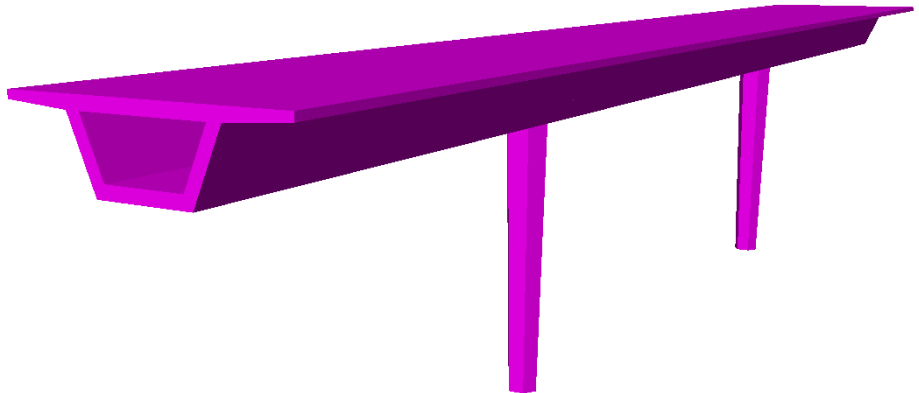


RC Frame Design to EN 1992-2

For LUSAS version:	21.0
For software product(s):	LUSAS <i>Civil & Structural</i> or LUSAS <i>Bridge</i> .
With product option(s):	RC Frame Design

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.

Design combinations are included in the model that is created from the file provided. These comprise ULS persistent combination Set B, SLS characteristic combination and SLS quasi-permanent combination for EN 1990 – 2002 Footbridges UK.

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, Design Combinations Enveloping, Member Utilisation Ratio, Reporting

Associated Files

Associated files can be downloaded from the user area of the LUSAS website.



- rc_frame_bridge.lvb** creates an initial model with general attributes assigned and load combinations and envelopes defined, ready to add reinforcement details and subsequently carry out a design check.
- rc_frame_bridge_modelling.lvb** creates a model with all reinforcement details defined and assigned in order to carry out a design check.

Modelling

Running LUSAS Modeller

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

Creating a Model


File
New...

- Enter a file name of **rc_frame_bridge**
- Use the default **User-defined** working folder.
- Ensure an Analysis type of **3D** is set.
- Click the **OK** button.




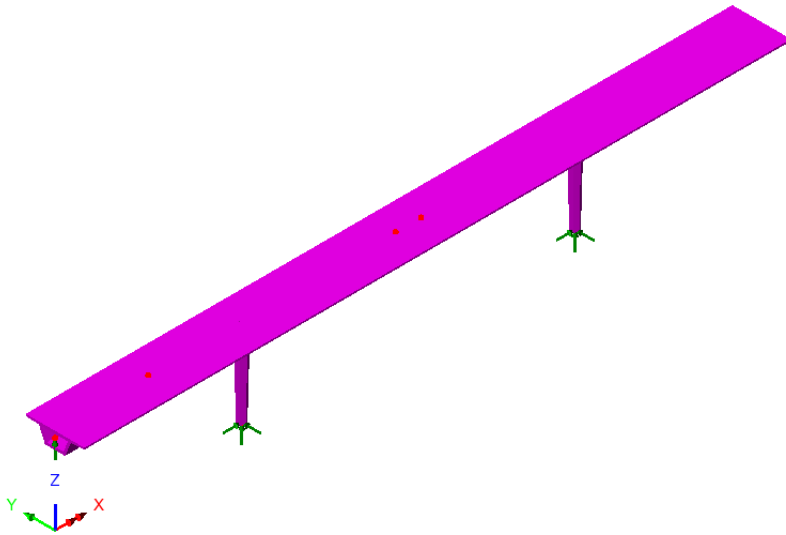
Note. There is no need to enter any other new model details when the intention is to run a script to build an initial model, since the contents of the script will overwrite any other settings made.

File
Script >
Run Script...

-  To create the model, open the read-only file **rc_frame_bridge.lvb** that was downloaded and placed in a folder of your choosing.

A model ready for defining member reinforcement is displayed.

-  Use the Isometric button to show the true section shapes, and then select again to return to the view of the unfleshed line beam model.




- Save the file.

File
Save


Defining member reinforcement

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 (D=0.8 (HSS D=0.8)/ D=0.55 (HSS D=0.55)** entry.

- In the ‘Reinforcement (only used 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.
- On the Longitudinal tab click in the first cell of the grid and select **New**. A section reinforcement dialog for the chosen 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.

Face data

- 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)

Row	Face	Actual cover to link	Nominal cover to bars	Allowance for link
1	All	0.03	Actual	0.01

Layers of rebar



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**. Click elsewhere in the grid top force the display to update. A row of bars will appear in the section for face 1.

RC Frame Design to EN 1992-2

- 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.

The screenshot shows the 'Section Reinforcement' dialog box. At the top, the 'Shape outline from' dropdown is set to '2:Pier (D=0.8 (+SS D=0.8)/D=0.55 (+SS D=0.55))'. Below this, there are tabs for 'Face Data', 'Layers of rebar', and 'Additional bars'. The 'Face Data' tab is active, showing a yellow hexagonal reinforcement layout with faces numbered 1 through 6. A '100%' zoom indicator is visible in the top right corner of the layout area.

Row	Face	Layer	No. of Bars	Gap	Bar diameter	Alternate bars	End bars	Start distance	End distance
1	1	1	4	0.0	0.02	Same dia	Omit end	Auto	Auto
2	2	1	4	0.0	0.02	Same dia	Omit end	Auto	Auto
3	3	1	4	0.0	0.02	Same dia	Omit end	Auto	Auto
4	4	1	4	0.0	0.02	Same dia	Omit end	Auto	Auto
5	5	1	4	0.0	0.02	Same dia	Omit end	Auto	Auto
6	6	1	4	0.0	0.02	Same dia	Omit end	Auto	Auto

At the bottom of the dialog, there are buttons for 'Convert row to additional bars', 'Sort', 'Add', and 'Delete'. The 'Name' field is set to 'PierSectionReinf' and has a '(new)' indicator. At the very bottom are 'OK', 'Cancel', 'Apply', and 'Help' buttons.

- Change the name to **PierSectionReinf** and click OK to return to the previous dialog.

Defining torsion / shear reinforcement

Line Reinforcement

Shear and torsion reinforcement
 Torsion/shear-closed links
 Shear legs local z
 Shear legs local y

Shear and torsion properties (Advanced)
 Auto

Longitudinal | Torsion links

	No. of Bars	Dia. of Bars	Spacing	Cover to Centre of Bar	Allowed to stretch	Length
1	1	0.01	0.2	0.035	<input checked="" type="checkbox"/>	N/A

Buttons: Insert, Delete, Move up, Move down

100%

Name: PierLineReinf (new)

Buttons: OK, Cancel, Apply, Help

- Back on the Line reinforcement dialog, select the option for **Torsion/shear-closed links**.
- Go to the Torsion links tab. If the outer faces can be assumed to have fully anchored reinforcing bars that can be taken as a torsional link, the No of Bars can be set to **1** (as in this case).
- Enter the Spacing as **0.2**.
- Change the Cover to Centre of Bar to **0.035**.
- Enter a name of **PierLineReinf** and click **OK**.
- Back on the Geometric Line dialog click **OK** to save the change.



Note. Shear checks are typically carried out using an equivalent rectangular section that represent the narrowest part of the web of a concrete section. For non-rectangular section shapes (such as the hexagonal piers used here) the use of the Auto option will allow a best estimate of equivalent shear and torsion properties to be automatically calculated and allow a design check for those members to take place

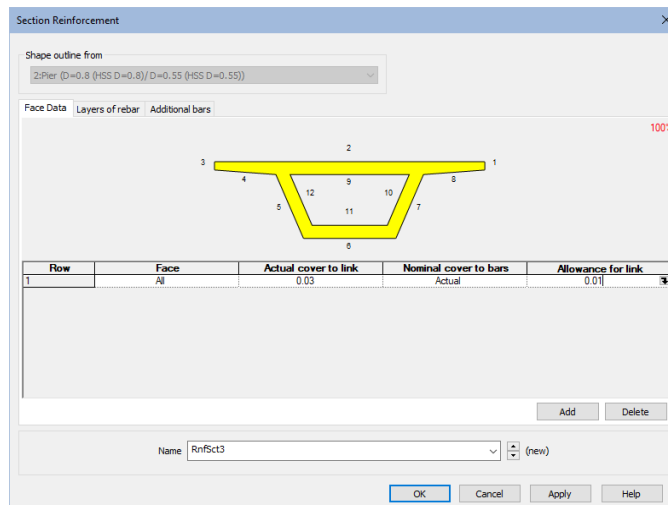
Defining deck reinforcement

- In the  treeview double-click the **DeckNominal (Deck (Sbox...))** entry.

- In the 'Reinforcement (only used for design checks)' panel click on the drop-down list button and select **New**. The Line Reinforcement dialog will be displayed.
- On the Longitudinal tab click in the first cell of the grid and select **New**. A section reinforcement dialog for the geometric line attribute is displayed.

Face data

- 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.

Defining layers of rebar

- 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 end bars defined for faces 4, 8, 9, 6 and 11.

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

Section Reinforcement

Shape outline from
1:DeckNominal (Deck)

Face Data Layers of rebar Additional bars

Row	Face	Layer	No. of Bars	Gap	Bar diameter	Alternate bars	End bars	Start distance	End distance
1	2	1	22	0.0	0.012	Same dia	Same dia	Auto	Auto
2	4	1	6	0.0	0.012	Same dia	Same dia	Auto	Auto
3	8	1	6	0.0	0.012	Same dia	Same dia	Auto	Auto
4	5	1	7	0.0	0.012	Same dia	Omit both	Auto	Auto
5	7	1	7	0.0	0.012	Same dia	Omit both	Auto	Auto
6	6	1	10	0.0	0.012	Same dia	Same dia	Auto	Auto
7	9	1	12	0.0	0.012	Same dia	Same dia	Auto	Auto
8	11	1	10	0.0	0.012	Same dia	Same dia	Auto	Auto
9	10	1	6	0.0	0.012	Same dia	Omit both	Auto	Auto

Convert row to additional bars Sort Add Delete

Name: DeckSectionNominalReinf (3)

OK Cancel Apply Help

- Change the name to **DeckSectionNominalReinf** and click **OK** to return to the Line Reinforcement dialog.

Defining shear and torsion reinforcement / properties

- Select the options to define **Torsion/shear-closed links**, **Shear legs local z** and **Shear legs local y**.



Note. When carrying out RC Frame design, shear and torsion checks are typically carried out using an equivalent rectangular section. All geometric line attributes assigned to a model that use a section shape for which a best-estimate of equivalent shear and torsion properties can be automatically calculated, will allow a design check for those members to take place. Members with sections that are not supported (such as this hollow box deck section) will require an equivalent shear and torsion section to be assigned to them for this to happen.

In this example, an equivalent rectangular section needs to be defined based upon the depth of the deck (1.25) and the average width of the box section part of the deck i.e. top width + bottom width / 2 = $(2.4+1.5)/2 = 1.95$

- In the Shear and torsion properties (Advanced) panel click on the drop-down list and select **New**.
- For Height (D) enter a value of **1.25**
- For Width (B) enter a value of **1.95**
- Select the **Hollow shape** check box.
- For tf1 enter a value of **0.2**
- Select the **tf1=tf2=tw1=tw2** check box option

Equivalent Shear and Torsion Section

Equivalent section dimensions

Height (D) - local z: 1.25

Width (B) - local y: 1.95

Hollow shape

tf1: 0.2

tf1=tf2=tw1=tw2

tf2: 0.2

tw1: 0.2

tw2: 0.2

Shear in z direction | Shear in y direction | Torsion

Effective web width

- Change the name to **EquivalentDeck** and click **OK** to return to the Line reinforcement dialog.

Line Reinforcement ✕

Shear and torsion reinforcement

 Torsion/shear-closed links
 Shear legs local z
 Shear legs local y

Shear and torsion properties (Advanced)

1:EquivalentDeck

Longitudinal

Transverse z

Transverse y

Torsion links

	No. of Bars	Dia. of Bars	Spacing	Cover to Centre of Bar	Allowed to stretch	Length
1	1	0.01	0.15	0.035	<input checked="" type="radio"/>	N/A

Insert

Delete

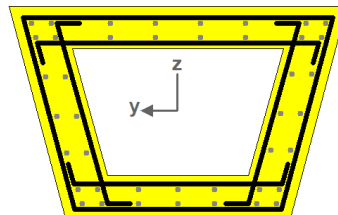
Move up

Move down

- On the Line reinforcement dialog, go to the **Torsion links** tab and enter the Spacing as **0.15**. Change the Cover to Centre of Bar to **0.035** and leave the value for Dia. of bars as **0.01**.
- Go to the Transverse z tab and enter the Spacing as **0.15** and leave the value for Dia. of bars as **0.01**
- Go to the Transverse y tab and enter the Spacing as **0.15** and leave the value for Dia. of bars as **0.01**
- Enter the name as **DeckLineNominalReinf** and click **OK**
- Back on the Geometric Line attribute dialog click **OK** to save the change.



Note. When defining links for box sections, if the 4 outer faces have fully anchored reinforcing bars that can be taken as a torsional link, then 'Torsion/shear closed links', No of Bars = 1 can be stated. If the 4 inner faces have reinforcing bars which can be used to carry vertical (z-) and horizontal (y-) shear, then 'Shear legs local z', 'No. of legs' = 2 and 'Shear legs local y'. No. of legs' = 2 can be stated.



Defining Concrete (Reinforced) Material

A general concrete material has already been assigned to the supplied model. However, for RC Frame Design a reinforced concrete material attribute must be defined and assigned to all lines representing concrete members. The ‘Reinforced Concrete’ material is a compound material that references both concrete and steel properties for a chosen design code in a single material attribute.

- Select Country (Region) **UK**.
- 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**.

The screenshot shows the 'RC Material' dialog box with the following settings:

- Country: Europe
- Design code: EN1992
- Concrete section: Iso1 (C40/50 | Concrete | EN1992-2:2005)
- Reinforcement: Iso2 (550B | Steel - Reinforcing bar | EN1992-2:2005)
- Name: Reinforced Concrete (new)

- 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 (EN1992)** 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...

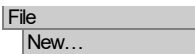
If the analysis fails, information relating to the nature of the error encountered can be written to an output file in addition to the text output window. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.


Rebuilding a Model

If it proves impossible to correct the errors reported, a file is provided to enable you to re-create the model from scratch and run an analysis successfully. You may download this file from the user area of the LUSAS website.



- rc_frame_bridge_modelling.lvb** carries out the modelling of the example.

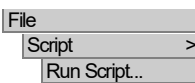


 Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **rc_frame_bridge**
- Use the default **User-defined** working folder.
- Ensure an Analysis type of **3D** is set.
- Click the **OK** button.



Note. There is no need to enter any other new model details when a script is run to build a model, since the contents of the script will overwrite any other settings made.



To recreate the model, select the file **rc_frame_bridge_modelling.lvb** that was downloaded and placed in a folder of your choosing.



Rerun the analysis to generate the results.

Viewing the Results

Analysis loadcase results are present in the  Treeview. A deformed mesh layer will be added automatically to the to the  Treeview.

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.

Meshing considerations

Design checks can be carried out at:

- **Averaged nodal** - results for nodes at the ends and within each beam element that is assigned to a member are averaged at each node.
- **Unaveraged nodal** - results for nodes at the ends and within each beam element that is assigned to a member are unaveraged at each node.
- **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

Design
RC Frame Design...

- From the country drop-down menu choose **UK**.
- From the Design code drop-down menu choose **BS EN 1992-2:2005/NA:2007**. The Nationally Determined Parameters (NDPs) used in the design checks for the code 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:

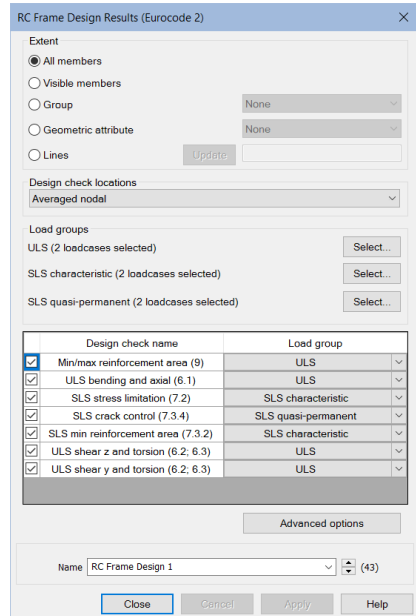
- For **Beam**, accept the default values for minimum and maximum reinforcement ratio of **1.3e-3** and **0.04** respectively, and change the name to **Deck**. Click **Apply**.
- Select **Column**, accept the default values for minimum and maximum reinforcement ratio of **2.0e-3** and **0.04** respectively, 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 drop down menu ensure that **Averaged nodal** is selected.
- In the Load Groups panel press the **Select** button for **ULS**.
- On the Select Loadcases dialog presented, select **ULS_P_B_S (Max)** and **ULS_P_B_S (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_C_S (Max)** and **SLS_C_S (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_S (Max)** and **SLS_QP_S (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.

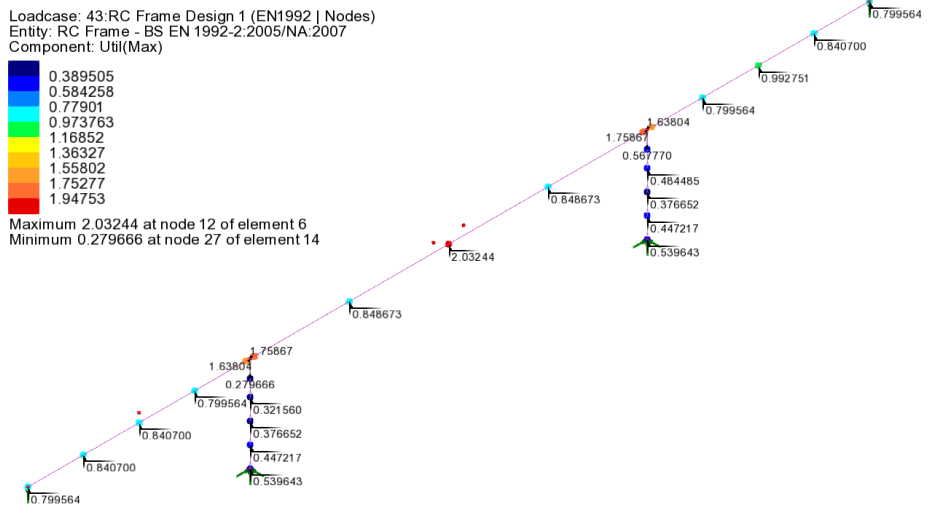


An RC Frame Design loadcase entry  named **RC Frame Design 1** will be added inside a Post processing / Design Check results folder structure in the  treeview.

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 EN1992-2:2005** 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 – BS EN1992-2:2005** and component **Util(max)** from the drop down lists. Select the **Values Display** tab and set the **Maxima** value to **100%** to see all values and click **OK** to finish. to see utilisation results for all members.




From the nodal results it can be seen that the utilisation values exceed 1.0 in the deck at both pier supports and also in the central span. Results for these locations will require further investigation. Utilisation values in the piers are less than 1.0.



Note. Because the design check locations were set to be **Averaged nodal**, 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.

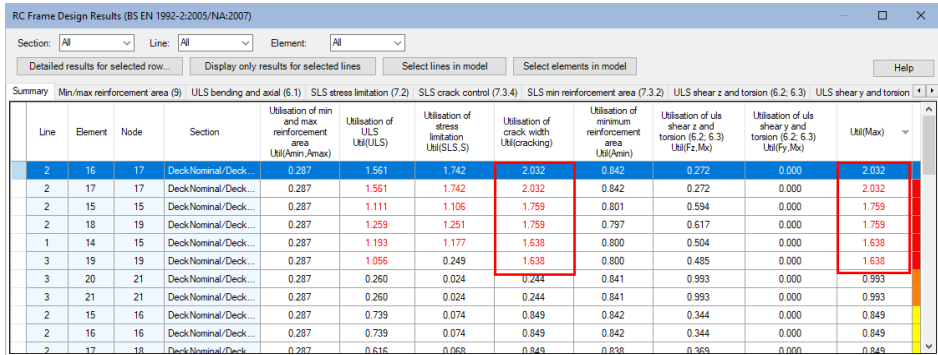
Viewing RC Frame Design Results

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, expand the table and click the **Util(Max)** label at the top of the last column to sort by Utilisation value in ascending descending value.



Line	Element	Node	Section	Utilisation of min and max reinforcement area Util(Amin,Amx)	Utilisation of ULS Util(ULS)	Utilisation of stress limitation Util(SLS.S)	Utilisation of crack width Util(cracking)	Utilisation of minimum reinforcement area Util(Amin)	Utilisation of uls shear z and torsion (6.2, 6.3) Util(Fz,Mx)	Utilisation of uls shear y and torsion (6.2, 6.3) Util(Fy,Mx)	Util(Max)
2	16	17	DeckNominal/Deck...	0.287	1.561	1.742	2.032	0.842	0.272	0.000	2.032
2	17	17	DeckNominal/Deck...	0.287	1.561	1.742	2.032	0.842	0.272	0.000	2.032
2	15	15	DeckNominal/Deck...	0.287	1.111	1.106	1.759	0.801	0.594	0.000	1.759
2	18	19	DeckNominal/Deck...	0.287	1.259	1.251	1.759	0.797	0.617	0.000	1.759
1	14	15	DeckNominal/Deck...	0.287	1.193	1.177	1.638	0.800	0.504	0.000	1.638
3	19	19	DeckNominal/Deck...	0.287	1.056	0.249	1.638	0.800	0.485	0.000	1.638
3	20	21	DeckNominal/Deck...	0.287	0.260	0.024	0.244	0.841	0.993	0.000	0.993
3	21	21	DeckNominal/Deck...	0.287	0.260	0.024	0.244	0.841	0.993	0.000	0.993
2	15	16	DeckNominal/Deck...	0.287	0.739	0.074	0.849	0.842	0.344	0.000	0.849
2	16	16	DeckNominal/Deck...	0.287	0.739	0.074	0.849	0.842	0.344	0.000	0.849
2	17	18	DeckNominal/Deck...	0.287	0.616	0.068	0.849	0.838	0.369	0.000	0.849

From the Summary table it can be seen that:

- For the deck sections the maximum values are for the ‘Utilisation of crack width Util(cracking)’ 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 would be seen for 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

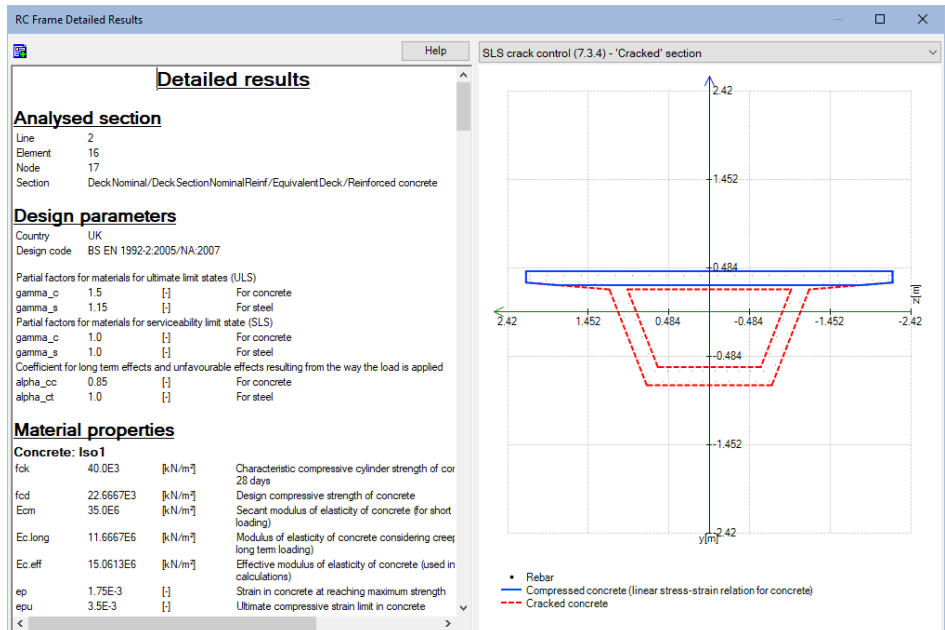
- On the main RC Frame Design Results summary table select the **SLS crack control (7.3.4)** tab.

- Expand the width of table and sort the **Utilisation of 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.

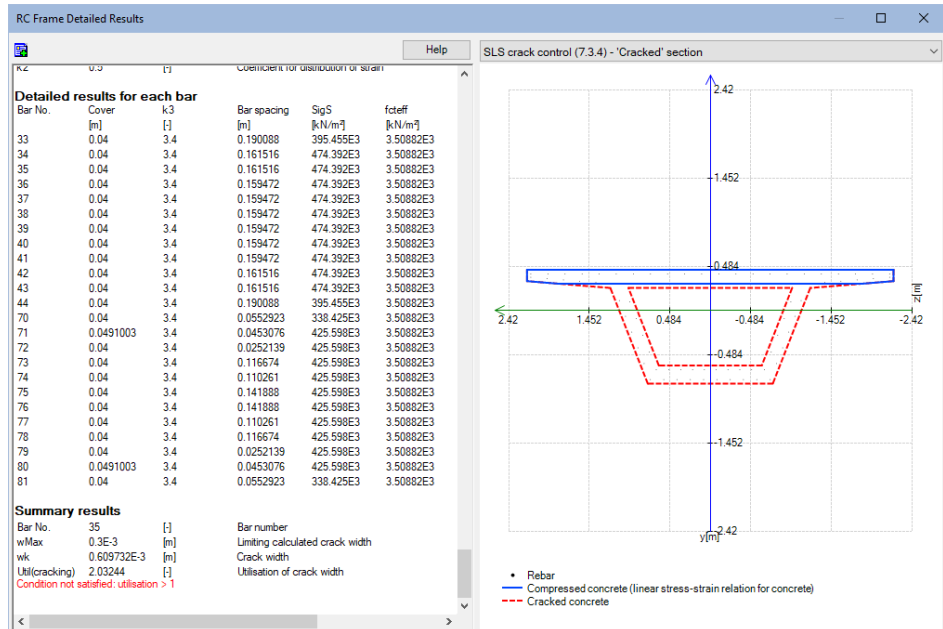
Line	Element	Node	Section	Loadcase	Primary component	Internal forces Fx(kN)	Internal forces My(kNm)	Internal forces Mz(kNm)	Internal forces Mx(kNm)	Limiting calculated crack width(in)	Crack width(in)	Utilitation of crack width Util(cracking)
2	16	17	DeckNominal/Deck...	SLS_OP_S	Fmax.My,Mz	-24.6325	-1.46133E3	0.0	1.46133E3	0.3E-3	0.609732E-3	2.032
2	16	17	DeckNominal/Deck...	SLS_OP_S	Fmin.My,Mz	-24.6325	-1.46133E3	0.0	1.46133E3	0.3E-3	0.609732E-3	2.032
2	16	17	DeckNominal/Deck...	SLS_OP_S	Fx.Mymax,Mz	-24.6325	-1.46133E3	0.0	1.46133E3	0.3E-3	0.609732E-3	2.032
2	16	17	DeckNominal/Deck...	SLS_OP_S	Fx.Mymin,Mz	-24.6325	-1.46133E3	0.0	1.46133E3	0.3E-3	0.609732E-3	2.032
2	16	17	DeckNominal/Deck...	SLS_OP_S	Fx.My,Mtmax	-24.6325	-1.46133E3	0.0	1.46133E3	0.3E-3	0.609732E-3	2.032
2	16	17	DeckNominal/Deck...	SLS_OP_S	Fx.My,Mtmin	-24.6325	-1.46133E3	0.0	1.46133E3	0.3E-3	0.609732E-3	2.032
2	17	17	DeckNominal/Deck...	SLS_OP_S	Fmax.My,Mz	-24.6325	-1.46133E3	0.0	1.46133E3	0.3E-3	0.609732E-3	2.032
2	17	17	DeckNominal/Deck...	SLS_OP_S	Fmin.My,Mz	-24.6325	-1.46133E3	0.0	1.46133E3	0.3E-3	0.609732E-3	2.032
2	17	17	DeckNominal/Deck...	SLS_OP_S	Fx.Mymax,Mz	-24.6325	-1.46133E3	0.0	1.46133E3	0.3E-3	0.609732E-3	2.032
2	17	17	DeckNominal/Deck...	SLS_OP_S	Fx.Mymin,Mz	-24.6325	-1.46133E3	0.0	1.46133E3	0.3E-3	0.609732E-3	2.032
2	17	17	DeckNominal/Deck...	SLS_OP_S	Fx.My,Mtmax	-24.6325	-1.46133E3	0.0	1.46133E3	0.3E-3	0.609732E-3	2.032
2	17	17	DeckNominal/Deck...	SLS_OP_S	Fx.My,Mtmin	-24.6325	-1.46133E3	0.0	1.46133E3	0.3E-3	0.609732E-3	2.032
2	15	15	DeckNominal/Deck...	SLS_OP_S	Fmax.My,Mz	-24.6325	1.92158E3	0.0	1.92158E3	0.3E-3	0.527601E-3	1.759
2	15	15	DeckNominal/Deck...	SLS_OP_S	Fmin.My,Mz	-24.6325	1.92158E3	0.0	1.92158E3	0.3E-3	0.527601E-3	1.759

Viewing detailed results for a deck location

- Select the first row 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 most of the bars to be in faces 6 and 11 (the lower and upper faces of the bottom of the box section).

- Close the Detailed Results dialog.

Viewing detailed results for a pier location

- On the RC Frame Results dialog select the **Summary** tab
- Click the **Util(max)** column to sort the values in descending order

RC Frame Design Results (BS EN 1992-2:2005/NA:2007)

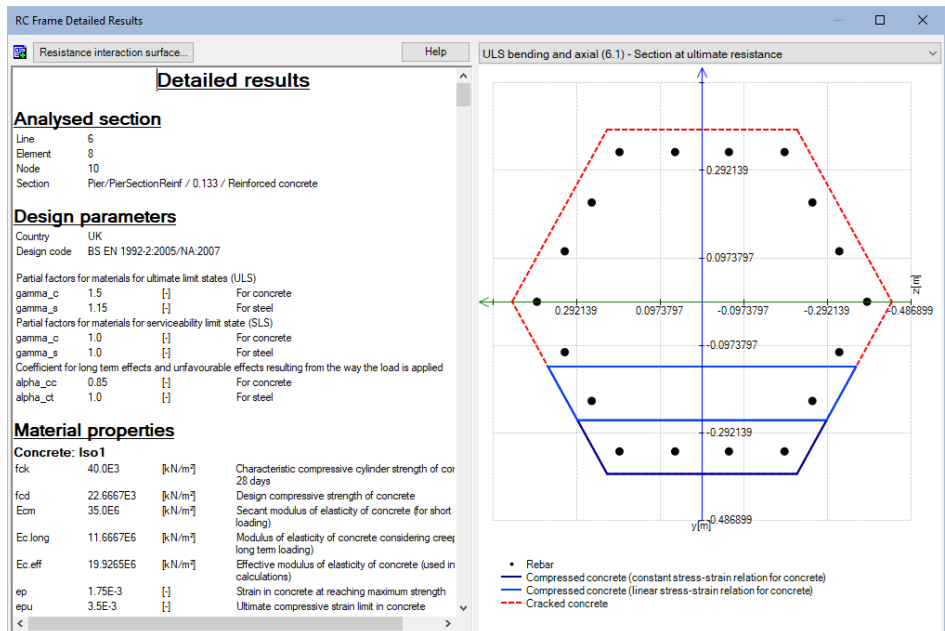
Section: All Line: All Element: All

Detailed results for selected row... Display only results for selected lines... Select lines in model... Select elements in model... Help

Line	Element	Node	Section	Utilisation of min and max reinforcement area (Util(Amin, Amax))	Utilisation of ULS (Util(ULS))	Utilisation of stress limitation (Util(SLS.S))	Utilisation of crack width (Util(cracking))	Utilisation of minimum reinforcement area (Util(Amin))	Utilisation of Uls shear z and torsion (6.2, 6.3) Util(Fz, Mz)	Utilisation of Uls shear y and torsion (6.2, 6.3) Util(Fy, My)	Util(Max)
3	22	22	DeckNominal/Deck...	0.287	0.415	0.043	0.493	0.841	0.293	0.000	0.841
1	11	11	DeckNominal/Deck...	0.287	0.000	0.000	0.000	0.800	0.210	0.000	0.800
1	13	14	DeckNominal/Deck...	0.287	0.409	0.097	0.494	0.800	0.353	0.000	0.800
1	14	14	DeckNominal/Deck...	0.287	0.409	0.097	0.494	0.800	0.353	0.000	0.800
3	19	20	DeckNominal/Deck...	0.287	0.306	0.072	0.494	0.800	0.334	0.000	0.800
3	20	20	DeckNominal/Deck...	0.287	0.306	0.072	0.494	0.800	0.334	0.000	0.800
3	22	23	DeckNominal/Deck...	0.287	0.000	0.000	0.000	0.800	0.205	0.000	0.800
6	8	10	Pier/PierSectionRei...	0.278	0.568	0.555	0.005	0.122	0.265	0.000	0.568
5	1	1	Pier/PierSectionRei...	0.540	0.000	0.183	0.000	0.055	0.172	0.000	0.540
6	2	2	Pier/PierSectionRei...	0.540	0.000	0.183	0.000	0.055	0.363	0.000	0.540
6	8	8	Pier/PierSectionRei...	0.322	0.477	0.484	0.000	0.100	0.276	0.000	0.484
6	6	8	Pier/PierSectionRei...	0.322	0.477	0.484	0.000	0.100	0.276	0.000	0.484
5	3	3	Pier/PierSectionRei...	0.447	0.100	0.182	0.000	0.067	0.162	0.000	0.447
5	1	3	Pier/PierSectionRei...	0.447	0.100	0.182	0.000	0.067	0.162	0.000	0.447
6	4	4	Pier/PierSectionRei...	0.447	0.208	0.307	0.000	0.068	0.312	0.000	0.447

- Scroll down the table and select the first entry for the pier (this should be Line 6 Element 8 Node 10) 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 in the top-right of the dialog.

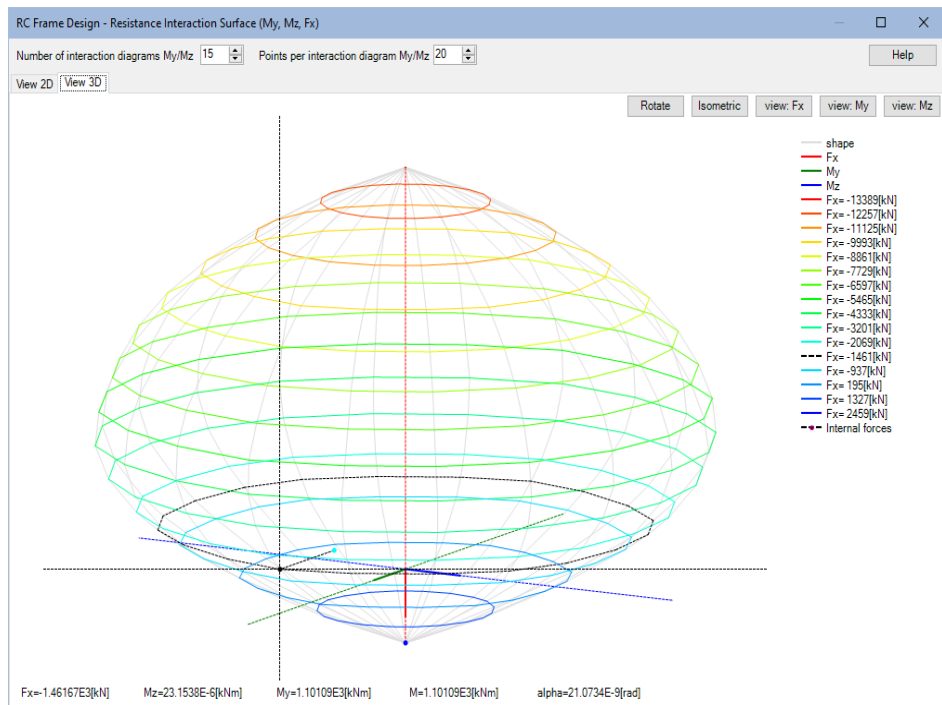
Interaction surface diagrams – for pier

For ULS results, interaction surface diagrams can be produced.

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

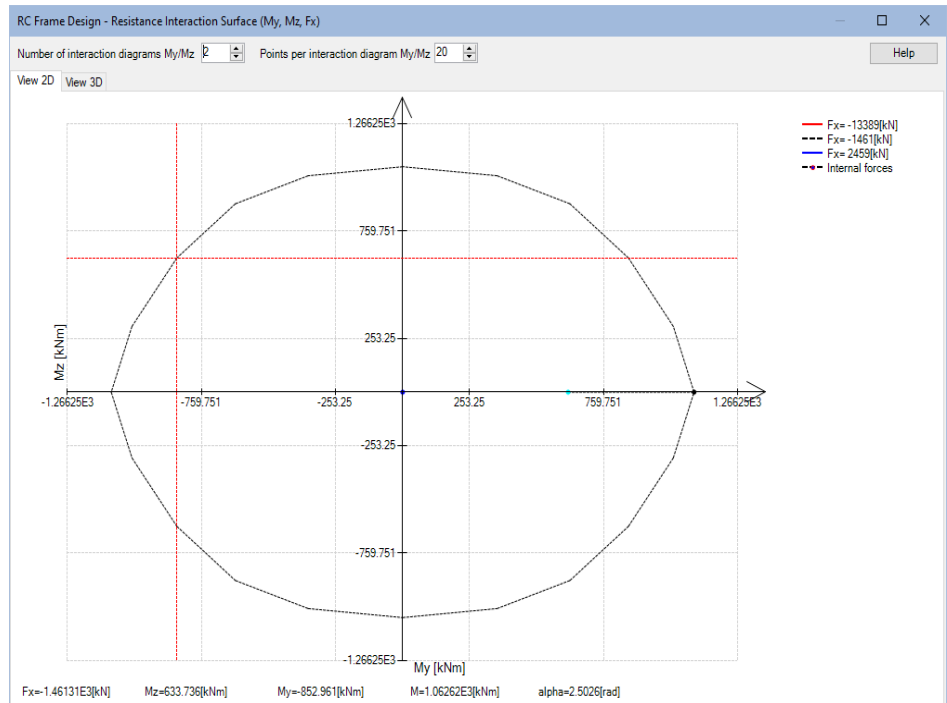
The interaction surface diagram for axial load F_x and moments M_y and M_z is displayed. By default a 2D view is shown. The black dotted line shows the M_y M_z contour for the applied axial force.

- 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.

- 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 need to be increased generally from 12mm diameter to 16mm diameter, and 20mm diameter bars will be used in the bottom two faces of the box section



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**.

Face Data
Layers of rebar
Additional bars

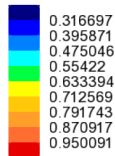
100%

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

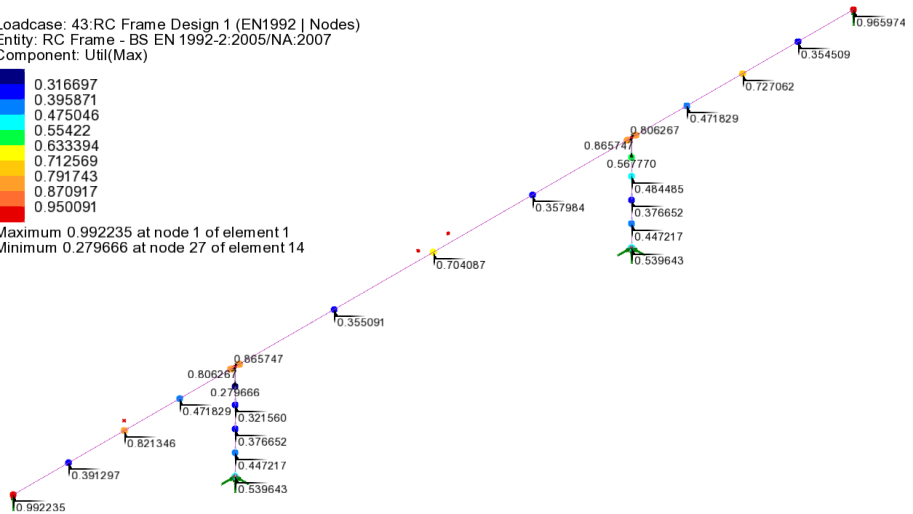
Convert row to additional bars
Sort
Add
Delete

The RC Frame design results will be re-calculated as shown below:

Loadcase: 43:RC Frame Design 1 (EN1992 | Nodes)
 Entity: RC Frame - BS EN 1992-2:2005/NA:2007
 Component: Util(Max)




Maximum 0.992235 at node 1 of element 1
 Minimum 0.279666 at node 27 of element 14



Utilisation values are now all less than 1.0 with the largest value of **0.992** 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 Averaged nodal to **Internal points**
- Change the name to **RC Frame Design 2** and click **OK**.



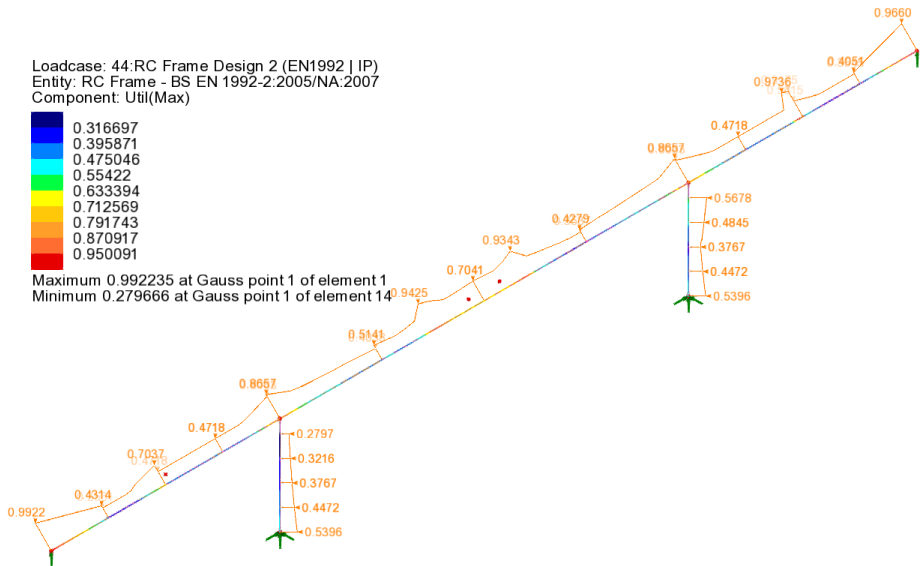
Note. This copies and creates a new loadcase (as opposed to creating a new one from scratch) and saves having to re-define the Load Groups.

- 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 display of 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.

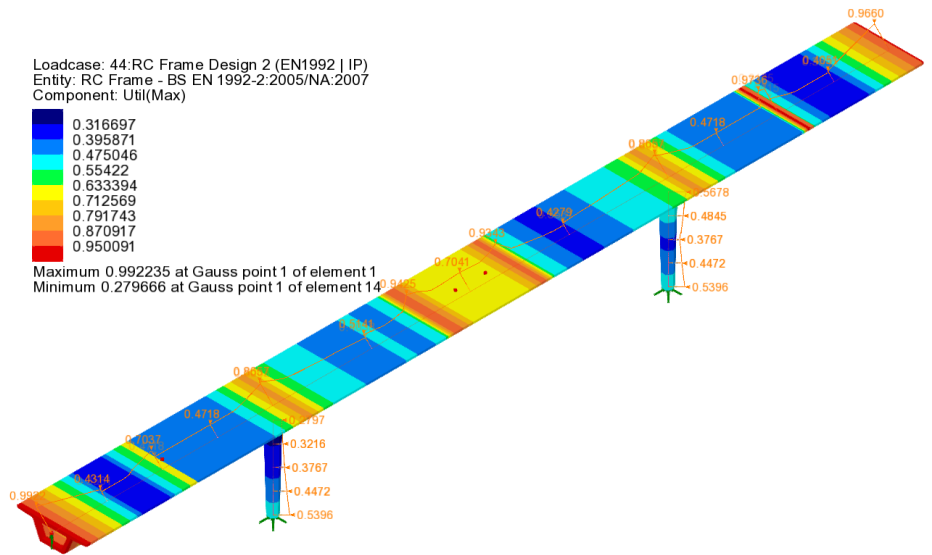


From this diagram the peak Utilisation value in the deck can be seen to be **0.992** as previously obtained.



Note. When plotting diagrams of results such as 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)**.

RC Frame Design Results (BS EN 1992-2:2005/NA:2007)

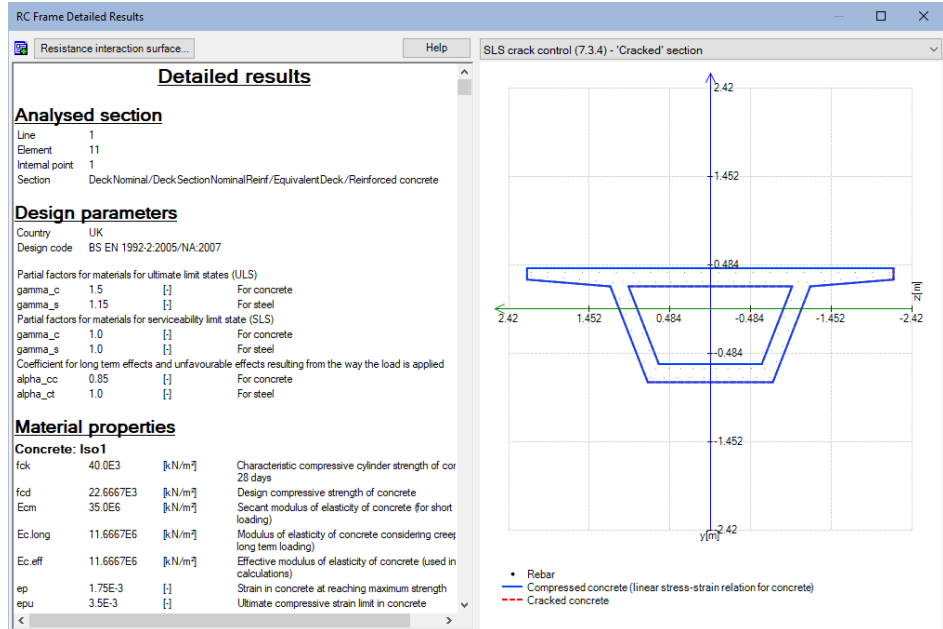
Section: All Line: All Element: All



Summary | Min/max reinforcement area (6.1) | ULS bending and axial (6.1) | SLS stress limitation (7.2) | SLS crack control (7.3.4) | SLS min reinforcement area (7.3.2) | ULS shear z and torsion (6.2, 6.3) | ULS sheary and torsion (6.2, 6.3) | Util(Max)

Line	Element	Internal point	Section	Utilisation of min and max reinforcement area Util(Amin,Amx)	Utilisation of ULS Util(ULS)	Utilisation of stress limitation Util(SLS,S)	Utilisation of crack width Util(SLS,S)	Utilisation of minimum reinforcement area Util(Amin)	Utilisation of uls shear z and torsion (6.2, 6.3) Util(Fz,Mx)	Utilisation of uls shear y and torsion (6.2, 6.3) Util(Fy,Mx)	Util(Max)
1	11	1	DeckNominal/Deck...	0.321	0.000	0.000	0.000	0.472	0.992	0.000	0.992
3	20	10	DeckNominal/Deck...	0.321	0.087	0.016	0.054	0.355	0.974	0.000	0.974
3	22	11	DeckNominal/Deck...	0.321	0.000	0.000	0.000	0.472	0.966	0.000	0.966
2	16	6	DeckNominal/Deck...	0.321	0.557	0.577	0.525	0.355	0.943	0.000	0.943
1	11	2	DeckNominal/Deck...	0.321	0.025	0.006	0.024	0.355	0.936	0.000	0.936
2	17	5	DeckNominal/Deck...	0.321	0.570	0.589	0.559	0.354	0.934	0.000	0.934
3	20	11	DeckNominal/Deck...	0.321	0.111	0.022	0.075	0.355	0.918	0.000	0.918
3	22	10	DeckNominal/Deck...	0.321	0.024	0.006	0.024	0.355	0.910	0.000	0.910
1	11	3	DeckNominal/Deck...	0.321	0.048	0.011	0.046	0.355	0.880	0.000	0.880
2	18	11	DeckNominal/Deck...	0.321	0.737	0.718	0.866	0.471	0.617	0.000	0.866
2	15	1	DeckNominal/Deck...	0.321	0.651	0.241	0.866	0.473	0.594	0.000	0.866

RC Frame Design to EN 1992-2

- 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 (7.3.4) – ‘Cracked’ Section**



- Click the **Add to report**  button at the top of the dialog.
- Choose **Yes** 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



Save the model file.

File
Save

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 longitudinally throughout the deck section, and with 20mm diameter bars in the lowest two faces of the box section.

If additional refinement of the reinforcement arrangement is required this may be achieved by adding more bars to a layer, 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

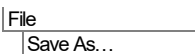
Two methods are available:

- [Option 1](#): Assigning a single geometric line attribute referencing all required section reinforcement to all lines representing 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


With the previously solved `rc_frame_bridge` model loaded:



Save the model as `rc_frame_bridge_optimise_option_1`

Define section reinforcement

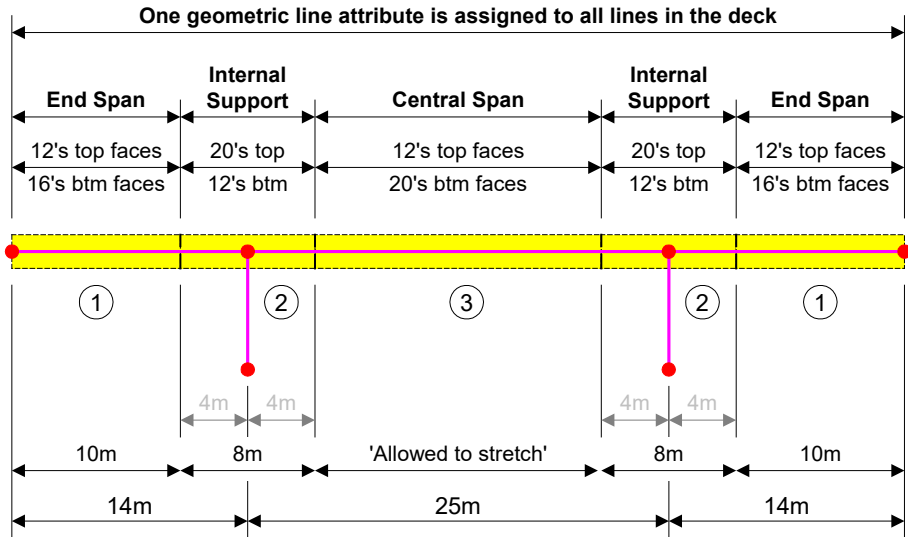
For this option separate section reinforcement utilities need to be created for each 'region' of the deck that is to be reinforced, and be given suitable names that specify their location, such as **EndSpan**, **InternalSupport**, and **CentralSpan**.

This can be done by going to the Utilities  treeview and editing the **DeckSectionNominalReinf** section utility, editing the layers or rebar details and changing the name to be **DeckSectionEndSpanReinf**, then repeating for the **DeckSectionInternalSupportReinf** and **DeckSectionCentralSpanReinf** arrangements and names required.

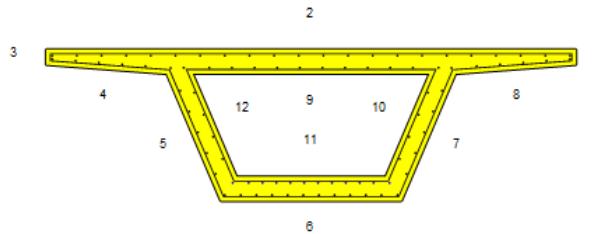
The following image shows a side elevation of the bridge and a typical scenario where different reinforcing bar arrangements can be specified for hogging at supports or

RC Frame Design to EN 1992-2

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.

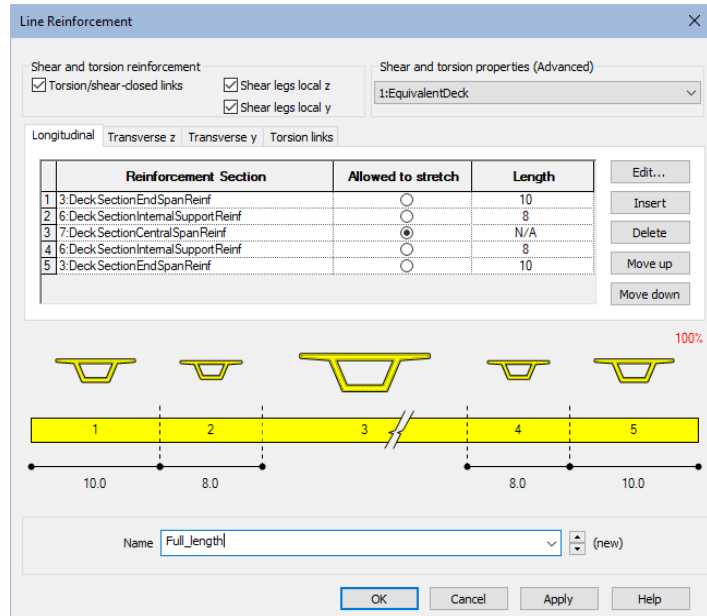


Faces	Bar diameters required		
	End Span	Internal Support	Central Span
2 and 9	12	20	12
6 and 11	16	12	20
All other faces	12	12	12

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.

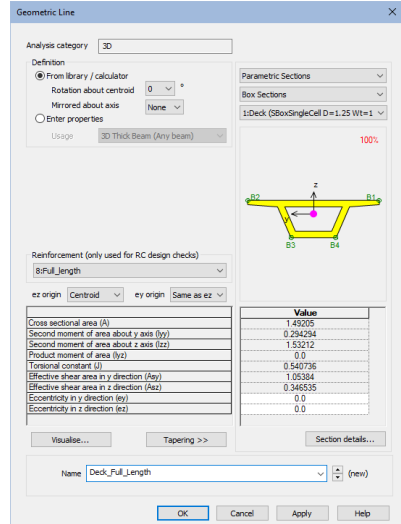


Note that the central span region is un-dimensioned and is 'Allowed to stretch' to fit the length of the deck that remains between the support regions.

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

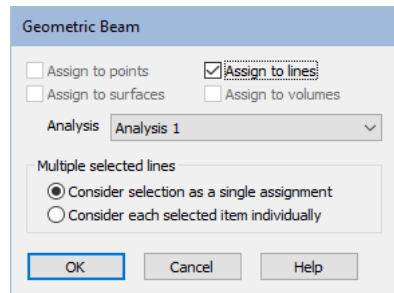
Geometric line attribute

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.

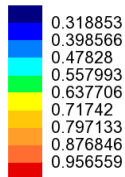


Select the **Solve Now** button and click **OK** to run the analysis.

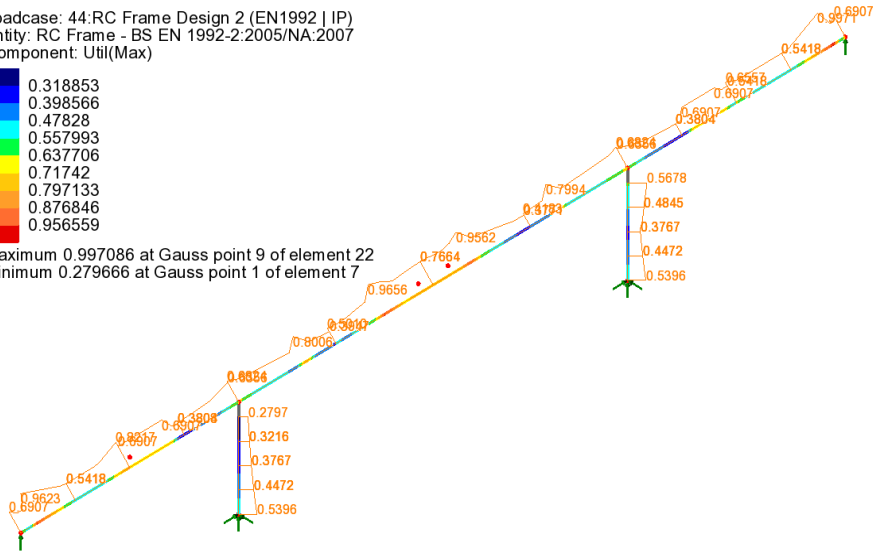
Viewing results of Util(Max)

Results for loadcase RC Frame Design 2 are shown:

Loadcase: 44:RC Frame Design 2 (EN1992 | IP)
 Entity: RC Frame - BS EN 1992-2:2005/NA:2007
 Component: Util(Max)



Maximum 0.997086 at Gauss point 9 of element 22
 Minimum 0.279666 at Gauss point 1 of element 7



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

With the previously solved `rc_frame_bridge` model loaded:

File
 Save As...



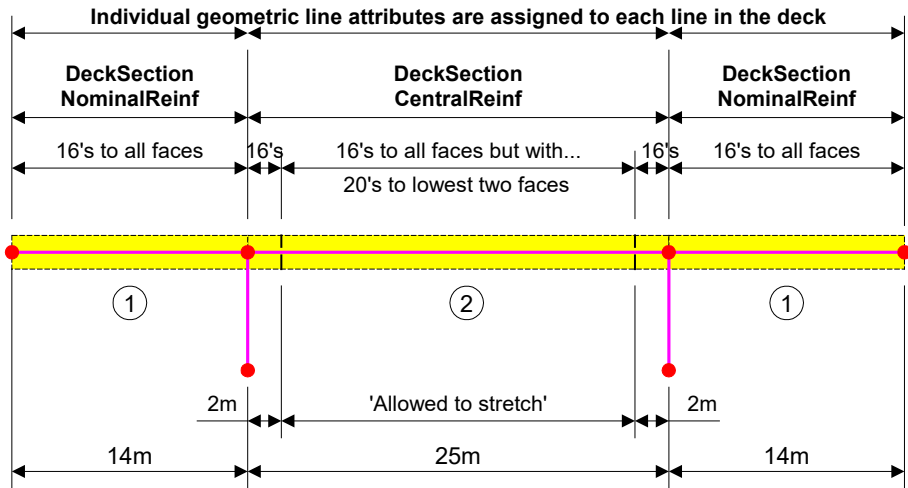
Save the model as `rc_frame_bridge_optimise_option_2`

Section reinforcement

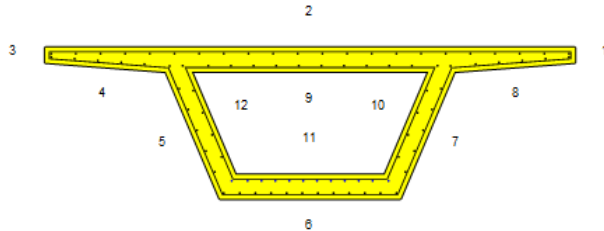
For this option separate section reinforcement utilities need to be created for each ‘member length’ in the deck that is to be reinforced and given names such as: **DeckSectionNominalReinf** and **DeckSectionCentralReinf**.

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

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

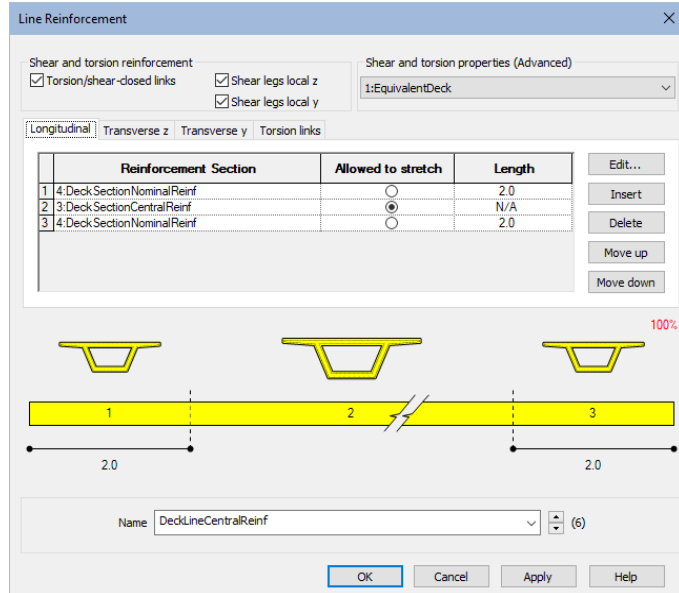
Line reinforcement dialog

A new line reinforcement utility called **DeckLineCentralReinf** is required for the central span.

This can be done by editing the **DeckLineNominalReinf** section utility, editing the reinforcement bar details as shown below, then typing a new name of **DeckLineCentralReinf**

Discussion on optimising the reinforcement in the deck

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, which has 20mm diameter bars in the two lowest faces in this span.



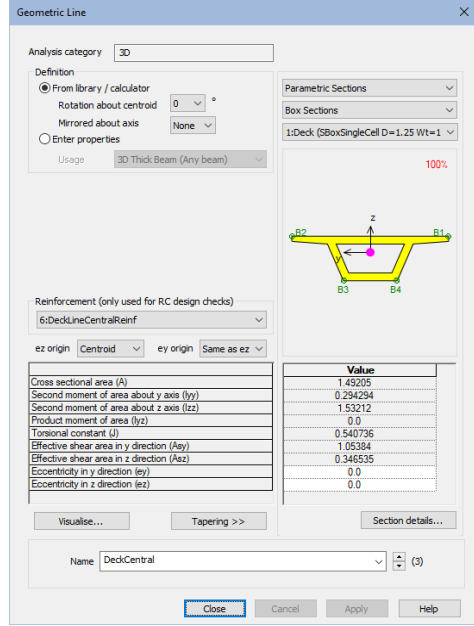
The central region is un-dimensioned and would be 'Allowed to stretch' to fit the remainder of the length of this span.

Geometric line attribute

The existing geometric line attribute for the deck **DeckNominal (SBox...)** should be edited and renamed to be **DeckCentral** and then be further edited to reference the **DeckLineCentralReinf** line reinforcement, as per the image shown right.

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

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

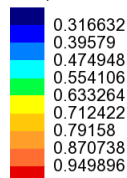


Select the **Solve Now** button and click **OK** to run the analysis.

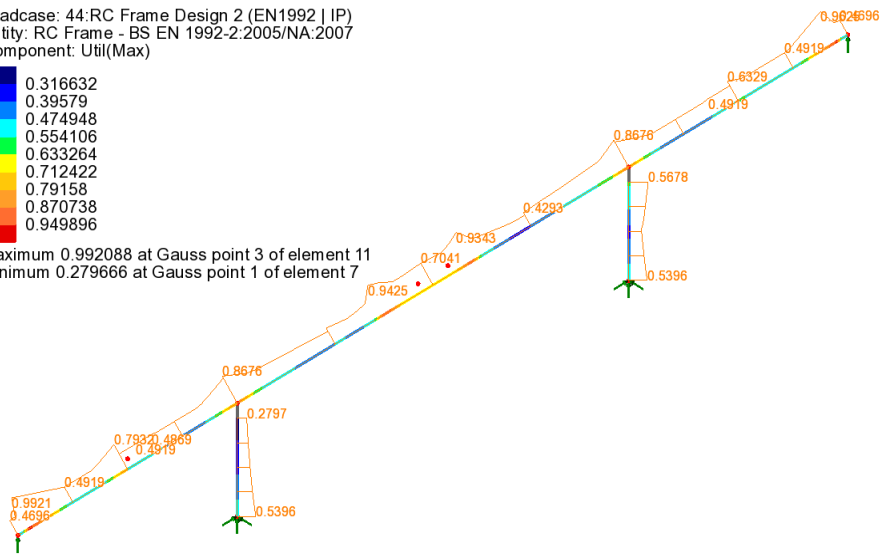
Viewing results of Util(Max)

Results for loadcase RC Frame Design 2 follow:

Loadcase: 44:RC Frame Design 2 (EN1992 | IP)
Entity: RC Frame - BS EN 1992-2:2005/NA:2007
Component: Util(Max)



Maximum 0.992088 at Gauss point 3 of element 11
Minimum 0.279666 at Gauss point 1 of element 7



Summary

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

