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 exceed	ds 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

\mathbf{T}

- □ **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

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.

- 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**

File Save As...

File

Open..

• Click **OK**

Feature Geometry

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

This completes the geometry definition for the bridge.

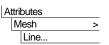
Select the isometric button to view the frame in 3D.

Meshing

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

- 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 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.



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

Geometry Line > By Sweeping...

Attributes

- Specify a rigid zone length of **0.8** and for the End of line <u>un-</u> <u>tick</u> 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**.

Line Mesh End Conditions	×
End releases Rigid zone	
Start of line	End of line
None	None
Specified length 0.8	Specified length 1.0
© Beam / column Specify	◎ Beam / column Specify
Rigid zone factor 1.0	Rigid zone factor 1.0
Include mass	Include mass
OK	Cancel Help

• 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

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

Tools	
Section Property	
Calculator	>
Box Section	

-	: imple section iomplex section		Exclude void	Ì	,	
Cell (details Number of cell	Is	1 ×		t yt	t 7/7
Box :	section dimension	ons			ŧ	++
н	1.25	Hm	0.0			<u> </u>
Wb	0.75	ть	0.2		b yl	/
Wt	1.2	Tt	0.2		, y	
W	0.0	Tw	0.2			
Cant	ilever slab dime	ensions		I+-₩b-→		
Wc	1.0	Tc1	0.2	Extra points locations	Calcu	lated prop
		Tc2	0.125		A	1.49205
Num	ber of extra poi	inte			Ixx	0.29429
- North	Top slab	110	0		Іуу	1.53212
	Bottom slab		0		Ixy	0.0
	Cantilever slab		0 ^		J [0.540736
	Web interior fa	ace	0		Asy	0.34653
	Web exterior f	face	0		Asx	1.05384
					yt	0.44918
ame	Deck			Load section Clear inputs		0.800813

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		
Secti	ion Property	
Calc	ulator	>
Mo	ore 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 sections before they can be assigned to the model.

Adding deck section to the Treeview

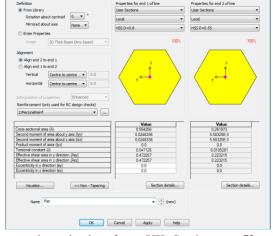
The standard sections dialog will appear.

- 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:

- 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.

Attributes Geometric > Section Library...

Attributes Geometric

Tapered

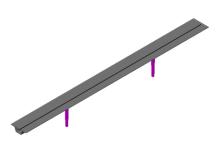
Section...

>

- 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 streeview onto them.



Material Properties

A	ttri	butes	
	N	laterial	>
		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 Streeview.
- Select the whole model and drag and drop the material attribute Iso1 (C40/50 Concrete EN1992-2:2005) from the selected Surfaces.

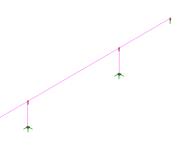
Supports

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.

8

- 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 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

- 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

- 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.

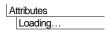
Service vehicle load

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...

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 <u>unchecked</u>. 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).

	3D					
	Ur	ntrans	formed loa	direction	Projection vector	or
Arbitrary	0	X	© XYZ		Project in lo	ad direction
⊙ Grid x 0	0	Y	Surface	e normal	Project for p	prestress
y O			0		X component	0.0
	C	Z			Y component	0.0
					Z component	1.0
X		Y		Z		Load
0.0		0.0		0.0		108.0

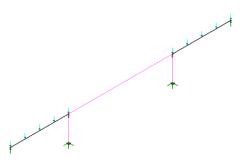
- Click on row 2 of the table and press the **Delete** key to remove this row.
- Change the name to **Live Horizontal** and click **Finish**.

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 \bigcirc 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 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 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 selection. Ensure that the **Project onto line** option is chosen and the loadcase **Central span gr1** is selected. Click **OK**.

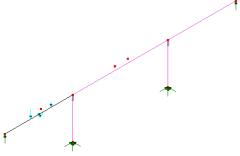
Service vehicle load

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 payement (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) = 0.55 m 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 do 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, the Live



- and drop drag Horizontal load from the streeview and click OK.
- Select the point at (25.55,0,6.55) and drag and drop the Accidental Vehicle load attribute from the streeview. 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 streeview. 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 $\sqrt[n]{}$ 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 streeview 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)

Section Reinforcer	nent			×
	0.75/ HSS D=0.5)	•		
Face Data Laye	rs of rebar Additional bars	8	*	100%
Row 1	Face Al	Actual cover to link 0.03	Nominal cover to bars Actual	Allowance for link 0.01
				Add Delete
	Name PierSectio	nReinf	•	(1)
			Close Cancel	Apply Help



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.

2.Pier (h	ISS D=0.8/HS	S D=0.55)			•				
Zirici (i	155 0 - 0.07 115	0.00			-				
ace Data	Layers of re	abar Additiona	al bars						
					6				1
				2)			
					3				
Row	Face	Layer	No. of Bars	Gap	3 Bar diameter	Alternate	End bars	Start distance	End distar
1	1	Layer 1	No. of Bars	Gap 0.0	Bar diameter		End bars Omit end		End distar
1	1 2	1	4	0.0	Bar diameter 0.02 0.02	bars Same dia Same dia	Omit end Omit end	distance Auto Auto	Auto Auto
1	1	1	4 4 4	0.0 0.0 0.0	Bar diameter 0.02 0.02 0.02 0.02	bars Same dia Same dia Same dia	Omit end Omit end Omit end	distance Auto Auto Auto	Auto Auto Auto
1 2 3 4	1 2 3 4	1 1 1 1	4	0.0 0.0 0.0 0.0 0.0	Bar diameter 0.02 0.02 0.02 0.02 0.02 0.02	bars Same dia Same dia Same dia Same dia	Omit end Omit end Omit end Omit end	distance Auto Auto Auto Auto	Auto Auto Auto Auto
1 2 3 4 5	1 2 3 4 5	1 1 1 1 1	4 4 4 4 4	0.0 0.0 0.0 0.0 0.0 0.0	Bar diameter 0.02 0.02 0.02 0.02 0.02 0.02 0.02 0.0	bars Same dia Same dia Same dia Same dia Same dia	Omit end Omit end Omit end Omit end Omit end	distance Auto Auto Auto Auto Auto Auto	Auto Auto Auto Auto Auto
1 2 3 4	1 2 3 4	1 1 1 1	4 4 4 4	0.0 0.0 0.0 0.0 0.0	Bar diameter 0.02 0.02 0.02 0.02 0.02 0.02	bars Same dia Same dia Same dia Same dia	Omit end Omit end Omit end Omit end	distance Auto Auto Auto Auto	Auto Auto Auto Auto
1 2 3 4 5	1 2 3 4 5	1 1 1 1 1	4 4 4 4 4	0.0 0.0 0.0 0.0 0.0 0.0	Bar diameter 0.02 0.02 0.02 0.02 0.02 0.02 0.02 0.0	bars Same dia Same dia Same dia Same dia Same dia	Omit end Omit end Omit end Omit end Omit end	distance Auto Auto Auto Auto Auto Auto	Auto Auto Auto Auto Auto
1 2 3 4 5	1 2 3 4 5	1 1 1 1 1	4 4 4 4 4	0.0 0.0 0.0 0.0 0.0 0.0	Bar diameter 0.02 0.02 0.02 0.02 0.02 0.02 0.02 0.0	bars Same dia Same dia Same dia Same dia Same dia	Omit end Omit end Omit end Omit end Omit end	distance Auto Auto Auto Auto Auto Auto	Auto Auto Auto Auto Auto

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

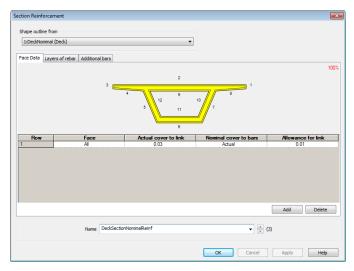
Line Reinforcement			×
Reinforcement Section	Allowed to stretch	Length	Edit
1 1:PierSectionReinf	۲	N/A	Insert
			Delete
			Move up
			Move down
]			
	\bigcirc		100%
	1 7		
Name PierLineReinf		• • (new)
	OK Canc	el Apply	Help

• 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:

	Vominal (Deck)				•					
Face Data	a Layers of r	ebar Additiona	l bars							
										10
					2					
			3				1			
			4		9		5			
					2 1	° // // _				
				5	11	/// ⁷				
					/					
					6					
					8					
						Alternate		Start	End	Т
Row	Face	Layer	No. of Bars	Gap	Bar diameter	bars	End bars	distance	distance	I
Row 1	2	1	22	0.0	0.012	bars Same dia	Same dia	distance Auto	distance Auto	
1	2	1	22 6	0.0	0.012	bars Same dia Same dia	Same dia Same dia	distance Auto Auto	distance Auto Auto	
1	2	1	22	0.0	0.012	bars Same dia	Same dia	distance Auto	distance Auto	
1 2 3 4	2 4 8 5	1	22 6 6 7	0.0	0.012	bars Same dia Same dia	Same dia Same dia	distance Auto Auto	distance Auto Auto	and and and and
1 2 3	2 4 8	1	22 6 6	0.0 0.0 0.0	0.012 0.012 0.012	bars Same dia Same dia Same dia	Same dia Same dia Same dia	distance Auto Auto Auto	distance Auto Auto Auto	
1 2 3 4	2 4 8 5	1 1 1 1	22 6 6 7	0.0 0.0 0.0 0.0	0.012 0.012 0.012 0.012 0.012	bars Same dia Same dia Same dia Same dia	Same dia Same dia Same dia Omit both	distance Auto Auto Auto Auto	distance Auto Auto Auto Auto	and and and and and and
1 2 3 4 5	2 4 8 5 7	1 1 1 1 1 1	22 6 6 7 7 7	0.0 0.0 0.0 0.0 0.0 0.0	0.012 0.012 0.012 0.012 0.012 0.012	bars Same dia Same dia Same dia Same dia Same dia	Same dia Same dia Same dia Omit both Omit both	distance Auto Auto Auto Auto Auto	distance Auto Auto Auto Auto Auto	and and and and and and and
1 2 3 4 5 6 7 8	2 4 8 5 7 6	1 1 1 1 1 1 1	22 6 7 7 10	0.0 0.0 0.0 0.0 0.0 0.0 0.0	0.012 0.012 0.012 0.012 0.012 0.012 0.012	bars Same dia Same dia Same dia Same dia Same dia Same dia	Same dia Same dia Same dia Omit both Omit both Same dia	distance Auto Auto Auto Auto Auto Auto	distance Auto Auto Auto Auto Auto Auto	and and and and and and and
1 2 3 4 5 6 7	2 4 8 5 7 6 9	1 1 1 1 1 1 1 1 1	22 6 7 7 10 12	0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0	0.012 0.012 0.012 0.012 0.012 0.012 0.012 0.012	bars Same dia Same dia Same dia Same dia Same dia Same dia	Same dia Same dia Same dia Omit both Omit both Same dia Same dia	distance Auto Auto Auto Auto Auto Auto Auto	distance Auto Auto Auto Auto Auto Auto Auto	and and and and and and and and
1 2 3 4 5 6 7 8	2 4 8 5 7 6 9 11	1 1 1 1 1 1 1 1 1 1	22 6 7 7 10 12 10	0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0	0.012 0.012 0.012 0.012 0.012 0.012 0.012 0.012 0.012	bars Same dia Same dia Same dia Same dia Same dia Same dia Same dia Same dia	Same dia Same dia Same dia Omit both Omit both Same dia Same dia Same dia	distance Auto Auto Auto Auto Auto Auto Auto Auto	distance Auto Auto Auto Auto Auto Auto Auto Auto	and and and and and and and and
1 2 3 4 5 6 7 8	2 4 8 5 7 6 9 11	1 1 1 1 1 1 1 1 1 1	22 6 7 7 10 12 10	0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0	0.012 0.012 0.012 0.012 0.012 0.012 0.012 0.012 0.012 0.012	bars Same dia Same dia Same dia Same dia Same dia Same dia Same dia Same dia	Same dia Same dia Same dia Omit both Omit both Same dia Same dia Same dia	distance Auto Auto Auto Auto Auto Auto Auto Auto	distance Auto Auto Auto Auto Auto Auto Auto Auto	and and and and and and and and
1 2 3 4 5 6 7 8	2 4 8 5 7 6 9 11		22 6 7 7 10 12 10	0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0	0.012 0.012 0.012 0.012 0.012 0.012 0.012 0.012 0.012 0.012	bars Same dia Same dia Same dia Same dia Same dia Same dia Same dia Same dia	Same dia Same dia Same dia Omt both Omt both Same dia Same dia Same dia Same dia Same dia Same dia	distance Auto Auto Auto Auto Auto Auto Auto Auto	distance Auto Auto Auto Auto Auto Auto Auto Auto	and and and and and and and and

- 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.

RC Material	3
Design code EN1992	•
Section	
Material	
Iso1 (C40/50 Concrete EN1992-2:2005)	
Reinforcement	
Material	
Iso2 (500B Steel - Reinforcing bar EN1992-2:2005)	-
	-
Name Reinforced Concrete EN1992	
OK Cancel Apply Help	

Attri	butes	
M	laterial	>
_	Concrete	
	(Reinforced)	

• 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 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.

E 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.

• Add the	loadcases	End spans	gr1 and	Central	span	gr1	to the	comt	ination.
For both	loadcases	change the	e Benefic	ial 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.

A	nalyses
	Envelope

Combination...

Analyses Smart

- 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

- 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).

Analyses Smart Combination... • For each included loadcase, edit the **Beneficial** and **Adverse** factors to be as shown in the dialog that follows:

				Variab	le loadcases	-1
Available			Includ	ed		
	*]	ID	Name	Beneficial factor	Adverse factor
			1	End spans SW	0.95	1.35
- 7:End span gr2			2	Central span SW	0.95	1.35
- 8:Pier gr2			3	End spans SDL	0.95	1.2
9:Central span gr2			4	Central span SDL	0.95	1.2
			10	gr1 combination (Max)	0	1
Post processing		>>	11	gr1 combination (Min)	0	1
10:gr1 combination (Max)			12	gr2 envelope (Max)	0	1.35
 11:gr 1 combination (Min) 16:SLS characteristic (Max) 	=	<<	13	gr2 envelope (Min)	0	1.35
- 17:SLS characteristic (Min) - 18:SLS QP (Max) - 19:SLS QP (Min) - 12:gr2 envelope (Max) - 13:gr2 envelope (Min)	-	Step	•	m		
Name ULS combination	1				▼ ▲ (new)	

• 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:

						ases to consider de loadcases	0
ailable 4:Central spa	an SDI		1	Include		Beneficial	Adverse
5:End spans				ID	Name	factor	factor
6:Central spa				1	End spans SW	1	1
7:End span o				2	Central span SW	1	1
8:Pier gr2	-	_		3	End spans SDL	1	1
9:Central sp	an or2			4	Central span SDL	1	1
- Post processing				12	gr2 envelope (Max)	0	1
- 10:gr 1 combi	ination (Max)		>>	13	gr2 envelope (Min)	0	
11:gr 1 combi				5	End spans gr1 Central span gr1	0	
	vination (Min) Max) Min) lope (Max)	4	Step 1	•	m		

• 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:

						to consider	0
Available			Includ	ed	Variable lo	adcases	-1
- 4:Central span SDL - 5:End spans gr1 - 6:Central span gr1 - 7:End span gr2 - 8:Pier gr2 - 9:Central span gr2 - 9:Central span gr2 - 0:org1 combination (Max) - 11:gr1 combination (Min) - 14:LUS combination (Max) - 15:LUS combination (Mix) - 15:SLS characteristic (Mix) - 15:SLS characteristic (Mix)	E	>>	ID 1 2 3 4	Name End spans SW Central span SW End spans SDL Central span SDL		Beneficial factor 1 1 1 1 1	Adverse factor 0 0 0
12:gr2 envelope (Max) 13:gr2 envelope (Min) Name SLS QP	-	Step 1 ×	•	m	.	<u> </u>	

• 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.

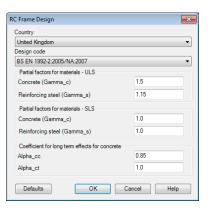


Note. Unlike nodal results, which only produce 'spot' contour results at each node, when plotting results for internal points the values obtained at each internal point can be interpolated to create line contour plots.

Checking against a Design Code

RC Frame Design...

- 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.



Attributes

Design

RC Frame Design

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.

_____ 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 selection.
- Select the two lines representing the piers, then drag and drop the attribute **Pier** from the treeview onto the selection.

General ULS SLS Creep Beam Minimum reinforcement ratio I.3E-3 Maximum reinforcement ratio O.04 Column Maximum reinforcement ratio I.3E-3 Maximum reinforcement ratio O.4 O.4 O.4 O.4 O.4 O.4 Maximum reinforcement ratio O.4 O.4 O.4 O.4 O.4 O.4 O.4 O.4 O.4 O.4 O.4 O.4 O.4 O.4 O.4 O.4 	Eurocode 2 RC Frame Design	
Minimum reinforcement ratio 1.3E.3 Maximum reinforcement ratio 0.04 Column	General ULS SLS Creep	
Maximum reinforcement ratio O.04 Column Minimum reinforcement ratio 2.0E-3	Beam	
Column Minimum reinforcement ratio	Minimum reinforcement ratio	1.3E-3
Minimum reinforcement ratio 2.0E-3	Maximum reinforcement ratio	0.04
	© Column	
Maximum reinforcement ratio	Minimum reinforcement ratio	2.0E-3
	Maximum reinforcement ratio	0.04
Name deck 🗸 🚔 (new)	Name deck	- A form
Name UELK (new)	Name ueux	▼ <u></u> (new)
OK Cancel Apply Help	OK Cancel A	pply Help

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 \bigcirc 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 – Quasipermanent.

RC Frame Design Results (Eurocode 2)	—										
Extent											
All members											
Visible members											
O Group None	-										
Geometric attribute	▼]										
C Lines Update											
Design check locations											
Nodes											
Internal points											
Inspection locations											
Load groups											
ULS Select											
SLS characteristic	Select										
SLS quasi-permanent	Select										
Design check name	Load group										
Min/max reinforcement area check											
ULS check (6.1)	ULS 🔻										
SLS (stress limitation) check (7.2)											
SLS (crack control) check (7.3.4)											
Min reinforcement area check (7.3.	SLS characteri 🔻										
Ac	dvanced options										
Name RC Frame Design euroo	cod 🔻 🚖 (21)										
OK Cancel A	pply Help										

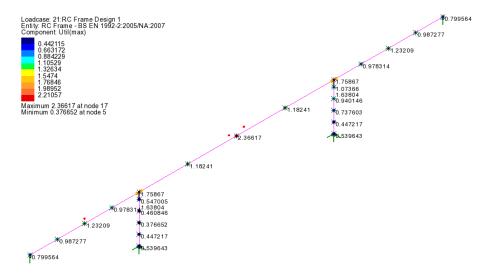
- On the Select Loadcases dialog presented select add SLS QP (Max) and SLS QP (Min) and add to the Included list. Click OK.
- On the RC Frame Design Results dialog accept the default name of **RC Frame Design 1** and click **OK** to finish.

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.

Detail	ed results fo	selecte	d row								F	lelp
ummary	Min/max reir	forceme	nt area	check	ULS check (6.1) SLS (str	ess limitation) check	(7.2) SLS (crac	k control) check (7.3.	4) Min reinforcem	ent area check (7.3.2)]	
	Line	Ele	ment	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	1	7	17	DeckNominal/Deck	0.286561	2.06171	2.27292	2.36617	0.843177	2.36617	1
2	2	1	16	17	DeckNominal/Deck	0.286561	2.06171	2.27292	2.36617	0.843177	2.36617	
3	2	1	15	15	DeckNominal/Deck	0.286561	1.24142	1.23026	1.75867	0.801848	1.75867	1
4	2	1	8	19	DeckNominal/Deck	0.286561	1.49091	1.48754	1.75867	0.798193	1.75867	1
5	3	1	9	19	DeckNominal/Deck	0.286561	1.15029	1.13709	1.63804	0.799564	1.63804	1
6	1	1	4	15	DeckNominal/Deck	0.286561	1.42513	1.39947	1.63804	0.799564	1.63804	1
7	1	1	3	13	DeckNominal/Deck	0.286561	0.815212	0.0901007	1.23209	0.8407	1.23209	1
8	3	2	20	21	DeckNominal/Deck	0.286561	0.648629	0.0670738	1.23209	0.8407	1.23209	1
9	1	1	2	13	DeckNominal/Deck	0.286561	0.798348	0.0870212	1.23209	0.8407	1.23209	1
10	3	2	21	21	DeckNominal/Deck	0.286561	0.648629	0.0670738	1.23209	0.8407	1.23209	1
11	2	1	16	16	DeckNominal/Deck	0.286561	1.07217	1.125	1.18241	0.843181	1.18241	1
12	2	1	8	18	DeckNominal/Deck	0.286561	0.79245	0.0873164	1.18241	0.839213	1.18241	1
13	2	1	15	16	DeckNominal/Deck	0.286561	1.07217	1.125	1.18241	0.843181	1.18241	1
14	2	1	7	18	DeckNominal/Deck	0.286561	0.79245	0.0873164	1.18241	0.839213	1.18241	1
15	6		8	10	Pier/PierSectionRei	0.277727	1.02911	1.07366	0.227407	0.121624	1.07366	1
16	3	2	21	22	DeckNominal/Deck	0.286561	0.60915	0.0706255	0.987277	0.8407	0.987277	1
17	3	1	2	22	DeckNominal/Deck	0.286561	0.60915	0.0706255	0.987277	0.8407	0.987277	1
10	1		2	12	Deck Nominal /Deck	0.286561	0 748047	0.0779061	0.987277	0.8407	0.987277	1

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.

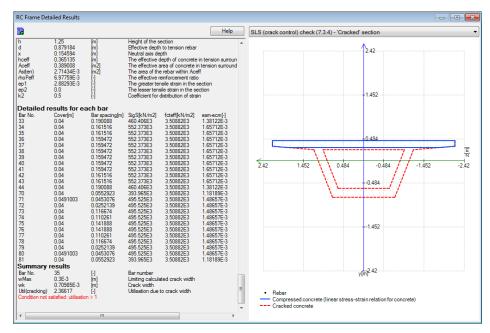
Detail	ed results for s	elected row											Help
iummary	Min/max reinfo	proement are	a check 🛛	ULS check (6.1) SLS (st	ress limitation) check	7.2) SLS (crack contro	ol) check (7.3.4)	Min reinforceme	nt area check (7	.3.2)			
	Line	Bement	Node	Section	Loadcase	Primary component	Internal forces Fx[kN]	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	Exmax, 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 OP	Ex 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.

RC Frame De	etailed Results			
8			Help	SLS (crack control) check (7.3.4) - 'Cracked' section
	E	Detail	ed results	2.42
	ed section		E	
Line Element Node Section	2 17 17 DeckNominal/De	ckSectionN	ominal Reinf/Iso 1	-1452
Design Country Design code	United Kingdom BS EN 1992-2:20			1,402
Partial factor gamma_c gamma_s	s for materials for ultim 1.5 [nate limit stat [-] [-]	es (ULS) For concrete For steel	0.484
gamma_s Coefficient fo alpha_cc alpha_ct	1.0 (orlong term effects an 0.85 (1.0 (-) -)	For steel ble effects resulting from the way the load is applied For concrete For steel	2.42 1.452 0.484 -0.484 -1.452 -2.42 -0.484
Materia Concrete	al propertie	S		\
fck fcd Ecm Ec.long Ec.eff ep epu fctm	40.0E3 [22.6667E3] 35.0E6 [11.6667E6] 1.142E6 [1.75E-3] 3.5E-3]	kN/m2] kN/m2] kN/m2] kN/m2] kN/m2] ·] kN/m2]	Characteristic compressive cylinder strength of cor Design compressive strength of concrete Secart modulus of elasticity of concrete for short Modulus of elasticity of concrete for short Britchive modulus of elasticity of concrete fuest Effective modulus of elasticity of concrete strength Ulimate compressive strain lim to incorrete Mean value of axial tensile strength of concrete	1.452
Creep pa Phi	20		Creep coefficient	ym ^{2.42}
	nlinear creep effect: r 0.45 [40.0E3 [18.0E3]	honlinearity r [-] [kN/m2] [kN/m2] [kN/m2] [liii]	Creep coefficient eed not be considered (k2Tck > abs(Sig q-p)) Non-inser creep limt parameter Characteristic compressive cylinder strength of cor The limt of the compressive stresses in the concre Compressive stresses in the concrete due to quasi	y(m)- ** • Rebar Compressed concrete (linear stress-strain relation for concrete) Cracked concrete

• 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

Det	ailed re	sults for se	elected row.								He	lp
mmar	y Min,	/max reinfor	cement area	a check	JLS check (6.1) SLS (str	ress limitation) check	(7.2) SLS (crac	control) check (7.3	.4) Min reinforcem	ent 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	
1	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	Ī
	5	3	19	19	DeckNominal/Deck	0.286561	1.15029	1.13709	1.63804	0.799564	1.63804	Ī
	6	1	14	15	DeckNominal/Deck	0.286561	1.42513	1.39947	1.63804	0.799564	1.63804	Ī
	7	1	13	13	DeckNominal/Deck	0.286561	0.815212	0.0901007	1.23209	0.8407	1.23209	Ī
-	8	3	20	21	DeckNominal/Deck	0.286561	0.648629	0.0670738	1.23209	0.8407	1.23209	Ī
	9	1	12	13	DeckNominal/Deck	0.286561	0.798348	0.0870212	1.23209	0.8407	1.23209	Ī
1	0	3	21	21	DeckNominal/Deck	0.286561	0.648629	0.0670738	1.23209	0.8407	1.23209	Ī
1	1	2	16	16	DeckNominal/Deck	0.286561	1.07217	1.125	1.18241	0.843181	1.18241	Ī
1	2	2	18	18	DeckNominal/Deck	0.286561	0.79245	0.0873164	1.18241	0.839213	1.18241	Ī
1	13	2	15	16	DeckNominal/Deck	0.286561	1.07217	1.125	1.18241	0.843181	1.18241	Í
1	4	2	17	18	DeckNominal/Deck	0.286561	0.79245	0.0873164	1.18241	0.839213	1.18241	Ĩ
1	15	6	8	10	Pier/PierSectionRei	0.277727	1.02911	1.07366	0.227407	0.121624	1.07366	ſ
1	6	3	21	22	DeckNominal/Deck	0.286561	0.60915	0.0706255	0.987277	0.8407	0.987277	I
1	7	3	22	22	DeckNominal/Deck	0.286561	0.60915	0.0706255	0.987277	0.8407	0.987277	1
1	8	1	12	12	DeckNominal/Deck	0.286561	0.748047	0.0779061	0.987277	0.8407	0.987277	ĺ
1	19	1	11	12	DeckNominal/Deck	0.286561	0.748047	0.0779061	0.987277	0.8407	0.987277	Í
2	20	1	13	14	DeckNominal/Deck	0.286561	0.73272	0.156822	0.978314	0.799564	0.978314	1

• On the RC Frame Results dialog select the Summary tab

- 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.

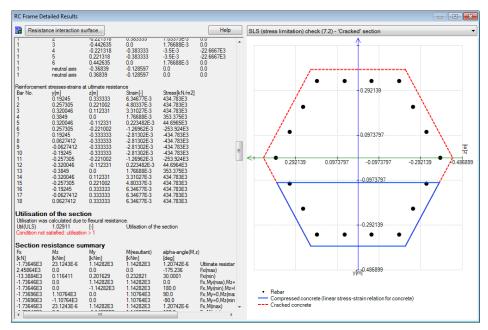


Note. When accessing detailed results from the Summary page the appropriate cross-section diagram showing the extent of the tension or compression in the section will need to be selected from the droplist.

• 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.

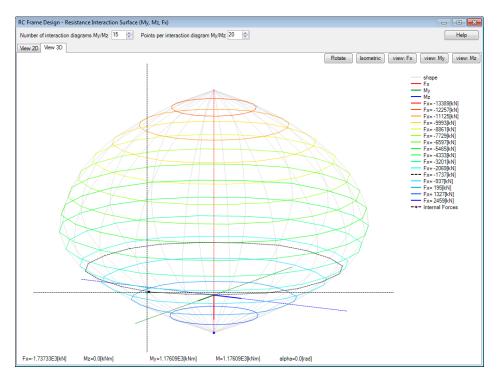
Interaction surface diagrams - for pier

For ULS results, interaction surface diagrams can be produced.

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

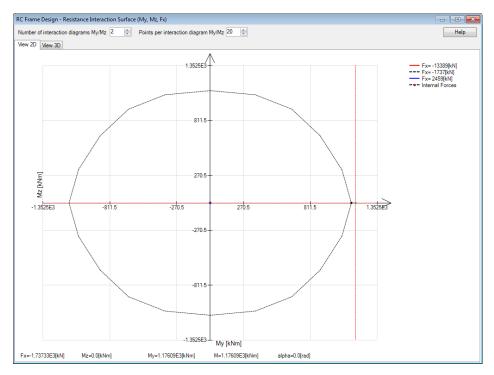
The interaction surface diagram for axial load Fx and moments My and Mz is displayed. By default a 2D view is shown. The black dotted line shows the My Mz contour for the applied axial force.

• 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.

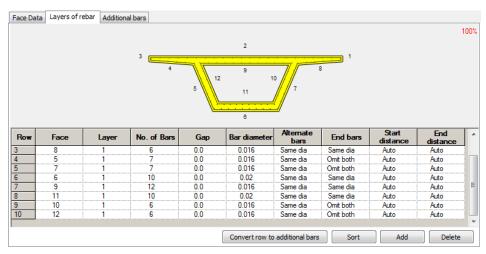


Note. When making changes to reinforcement quantities there is no need to re-solve the model. The RC Frame Design calculations will be automatically re-run according to the changes made. When making changes to geometric attributes a model must be re-solved.

Modifying the reinforcement in the deck

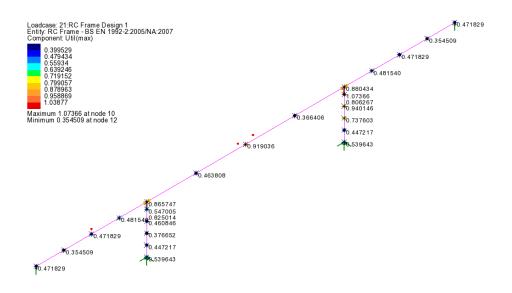
For the deck, the bar diameter will be changed from 12mm to 16mm throughout, with the exception of faces 6 and 11, which will have 20mm diameter bars. The section reinforcement is modified from the Utilities $\sqrt{2}$ treeview.

- 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**.

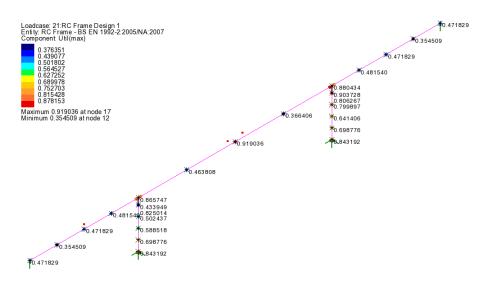
The RC Frame design results will be re-calculated, showing that all deck utilisation values are now less than 1.0. One of the piers however still shows a utilisation value that exceeds 1.0. To attempt to reduce this value the pier reinforcement will be modified.



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.

Face Data	Layers of re	bar Additiona	l bars	1	8 ••••••••••••••••••••••••••••••••••••				100%
				2	· · · · · · · · · · · · · · · · · · ·	>			
Row	Face	Laver	No. of Bars	Gap	3 Bar diameter	Alternate	End bars	Start	End distance
Row	Face	Layer	No. of Bars	Gap	Bar diameter	bars	End bars	distance	
1	1	Layer	5	0.0	Bar diameter	bars Same dia	Omit end	distance Auto	Auto
1	1	Layer 1 1	5	0.0	Bar diameter 0.025 0.025	bars Same dia Same dia	Omit end Omit end	distance Auto Auto	Auto Auto
1 2 3	1	Layer 1 1 1	5 5 5	0.0 0.0 0.0	Bar diameter 0.025 0.025 0.025	bars Same dia Same dia Same dia	Omit end Omit end Omit end	distance Auto Auto Auto	Auto Auto Auto
1 2 3 4	1 2 3 4	Layer 1 1 1	5 5 5 5 5	0.0 0.0 0.0 0.0	Bar diameter 0.025 0.025 0.025 0.025 0.025	bars Same dia Same dia Same dia Same dia	Omit end Omit end Omit end Omit end	distance Auto Auto Auto Auto	Auto Auto Auto Auto
Row 1 2 3 4 5 6	1	Layer 1 1 1 1 1	5 5 5	0.0 0.0 0.0	Bar diameter 0.025 0.025 0.025	bars Same dia Same dia Same dia	Omit end Omit end Omit end	distance Auto Auto Auto	Auto 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**.



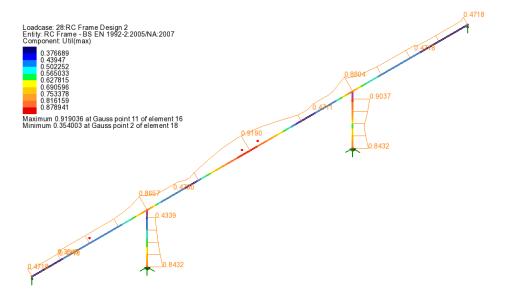
Note. This copies and creates a new loadcase (as opposed to creating a new one from scratch) and saves having to re-define the Load Groups.

• 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.

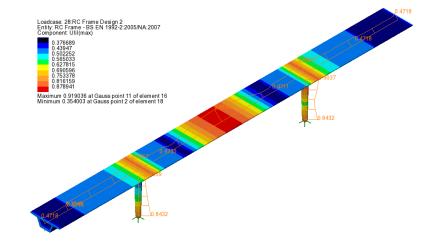


From this diagram the peak Utilisation value in the deck can be seen to be **0.919** as previously obtained.



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, so	rt by values	of Util(max).
---	----------------	-----------	--------------	---------------

		sults for se	elected row								He
mr	mary Min/	'max reinfo	rcement area	check U	LS check (6.1) SLS (str	ress limitation) check	: (7.2) SLS (crad	k control) check (7.3	3.4) Min reinforcem	ent area check (7.3.2)	
		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

RC Frame Detai	iled Results						
Resistanc	e interaction surface		Help	SLS (crack contr	rol) check (7.3.4) - 'Crack	ed' section	
	De	tailed results	A E			2.42	
Analyse	d section						
Line Element	2						
Internal point	11	ectionNominalReinf/Iso1					
		ection/vominal/veint/iso i				1.452	
	parameters						
	United Kingdom BS EN 1992-2:2005/N	VA:2007					
	r materials for ultimate I						
	1.5 [·] 1.15 [·]	For concrete For steel				0.484	
Partial factors fo	r materials for serviceal	bility limit state (SLS)				and the second	
	1.0 [-] 1.0 [-]	For concrete For steel		<	11		
Coefficient for lo	ng term effects and un	favourable effects resulting from the w	ay the load is applied	2.42	1.452 0.484	-0.484	-1.452 -2.42
	0.85 [-]	For concrete For steel			11	11	
		101000			\. <u>\</u>	-0.484	
	properties				Aller Aller		
Concrete: Is fok	so1 40.0E3 kN/r	n21 Characteristic compressive	and in days where with a firmer				
fcd	22.6667E3 [kN/r	m21 Design compressive strend	th of concrete				
	35.0E6 [kN/r 11.6667E6 [kN/r		ty of concrete (for short			-1 452	
Ec.eff	16.1155E6 kN/r	m2] Effective modulus of elasti	city of concrete (used in				
	1.75E-3 [·]	Strain in concrete at reach					
	3.5E-3 [-] 3.50882E3 [kN/r	12 Ultimate compressive strain 12 Mean value of axial tensile					
Сгеер рага	meters						
	2.0 [-]	Creep coefficient				y[m] ^{2.42}	
		nearity need not be considered (k2*fck	:>abs(Sig.q-p))				
	0.45 [-] 40.0E3 [kN/r	Non-linear creep limit para	meter	 Rebar 			
	40.0E3 [kN/r 18.0E3 [kN/r				ssed concrete (linear stress	s-strain relation for concre	ste)
	-2.48365E3 [kN/r	m2] Compressive stresses in th		Cracked	d concrete		
•	1	"	+				

- 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 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.

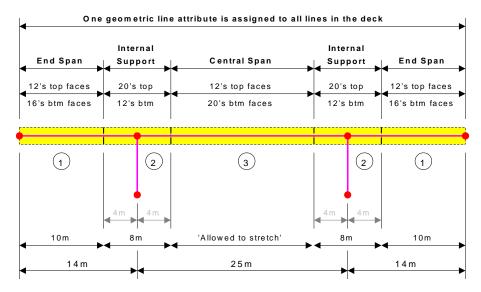
Option 1: Assigning a single geometric line attribute to all lines of the deck

Define section reinforcement

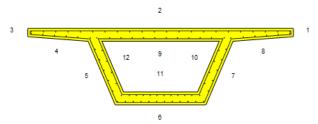
For this option separate section reinforcement utilities need to be created for each 'region' of the deck that is to be reinforced, and be given suitable names that specify their location, such as **EndSpan**, **InternalSupport**, and **CentralSpan**.

This can be done by going to the Utilities \checkmark 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	I	Bar diameters require	d
	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.

Reinfo	preement Section	Allowed to stretch	Length	Edit
1 7:DeckSectionEndS		0	10	Insert
2 5:DeckSectionIntern		Ó	8	
3 6:DeckSectionCentre		۱	N/A	Delete
4 5:DeckSectionIntern	alSupportReinf	<u> </u>	8	
5 7:DeckSectionEndS	panReinf	<u> </u>	10	Move u
				Move do
1	2	3	4	5
1	2	3	4	
•	2	3	4	
1	2	3	4	
•		3		5
• 10.0		3	8.0	5

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

Analysis category 3D	
Definition	16
From Library	User Sections
Rotation about centroid 0 •	Local
Mirrored about axis None 🔻	Deck
C Enter Properties	Deck
Usage 3D Thick Beam (Any beam) *	100%
	z
	1
Reinforcement (only used for RC design checks)	
8:Eul length	
8:Ful_length	
,	Value
Cross sectional area (A)	1.49205
Cross sectional area (A) Second moment of area about y axis (lyy)	
Cross sectional area (A) Second moment of area about y axis (hyy) Second moment of area about z axis (Izz) Product moment of area (hz)	1.49205 0.294294 1.53212 0.0
Cross sectional area (A) Second moment of area about y axis (lyy) Second moment of area about x axis (lz1) Product moment of area (ly2) Tomional constant (J)	1.49205 0.294294 1.53212 0.0 0.540736
Cross sectional area (A) Second moment of area about y axis (by) Second moment of area about y axis (by) Product moment of area (by) Tereisrani constant (J) Effective shear area in y direction (Agy)	1 49205 0 294294 1.53212 0 0 0.540736 1.05384
Cross sectional area (A) Second moment of area about y axis (by) Second moment of area about z axis (bz) Product moment of area (by) Troincian constants, y (afrection (Aay) Effective shear area in x (afrection (Aax)	1 49205 0 294294 1.53212 0 0 0.540736 1.05384 0.346535
Cross sectional area (A) Second moment of area about y axis (by) Second moment of area about y axis (by) Product moment of area (by) Tereisrani constant (J) Effective shear area in y direction (Agy)	1 49205 0 294294 1.53212 0 0 0.540736 1.05384
Cross sectional area (A) Second moment of area alond y axis (by) Second moment of area alond y axis (by) Pocular moment of area (by) Torisonal constant (J) Torisonal constant (J) Effective alear area in y direction (Asy) Effective alear area in y direction (Asy) Effective alear area in z direction (Asy)	1 49205 0 294294 1.53212 0.0 0.540736 1.05384 0.346335 0.0
Cross sectional area (A) Second moment of area alond y axis (by) Second moment of area alond y axis (by) Pocular moment of area (by) Torisonal constant (J) Torisonal constant (J) Effective alear area in y direction (Asy) Effective alear area in y direction (Asy) Effective alear area in z direction (Asy)	1 49205 0 294294 1.53212 0.0 0.540736 1.05384 0.346335 0.0
Cross sectional area (A) Second moment of area about y axis (by) Second moment of area about y axis (by) Product moment of area (by) Tomoral constant (J) Effective shares area (r) decision (Ap) Effective shares area (r) decision (Ap) Effective shares area (r) decision (Ap) Eccentroly (r) of decision (by) Eccentroly (r) of decision (by)	14205 02424 153212 0.0 0.540736 0.540736 0.346535 0.0 0.0 0.0 Section details
Cross sectional area (A) Second moment of area about y axis (by) Second moment of area about y axis (by) Product moment of area (by) Tomoral constant (J) Effective shares area (r) decision (Ap) Effective shares area (r) decision (Ap) Effective shares area (r) decision (Ap) Eccentroly (r) of decision (by) Eccentroly (r) of decision (by)	1.4205 0.24224 1.53212 0.0 0.540736 1.05384 0.24535 0.0 0.0
Cross sectional area (A) Second moment of area about y axis (by) Second moment of area about y axis (by) Texture moment of area about y axis (by) Texture about area (by) Effective about area as (by) Effective about area as (before) (by) Eccentricity in y direction (by) Eccentricity in y direction (by) Woundee Tapering >>	14205 02424 153212 0.0 0.540736 0.540736 0.346535 0.0 0.0 0.0 Section details
Cross sectional area (A) Second moment of area about y axis (by) Second moment of area about y axis (by) Texture moment of area about y axis (by) Texture about area (by) Effective about area as (by) Effective about area as (before) (by) Eccentricity in y direction (by) Eccentricity in y direction (by) Woundee Tapering >>	14205 02424 153212 0.0 0.540736 0.540736 0.346535 0.0 0.0 0.0 Section details

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 deassigned from the selection.

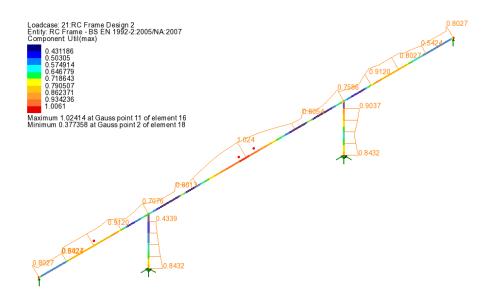
Then, with the lines in the deck still selected, assign the geometric line attribute **Full_length** to those lines, ensuring (very importantly) that the option to **Consider selection as a single assignment** is selected.

Geometric Beam
Assign to points Assign to lines
Analysis Analysis 1
Multiple selected lines
OK Cancel Help

Util(max)

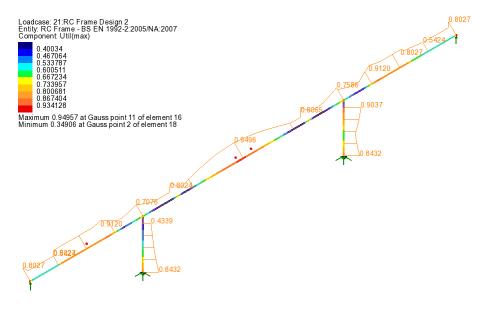
Once the revised geometric line attribute had been assigned, the RC Frame design results will update to show values for **Util(max)** in the deck.

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.

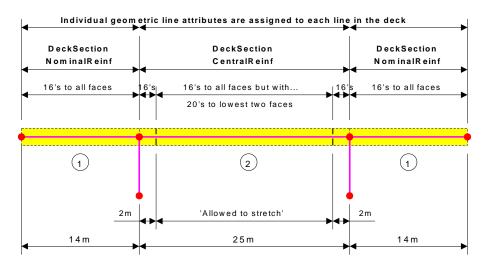
Option 2: Assigning individual geometric beam attributes to each line of the deck

Section reinforcement

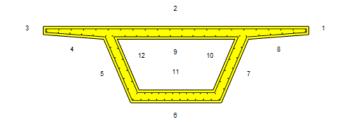
For this option separate section reinforcement utilities need to be created for each 'member length' in the deck that is to be reinforced, and given names such as: **DeckSectionNominalReinf** and **DeckSectionCentralReinf**.

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 diameter	s 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.

- 1	Reinforcement Section	Allowed to stretch	Length	Edit
	3:DeckSectionNominalReinf	0	2	Insert
	6:DeckSectionCentralSpanReinf	۲	N/A	
3	3:DeckSectionNominalReinf	<u> </u>	2	Delete
				Move up
				Move down
				100
			_	
			7	
			7	
				3
		2		3
		2 1		3
•		2		
•	1	2 1		3
•		2		
•		2		
•		2 1		2.0

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 deassigned from any lines prior to reassigning a different geometric line attribute.

With reference to the above note, deassign DeckFullLength line attribute from all lines in the model

Analysis category 3D	
Definition	
From Library	User Sections
Rotation about centroid 0 🔹 °	Local
Mirrored about axis	
	Deck
Enter Properties	
Usage 3D Thick Beam (Any beam) 🔻	100%
	z
Reinforcement (only used for RC design checks)	
9:DeckLineCentralReinf	Value
(9:DeckLineCentralReinf (Cross sectional area (A)	1.49205
(9:DeckLineCentralReinf (Cross sectional area (A) Second moment of area about y axis (lyy)	1.49205 0.294294
(9:Dedd.ineCentralReinf (Cross sectional area (Å) Second moment of area about y axis (lyy) Second moment of area about z axis (lzz)	1.49205 0.294294 1.53212
(9:DedLineCentraReinf ♥ (Gross sectional area (A) Second moment of area about y axis (lyy) Second moment of area baout z axis (lz2) Product moment of area (lyz2)	1.49205 0.294294 1.53212 0.0
9:DedklineCentraRenf ✓ Cross sectional area (A) Second moment of area about y axis (lyy) Second moment of area about z axis (lyz) Product moment of area (lyz) Product moment of area (lyz) Toxinonial constant (l)	1.49205 0.294294 1.53212 0.0 0.0 0.540736
9:DeddineCentraReinf Cross sectional area (A) Second moment of area about y axis (by) Second moment of area (b) Toxisonal constant (J) Tectivershare area in y direction (Asy)	1.49205 0.294294 1.5212 0.0 0.540736 1.05384
9:DedLineCentraReinf (Second moment of area shout y asis (hy) Second moment of area shout z asis (hy) Second moment of area shout z asis (hz) Product moment of area shout z asis (hz) Tectional constant (j) Effective shear area in z direction (Apr) Effective shear area in z direction (Apr)	1.49205 0.294294 1.52212 0.0 0.540736 1.05384 0.346535
9:DedLineCentraReinf Cross sectional area (A) Second moment of area about y axis (by) Second moment of area (b) Toxisional constant (J) Effective shear area in y direction (Asy) Effective shear area in y direction (Asy) Effective shear area in y direction (Max) Corenticity in y indexton (b)	1.49205 0.294294 1.53212 0.0 0.540736 1.05304 0.346535 0.0
(stDeckLineCentraReinf ↓ Conse sectional area (A) ↓ Second moment of area shout y asis (by) ↓ Second moment of area shout z asis (by) ↓ Second moment of area shout z asis (by) ↓ Totional contract of area shout z asis (by) ↓ Totional contract of area shout z asis (by) ↓ Effective shear area in y direction (Ap) ↓ Effective shear area in z direction (Ap) ↓	1.49205 0.294294 1.52212 0.0 0.540736 1.05384 0.346535
9xDedLineCentraReinf Cross sectional area (A) Second moment of area about y axis (ty) Second moment of area about z axis (ty) Toxinent contrast (ty) Toxinent contrast (ty) Effective share area in z direction (Apr) Eccentricity in y direction (by) Eccentricity in z direction (by)	1 43205 0.2%424 1 53212 0 0 544736 1 5534 0.346355 0.0 0 0
9:DeddineCentraReinf Cross sectional area (A) Second moment of area about y axis (by) Second moment of area (b) Toxisonal constant (J) Effective shear area in y direction (Asy) Effective shear area in y direction (Asy) Effective shear area in y direction (Pas)	1.49205 0.294294 1.53212 0.0 0.540736 1.05304 0.346535 0.0
9:DedLineCentraReinf Second moment of area about y axis (ty) Second moment of area about z axis (ty) Second moment of area about z axis (ty) Totoriand constants (ty) Totoriand constants of (ty) Technic hear area area in z direction (Aar) Elective hear area in z direction (Aar) Eccentricity in z direction (Ayr) Eccentricity in z direction (Ayr) Eccentricity in z direction (Agr)	1 43205 0.2%424 1 53212 0 0 544736 1 5534 0.346355 0.0 0 0
9:DedLineCentraReinf Second moment of area about y axis (ty) Second moment of area about z axis (ty) Second moment of area about z axis (ty) Totoriand constants (ty) Totoriand constants of (ty) Technic hear area area in z direction (Aar) Elective hear area in z direction (Aar) Eccentricity in z direction (Ayr) Eccentricity in z direction (Ayr) Eccentricity in z direction (Agr)	1 43205 0.2%424 1 53212 0 0 544736 1 5534 0.346355 0.0 0 0

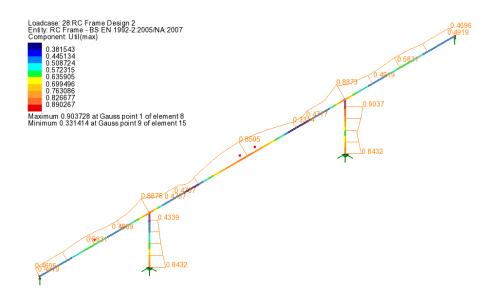
Then assign the **DeckLineCentral** line attribute to the line representing the central span

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.