

Supports

Attributes
Support…

- Select **Fixed** for Translation in **Z**. Enter the name as **Fixed Translation Z** and click **Apply**.
- Change the Translation in **X** and **Y** to **Fixed** and enter the name as **Fixed Translation All** and click **OK**.

- Press the button to turn off the fleshing.
- Select the point at each end of the end spans of the bridge deck, then drag and drop the **Fixed Translation Z** support attribute from the treeview onto the selected features.



- Select the two points at the base of the piers, then drag and drop the **Fixed Translation All** support attribute from the treeview onto the selected features.
- Click in a blank part of the view window to deselect all points.

Defining the Loading

EN 1991-2:2003 requires two load types (gr1 and gr2) to be considered in addition to self weight. Load type gr1 comprises a uniformly distributed load and a horizontal force component, and load type gr2 comprises a service vehicle load and a horizontal force component. The horizontal force to be applied in both cases is the larger of either 10% of the total UDL on the footbridge or 60% of the service vehicle load, which in this example is the greater. Separate loading types are defined for each loading condition. These are then combined at the results processing stage.

Self weight

Attributes
Loading…

- In the Structural panel select **Body Force** and click **Next**.
- Enter a value for Linear acceleration in Z of **-9.81**. Change the name to **Self Weight** and click **Finish**.

Distributed load

Attributes
Loading…

- In the Structural panel select **Global Distributed** and click **Next**.
- For the surfacing load, select **Per unit length** and enter a value in **Z** Direction of **-6**. Change the name to **Superimposed Dead** and click **Apply**.
- For the pedestrian loading, change the value in **Z** Direction to **-20**. Change the name to **Live Vertical** and click **Finish**.

Service vehicle load

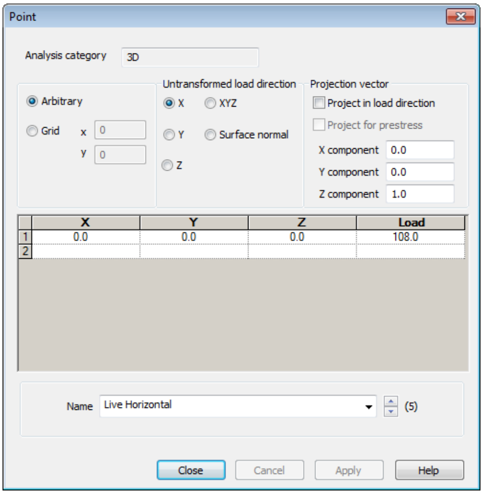
The service vehicle load model as defined in the EN 1991-2:2003 UK National Annex is adapted for use on this beam model. The specified axle loadings of 115kN and 65kN are defined for only two points, 3m apart, specified about a mid-point origin.

Attributes
Loading…

- In the Discrete panel select **Point** and click **Next**.
- In the first line of the grid enter the following values for X, Y and Z respectively: **-1.5**, **0.0** and **0.0**. Enter a value of **-115** in the load column.
- Press the tab key to add a new row to the grid and enter the following values for X, Y and Z: **1.5**, **0.0** and **0.0**. Enter a value of **-65** in the load column.
- Change the name to **Accidental Vehicle** and press **Apply**.

Horizontal component

The horizontal loading force required is defined in an untransformed load direction.

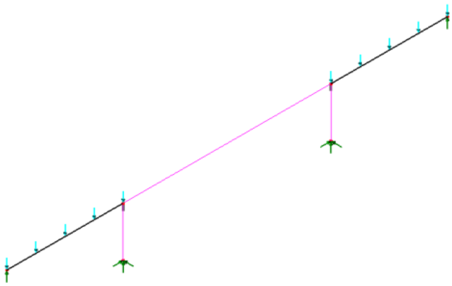


- Change the **Untransformed load direction** to **X** so that the loading will apply along the axis of the beam.
- Ensure the **Project in load direction** option is unchecked. The **X** component and **Y** component should both be set to **0**. The **Z** component should be set to **1.0**.
- Change the **X** value on the first row of the table to **0** and the load to be **108** (60% of the service vehicle load).
- Click on row 2 of the table and press the **Delete** key to remove this row.
- Change the name to **Live Horizontal** and click **Finish**.

Assigning the Loading

The loading now needs to be assigned to the spans. This is done to a limited extent to develop the most adverse effects.

Self weight



- In the treeview right click on **Loadcase 1**, select the **Rename** option and change the loadcase name to **End spans SW**.
- Select the two end spans, then drag and drop the structural loading attribute **Self Weight** from the treeview onto the selection. Click **OK**.
- Select the central span and both piers, then drag and drop the attribute **Self Weight** from the treeview onto the selection. On the Loading Assignment dialog over-type a new loadcase name to be **Central span SW** and click **OK**.

Superimposed dead load

- Select the two end spans, then drag and drop the attribute **Superimposed Dead** from the treeview onto the selection. Over-type to change the loadcase name to **End spans SDL** and click **OK**.
- Select the central span, then drag and drop the attribute **Superimposed Dead** from the treeview onto the selection. Over-type to change the loadcase name to **Central span SDL** and click **OK**.

Live load

- Select the two end spans, then drag and drop the attribute **Live Vertical** from the treeview onto the selection. Change the loadcase name to **End spans gr1** and click **OK**.
- Select the central span, then drag and drop the attribute **Live Vertical** from the treeview onto the selection. Change the loadcase name to **Central span gr1** and click **OK**.
- Select the point at the top of the left hand pier, then drag and drop the attribute **Live Horizontal** from the treeview onto the selection. Ensure that the **Project onto line** option is chosen and the loadcase **Central span gr1** is selected. Click **OK**.

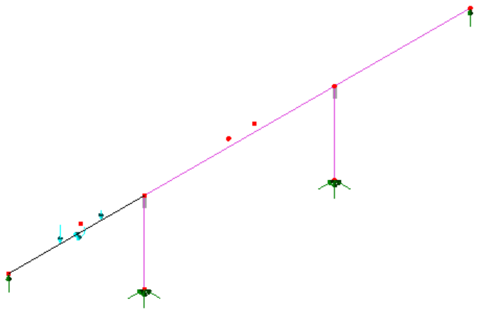
Service vehicle load

A moving load analysis would normally be carried out to find the worst–case loading position of the service vehicle for the end span, pier and central span locations. But for this example, the mid-span location of the end span, and two defined locations on the central span will be used.

Note. To comply with EN1991-2:2003 the service vehicle load needs to be applied at pavement (top of surfacing) level. This would be at a distance of 0.45 (distance from centroid to top of deck) + an allowance for (0.1) = 0.55m above the line representing the box section, which itself was defined at a Z coordinate of 6m, resulting in a value of 6.55m.

Geometry
Point >
By Coords...

- Enter coordinates of (7**,0,6.55**) and using the **Tab** key to move to the next entry field on the dialog enter additional points of (**25.55,0,6.55**) and (**27.45,0,6.55**). With all the coordinates entered click the **OK** button.



- Select the point at (**7,0,6.55**) and drag and drop the **Accidental Vehicle** load attribute from the treeview. Ensure that the **Project onto line** option is selected. Over-type a new loadcase name of **Endspan gr2** and click **OK**.

- With the same point selected, drag and drop the **Live Horizontal** load from the treeview and click **OK**.

- Select the point at (**25.55,0,6.55**) and drag and drop the **Accidental Vehicle** load attribute from the treeview. Ensure that the Project onto line option is selected. Change the loadcase to **Pier gr2** and click **OK**.

- With the same point selected, drag and drop the **Live Horizontal** load from the treeview and click **OK**.

- Select the point at (**27.45,0,6.55**) and drag and drop the **Accidental Vehicle** load attribute from the treeview. Ensure that the Project onto line option is selected. Change the loadcase to **Central span gr2** and click **OK**.

- With the same point selected Drag and drop the **Live Horizontal** load from the treeview and click **OK**.

Saving the model

File
Save As

Save the model file.

The model is now ready for adding reinforcement details.

Defining member reinforcement

Continue here if you are using the supplied bridge model.

Defining deck/pier (or beam/column) member reinforcement requires specifying **Section Reinforcement** for the member cross-section, and **Line Reinforcement**, which specifies the length(s) over which the section reinforcement arrangements act.

Section reinforcement is a utility object stored as an entry in the Utilities Treeview and cannot be assigned to lines in the model directly. Section reinforcement utilities are referenced by line reinforcement utilities. Line reinforcement utilities, in turn, are referenced by geometric line attributes (that define the concrete section size/shape and properties for a member).

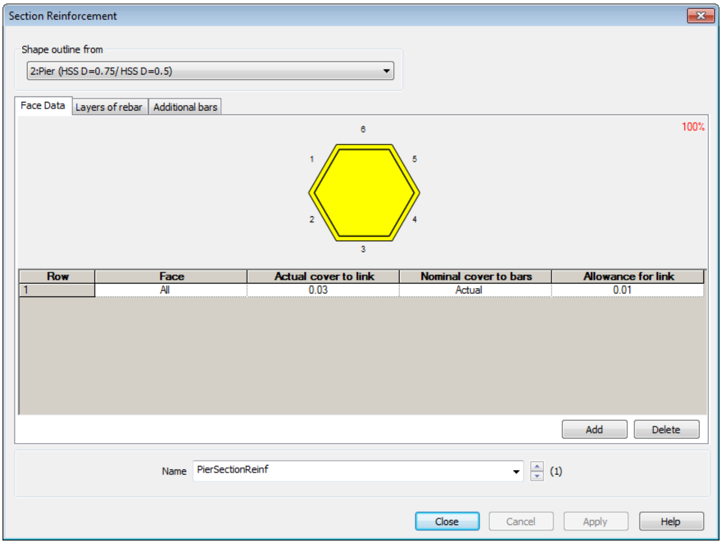
Reinforcement can be defined either by accessing the geometric line properties dialog and in the 'Reinforcement (for RC design checks)' panel clicking on the drop-down list button and selecting **New**, or by defining section and line reinforcement from the **Utilities > Reinforcement** menu item, and then referencing these utilities in the relevant dialogs. For this example, the former method will be shown.

Defining pier reinforcement

- In the treeview double-click the **Pier (HSS D=0.8/HSS D=0.55)** entry.
- In the 'Reinforcement (for RC design checks)' panel click on the drop-down list button and select **New**). The Line Reinforcement dialog will be displayed. This is used to specify how individual section reinforcement arrangements apply over the length of a line representing a concrete member. For this example the reinforcement will act over the whole length of a line.
- Click in the first cell of the grid and select **New**. A section reinforcement dialog for the newly created geometric line attribute is displayed.

For all, or individual faces of the section, the actual and nominal covers, and allowance for links, and reinforcement in the section must be stated.

- With the Face Data tab active, change the Actual cover to link to **0.03** and the Allowance for link to **0.01** (because the model units are metres)



Note. Layers of reinforcement are defined for each numbered face in a cross-section. Faces forming the external perimeter of a section are numbered anti-clockwise. References to 'Start' and 'End' with regard to bar diameters and distances stated in the grid relate to the start and end of a face. So face 2 starts at the end of face 1, and ends at the start of face 3.

- Select the **Layers of rebar** tab and change the **No. of bars** to **4** and the **Bar diameter** to **0.020**. A row of bars will appear in the section for face 1.
- From the drop-down menu for **End bars** change the entry to **Omit End** and click elsewhere in the grid to force the display to update.

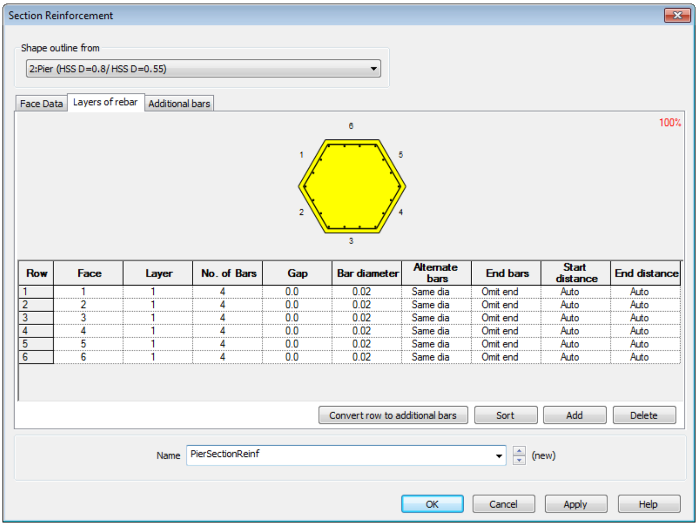
Note. Bar diameters must be entered in model units, so enter 0.016 for 16mm bars if metres are in use.

Note. ‘Omit End’ is specified because the last bar position of the run of bars in this face will be taken by the first bar in the run of bars in the next face.

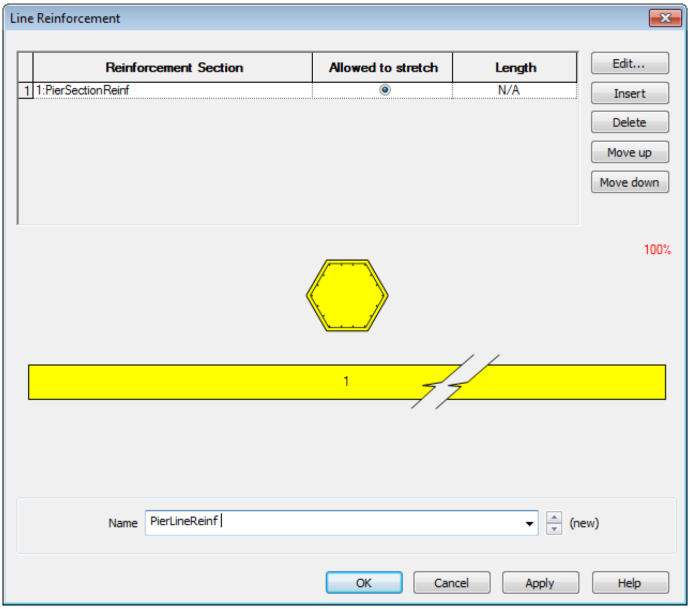
- Click the **Add** button. This duplicates the previous row. (The newly added row of bars will temporarily clash with the previously added row – as shown by being drawn in red.)
- Change number of the **Face** for the new entry to **2** and click elsewhere in the grid to force the display to update.
- Add four more rows, changing the number of the **Face** each time to **3, 4, 5** and **6** respectively.

- After entering Face 6 click anywhere else in the grid to update the bar display.

This results in the following section reinforcement arrangement.



- Change the name to **PierSectionReinf** and click **OK.**

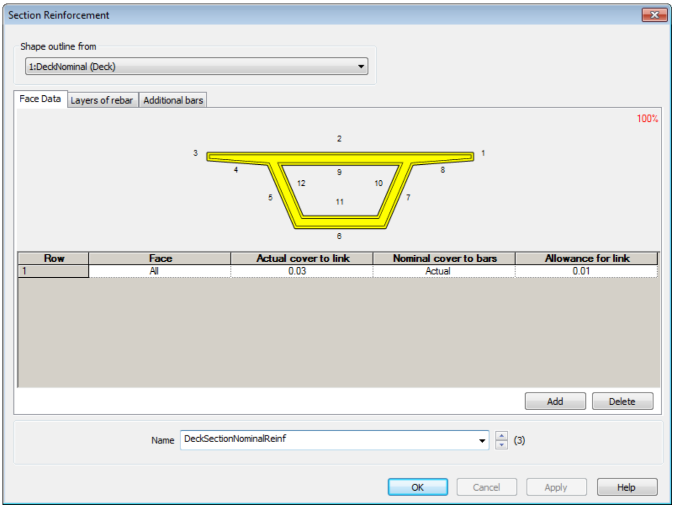


- Back on the Line reinforcement dialog, enter a name of **PierLineReinf** and click **OK**.

- Back on the Geometric Line dialog click **OK** to save the change.

Defining deck reinforcement

- In the treeview double-click the **Deck Nominal (deck)** entry.
- In the ‘Reinforcement (for RC design checks)’ panel click on the drop-down list button and select **New**). The Line Reinforcement dialog will be displayed.
- Click in the first cell of the grid and select **New**. A section reinforcement dialog for the newly created geometric line attribute is displayed.
- With the Face Data tab active, change the Actual cover to link to **0.03** and the Allowance for link to **0.01**



Note. Faces forming the external perimeter of a section are numbered anti-clockwise. Faces forming the internal void of a section are numbered clockwise.

- Select the **Layers of rebar** tab and for the initial row in the grid change the Face number from 1 to **2** (because no bars are required for Face 1). Change the **No. of bars** to **22** and the **Bar diameter** to **0.012**
- Click anywhere else in the grid to update the bar display in the section.

Note. Section reinforcement can be inspected by zooming and panning the visualised section. A changing cursor image indicates whether the facility is enabled or not. If necessary, click in the panel to activate this facility. Use the mouse wheel to zoom in and out. Click and hold-down the left mouse button, or click and hold-down the mouse wheel to pan.

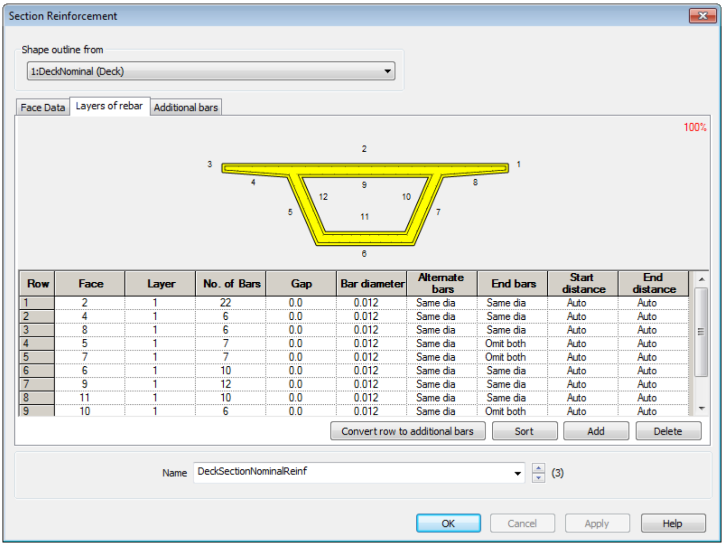
- Click the **Add** button. Change number of the **Face** for the new entry to **4** and No. of Bars to **6**.
- Click **Add**. Change number of the **Face** for the new entry to **8**.

Note. Whilst faces are numbered in order, reinforcement can be defined for faces in any order. The symmetry of a section may mean that it is easier to define similar faces one after each other, as in this case for faces 4 and 8.

- Click **Add**. Change number of the Face for the new entry to **5** and No. of Bars to **7**. Change the End bars entry to **Omit both**.
- Click **Add**. Change number of the Face for the new entry to **7**.
- Click **Add**. Change number of the Face for the new entry to **6** and No. of bars to **10**. Change the End bars entry to **Same dia**.
- Click **Add**. Change number of the Face for the new entry to **9** and No. of bars to **12**.
- Click **Add**. Change number of the Face for the new entry to **11** and No. of bars to **10**.
- Click **Add**. Change number of the Face for the new entry to **10** and No. of bars to **6**. Change the End bars entry to **Omit both**.
- Click **Add**. Change number of the Face for the new entry to **12**.

Note. End bars have been omitted in faces 5, 7, 10 and 12 since these bar positions will be taken by the ends bars defined for faces 4, 8, 9, 6 and 11.

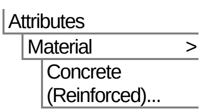
The initial entries for the section reinforcement dialog should look like this:



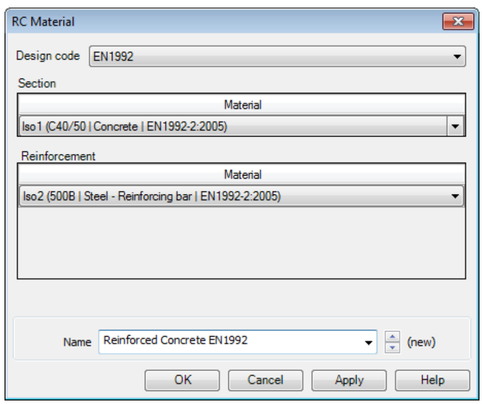
- Change the name to **DeckSectionNominalReinf** and click **OK.**
- Back on the Line reinforcement dialog change the name to **DeckLineNominalReinf** and click **OK**
- Back on the Geometric Line attribute dialog click **OK** to save the change.

Defining Concrete (Reinforced) Material

Whilst a general concrete material was previously assigned to the model, for RC Frame Design a Concrete (Reinforced) material attribute must be defined and assigned to all lines representing concrete members. The Concrete (Reinforced) material is a compound material that references both concrete and steel properties for a chosen design code in a single material attribute.



- Select **EN1992** from the droplist for Design code
- For Section select **Iso1 (C40/50 Concrete EN1992-2:2005)** from the drop-list.
- For Reinforcement select **New**. In the Material Library dialog that appears select Standard **EN1992-2:2005** and Grade **500B.** Accept the default name and click **OK.**



- Back on the RC Material dialog enter the name as **Reinforced Concrete** and click **OK**.

Assigning the material

- Select all lines in the bridge model (press **Ctrl + A** keys).
- Drag and drop the **Reinforced Concrete** material attribute from the treeview onto the selected lines.

The model is now complete and ready for design code checking.

Design code checking is carried out in LUSAS as a results processing operation following the solving of an analysis model. So, first the model must be solved.

Running the Analysis

With the model loaded:

Select the **Solve Now** button from the toolbar and click **OK** to run the analysis.

A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

If the analysis is successful...

Analysis loadcase results are added to the Treeview.

Additional files will be created in the directory where the model file resides, including:

- ❑ **rc_frame_bridge.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- ❑ **rc_frame_bridge.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results processing to take place.

If the analysis fails...

If the analysis fails, information relating to the nature of the error encountered can be written to an output file in addition to the text output window. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

File
Script >
Run Script...

To recreate the model, select the file **rc_frame_bridge_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Rerun the analysis to generate the results.

Continue here if you have run the supplied script

Viewing the Results

Analysis loadcase results are present in the Treeview. Appropriate load combinations and envelopes must now be defined in preparation for a design check.

General note relating to design checks

Note. Effects of deformed geometry, and whether a first order analysis or second order analysis should be used, are not currently considered in the design checks carried out. Second order effects need to be accounted for manually.

Defining combinations and envelopes

First, a smart combination is defined to combine the effects of load group gr1 on the different spans.

Analyses
Smart Combination...

- Add the loadcases **End spans gr1** and **Central span gr1** to the combination. For both loadcases change the Beneficial factor to 0 and change the Adverse factor to 1.35.
- Change the name to **gr1 combination** and click **OK**.

Next, an envelope is defined to determine the most onerous effects of the different gr2 loadcases.

Analyses
Envelope...

- Add the loadcases **End span gr2**, **Pier gr2** and **Central span gr2** to the envelope.
- Change the name to **gr2 envelope** and click **OK**.

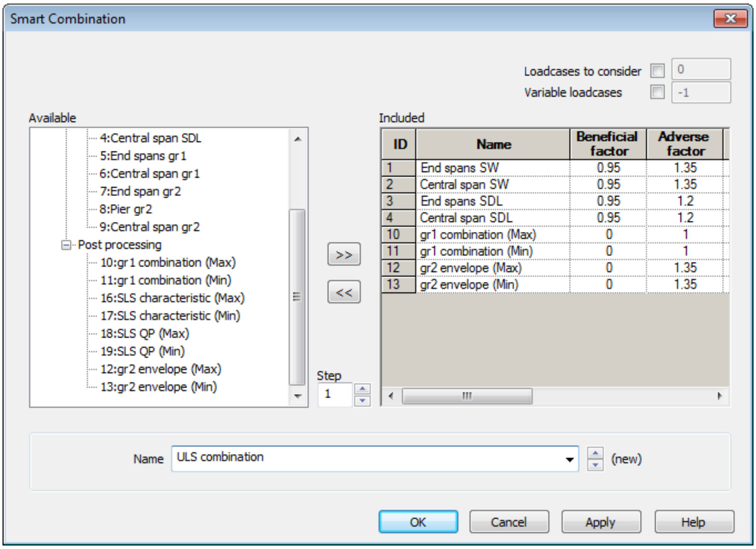
Finally, smart combinations are defined to create the required ultimate and serviceability limit state combinations as set out in EN1990.

ULS combination

Analyses
Smart Combination...

- Add the loadcases **End spans SW**, **Central span SW**, **End spans SDL** and **Central span SDL** to the combination.
- Add the combinations **gr1 combination (Max)** and **gr1 combination (Min)**.
- Add the envelopes **gr2 envelope (Max)** and **gr2 envelope (Min)**.

- For each included loadcase, edit the **Beneficial** and **Adverse** factors to be as shown in the dialog that follows:



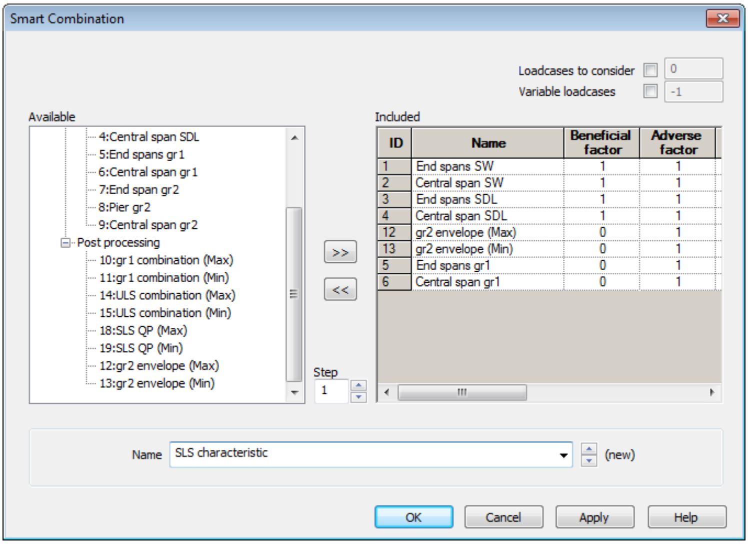
- Change the name to **ULS combination** and click **Apply**.

SLS Characteristic

- Remove **gr1 combination (Max)** and **gr1 combination (Min)** from the list of included loadcases.

- Add the loadcases **Endspans gr1** and **Central span gr1**.

- For each included loadcase, edit the **Beneficial** and **Adverse** factors to be as shown in the dialog that follows:

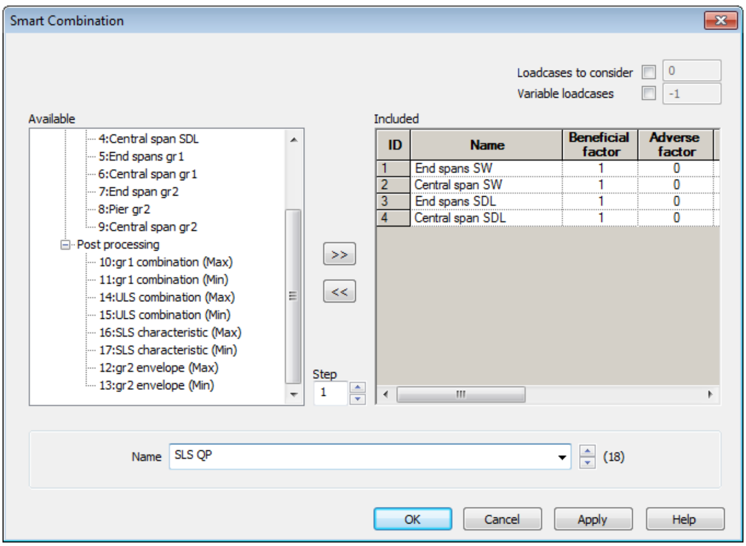


- Change the name to **SLS characteristic** and click **Apply**.

SLS Quasi-permanent

- With the SLS characteristic dialog active, remove the **gr2 envelope (Max)**, **gr2 envelope (Min)**, **Endspans gr1** and **Central span gr1** loadcases from the table.

The dialog should look like this:



- Change the name to **SLS QP** (signifying Quasi-Permanent) and click **OK**.

Meshing considerations

Design checks can be carried out at:

- **Nodes** - as used by this example, where the number of nodal results varies within each element according to the mesh discretisation used.
- **Internal points** - nine equally-spaced internal points along each element with two end points coinciding with the nodal locations of the element.
- **Inspection locations** - a user-defined position of interest on a model for results extraction / viewing purposes.

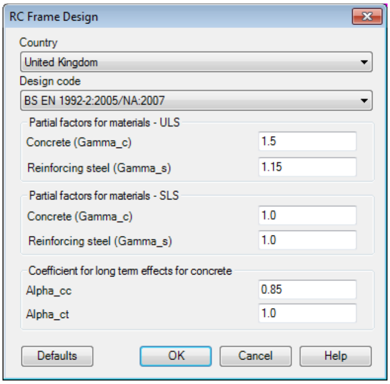
Modelling using more elements (and therefore more nodes and internal points) assigned to lines allows a more detailed assessment but requires a longer calculation time. Using the default of four line mesh divisions is suggested for initial checking.

In this example, RC Frame design calculations are carried out at the nodes. When suitable reinforcement requirements are determined, a final assessment could be carried out using internal points to confirm the steel reinforcement specified.

Note. Unlike nodal results, which only produce ‘spot’ contour results at each node, when plotting results for internal points the values obtained at each internal point can be interpolated to create line contour plots.

Checking against a Design Code

- From the country drop-down menu choose **United Kingdom**.
- From the Design code drop-down menu choose **BS EN 1992-2:2005/NA:2007**. The NDPs used in the design checks are shown on the dialog.
- Click the **OK** button to finish.



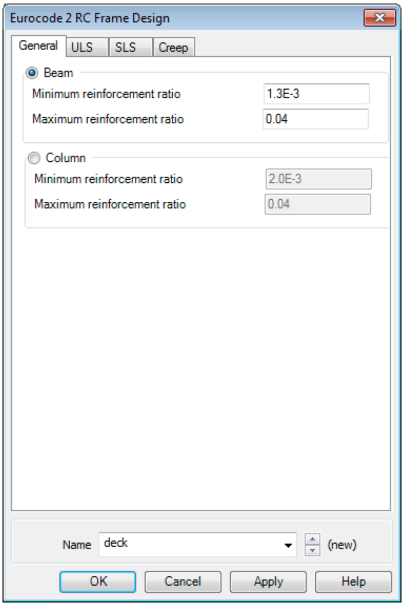
Defining and Assigning RC Frame Design Attributes

Reinforced Concrete frame design attributes are used to specify additional information about model line features that represent concrete members in order to enable RC frame design checks to be carried out.

Note. Since design attributes have no impact on a structural analysis they can be assigned after the analysis has been run. Changes can also be made to a design attribute (e.g. changing the minimum reinforcement ratio) without re-running the analysis.

Attributes
Design
RC Frame Design

The RC frame design attribute dialog is displayed:

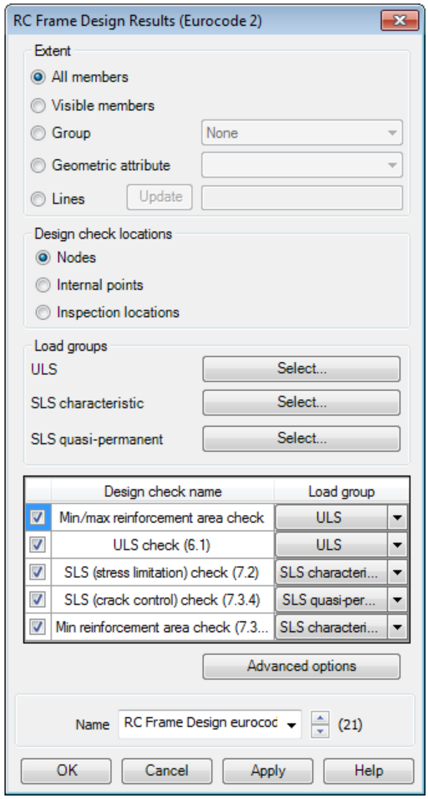


- For **Beam**, accept the default values for minimum and maximum reinforcement ratio of **1.3e-3** and **0.04** respectively, and change the name to **Deck**. Click **Apply**.
- Select **Column**, accept the default values for minimum and maximum reinforcement ratio of **2.0e-3** and **0.04** respectively, and change the name to **Pier** and click **OK**.
- Select the three lines representing the deck, then drag and drop the attribute **Deck** from the treeview onto the selection.
- Select the two lines representing the piers, then drag and drop the attribute **Pier** from the treeview onto the selection.

Creating RC Frame Design Results

Design
RC Frame Design Results...

On this dialog, selections can be made to specify which parts of a model should be checked for which load groups. From the resulting entry in the treeview the design check results can be viewed and tables of results can be obtained.



- In the Extent panel, select **All members**.
- In the Design check locations panel ensure that **Nodes** is selected.
- In the Load Groups panel press the **Select** button for **ULS**.
- On the Select Loadcases dialog presented, select **ULS Combination (Max)** and **ULS Combination (Min)** and add to the Included list. Click **OK**.
- In the Load Groups panel press the **Select** button for **SLS – Characteristic**.
- On the Select Loadcases dialog presented select **SLS Characteristic (Max)** and **SLS Characteristic (Min)** and add to the Included list. Click **OK**.
- In the Load Groups panel press the **Select** button for **SLS – Quasi-permanent**.
- On the Select Loadcases dialog presented select add **SLS QP (Max)** and **SLS QP (Min)** and add to the Included list. Click **OK**.
- On the RC Frame Design Results dialog accept the default name of **RC Frame Design 1** and click **OK** to finish.

An RC Design loadcase entry named **RC Frame Design 1** will be added inside a Post processing folder in the treeview.

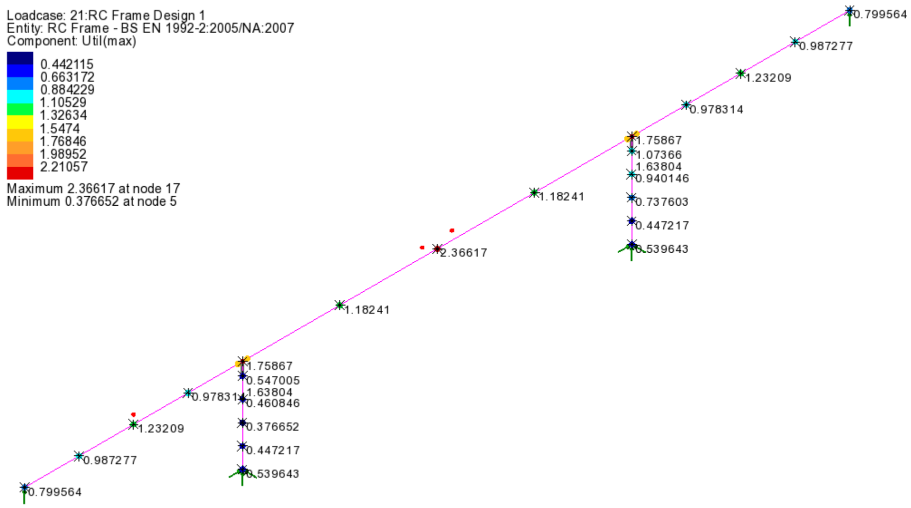
Plotting Results Contours and Values of Utilisation Ratio

- In the treeview right-click on the loadcase **RC Frame Design 1** and select **Set Active**.
- With no features selected, click the right-hand mouse button in a blank part of the active window and select the **Contours** option to add the contours layer to the Treeview.

- On the Contour properties dialog, select entity **RC Frame – BS EN1992** and component **Util(max)** from the drop down lists and click **OK** to see utilisation results for all members.

- With no features selected, click the right-hand mouse button in a blank part of the active window and select the **Values** option to add the values layer to the Treeview.

- On the Values properties dialog, select entity **RC Frame – EN1992** and component **Util(max)** from the drop down lists and click **OK** to see utilisation results for all members. Select the **Values Display** tab and set the **Maxima** value to **100%** to see all values, and click **OK** to finish.



From the nodal results it can be seen that the utilisation values exceed 1.0 in both the end spans, in the central span, and also at the tops of the both piers, so results for these locations will require further investigation.

Note. Because the design check locations were set to be **Nodes**, spot contours results are shown at the location of the element nodes only. For clarity, contours for coincident nodes (as at the intersections of the deck and piers) are shown drawn slightly offset from their true positions.

Viewing RC Frame Design Results

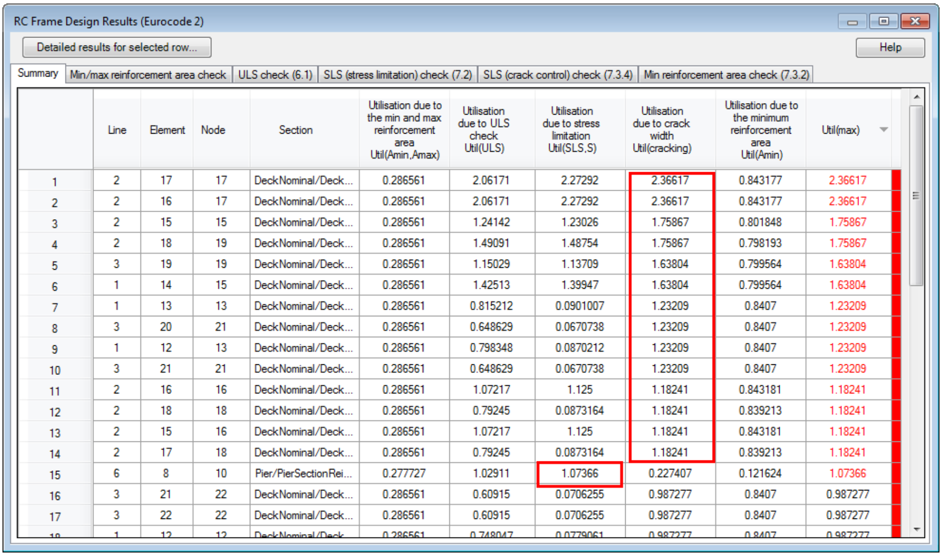
Detailed RC frame design results can also be obtained in a tabular format consisting of a summary page and separate tabbed pages for each of the design checks requested in the RC Frame Design Check dialog.

- In the treeview right click on loadcase **RC Frame Design 1** and select **Show Results**.

Summary results

- With the **Summary** tab selected, click the **Util(Max)** label at the top of the last column to sort by Utilisation value.

Note. The RC Frame Design Results dialog can be expanded to show the full width of column data.



RC Frame Design Results (Eurocode 2)

Detailed results for selected row... | Help

Summary | Min/max reinforcement area check | ULS check (6.1) | SLS (stress limitation) check (7.2) | SLS (crack control) check (7.3.4) | Min reinforcement area check (7.3.2)

	Line	Element	Node	Section	Utilisation due to the min and max reinforcement area Util(Amin,Amax)	Utilisation due to ULS check Util(ULS)	Utilisation due to stress limitation Util(SLS,S)	Utilisation due to crack width Util(cracking)	Utilisation due to the minimum reinforcement area Util(Amin)	Util(max)
1	2	17	17	DeckNominal/Deck...	0.286561	2.06171	2.27292	2.36617	0.843177	2.36617
2	2	16	17	DeckNominal/Deck...	0.286561	2.06171	2.27292	2.36617	0.843177	2.36617
3	2	15	15	DeckNominal/Deck...	0.286561	1.24142	1.23026	1.75867	0.801848	1.75867
4	2	18	19	DeckNominal/Deck...	0.286561	1.49091	1.48754	1.75867	0.798193	1.75867
5	3	19	19	DeckNominal/Deck...	0.286561	1.15029	1.13709	1.63804	0.799564	1.63804
6	1	14	15	DeckNominal/Deck...	0.286561	1.42513	1.39947	1.63804	0.799564	1.63804
7	1	13	13	DeckNominal/Deck...	0.286561	0.815212	0.0901007	1.23209	0.8407	1.23209
8	3	20	21	DeckNominal/Deck...	0.286561	0.648629	0.0670738	1.23209	0.8407	1.23209
9	1	12	13	DeckNominal/Deck...	0.286561	0.798348	0.0870212	1.23209	0.8407	1.23209
10	3	21	21	DeckNominal/Deck...	0.286561	0.648629	0.0670738	1.23209	0.8407	1.23209
11	2	16	16	DeckNominal/Deck...	0.286561	1.07217	1.125	1.18241	0.843181	1.18241
12	2	18	18	DeckNominal/Deck...	0.286561	0.79245	0.0873164	1.18241	0.839213	1.18241
13	2	15	16	DeckNominal/Deck...	0.286561	1.07217	1.125	1.18241	0.843181	1.18241
14	2	17	18	DeckNominal/Deck...	0.286561	0.79245	0.0873164	1.18241	0.839213	1.18241
15	6	8	10	Pier/PierSectionRei...	0.277727	1.02911	1.07366	0.227407	0.121624	1.07366
16	3	21	22	DeckNominal/Deck...	0.286561	0.60915	0.0706255	0.987277	0.8407	0.987277
17	3	22	22	DeckNominal/Deck...	0.286561	0.60915	0.0706255	0.987277	0.8407	0.987277
18	1	12	12	DeckNominal/Deck...	0.286561	0.748047	0.0779061	0.987277	0.8407	0.987277

From the Summary table it can be seen that:

- For the deck sections the maximum values are for the ‘Utilisation due to crack width Util(cracking)’ design check.

- For the pier section the maximum value is for the ‘Utilisation due to stress limitation Util(SLS,S)’ design check.

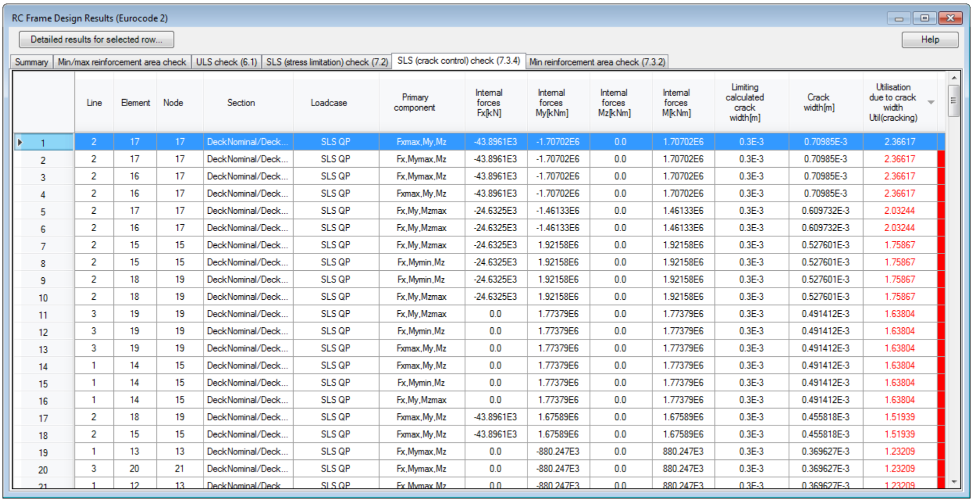
These results are to be investigated further by selecting the appropriate tabs.

Note. The Section names in the table are formed from the names of the Geometric line attribute and the referenced line reinforcement attribute. When tabulating details for tapered members (as in the pier sections) the Section name will also include a parametric distance value to indicate where along the length of line the section applies.

Viewing detailed results for a deck location

SLS (Crack Control) Check results

- On the main RC Frame Design Results summary table select the **SLS (Crack Control) Check (7.3.4)** tab.
- Expand the width of table and sort the **Utilisation due to crack width Util(cracking)** column in descending order.
- The table shows the Load combination, Primary component and related force and moment data causing the crack width stated.


RC Frame Design Results (Eurocode 2)

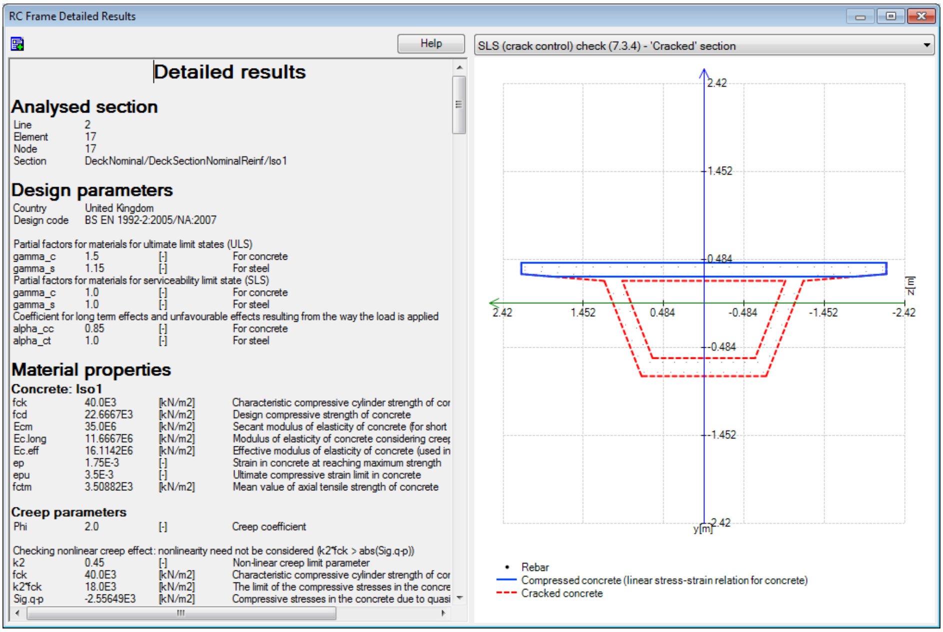
Detailed results for selected row... | Help

Summary | Min/max reinforcement area check | ULS check (6.1) | SLS (stress limitation) check (7.2) | SLS (crack control) check (7.3.4) | Min reinforcement area check (7.3.2)

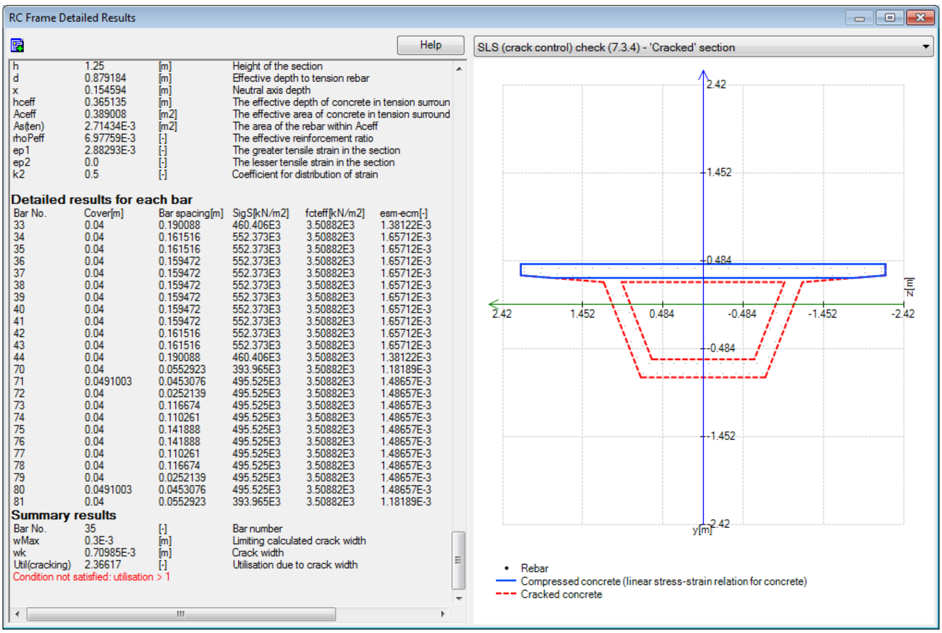
	Line	Element	Node	Section	Loadcase	Primary component	Internal forces Fx[kN]	Internal forces My[kNm]	Internal forces Mz[kNm]	Internal forces M[kNm]	Limiting calculated crack width[m]	Crack width[m]	Utilisation due to crack width Util(cracking)
1	2	17	17	DeckNominal/Deck...	SLS QP	Fxmax,My,Mz	-43.8961E3	-1.70702E6	0.0	1.70702E6	0.3E-3	0.70985E-3	2.36617
2	2	17	17	DeckNominal/Deck...	SLS QP	Fx,Mymax,Mz	-43.8961E3	-1.70702E6	0.0	1.70702E6	0.3E-3	0.70985E-3	2.36617
3	2	16	17	DeckNominal/Deck...	SLS QP	Fx,Mymax,Mz	-43.8961E3	-1.70702E6	0.0	1.70702E6	0.3E-3	0.70985E-3	2.36617
4	2	16	17	DeckNominal/Deck...	SLS QP	Fxmax,My,Mz	-43.8961E3	-1.70702E6	0.0	1.70702E6	0.3E-3	0.70985E-3	2.36617
5	2	17	17	DeckNominal/Deck...	SLS QP	Fx,My,Mzmax	-24.6325E3	-1.46133E6	0.0	1.46133E6	0.3E-3	0.609732E-3	2.03244
6	2	16	17	DeckNominal/Deck...	SLS QP	Fx,My,Mzmax	-24.6325E3	-1.46133E6	0.0	1.46133E6	0.3E-3	0.609732E-3	2.03244
7	2	15	15	DeckNominal/Deck...	SLS QP	Fx,My,Mzmax	-24.6325E3	1.92158E6	0.0	1.92158E6	0.3E-3	0.527601E-3	1.75867
8	2	15	15	DeckNominal/Deck...	SLS QP	Fx,Mymin,Mz	-24.6325E3	1.92158E6	0.0	1.92158E6	0.3E-3	0.527601E-3	1.75867
9	2	18	19	DeckNominal/Deck...	SLS QP	Fx,Mymin,Mz	-24.6325E3	1.92158E6	0.0	1.92158E6	0.3E-3	0.527601E-3	1.75867
10	2	18	19	DeckNominal/Deck...	SLS QP	Fx,My,Mzmax	-24.6325E3	1.92158E6	0.0	1.92158E6	0.3E-3	0.527601E-3	1.75867
11	3	19	19	DeckNominal/Deck...	SLS QP	Fx,My,Mzmax	0.0	1.77379E6	0.0	1.77379E6	0.3E-3	0.491412E-3	1.63804
12	3	19	19	DeckNominal/Deck...	SLS QP	Fx,Mymin,Mz	0.0	1.77379E6	0.0	1.77379E6	0.3E-3	0.491412E-3	1.63804
13	3	19	19	DeckNominal/Deck...	SLS QP	Fxmax,My,Mz	0.0	1.77379E6	0.0	1.77379E6	0.3E-3	0.491412E-3	1.63804
14	1	14	15	DeckNominal/Deck...	SLS QP	Fxmax,My,Mz	0.0	1.77379E6	0.0	1.77379E6	0.3E-3	0.491412E-3	1.63804
15	1	14	15	DeckNominal/Deck...	SLS QP	Fx,Mymin,Mz	0.0	1.77379E6	0.0	1.77379E6	0.3E-3	0.491412E-3	1.63804
16	1	14	15	DeckNominal/Deck...	SLS QP	Fx,My,Mzmax	0.0	1.77379E6	0.0	1.77379E6	0.3E-3	0.491412E-3	1.63804
17	2	18	19	DeckNominal/Deck...	SLS QP	Fxmax,My,Mz	-43.8961E3	1.67589E6	0.0	1.67589E6	0.3E-3	0.455818E-3	1.51939
18	2	15	15	DeckNominal/Deck...	SLS QP	Fxmax,My,Mz	-43.8961E3	1.67589E6	0.0	1.67589E6	0.3E-3	0.455818E-3	1.51939
19	1	13	13	DeckNominal/Deck...	SLS QP	Fx,Mymax,Mz	0.0	-880.247E3	0.0	880.247E3	0.3E-3	0.369627E-3	1.23209
20	3	20	21	DeckNominal/Deck...	SLS QP	Fx,Mymax,Mz	0.0	-880.247E3	0.0	880.247E3	0.3E-3	0.369627E-3	1.23209
21	1	12	13	DeckNominal/Deck...	SLS QP	Fx,Mymax,Mz	0.0	-880.247E3	0.0	880.247E3	0.3E-3	0.369627E-3	1.23209

Viewing detailed results for a deck location

- Select **row 1** in the SLS (crack control) check table and then click the **Detailed results for selected row** button at the top of the dialog. Double clicking the row will also give the detailed results.



- Scroll to the bottom of the detailed results for the selected row to view the data relating to the Utilisation being exceeded.

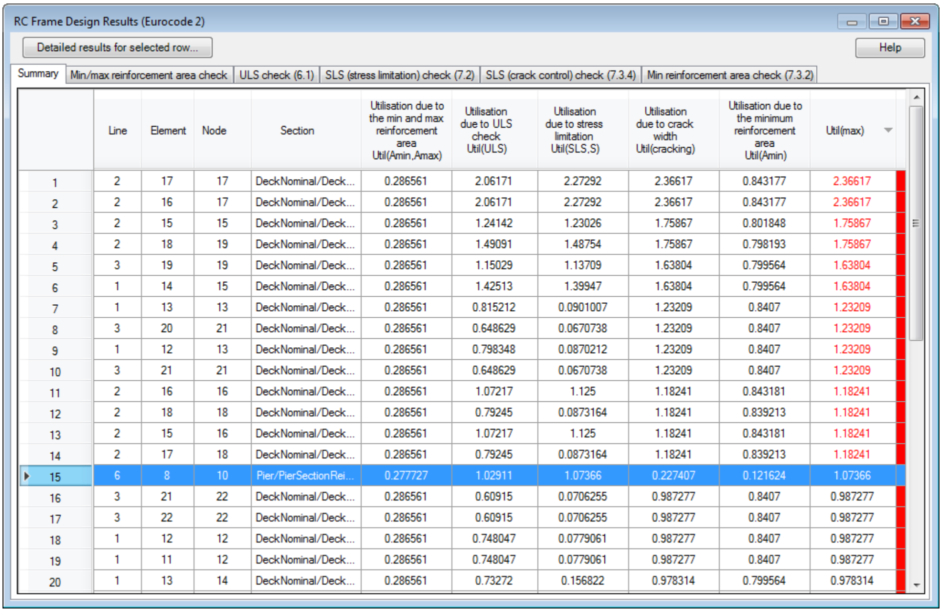


Under the ‘Detailed results for each bar’ heading the bars for which the crack width check has been applied are listed. The location of these bars (bars 33-44 and 70-81) can be found under the ‘Section Definition’ ‘Reinforcement’ heading further up the listing, which shows them to be in faces 6 and 11.

- **Close** the Detailed Results dialog.

Viewing detailed results for a pier location

- On the RC Frame Results dialog select the **Summary** tab



	Line	Element	Node	Section	Utilisation due to the min and max reinforcement area Util(Amin,Amax)	Utilisation due to ULS check Util(ULS)	Utilisation due to stress limitation Util(SLS,S)	Utilisation due to crack width Util(cracking)	Utilisation due to the minimum reinforcement area Util(Amin)	Util(max)
1	2	17	17	DeckNominal/Deck...	0.286561	2.06171	2.27292	2.36617	0.843177	2.36617
2	2	16	17	DeckNominal/Deck...	0.286561	2.06171	2.27292	2.36617	0.843177	2.36617
3	2	15	15	DeckNominal/Deck...	0.286561	1.24142	1.23026	1.75867	0.801848	1.75867
4	2	18	19	DeckNominal/Deck...	0.286561	1.49091	1.48754	1.75867	0.798193	1.75867
5	3	19	19	DeckNominal/Deck...	0.286561	1.15029	1.13709	1.63804	0.799564	1.63804
6	1	14	15	DeckNominal/Deck...	0.286561	1.42513	1.39947	1.63804	0.799564	1.63804
7	1	13	13	DeckNominal/Deck...	0.286561	0.815212	0.0901007	1.23209	0.8407	1.23209
8	3	20	21	DeckNominal/Deck...	0.286561	0.648629	0.0670738	1.23209	0.8407	1.23209
9	1	12	13	DeckNominal/Deck...	0.286561	0.798348	0.0870212	1.23209	0.8407	1.23209
10	3	21	21	DeckNominal/Deck...	0.286561	0.648629	0.0670738	1.23209	0.8407	1.23209
11	2	16	16	DeckNominal/Deck...	0.286561	1.07217	1.125	1.18241	0.843181	1.18241
12	2	18	18	DeckNominal/Deck...	0.286561	0.79245	0.0873164	1.18241	0.839213	1.18241
13	2	15	16	DeckNominal/Deck...	0.286561	1.07217	1.125	1.18241	0.843181	1.18241
14	2	17	18	DeckNominal/Deck...	0.286561	0.79245	0.0873164	1.18241	0.839213	1.18241
15	6	8	10	Pier/PierSectionRei...	0.277727	1.02911	1.07366	0.227407	0.121624	1.07366
16	3	21	22	DeckNominal/Deck...	0.286561	0.60915	0.0706255	0.987277	0.8407	0.987277
17	3	22	22	DeckNominal/Deck...	0.286561	0.60915	0.0706255	0.987277	0.8407	0.987277
18	1	12	12	DeckNominal/Deck...	0.286561	0.748047	0.0779061	0.987277	0.8407	0.987277
19	1	11	12	DeckNominal/Deck...	0.286561	0.748047	0.0779061	0.987277	0.8407	0.987277
20	1	13	14	DeckNominal/Deck...	0.286561	0.73272	0.156822	0.978314	0.799564	0.978314

- Click the **Util(max)** column to sort the values in descending order

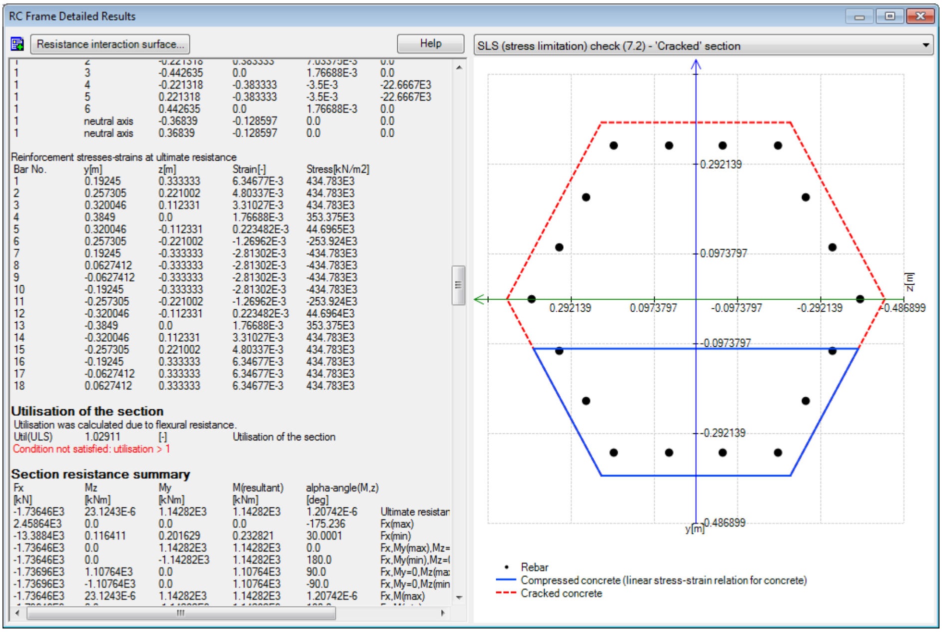
- Select **row 15** (the row containing the highest utilisation value for the pier) and then click the **Detailed results for selected row** button at the top of the dialog.

Detailed results will be displayed.

Note. When accessing detailed results from the Summary page the appropriate cross-section diagram showing the extent of the tension or compression in the section will need to be selected from the droplist.

- Scroll down the Detailed Results panel, and in the SLS Stress Limitation Check (7.2) a statement will be made according to whether a section is cracked or uncracked. For this example it states: “*The section is cracked - the stresses in the* *concrete assuming 'uncracked' concrete are larger than fctm*" – so a Cracked concrete section diagram should be drawn.

- Select **SLS Stress Limitation Check (7.2) – 'Cracked Section'** from the droplist on the right-hand side of the dialog.



- Scroll further down the detailed results for the selected row to view the data relating to the Utilisations being exceeded, and the "Condition not fulfilled: utilisation > 1" statement.

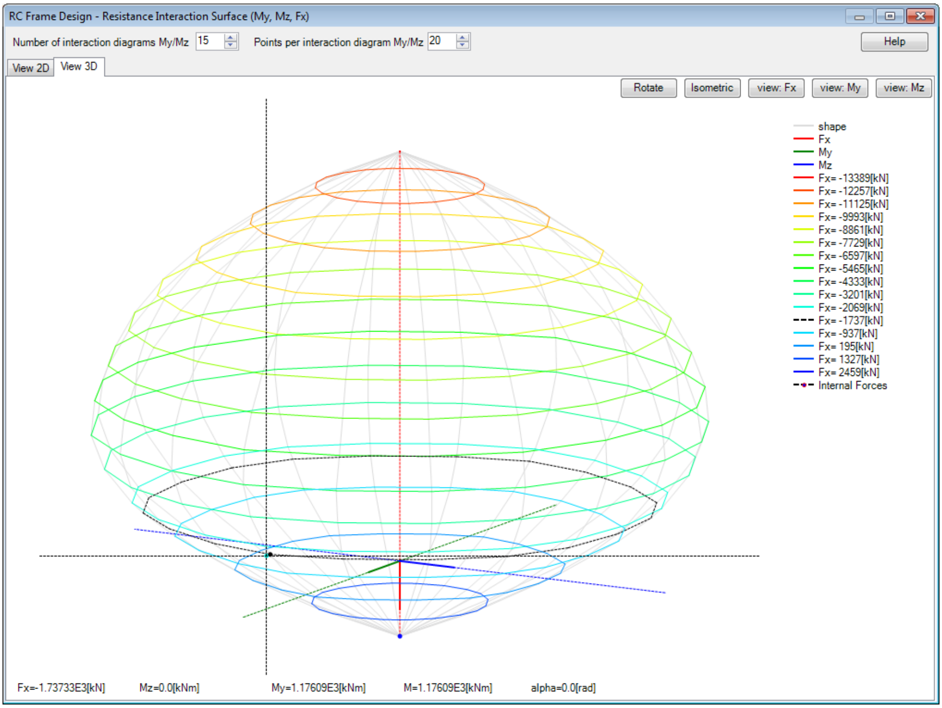
Interaction surface diagrams – for pier

For ULS results, interaction surface diagrams can be produced.

- Click the **Resistance Interaction Surface** button at the top of the dialog.

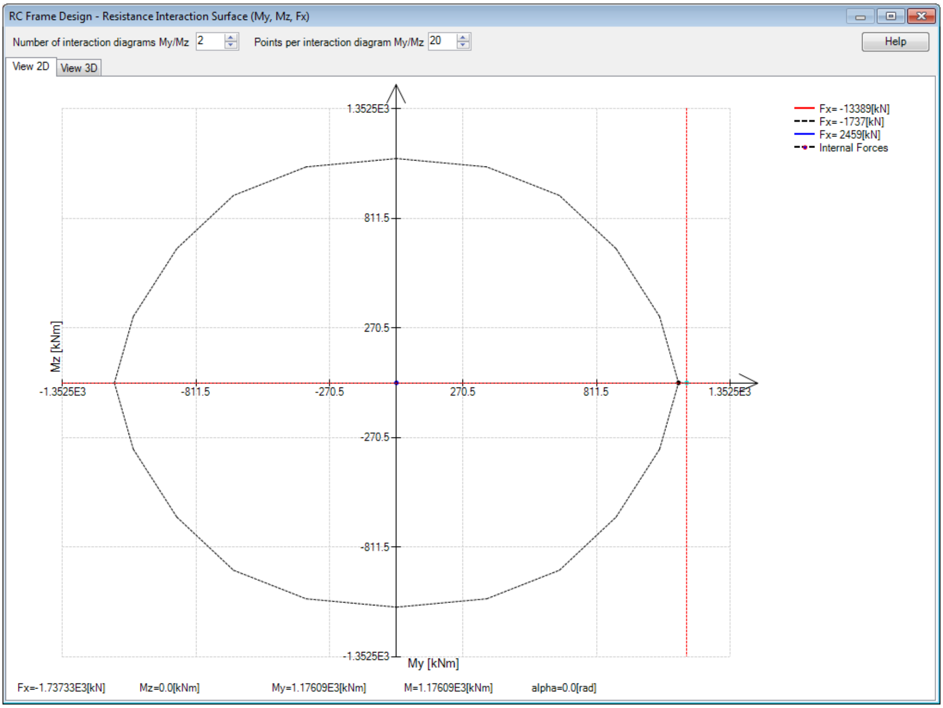
The interaction surface diagram for axial load Fx and moments My and Mz is displayed. By default a 2D view is shown. The black dotted line shows the My Mz contour for the applied axial force.

- Click the **View 3D** button to see the 3D interaction surface diagram.



Isolating a value for applied axial force

- With the **View 2D** tab selected change the Number of interaction diagrams **My/Mz** to **2** (and click elsewhere on the dialog to update the image).



This isolates the My Mz plot for the applied axial force in the selected row. The extent to which the force is outside the resistance surface can now be more easily seen.

- Close the Resistance Interaction Surface dialog.
- Close the Detailed Results dialog.

Changes required following the initial design check

As a result of the initial design check:

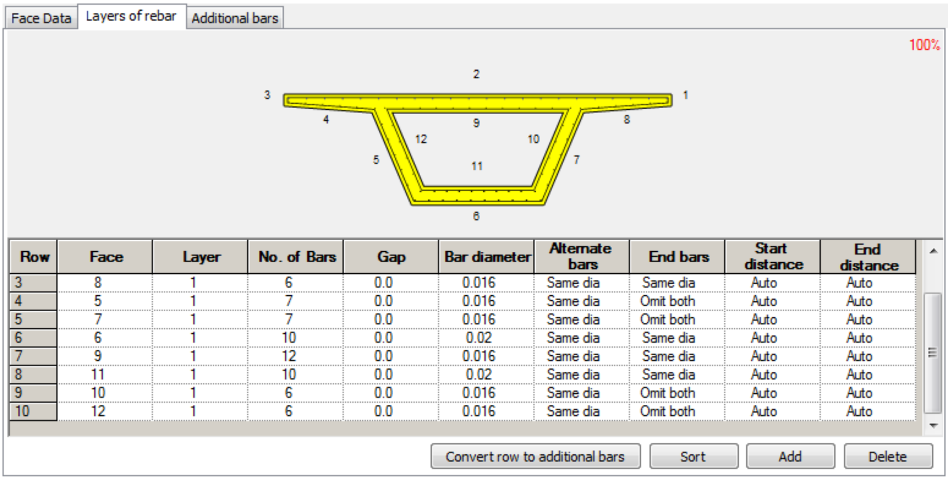
- ❑ **The reinforcing bars in the deck section will be increased generally from 12mm diameter to 16mm diameter, but 20mm diameter bars will be used in the bottom two faces of the box section**
- ❑ **The main reinforcing bars in the piers will be increased from 20mm diameter to 25mm diameter.**

Note. When making changes to reinforcement quantities there is no need to re-solve the model. The RC Frame Design calculations will be automatically re-run according to the changes made. When making changes to geometric attributes a model must be re-solved.

Modifying the reinforcement in the deck

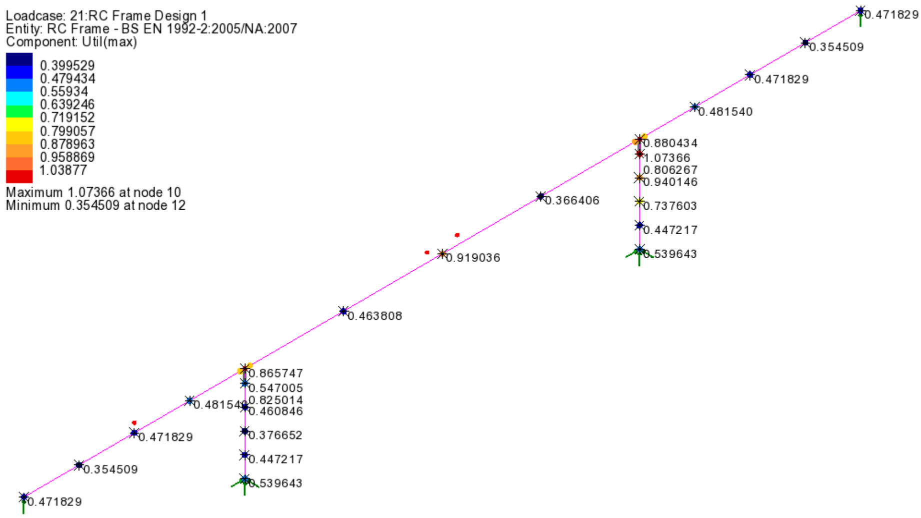
For the deck, the bar diameter will be changed from 12mm to 16mm throughout, with the exception of faces 6 and 11, which will have 20mm diameter bars. The section reinforcement is modified from the Utilities treeview.

- In the treeview double click **DeckSectionNominalReinf**.
- Select the **Layers of rebar** tab and change the Bar diameter to **0.016** for each face except faces 6 and 11.
- For faces 6 and 11 change the bar diameter to **0.020** and click **OK**.



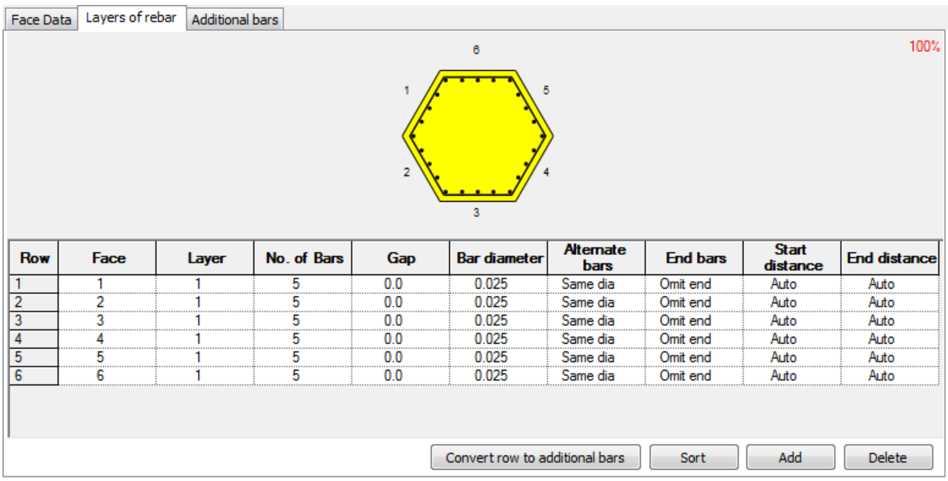
Row	Face	Layer	No. of Bars	Gap	Bar diameter	Alternate bars	End bars	Start distance	End distance
3	8	1	6	0.0	0.016	Same dia	Same dia	Auto	Auto
4	5	1	7	0.0	0.016	Same dia	Omit both	Auto	Auto
5	7	1	7	0.0	0.016	Same dia	Omit both	Auto	Auto
6	6	1	10	0.0	0.02	Same dia	Same dia	Auto	Auto
7	9	1	12	0.0	0.016	Same dia	Same dia	Auto	Auto
8	11	1	10	0.0	0.02	Same dia	Same dia	Auto	Auto
9	10	1	6	0.0	0.016	Same dia	Omit both	Auto	Auto
10	12	1	6	0.0	0.016	Same dia	Omit both	Auto	Auto

The RC Frame design results will be re-calculated, showing that all deck utilisation values are now less than 1.0. One of the piers however still shows a utilisation value that exceeds 1.0. To attempt to reduce this value the pier reinforcement will be modified.

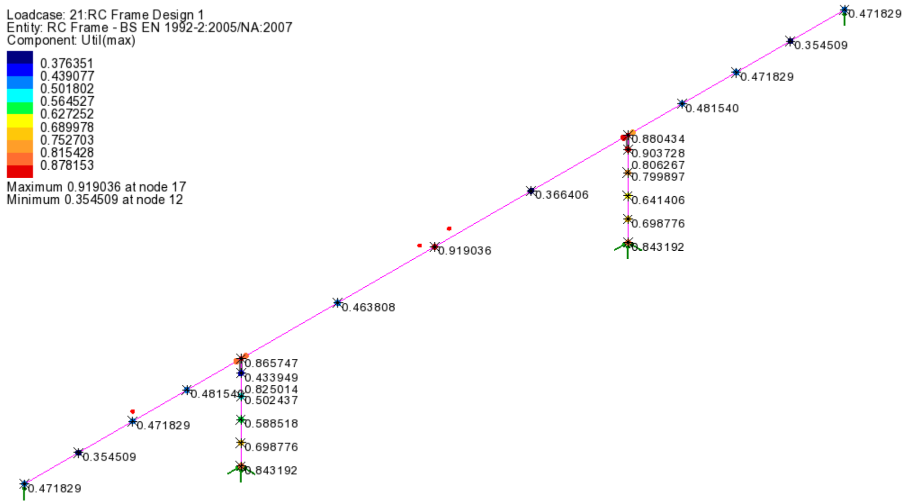


Modifying the reinforcement in the piers

- In the treeview double click **PierSectionReinf**.
- Select the **Layers of rebar** tab and change the Bar diameter to **0.025** for each face and click **OK**. The RC Frame design results will be re-calculated.



Row	Face	Layer	No. of Bars	Gap	Bar diameter	Alternate bars	End bars	Start distance	End distance
1	1	1	5	0.0	0.025	Same dia	Omit end	Auto	Auto
2	2	1	5	0.0	0.025	Same dia	Omit end	Auto	Auto
3	3	1	5	0.0	0.025	Same dia	Omit end	Auto	Auto
4	4	1	5	0.0	0.025	Same dia	Omit end	Auto	Auto
5	5	1	5	0.0	0.025	Same dia	Omit end	Auto	Auto
6	6	1	5	0.0	0.025	Same dia	Omit end	Auto	Auto



Utilisation values are now all less than 1.0 with the largest value of **0.919** occurring in the deck. Whilst the reinforcement used has not been optimised, the bridge has effectively passed the design check.

Verification of reinforcement specified

A final assessment can be carried out by considering the results at internal points all the beams/piers instead at nodes. A separate RC Frame design loadcase will be created for this.

- In the treeview double click on the **RC Frame Design 1** loadcase.
- Change the Design check locations from Nodes to **Internal points**
- Change the name to **RC Frame Design 2** and click **OK**.

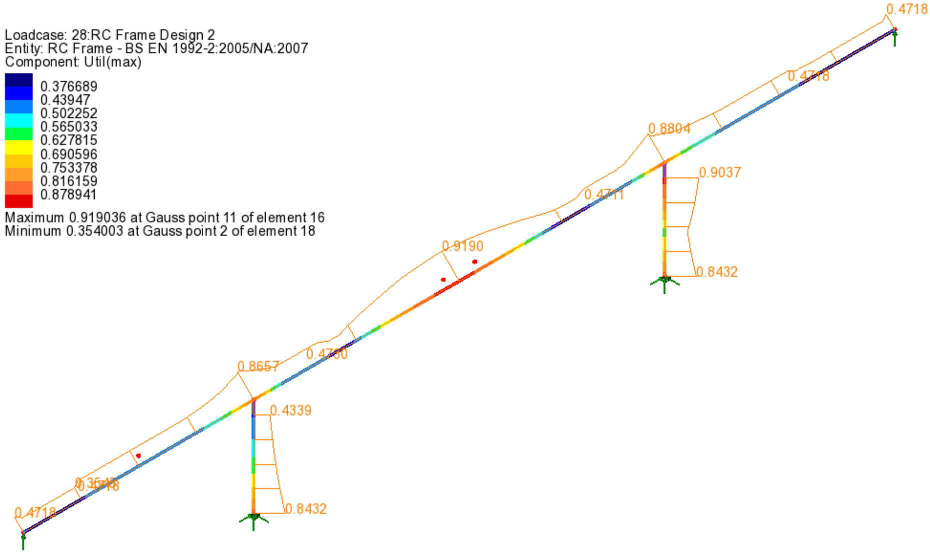
Note. This copies and creates a new loadcase (as opposed to creating a new one from scratch) and saves having to re-define the Load Groups.

- In the treeview right-click on the loadcase **RC Frame Design 2** and select **Set Active**. If asked, click **No** to not save results.

As an alternative to plotting values, diagrams of Util(max) can be drawn.

- In the Treeview turn-off the **Values** layer.
- With no features selected, click the right-hand mouse button in a blank part of the active window and select the **Diagrams** option to add the diagrams layer to the Treeview.

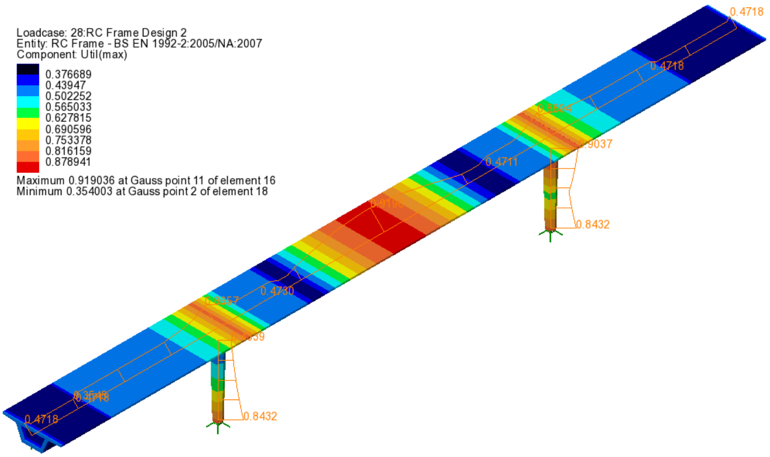
- On the Diagrams properties dialog, ensure component **Util(max)** is selected for **Internal points** and click **OK** to see the utilisation results for all members.



From this diagram the peak Utilisation value in the deck can be seen to be **0.919** as previously obtained.

Note. When plotting diagrams of results such those for values of utilisation that have no direction component, diagrams are drawn by flat-to-screen by default.

- Press the button to see the contours on a fleshed model.

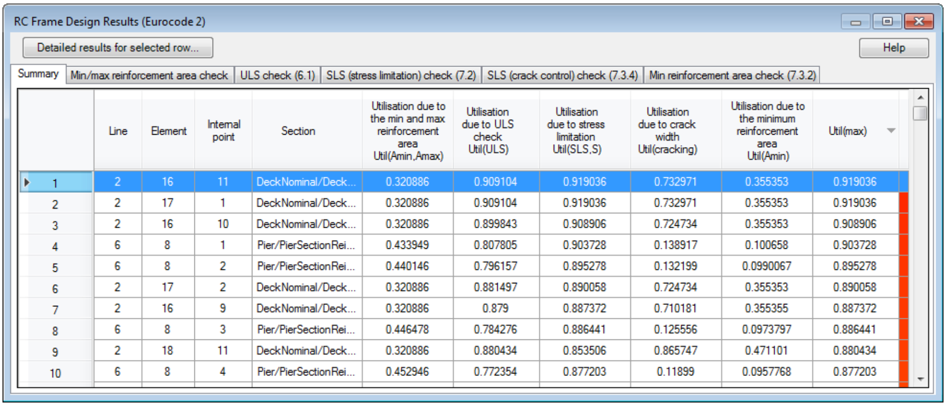


Note. Unlike nodal results, which only produce contour results at singular node, when plotting results for internal points the values obtained at each internal point can be interpolated to create line contours.

Creating a report

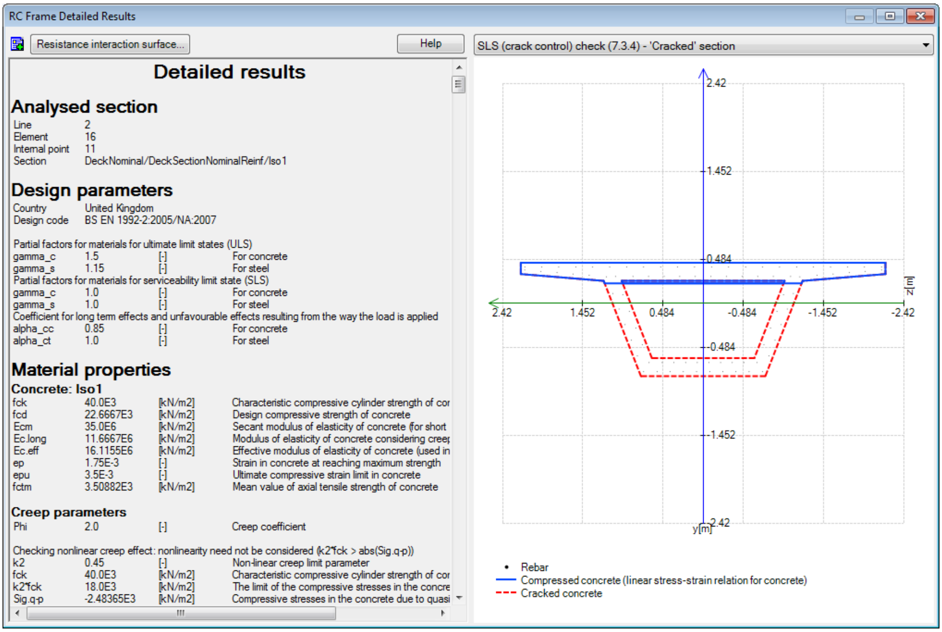
RC Frame design checks can be added as a chapter to a model report. To illustrate how this is done, a report is to be created for the location of Util(max) in the central span.

- In the treeview right-click on **RC Frame Design 2** and select **Show Results**.
- On the summary table, sort by values of **Util(max)**.



	Line	Element	Internal point	Section	Utilisation due to the min and max reinforcement area Util(Amin,Amax)	Utilisation due to ULS check Util(ULS)	Utilisation due to stress limitation Util(SLS,S)	Utilisation due to crack width Util(cracking)	Utilisation due to the minimum reinforcement area Util(Amin)	Util(max)
1	2	16	11	DeckNominal/Deck...	0.320886	0.909104	0.919036	0.732971	0.355353	0.919036
2	2	17	1	DeckNominal/Deck...	0.320886	0.909104	0.919036	0.732971	0.355353	0.919036
3	2	16	10	DeckNominal/Deck...	0.320886	0.899843	0.908906	0.724734	0.355353	0.908906
4	6	8	1	Pier/PierSectionRei...	0.433949	0.807805	0.903728	0.138917	0.100658	0.903728
5	6	8	2	Pier/PierSectionRei...	0.440146	0.796157	0.895278	0.132199	0.0990067	0.895278
6	2	17	2	DeckNominal/Deck...	0.320886	0.881497	0.890058	0.724734	0.355353	0.890058
7	2	16	9	DeckNominal/Deck...	0.320886	0.879	0.887372	0.710181	0.355355	0.887372
8	6	8	3	Pier/PierSectionRei...	0.446478	0.784276	0.886441	0.125556	0.0973797	0.886441
9	2	18	11	DeckNominal/Deck...	0.320886	0.880434	0.853506	0.865747	0.471101	0.880434
10	6	8	4	Pier/PierSectionRei...	0.452946	0.772354	0.877203	0.11899	0.0957768	0.877203

- Select **Row 1** (containing the highest utilisation value for the deck) and then click the **Detailed Results for Selected Row** button at the top of the dialog. Double clicking the row will also give the detailed results.
- On the RC Frame Detailed Results dialog, and from the drop-list on the top-right of the dialog, select **SLS (Crack Control) Check (7.3.4) – ‘Cracked’ Section**



- Click the **Add to report** button at the top of the dialog.
- Choose **OK** to create a report.
- Enter the title as **Pedestrian Bridge** and click **OK**.
- On the dialog that appears click **OK**.
- Close the RC Frame Detailed Results and RC Frame Design Results dialogs.
- In the Reports treeview right click on **Rpt1** and select **View Report**.

A PDF of the report is created. This can be viewed, saved and printed as required.

Save the model

File
Save

Save the model file.

This completes the example.

Discussion on optimising the reinforcement in the deck

As shown, the footbridge passes the design code check when main steel reinforcement of 16mm diameter bars are generally used throughout the deck section (with 20mm diameter in the lowest two faces of the box section), and 25mm diameter bars are used throughout each pier.

If additional refinement of the reinforcement arrangement is required this may achieved by adding additional bars locally where needed, or by changing bar diameters, and defining the lengths over which selected bar diameters act.

Optimising the reinforcement in the deck

Two methods are available:

- **Option 1: Assigning a single geometric line attribute referencing all required section reinforcement to all lines of the deck**.
- **Option 2: Assigning individual geometric line attributes to each line of the deck**

These options are explained in the following pages.

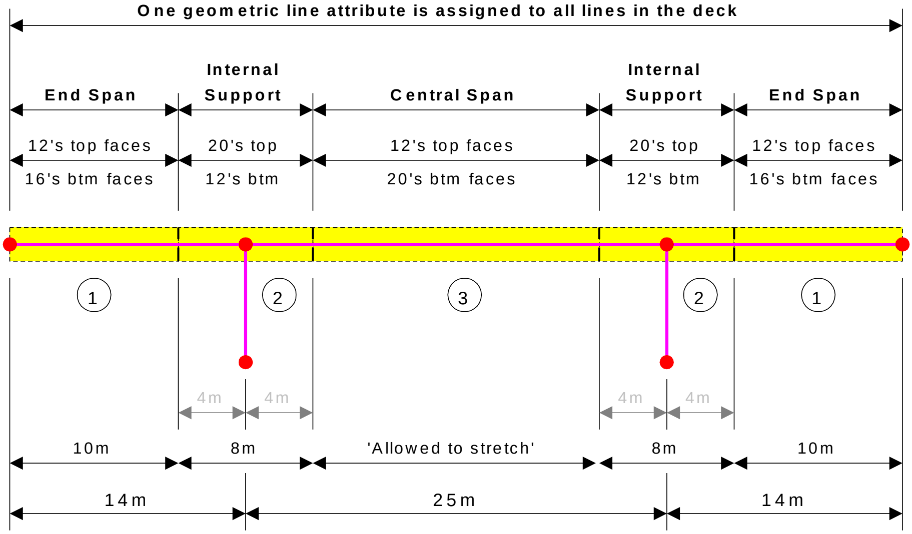
Option 1: Assigning a single geometric line attribute to all lines of the deck

Define section reinforcement

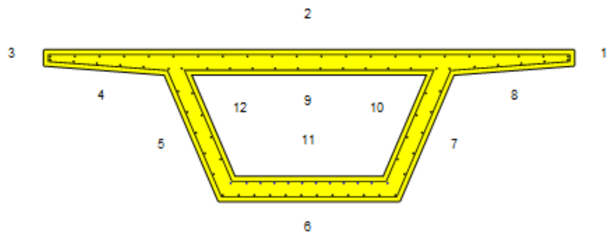
For this option separate section reinforcement utilities need to be created for each 'region' of the deck that is to be reinforced, and be given suitable names that specify their location, such as **EndSpan**, **InternalSupport**, and **CentralSpan**.

This can be done by going to the Utilities treeview and editing the **DeckSectionNominalReinf** section utility, editing the reinforcement bar details and changing the name to be **DeckSectionEndSpanReinft**, then repeating for the Internal Support and Central Span arrangements and names required.

The following image shows a side elevation of the bridge and a typical scenario where different reinforcing bar arrangements can be specified for hogging at supports or bending in spans. See the accompanying table for bar diameters required throughout the deck.



For each section reinforcement utility, the following bar diameters are defined for the faces stated.

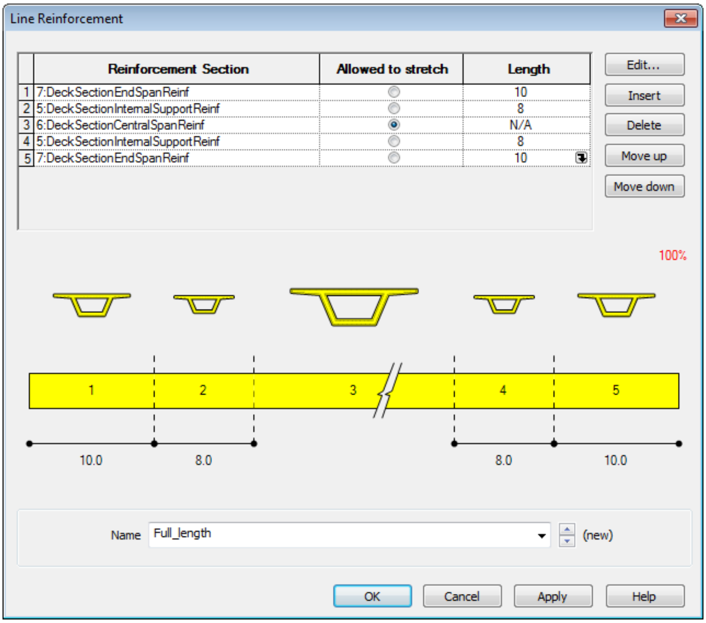


Faces	Bar diameters required		
	End Span	Internal Support	Central Span
2 and 9	**12**	**20**	**12**
6 and 11	**16**	**12**	**20**
All other faces	**12**	**12**	**12**

Define line reinforcement

A single line reinforcement utility named **Full_length** also needs to be defined. This will reference all three section reinforcement utilities, and also include the lengths over which each reinforcement section would apply.

This can be done by editing the **DeckLineNominalReinf** line utility, inserting additional reinforcement section details, specifying lengths, and changing the name to be **Full_length**, as shown on the following image, before saving the definition.

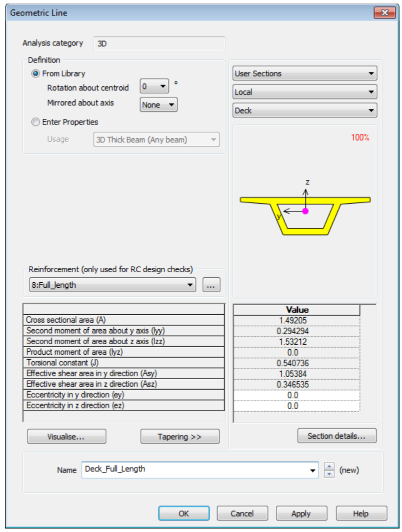


Note that the central span region is un-dimensioned and is 'Allowed to stretch' to fit the length of the deck that remains between the support regions.

The bridge is symmetrical hence the re-use of the Internal Support and End Span reinforcement section utilities in the line reinforcement definition.

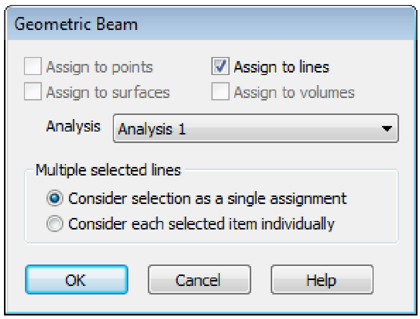
Geometric line attribute

In the Attributes treeview the existing geometric line attribute for the deck 'DeckNominal (deck)' needs to be edited to reference the **Full_length** line reinforcement utility, and given a new name of **Deck_Full_Length**, before saving the definition.



All three lines representing the deck then need to be selected, and the existing Geometric line assignment **DeckNominal (deck)** that is assigned to them, must be de-assigned from the selection.

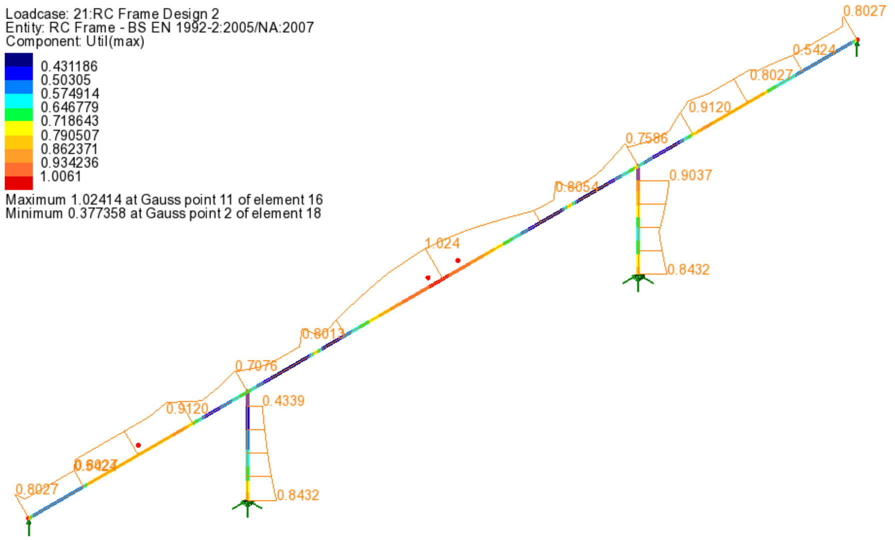
Then, with the lines in the deck still selected, assign the geometric line attribute **Full_length** to those lines, ensuring (very importantly) that the option to **Consider selection as a single assignment** is selected.



Util(max)

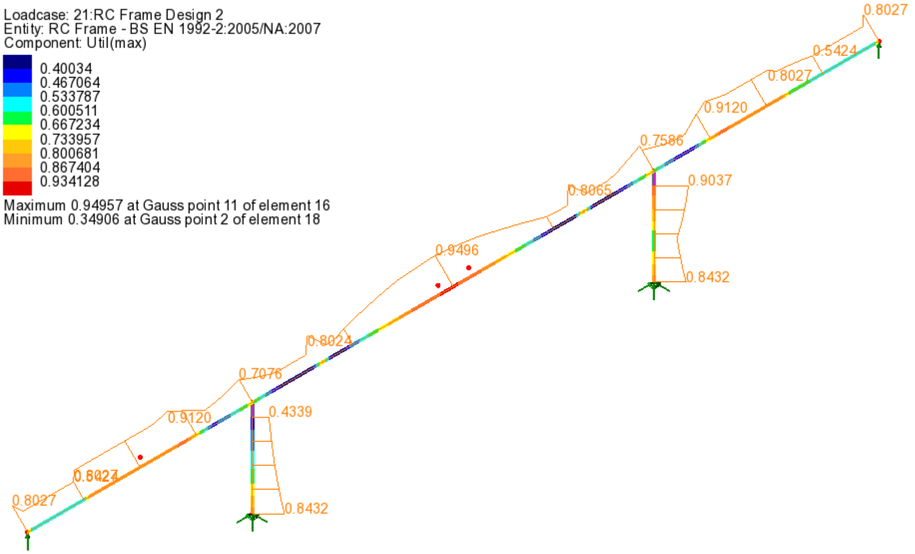
Once the revised geometric line attribute had been assigned, the RC Frame design results will update to show values for **Util(max)** in the deck.

Results for loadcase RC Frame Design 2 are shown:



This, however, shows a utilisation of just great than 1.0 in the central span.

By re-visiting the **DeckSectionCentralSpanReinft** utility and increasing the number of bars in faces 6 and 11 from 10 bars to 11 bars the utilisation will reduce to less than 1.0 as shown on the following image.



When compared to the values that previously passed the initial design check, an increase in the maximum utilisation in the central and end spans is seen.

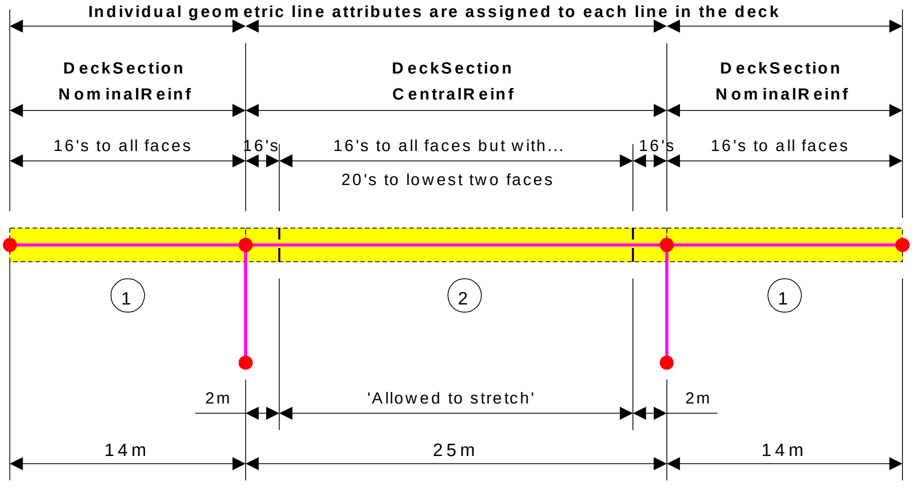
Option 2: Assigning individual geometric beam attributes to each line of the deck

Section reinforcement

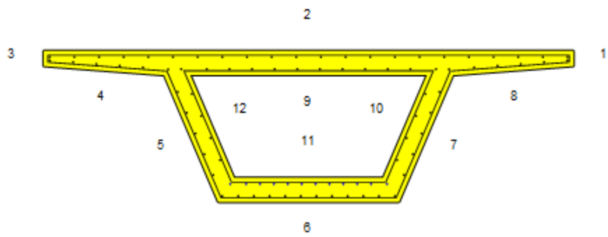
For this option separate section reinforcement utilities need to be created for each 'member length' in the deck that is to be reinforced, and given names such as: **DeckSectionNominalReinf** and **DeckSectionCentralReinf**.

This can be done by editing the **DeckSectionNominalReinf** section utility, editing the reinforcement bar details, then also creating another section utility named **DeckSectionCentralReinft**.

The following image shows a typical scenario where individual geometric line attributes are assigned to individual lines representing spans, and where in one span more than one reinforcement section arrangement is required. See the accompanying table for bar diameters required for each.



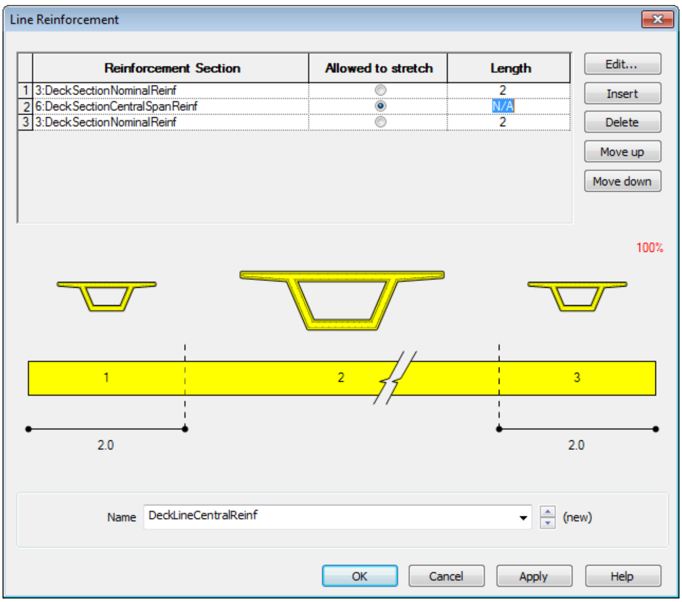
For each section reinforcement utility, the following bar diameters are defined for the faces stated:.



Faces	Bar diameters required	
	DeckSectionNominalReinf	DeckSectionCentralReinf
6 and 11	**16**	**20**
All other faces	**16**	**16**

Line reinforcement dialog

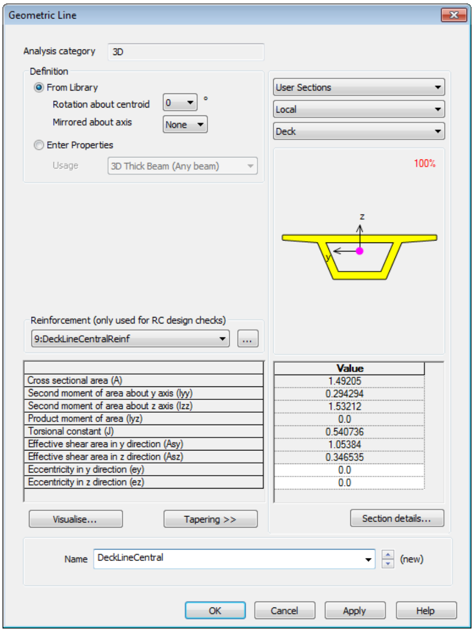
A new line reinforcement utility called **DeckLineCentralReinf** is required for the central span. This will reference the **DeckSectionNominalReinf** section reinforcement for a 2m region at each end of the span, but require a new reinforcement section **DeckSectionCentralReinf** for the remaining middle portion of the span, specifying 20mm diameter bars in the two lowest faces in this span.



The central region is un-dimensioned and would be 'Allowed to stretch' to fit the remainder of the length of this span.

Geometric line attribute

The existing geometric line attribute for the deck should be copied and renamed to be **DeckLineCentral**, and then be edited to reference the **DeckLineCentralReinf** line reinforcement, as per the image right.



Note. Geometric line attributes that have previously been assigned to a series of lines as a single assignment by using the '**Consider selection as a single assignment**' option (as done earlier in this example), will need to be de-assigned from any lines prior to re-assigning a different geometric line attribute.

With reference to the above note, de-assign DeckFullLength line attribute from all lines in the model

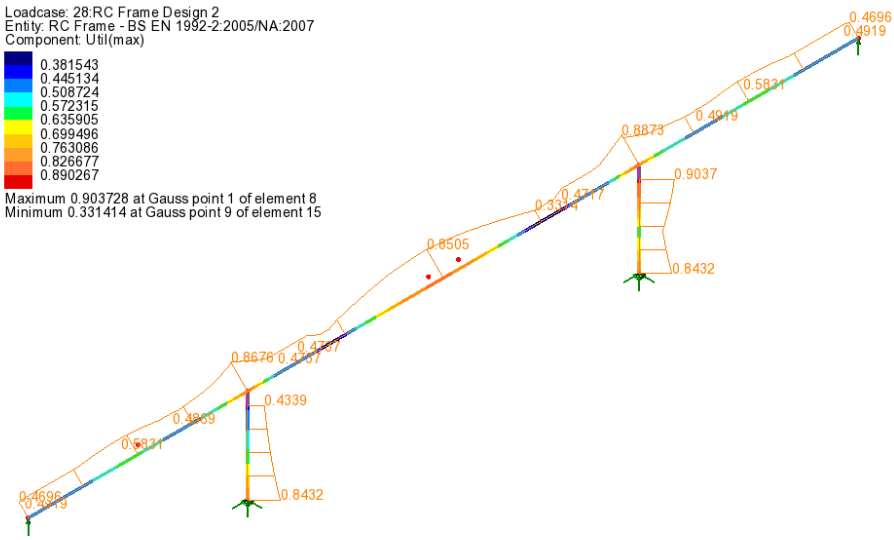
Then assign the **DeckLineCentral** line attribute to the line representing the central span

And finally, assign the **DeckNominal** line attribute to the lines representing the end spans

Util(max)

As the revised geometric line attributes are assigned, the RC Frame design results will update to show values for **Util(max)** in the deck.

Results for loadcase RC Frame Design 2 are shown:



Summary

Individual line attributes (referencing line and section reinforcement arrangements) can be assigned to individual lines representing concrete members, as in this latter discussion, or a single line attribute may be assigned to a series of lines that represent concrete members, as in the former discussion.