

Assessment of a Composite Bridge Deck to Eurocodes

For LUSAS version:	22.0 or above
For software product(s):	LUSAS Bridge or LUSAS Bridge plus
With product option(s):	Steel and Composite Deck Designer (PontiEC4 – V3.5.2 or above)
With additional software	Microsoft Excel installed

Description

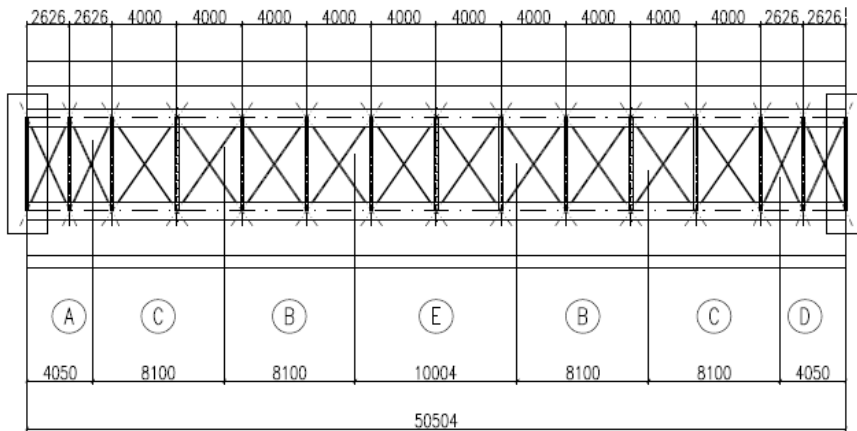
The global analysis and the assessment of the main girders of a two span continuous bridge is to be carried out using the Steel and Composite Deck Designer (PontiEC4) software option and LUSAS Bridge.

Because the primary aim of this particular example is to show the use of the PontiEC4 Composite Deck Designer and the interaction between it and LUSAS, no step-by-step instruction is provided for construction of the supplied LUSAS model that is used in this example. A separate example ‘Composite Highway Bridge Design’ shows the LUSAS model building requirements to allow a design check with PontiEC4 to be carried out.

The bridge has a symmetrical composite two-girder structure with two spans of 50.504m. For the purposes of the example the following geometrical simplifications have been made:

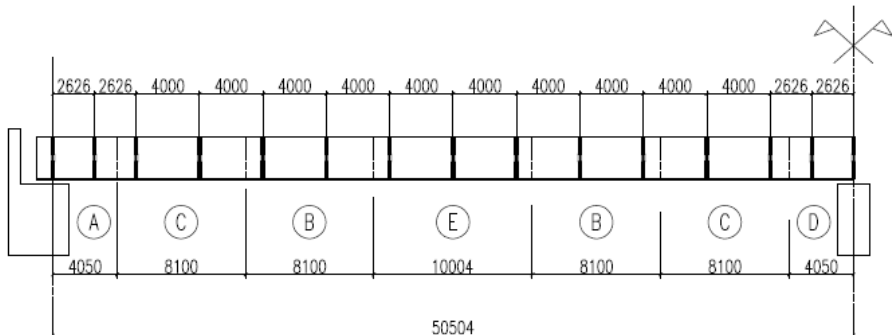
- ☐ The horizontal alignment is straight
- ☐ The top face of the deck is flat
- ☐ The bridge is straight, that is, not skewed
- ☐ The structural steel main girders are of a constant depth of 2700mm

Plan, elevation and typical cross section views of the bridge deck are shown on the following pages.



Plan of left-hand span.

(Note that transverse ladder bracing is modelled; X-bracing is not modelled.)



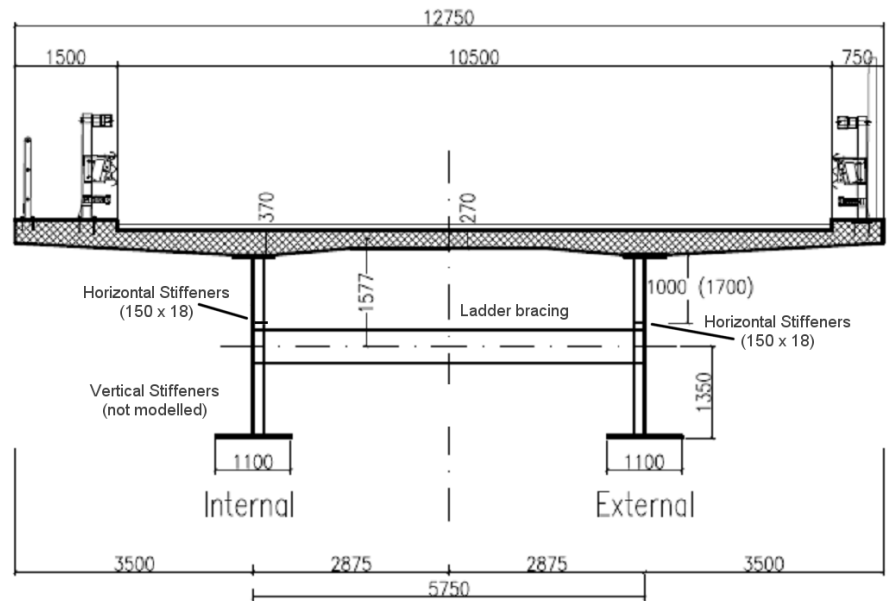
Elevation of left-hand span. Right-hand span is similar but handed.



Note.

- The girder shown above is formed from five **Segments** (lengths having the same beam section size) that are labelled A, B, C, D, and E.
- **Sections** are locations defined within a segment at which geometric properties are defined or calculated.

Both terms are used throughout this example.



Typical cross section

The total slab width is 12.75m. The centre-to-centre spacing between the main girders is 5.75m and the slab cantilever either side is 3.5m long. The main girder dimensions, for each segment, are summarized in the following table.

Table of main girder dimensions

Segment name	Top flange width (mm)	Top flange thickness (mm)	Height of beam (mm)	Web thickness (mm)	Bottom flange width (mm)	Bottom flange thickness (mm)
A	600	20	2700	20	1100	20
C	600	30	2700	22	1100	40
B	600	30	2700	18	1100	40
E	600	30	2700	18	1100	50
D	1100	40	2700	25	1200	50

The slab thickness varies from 0.37m over the main girders to 0.27m at its free edges and at its axis of symmetry. For simplification reasons, the actual slab cross-section of a half-deck is modelled by a rectangular area, which is the same as the actual width (i.e. 12.75/2 m), and a second rectangular area modelling the concrete haunch, which has the same width as the upper structural steel flange. The equivalent thickness of slab is 273mm and the equivalent height of the haunch is 75mm.

Steel and Composite Deck Designer (PontiEC4)

The Steel and Composite Deck Designer (PontiEC4) is a software option that carries out comprehensive calculations for multiple sections on steel/composite bridge decks to the Eurocodes. Force and moment results for selected bridge deck sections are provided by a corresponding LUSAS model, and loadcase combinations defined within LUSAS are associated with design limit states and phases defined in PontiEC4.

Design calculations covering ULS bending, stress, shear and interaction; SLS stress, web breathing and cracking, and fatigue checks for main members and connectors are carried out. Results, output in tabbed dialogs, visually show values that pass or fail. Graphs and a report containing all input data and output with references to the Eurocode clauses can be easily created.

In the Steel and Composite Deck Designer (PontiEC4) the units in use are always indicated in the input and output dialogs, and in general forces are expressed in N (Newton), lengths in mm (millimetres), and moments expressed in Nm (Newton Metres). The units used for the LUSAS model will therefore be N, m, kg, s, C.

Keywords

Steel, Composite, Deck, Designer, EC4, Design, Checking, Import, Export, ULS, SLS, Graphing, Detailed design.

Associated files

These are provided as part of a PONTI EC4 installation.



- ❑ **Composite_Bridge_Deck_Initial.csv** – Contains initial material and geometric data / definitions for use with PontiEC4
- ❑ **Materials_Sections_Rev0.vbs** - A file created by PontiEC4 containing materials and geometric section data to import into LUSAS. A version of this file is supplied also, but is for use only in case of difficulties in manually creating the appropriate data
- ❑ **Composite_Bridge_Deck.mdl** – A supplied LUSAS model file that is to be used to analyse the structure. After solving this model, analysis results are exported from LUSAS or use in PontiEC4 to undertake design checks.
- ❑ **Export to ponti EC4.xls** – File created by the LUSAS Export to Composite Deck Design dialog containing data to import into PontiEC4. Supplied file is only for use in case of problems in manually creating the appropriate data in the previous section of the example.
- ❑ **Composite_Bridge_Deck_Optimized.csv** – An input file for PontiEC4 with updated steel flange values for segments D and C.

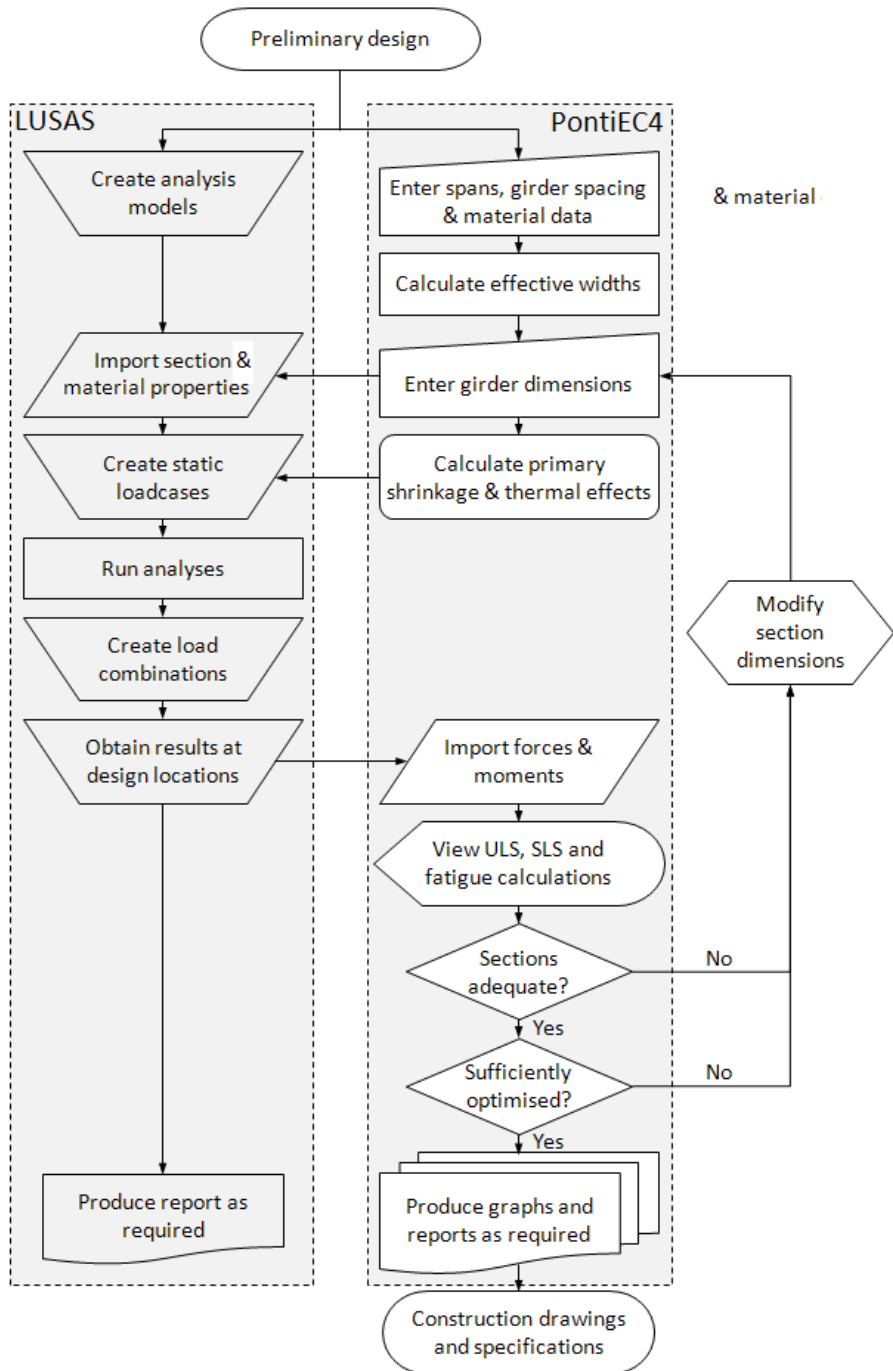
- ❑ **Sections_rev1.vbs** –A file created by PontiEC4 containing revised section data to import into LUSAS to update the geometric properties in the LUSAS model.
- ❑ **Composite_Bridge_Deck_Optimized.mdl** – A supplied LUSAS model file that is to be used to analyse the structure. After solving this model, analysis results are exported from LUSAS or use in PontiEC4 to undertake design checks.
- ❑ **Export to ponti EC4 rev1.xls** – File created by the LUSAS Export to Composite Deck Design dialog containing data to import into PontiEC4. Supplied file is only for use in case of problems in manually creating the appropriate data in the previous section of the example.

Method and Objectives

The main steps in using the Steel and Composite Deck Designer in conjunction with LUSAS can be summarized as follows:

- ❑ **Step 1: Define material and geometric properties in the PontiEC4 Steel and Composite Deck Designer. All effective section properties are calculated.**
- ❑ **Step 2: Export the geometric and material properties from the PontiEC4 Steel and Composite Deck Designer for use with a corresponding user-defined LUSAS model.**
- ❑ **Step 3: Use LUSAS to calculate the forces and moments for all the limit state combinations.**
- ❑ **Step 4: Export the force and moment data from LUSAS for use in the PontiEC4 Steel and Composite Deck Designer.**
- ❑ **Step 5: Import force and moment data into PontiEC4 Steel and Composite Deck Designer**
- ❑ **Step 6: Assess the results and carry out detailed design checks in PontiEC4**
- ❑ **Step 7: Optimize the structure (by making adjustments to section sizes if necessary)**

The full interaction between LUSAS Bridge and the PontiEC4 Steel and Composite Deck Designer can be seen by referring to the following flowchart.



File types used / generated / saved

LUSAS	PontiEC4
LUSAS Model data (.mdl)	PontiEC4 dialog values and settings (.csv)
Forces and Moment data calculated by LUSAS for import into PontiEC4 (.xls)	Effective geometric section properties data calculated by PontiEC4 for import into LUSAS (.vbs)
Saved values and settings made on the Export Force and Moment dialog (.inp)	PontiEC4 analysis log file for all section calculations (.log)

Step 1: Define material and geometric properties in PontiEC4

This section covers the definition of material and geometric properties in the PontiEC4 Composite Deck Designer in order to calculate effective section properties. Once done, the beam cross section and material properties are exported to a VBS file for subsequent importing into LUSAS.

Defining material properties

To define the required data for this worked example three options are available, according to what it is that you need to achieve from carrying out this example. Values and settings can be entered into dialogs in three ways:

- ❑ **(Option 1) Manually entering all material and geometric data** (so that an appreciation is obtained for all aspects of using and entering data into the various dialogs). This will obviously take the longest of the three options to input. See the heading '*Option 1: Manually entering all material and geometric data*' for this option.
- ❑ **(Option 2) Loading the material dialog with default values and settings** This option reduces the material data input, but requires careful checking and updating of non-default values and settings in the material dialog as required by the example. Geometric data will still need to be entered. See the heading '*Option 2: Loading the material dialog with default values*' for this option.
- ❑ **(Option 3) Loading all material and geometry data from a csv file** (to automatically populate the material and geometry dialogs with all the initial values and settings required by the example). This is the fastest option and would be used if it was simply required to see the main processes and steps involved in order to view the results obtained for this example and its accompanying LUSAS model. See the heading '*Option 3: Loading all material and initial geometric data from a csv file*', that follows, for this option.

The example is written assuming that manual entering of all values is done. (Option 1 above), but using Option 2 or Option 3 will save time in entering data.

The text that follows, having a grey background, shows the steps required for the options stated. **The options are listed in reverse order to the above, according to the amount of data entry required, with the least amount of data entry first.**

The example is written to manually enter all required data so that an understanding of the data entry can be appreciated.

Option 3: Loading all material and initial geometry data from a csv file:

A file is supplied if it is desired to automatically populate the material and geometry dialogs with the initial values required by the example (as opposed to manually defining the values required).

Associated file



- **Composite_Bridge_Deck_Initial.csv** – Contains initial material and geometric data / definitions for use with PontiEC4. To use this file:

Run the Steel and Composite Deck Designer (PontiEC4)

- Run the Steel and Composite Deck Designer (PontiEC4)
- Select the **File > Open** menu item, and browse in the \<PontiEC4 Installation Folder>\Help\Examples\Composite_Bridge_Deck folder for the **Composite_Bridge_Deck_Initial.csv** file and click **Open**. All dialogs will be filled with data initially required by the example.
- Expand the size of the PontiEC4 window and select the **File> Save as** menu item and **save the PontiEC4 dialog data to a working project folder different from the PontiEC4 installation folder**, (this would normally be a project folder in which a LUSAS model would also be created) with the name **Composite_Bridge_Deck.csv**
- Continue at Step 2 in the example entitled '*Step 2: Export geometric properties from PontiEC4 to LUSAS*'

Option 2: Loading the material dialog with default values and settings:

Run the Steel and Composite Deck Designer (PontiEC4)

- Run the Steel and Composite Deck Designer (PontiEC4)
- Select the **File> New** menu item to create a new PontiEC4 project.
- On the Materials dialog, press the **Defaults** button.

- Select the **File> Save as** menu item to save the PontiEC4 dialog data to a working project folder different from the PontiEC4 installation folder, (this would normally be a project folder in which a LUSAS model would also be created) with the name **Composite_Bridge_Deck.csv**
- Continue at the heading entitled '*Defining material properties*', *below*, checking that each dialog panel contains the values that are shown in this example.

Option 1: Manually entering all material and geometric data:

Run the Steel and Composite Deck Designer (PontiEC4)

- Run the Steel and Composite Deck Designer (**PontiEC4**)
- Select the **File> New** menu item to create a new PontiEC4 project.
- Select the **File> Save as** menu item to save the PontiEC4 dialog data to a working project folder different from the PontiEC4 installation folder (this would normally be a project folder in which a LUSAS model would also be created), with the name **Composite_Bridge_Deck.csv**



Note. All data entered in the various PontiEC4 dialogs will be saved to this file. Previously created and saved data, or supplied data (held in .csv files) can be loaded into the PontiEC4 dialogs by using the File> Open menu item.

Defining material properties

The material dialog is immediately available when you start a new project or when you open an existing one. The material dialog is organized into separate panels; Concrete, Concrete age, Steel etc.

- For each panel on the dialog enter the material data shown in the following images.

Concrete slab

Concrete slab	
Strength f_{ck} (N/mm ²)	33.2
Strength $f_{ct,ef}$ (N/mm ²)	0
Partial factor γ_c	1.5
Cement class	N
Aggregate type	Quartzite
Coeff. of thermal expansion	1E-05
Max. crack width w_k (mm)	0.2

Concrete age

Concrete age	
At onset of drying shrinkage t_s (day)	2
At time considered t (day)	25550
See note 1	
When perm. load is applied to (day)	30
See note 2	
When shrink. load is applied to (day)	2
See note 3	
When imposed d. are applied to (day)	30
Permanent creep multiplier Ψ_{iL}	1.1
Shrinkage creep multiplier Ψ_{iL}	0.55
Deformations creep multiplier Ψ_{iL}	1.5

Notes:

1. Age of concrete when permanent load is applied
2. Age of concrete when shrinkage load is applied
3. Age of concrete when imposed deformations are applied.

Modular ratios

Modular ratios

☒ Automatic calculation ☐ Direct input

n0

nL permanent loads

nL shrinkage

nL imposed deformation

Environment

Environment

Exposed area (mm²)

Exposed perimeter (mm)

Relative humidity (%)

Imposed strain in the slab

Imposed strain in the slab

☒ Automatic calculation ☐ Direct input

Shrinkage Deformation

Shrinkage comb. factor

Temperature difference (°C)

Temperature comb. factor

Shear connection

Shear connection

Ultimate strength f_u (N/mm²)

Reference values for fatigue strength at 2E6 cycles

shear stress $\Delta\tau_c$ (N/mm²)

normal stress $\Delta\sigma_c$ (N/mm²)

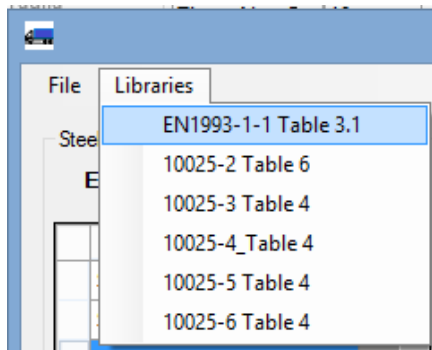
Partial factors:

γ_v k_s (SLS)

γ_{ff} $\gamma_{Mf,s}$

Steel

- Click on the  button, to open the Database of Structural Steel, and on Libraries menu select *EN1993-1-1 Table 3.1* as shown:



- Select S355 and tick all component checkboxes in the *Section component* region, as in the picture below, then click OK.

Database of Structural Steel

File Libraries

Steel Grade Database

EN1993-1-1 Table 3.1

Edit database ☐

SteelGrade	> t (mm)	≤ t (mm)	f _y (N/mm ²)	> t (mm)	≤ t (mm)	f _u (N/mm ²)
S235	0	40	355	0	40	510
S275	40	80	335	40	80	470
S355						
S450						
S275 N/NL						
S355 N/NL						
S420 N/NL						
S460 N/NL						
S275 M/ML						
S355 M/ML						
S420 M/ML						
S460 M/ML						
S235 W						

Section component

☒ Apply to Top Flange ☒ Apply to Web ☒ Apply to Bottom Flange

Ok Cancel

- Define all other values as shown below

Steel

Modulus of elasticity (N/mm²) 210000 Poisson's ratio v 0.3

.. structural steel

Steel structural Library

Top Flange S355

Web S355

Bottom Flange S355

Partial factors:

γ_{M0} 1.05 γ_F 1

γ_{M1} 1.1 γ_{Mf} 1.35

γ_{Mser} 1

.. reinforcement steel

f_{yk} (N/mm²) 450

Fatigue strength:

$\Delta\sigma_{Rsk}$ (N/mm²) 162.50

Partial factors:

γ_s 1.15

γ_F 1

γ_{Mf} 1.15

Fatigue

FATIGUE. Damage equivalent factors

	STRUCTURAL STEEL	SHEAR STUDS	REINFORC. BARS
-for damage effects induced by the traffic	λ_1 (*)	$\lambda_{v,1}$ 1.550	$\lambda_{s,1}$ (*)
-for the traffic composition	λ_2 0.928	$\lambda_{v,2}$ 0.953 ...	$\lambda_{s,2}$ 0.000
-for design life of the structure	λ_3 1.000	$\lambda_{v,3}$ 1.000 ...	$\lambda_{s,3}$ 0.000
-for effects of the heavy traffic on the other slow lanes	λ_4 1.000	$\lambda_{v,4}$ 1.000	$\lambda_{s,4}$ 1.000
			ϕ_{Fat} 0.000

(*) Values depending on the section position (input in the -Geometry- dialog window)

- Note that λ_2 can be calculated from selections made in the dialog that appears when you click on the  button next to the $\lambda_{v,2}$ input field. In this case, make the selections as shown below:

Damage equivalent factor LAMBDA2 for road bridge

☐ ☐ ☐

Calculations

$$\lambda_2 = \frac{Q_{ml}}{Q_0} \left(\frac{N_{Obs}}{N_0} \right)^{1/5} \quad Q_{ml} = \left(\frac{\sum n_i Q_i^5}{\sum n_i} \right)^{1/5}$$

$$\lambda_{v2} = \frac{Q_{ml}}{Q_0} \left(\frac{N_{Obs}}{N_0} \right)^{1/8} \quad Q_{ml} = \left(\frac{\sum n_i Q_i^8}{\sum n_i} \right)^{1/8}$$

$\lambda_2 = 0.928$ $\lambda_{v2} = 0.953$

Qo = 480 kN (weight of FML3)

No = 0.5E6

Nobs = 5E+5 (Cfr. Tab. 4.5)

Qml = 445.4 kN (Cfr. Tab. 4.7)

Qmlv = 457.4 kN (Cfr. Tab. 4.7)

Number of observations

Table 4.5(n) - Indicative number of heavy vehicles expected per year and per slow lane. EN 1991-2:2003 (E)

Traffic categories		N _{Obs} per year and per slow lane
1	☐ Roads and motorways with 2 or more lanes per direction with high flow rates of lorries	$2,0 \times 10^6$
2	☒ Roads and motorways with medium flow rates of lorries	$0,5 \times 10^6$
3	☐ Main roads with low flow rates of lorries	$0,125 \times 10^6$
4	☐ Local roads with low flow rates of lorries	$0,05 \times 10^6$
☐ User Calculate		

Heavy vehicle distribution

Table 4.7 - Set of equivalent lorries. EN 1991-2:2003 (E)

	 Q ₁ = 200 kN	 Q ₂ = 310 kN	 Q ₃ = 490 kN	 Q ₄ = 390 kN	 Q ₅ = 450 kN	
☒	20%	5%	50%	15%	10%	Long distance
☐	40%	10%	30%	15%	5%	Medium distance
☐	80%	5%	5%	5%	5%	Local traffic
☐						User Calculate

OK Exit

- Note that λ_3 can be calculated from data entered in the dialog that appears when you click on the button next to the $\lambda_{v,3}$ input field. In this case, input the data below:

Damage equivalent factor LAMDA3 for road bridge

☐ ☐ ☐

Design life of bridge (years) 100

$$\lambda_{v,3} = \left(\frac{t_{Ld}}{100} \right)^{1/8} = 1$$

$$\lambda_3 = \left(\frac{t_{Ld}}{100} \right)^{1/5} = 1$$

OK Exit

This completes the definition of the materials.

Save the data entered

- Select the **File> Save** menu item to save the PontiEC4 dialog data to the current working project folder.

Defining Geometric properties

Prior to defining the geometric section properties for the spans, the span lengths and the effective concrete slab width (b_{eff}) of each section, defined at various distances along the section, need to be defined.

Defining shear lag slab and flanges

- To input data for the effective slab width calculation, select the **Utilities> Shear lag slab and flanges** menu item.
- Input an array of the bridge spans (the span lengths) in the text box in the top right corner: **50.504,50.504** and click on the **-->** button. The X distance where the effective slab width changes will be automatically set up in the table
- For each row input the flange widths $b1^*$, $b2^*$, $b0$ as **2875, 3500, 500** respectively.



Note. Values in the table can be selected and copied and pasted into other cells by using the context menu.

- In the **Type** column ensure that the code **0** is used for the first and last section, **1** for the sagging regions, and **2** for the middle support.
- Click the **Calculate** button to calculate b_{eff} ,

Data for shear lag of concrete and steel flanges (EN1994-2 5.4.1.2, EN 1993-1-5 3.2.1)

Key:

- $L_e = 0,85 L_1$ for $b_{eff,1}$
- $L_e = 0,25(L_1 + L_2)$ for $b_{eff,1}$
- $L_e = 0,70 L_2$ for $b_{eff,1}$
- $L_e = 2 L_3$ for $b_{eff,2}$

X (m)	b1* (mm)	b2* (mm)	b0 (mm)	Type	b _{eff} (mm)	L _e (m)	b _{e1} (mm)	b _{e2} (mm)	beta1	beta2
0	2.875	3.500	500	0	5.878	42.93	2.625	3.250	0.959	0.880
12.626	2.875	3.500	500	1	6.375	42.93	2.625	3.250	1.000	1.000
37.878	2.875	3.500	500	1	6.375	42.93	2.625	3.250	1.000	1.000
50.504	2.875	3.500	500	2	6.282	25.25	2.625	3.157	1.000	1.000
63.13	2.875	3.500	500	1	6.375	42.93	2.625	3.250	1.000	1.000
88.382	2.875	3.500	500	1	6.375	42.93	2.625	3.250	1.000	1.000
101.0...	2.875	3.500	500	0	5.878	42.93	2.625	3.250	0.959	0.880

Calculate Exit

- Click the **Exit** button to close the dialog and return to the material dialog



Note. Values of effective length L_e can be used to manage the shear lag of the flanges.

Defining geometry for segment A

The geometric data for each segment and/or section along the bridge now needs to be defined starting with segment A.

- From the main menu select **Windows> 2 Geometry** to display the Geometry dialog.

Note at this stage it is not necessary to define all the sections in each segment that will be checked; it will be enough to define just one section for each segment, and input a distance to automatically obtain the corresponding effective width of slab from the shear lag slab table previously defined.



Note. No entered data is saved with a segment name until the **Add to list** button is clicked. This is done at the end of this section of the example.

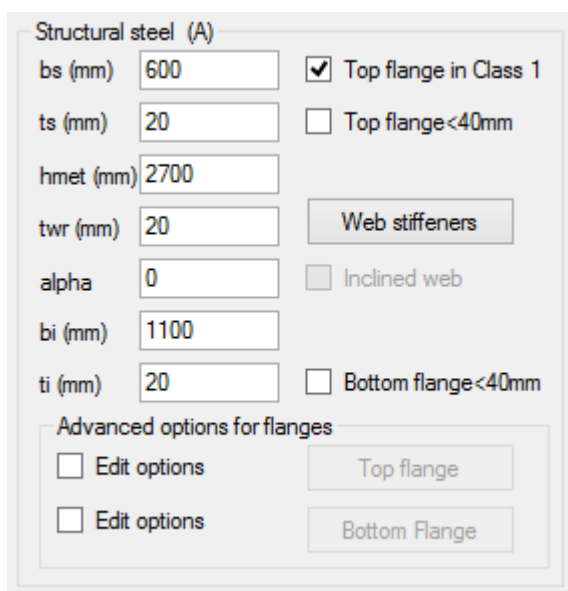
Segment name, sections, x-distances

- Input the data for the first Segment as shown in the following image.

The image shows a software dialog box titled 'Geometry'. It contains two input fields. The first field is labeled 'Segment name' and contains the text 'A'. The second field is labeled 'Sections (eg. Sec1,Sect2...) X (m) (es. X1,X2,...)' and contains the text 'S1' followed by a space and '0'.

Structural steel plate details

- With reference to the main table of girder dimensions, enter the following values:



Structural steel (A)

bs (mm) 600 ☒ Top flange in Class 1

ts (mm) 20 ☐ Top flange <40mm

hmet (mm) 2700

twr (mm) 20 **Web stiffeners**

alpha 0 ☐ Inclined web

bi (mm) 1100

ti (mm) 20 ☐ Bottom flange <40mm

Advanced options for flanges

☐ Edit options **Top flange**

☐ Edit options **Bottom Flange**

- Make sure that **Top flange in Class 1** is checked (ticked).

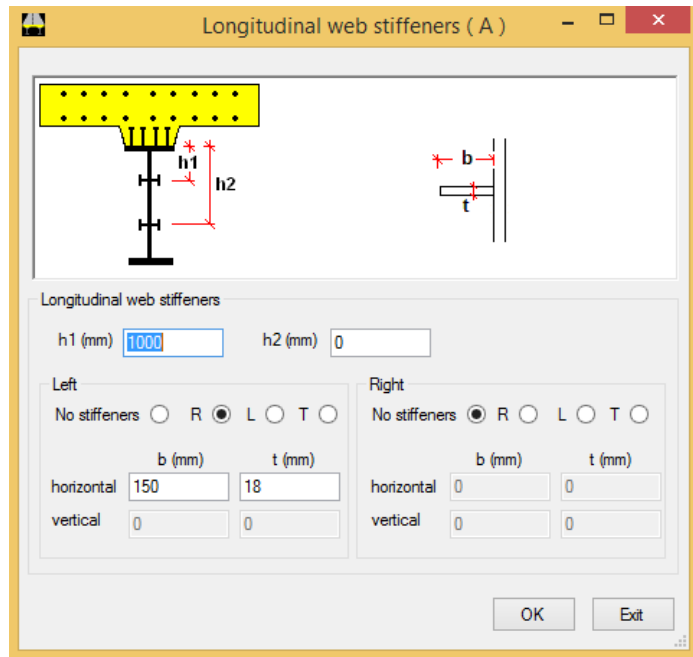


Note. The checkboxes for Top / Bottom flange <40mm should be only checked (ticked) for flanges that have a total thickness above 40mm, when they are made up from a number of plates each of a thickness less than 40mm. When done, ultimate and yielding tension for steel below 40mm thickness will be used.

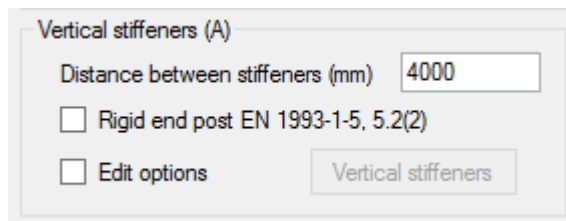
Longitudinal web stiffeners details

Properties of longitudinal stiffeners are necessary to perform web panel buckling calculations.

- Click the **Web stiffeners** button and input data as shown in the following image, then click **OK**.



Vertical web stiffener distance



Note. A vertical stiffeners check is optional and is not carried out in the example. Specifying the distance between vertical stiffeners is the only mandatory value in this panel.

Slab concrete

- In the Slab concrete panel right-click inside the bcls field and select **Calculate beff**. The width of slab will be computed using data previously defined in the shear lag table. Input other data as shown in the following image.

Slab concrete (A)			
bcls (mm)	5877.67	tcls (mm)	273
b1 (mm)	600	bsx (mm)	2766.96
hcop (mm)	75	☐ Consider haunch	

For the purposes of this example the section properties of the concrete haunch will be ignored in any calculations made, so it is left un-checked.

Reinforcing bars

Reinforcing bars (A)			
	bar diameter (mm)	bar spacing (mm)	bar cover (mm)
top layer	16	100	40
bottom layer	16	100	40

Shear connection

Shear connection (A)			
n (n°/m)	20	diameter (mm)	22
		height (mm)	260
Just class1 and 2 sections in the plastic zones			
Distance elastic-plastic section for ULS-min.		L (m)	0
Resulting compression in the concrete slab, at L from current section, for ULS M-min.		Fx (N)	0.000E+000

Fatigue data

Fatigue (A)			
<i>Damage equivalent factor (traffic)</i>	Steel (Bending)	λ_1	2.145 ?
	Steel (Shear)	λ_1	2.448
	Bars	$\lambda_{s,1}$	0.000
Traffic loading factor (Reinforcing bars)			0.000
Detail categories data (A)			

- λ_1 can be calculated automatically by clicking the Question mark button next to the input field and then, on the Damage equivalent factor dialog, select

the radio button for the **midspan section** and enter **50.5** for the Span length for moments and **20.2** for the Span length for shears.

Damage equivalent factor LAMDA1 for road bridge

Figure 9.7: Location of midspan or support

λ_1 , 9.5.2 (2) EN 1993-2, 2006(E)

			Bending moment	Shear force
at midspan		$2.55 - 0.7 (L-10) / 70$	L = length of span under consideration	L = 0.4 * span under consideration
at support	L < 30 m	$2.00 - 0.3 (L-10) / 20$	L = the mean of two adjacent spans	L = length of span under consideration
	L ≥ 30 m	$1.70 + 0.5 (L-30) / 50$		

Span length for moments (m) $\lambda_1 = 2.145$

Span length for shears (m) $\lambda_1 = 2.448$

OK Exit

Detail categories

- In the Fatigue data panel, select the **Detail categories data** button and input the data as shown on the following image, then click **OK**.

Data for fatigue verification of structural part (A)

Top flange	$\Delta\sigma_R$ (N/mm ²)	125	Table 8.1: Plain members and mechanically fastened joints (EN 1993-1-9)						
Bottom flange	$\Delta\sigma_R$ (N/mm ²)	125	Table 8.1: Plain members and mechanically fastened joints (EN 1993-1-9)						
Web	$\Delta\sigma_R$ (N/mm ²)	100	Table 8.1: Plain members and mechanically fastened joints (EN 1993-1-9)						
Top flange joint	$\Delta\sigma_R$ (N/mm ²)	112	t1 (mm)	20	t2 (mm)	0	e (mm)	0	Table 8.3: Transverse butt welds (EN 1993-1-9)
Bottom flange joint	$\Delta\sigma_R$ (N/mm ²)	112	t1 (mm)	20	t2 (mm)	0	e (mm)	0	Table 8.3: Transverse butt welds (EN 1993-1-9)
Web-Top flange	$\Delta\sigma_R$ (N/mm ²)	112	Table 8.2: Welded built-up sections (EN 1993-1-9)						
Web-Bottom flange	$\Delta\sigma_R$ (N/mm ²)	112	Table 8.2: Welded built-up sections (EN 1993-1-9)						
Vertical stiffener-Web	$\Delta\sigma_R$ (N/mm ²)	80	Table 8.4: Weld attachments and stiffeners (EN 1993-1-9)						
Vertical stiffener-Top flange	$\Delta\sigma_R$ (N/mm ²)	80	Table 8.4: Weld attachments and stiffeners (EN 1993-1-9)						
Vertical stiffener-Bottom flange	$\Delta\sigma_R$ (N/mm ²)	80	Table 8.4: Weld attachments and stiffeners (EN 1993-1-9)						
Horizontal stiffener-Web	$\Delta\sigma_R$ (N/mm ²)	80	Table 8.4: Weld attachments and stiffeners (EN 1993-1-9)						

OK Cancel

Adding data for segment A to the segment treeview

- Finally, click on the **Add to list** button located under the segment treeview to save all the data previously entered for Segment A.
- Before entering data for other segments, in the segment treeview **double-click** on the name of the segment A. All data previously input is now available for use in defining the next section.



Note. Double-clicking on a segment name in the Segment treeview panel will switch PontiEC4 from EDIT MODE to INPUT MODE. The status bar at the bottom-left of the screen will always show what mode is being used at any time.

Defining Segment C

With the status bar stating 'INPUT GEOMETRY:'

- In the Segment name field delete **A** and type **C**. Then, with reference to the following image, change or ensure that the dialog fields marked have entries as shown. Remember that the values of **bcls** and **bsx** can be calculated automatically by selecting **Calculate beff** from the context menu for both fields.

- When complete, click the **Add to list** button to save the data entered.

Defining Segment B

- Ensure that the dialog fields marked have the entries shown. When complete click the **Add to list** button to save the segment data entered.

Geometry

Segment name: B

Sections (eg. Sec1.Sect2...) X (m) (es. X1.X2...): S1 16.2

Structural steel (B)

bs (mm): 600 ☒ Top flange in Class 1

ts (mm): 30 ☐ Top flange <40mm

hmet (mm): 2700

twr (mm): 18 ☐ Web stiffeners

alpha: 0 ☐ Inclined web

bi (mm): 1100

ti (mm): 40 ☐ Bottom flange <40mm

Advanced options for flanges

☐ Edit options Top flange

☐ Edit options Bottom flange

Vertical stiffeners (B)

Distance between stiffeners (mm): 4000

☐ Rigid end post EN 1993-1-5, 5.2(2)

☐ Edit options Vertical stiffeners

Fatigue (B)

Damage equivalent factor (traffic):

Steel (Bending) λ_1 : 2.145

Steel (Shear) λ_1 : 2.448

Bars $\lambda_{s,1}$: 0.000

Traffic loading factor (Reinforcing bars): 0.000

Slab concrete (B)

bcls (mm): 6375 tcls (mm): 273

b1 (mm): 600 bxs (mm): 2875

hcop (mm): 75 ☐ Consider haunch

Reinforcing bars (B)

bar diameter (mm): bar spacing (mm): bar cover (mm)

top layer: 16 100 40

bottom layer: 16 100 40

Shear connection (B)

n (n°/m): 20 diameter (mm): 22 height (mm): 260

Just class 1 and 2 sections in the plastic zones

Distance elastic-plastic section for ULS-min. L (m): 0

Resulting compression in the concrete slab, at L from current section, for ULS M-min. Fx (N): 0.000E+000

Diagram: A cross-section diagram of a composite bridge deck. It shows a top flange with width 'bs', thickness 'ts', and total width 'bcls'. The web has thickness 'twr' and height 'hmet'. The bottom flange has width 'bi' and thickness 'ti'. The concrete slab has thickness 'tcls' and width 'bxs'. The total height of the concrete slab is 'hcop'. The diagram also shows the reinforcement bars (bs, ts, twr, hwt, bi, ti) and the shear connection (n, diameter, height).

Buttons: Add to list, Clear form

Segment E

- Ensure that the dialog fields marked have the entries shown. When complete click the **Add to list** button to save the segment data entered.

Geometry

Segment name: E

Sections (eg. Sec1.Sect2...) X (m) (es. X1.X2...): S1 25.252

Structural steel (E)

bs (mm): 600 ☒ Top flange in Class 1

ts (mm): 30 ☐ Top flange <40mm

hmet (mm): 2700

twr (mm): 18 ☐ Web stiffeners

alpha: 0 ☐ Inclined web

bi (mm): 1100

ti (mm): 50 ☐ Bottom flange <40mm

Advanced options for flanges

☐ Edit options Top flange

☐ Edit options Bottom flange

Vertical stiffeners (E)

Distance between stiffeners (mm): 4000

☐ Rigid end post EN 1993-1-5, 5.2(2)

☐ Edit options Vertical stiffeners

Fatigue (E)

Damage equivalent factor (traffic):

Steel (Bending) λ_1 : 2.145

Steel (Shear) λ_1 : 2.448

Bars $\lambda_{s,1}$: 0.000

Traffic loading factor (Reinforcing bars): 0.000

Slab concrete (E)

bcls (mm): 6375 tcls (mm): 273

b1 (mm): 600 bxs (mm): 2875

hcop (mm): 75 ☐ Consider haunch

Reinforcing bars (E)

bar diameter (mm): bar spacing (mm): bar cover (mm)

top layer: 16 100 40

bottom layer: 16 100 40

Shear connection (E)

n (n°/m): 20 diameter (mm): 22 height (mm): 260

Just class 1 and 2 sections in the plastic zones

Distance elastic-plastic section for ULS-min. L (m): 0

Resulting compression in the concrete slab, at L from current section, for ULS M-min. Fx (N): 0.000E+000

Diagram: A cross-section diagram of a composite bridge deck, similar to the one in the previous image, but with the segment name 'E' and the section 'S1' set to '25.252'.

Buttons: Add to list, Clear form

Segment D

- With reference to the following image change or ensure that the dialog fields highlighted have the entries shown.

Geometry

Segment name: D
Sections (eg. Sec1, Sect2...): X (m) (es. X1, X2...): 50.504

Structural steel (D)
 bs (mm): 1100 ☒ Top flange in Class 1
 ts (mm): 40 ☐ Top flange < 40mm
 hmet (mm): 2700
 twr (mm): 25
 alpha: 0 ☐ Inclined web
 bi (mm): 1200
 ti (mm): 50 ☐ Bottom flange < 40mm
 Advanced options for flanges:
☐ Edit options
☐ Edit options

Slab concrete (D)
 bcls (mm): 6281.5 ☐ tcls (mm): 273
 b1 (mm): 1100 ☐ bsx (mm): 2875
 hcop (mm): 75 ☐ Consider haunch

Reinforcing bars (D)
 bar diameter (mm) bar spacing (mm) bar cover (mm)
 top layer: 26 100 40
 bottom layer: 20 100 40

Fatigue (D)
 Damage equivalent factor (traffic):
 Steel (Bending) λ_1 : 1.905 ?
 Steel (Shear) λ_1 : 1.905
 Bars $\lambda_{s,1}$: 0.000
 Traffic loading factor (Reinforcing bars): 0.000

Shear connection (D)
 n (n²/m): 20 diameter (mm): 22 height (mm): 260
 Just class 1 and 2 sections in the plastic zones
 Distance elastic-plastic section for ULS-min. L (m): 0
 Resulting compression in the concrete slab, at L from current section, for ULS M-min. Fx (N): 0.000E+000

- Click on the Question mark button next to the Steel (Bending moment) field and specify a **support section** and a Span length for shears of **50.5** as shown in the following image.

Damage equivalent factor LAMDA1 for road bridge

Figure 9.7: Location of midspan or support

λ_1 , 9.5.2 (2) EN 1993-2, 2006(E)

			Bending moment	Shear force
at midspan		$2.55 - 0.7 (L-10) / 70$	L = length of span under consideration	L = 0.4 * span under consideration
at support	L < 30 m	$2.00 - 0.3 (L-10) / 20$	L = the mean of two adjacent spans	L = length of span under consideration
	L ≥ 30 m	$1.70 + 0.5 (L-30) / 50$		

Span length for moments (m) $\lambda_1 = 1.905$

Span length for shears (m) $\lambda_1 = 1.905$

OK Exit

- When complete click the **Add to list** button to save the segment data entered.

Save the entered data

- Select the **File> Save** menu item to save the PontiEC4 dialog data to the current working project folder.

Step 2: Export geometric properties from PontiEC4 to LUSAS

Prior to exporting geometric properties PontiEC4 needs to check that all the data that has been input is valid and then run part of the EC4 code check to calculate the required geometric properties of each defined cross section. This is automatically done when requesting to view the Results window/dialog and should always be done prior to exporting properties subsequently.

- On the main PontiEC4 menu select **Window> 4 Results** to set the **Results** dialog active. After a short delay whilst a partial code check is run, the geometric properties of each cross section will be calculated. This can be confirmed by selecting a Cross-section and Design combination with the **Geometric properties 2** tab active.

Results

Cross-sections and design combinations

	Fund. ULS	Mmax
A_S1		
C_S1		
B_S1		
E_S1		
D_S1		

Forces and Moments

Phase	N	V	M	T
1	0.00E+000	0.00E+000	0.00E+000	0.00E+000
2a	0.00E+000	0.00E+000	0.00E+000	0.00E+000
2b	0.00E+000	0.00E+000	0.00E+000	0.00E+000
Rit. Iso	0.00E+000	0.00E+000	0.00E+000	0.00E+000
2c	0.00E+000	0.00E+000	0.00E+000	0.00E+000
3a	0.00E+000	0.00E+000	0.00E+000	0.00E+000
DT. Iso	0.00E+000	0.00E+000	0.00E+000	0.00E+000
3b	0.00E+000	0.00E+000	0.00E+000	0.00E+000
Totale	0.00E+000	0.00E+000	0.00E+000	0.00E+000

Primary effects of Shrinkage and Thermal action

	ϵ	N	M	γ_v
Shrinkage	-3.366E-4	-7.04E+6	-5E+6	0
Thermal var.	0E+00	0E+00	0E+00	0

Additional bending moment induced by neutral axis shift

	Phase 1	Phase 2a	Phase 2b	Phase 2c	Phase 3a	Phase 3b
Cracked	0E+00	0E+00	0E+00	0E+00	0E+00	0E+00
Uncracked	0E+00	0E+00	0E+00	0E+00	0E+00	0E+00

Geometric properties 2

Plastic check | Stresses | Shear | Geometric properties 0 | Geometric properties 1 | **Geometric properties 2** | Domains Mpt-N | Studs. ULS, SLS | SLS. Web Breathing | FLS steel | FLS bars | Stiffeners

Web compres. zone, reduced for local and global buckling

Total reduction for cross section

Gross section geometric properties

	Phase 1	Phase 2a	Phase 2b	Phase 2c	Phase 3	Cracked
A	8.72E+4	1.994E+5	2.104E+5	1.823E+5	3.679E+5	1.108E+5
xG	1196.33	2161.29	2200.72	2091.17	2504.97	1562.09
DeltaxG	--	--	--	--	--	--
ly	9.036E+10	2.355E+11	2.414E+11	2.249E+11	2.879E+11	1.453E+11
Wy.0	-7.553E+7	-1.089E+8	-1.097E+8	-1.075E+8	-1.149E+8	-9.301E+7
Wy.1	-7.682E+7	-1.1E+8	-1.107E+8	-1.086E+8	-1.159E+8	-9.421E+7
Wy.3	6.09E+7	4.539E+8	5.037E+8	3.819E+8	1.645E+9	1.3E+8
Wy.4	6.009E+7	4.371E+8	4.835E+8	3.693E+8	1.476E+9	1.277E+8
Wy.5	--	3.837E+8	4.204E+8	3.288E+8	1.066E+9	1.198E+8
Wy.6	--	3.602E+8	3.93E+8	3.107E+8	9.286E+8	1.16E+8
Wy.7	--	2.781E+8	2.991E+8	2.453E+8	5.723E+8	1.005E+8
Wy.8	--	2.655E+8	2.849E+8	2.35E+8	5.302E+8	9.777E+7
Sy.1	2.61E+7	4.733E+7	4.82E+7	4.579E+7	5.489E+7	3.415E+7
Sy.2	3.994E+7	9.318E+7	9.575E+7	8.868E+7	1.166E+8	5.793E+7
Sy.3	1.792E+7	9.049E+7	9.345E+7	8.522E+7	1.163E+8	4.543E+7
Sy.4	-3.725E+9	8.414E+7	8.758E+7	7.803E+7	1.141E+8	3.189E+7

☒ Gross cross sec.
☐ Effective cross sec.

- When complete, select **File> Export geometric properties to LUSAS**

Export geometric properties to LUSAS

File name
C:\PROJECTS\CBDD\esempio\new\Materials_Sections_Rev0.vbs

Section type

☐ Composite sections
☒ Girders with top slab
☐ Flanges, Web
☐ Bridge wizard section

Eccentricity

☐ Distance middle plane of slab - neutral axis
☒ Distance extrados of the metal beam - neutral axis
☐ Null

Sections by phases

☒ Export Phase 1
☒ Export Phase 2a
☒ Export Phase 2b
☒ Export Phase 2c
☒ Export Phase 3a, 3b
☒ Export Cracked
☒ Export material

Segment definition

☒ Export one section for each segment
☐ Export all sections in the segment
☐ Export segment as tapered

Slab width constant

☐ Use maximum value
☐ Use average value
☒ Use minimum value

- On the dialog presented browse to a suitable project folder and enter a filename of “**Materials_Sections_Rev0.vbs**”.
- Select the **Girders with top slab** option.
- In the Eccentricity panel, select **Distance extrados of the metal beam - neutral axis**
- In the slab width, beam height panel, select **Assume slab width constant and use minimum value**,
- Finally, ensure the **Export material** check box is ticked, and click **OK** to finish.

The file created can be imported into LUSAS to create new, or overwrite any similarly named geometric and material properties.

Discussion: Modelling and Analysis with LUSAS

Because the primary aim of this particular example is to show the use of the PontiEC4 Composite Deck Designer and the interaction between it and LUSAS, no step-by-step instruction is provided for construction of the supplied LUSAS model that is used in this example. A separate example ‘Composite Highway Bridge design’ shows the LUSAS model building requirements to allow a design check with PontiEC4 to be carried out.

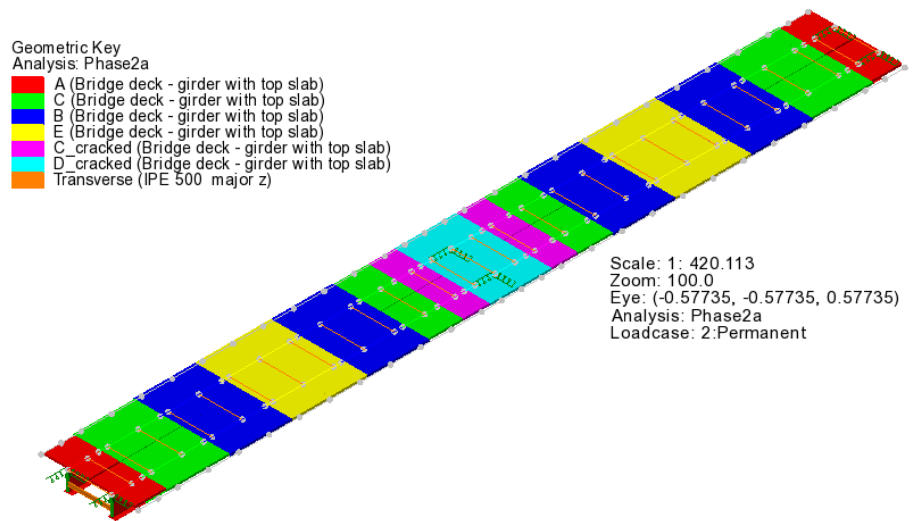
Several modelling approaches are possible in order to analyse a structure and obtain the forces and moments required for a design check of the composite sections. Possible options include grillage, ribbed-plate, 3D shell models, or line beam methods. Users can choose the most suitable approach to their method of working, and obtain forces and moments as required.

Modelling

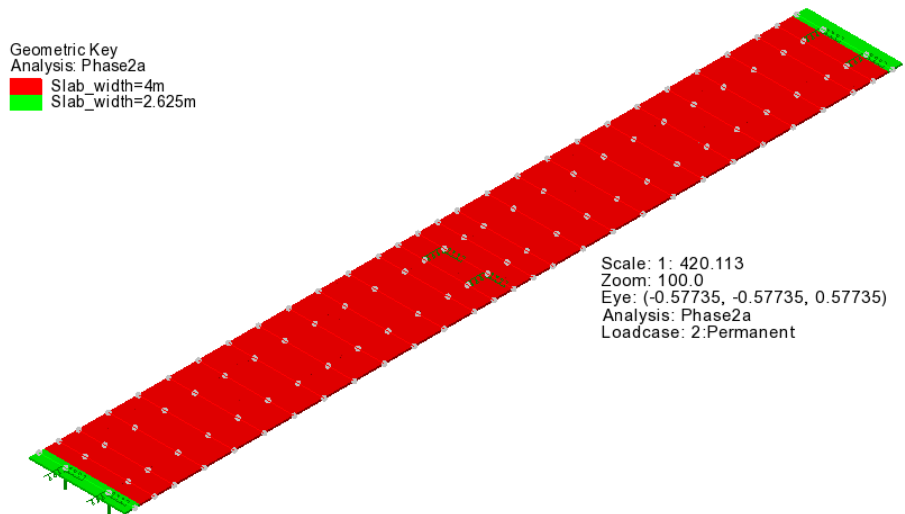
The model supplied for use with this example is a grillage model. Longitudinal grillage members have been placed at the location of the two girders and at the deck edges. Transverse members have been placed at the locations of the transverse beams and at the deck ends. Between the main girders, coincident lines are used to model both the transverse beams and the transverse segments of the slab deck separately.

Meshing

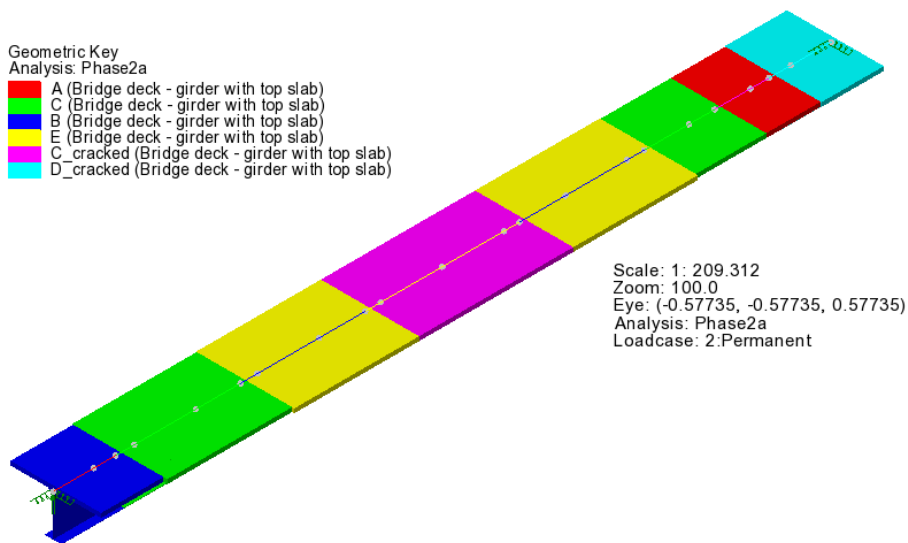
BMI21 elements have been used throughout, and all mesh lies on the extrados of the metal beam: the cross sections of the main girders and transverse beams are given an appropriate eccentricity within LUSAS. The edge longitudinal lines have a ‘Null’ mesh (element type = None) and are included in the model to enable processing of the discrete loads. The following figures show the model mesh with rendering.



Longitudinal and transverse beams



Transverse slabs



Details of half external beam

Geometric properties

Geometric properties are imported into LUSAS from PontiEC4 for each of the design phases considered. Eccentricities for sections are calculated as part of the PontiEC4 export process, with the extrados of the metal beam sitting on the nodal line if the appropriate option was selected during export. In the LUSAS model supplied, these section properties have already been assigned to corresponding lines of the model. Note that re-importing properties of the same name that have already been assigned to a model simply overwrites the existing definitions.

Material properties

Material properties, defined in PontiEC4, are imported into the LUSAS model as base Isotropic material properties and Bridge Deck (Grillage) material. These attributes have already been assigned to corresponding lines of the model.

Supports

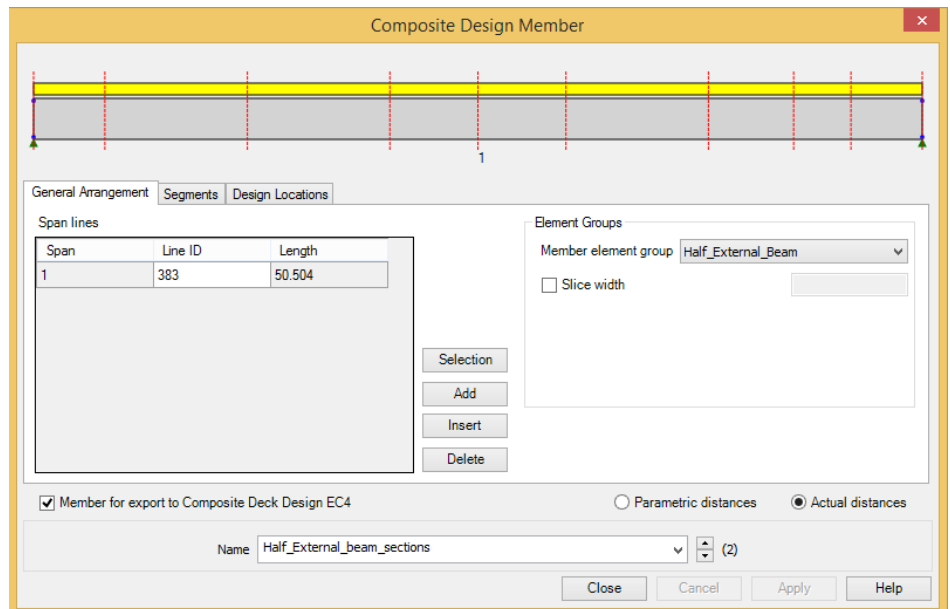
Span lengths, whilst defined in PontiEC4, are not imported into LUSAS to derive support positions. Instead, suitable supports must be defined within LUSAS and assigned to the relevant features of the LUSAS model. In this worked example all supports are fixed in the Z-direction and have a spring stiffness of 3.03E6 in the X and Y directions.

Composite design member

The composite design member is not imported into LUSAS, so to identify the sections at which design checks will be undertaken in PontiEC4, it must be defined within

LUSAS by using the ‘Utilities > Design > Composite Design Member’ menu item. The relevant data (Spans, Segments and Design locations) are shown in the figures below. Given the symmetry of the structure, only the first span and data for half of the external beam length is described in the Composite Design member (CDM). The elements to slice are collected in a dedicated group named ‘Half_External_Beam’. The CDM utility is also used for the definition of Slice resultants beam/shell and Influence surfaces for the DMI and VLO analysis.

When defining a design member, ensure the option ‘Member for export to Composite Deck Design EC4’ is checked to switch to the suitable display mode, and ensure ‘Actual distances’ is selected too.



General Arrangement Tab – First span and half external beam

For each Segment, the following Design section name and relative Segment length would be input.

Segment No.	Design section	Segment length
1	1:A	4.05
2	2:C	8.1
3	3:B	8.1
4	4:E	10.004
5	3:B	8.1
6	2:C	4.844
7	7:C_cracked	3.256
8	8:D_cracked	4.05

Composite Design Member

General Arrangement Segments Design Locations

Segments represent a portion of the span defined by a constant or tapered cross section

Span 1

Seg No.	Design Section	Segment Length
1	1:A	4.05
2	2:C	8.1
3	3:B	8.1
4	4:E	10.004
5	3:B	8.1
6	2:C	4.844
7	7:C_cracked	3.256

☒ Member for export to Composite Deck Design EC4 ☐ Parametric distances ☒ Actual distances

Name: Half_External_beam_sections (2)

Buttons: Close, Cancel, Apply, Help

Segment Tab – Design Section and Segment Length

Design locations (where design checks will be carried out) can be specified to be at segment ends and, additionally, at distances measured from the start of each span

Composite Design Member

General Arrangement Segments Design Locations

Each section design location requires additional slices to be taken in the unbraced length to determine moment shapes, these can cause the design checks to slow down for large models

Section design locations

☒ At segment ends

Additional section design locations

Span	Location
1	25.252

☐ None ☒ Specify location ☐ Specify spacing ☐ Specify number

Buttons: Add, Insert, Delete

☒ Member for export to Composite Deck Design EC4 ☐ Parametric distances ☒ Actual distances

Name: Half_External_beam_sections (2)

Buttons: Close, Cancel, Apply, Help

Design Locations Tab – Sections to Check in PontiEC4

Analysis

The LUSAS multiple analysis facility is used, with separate analyses being defined to represent the four design phases that need to be considered, and two specialized analysis to handle traffic loading optimization. These are:

1. **Phase 1** – Where **the slab is present just as weight**; the cross sections are only steel and not composite. The concreting order of the slab segments has been ignored, supposing that the concreting of the slab occurs at the same time.
2. **Phase 2a** - The composite section has been calculated using the **long term** properties of the concrete, and the long term loadings considered are the **permanent** ones.
3. **Phase 2b** - The composite section has been calculated using the **long term** properties of the concrete, and the long term loading considered is the **shrinkage of the concrete**.
4. **Phase 3** - The composite section has been calculated using the **short term** properties of the concrete. **All variable actions** are considered in this analysis. A series of loadcases has been created to represent thermal action, fatigue vehicle FLM3 and wind effects.
5. **DMI Analysis** - The composite section has been calculated using the **short term** properties of the concrete. The influence surfaces of Moments and Shear forces have been assigned to the sections to check in PontiEC4, by definition of a **Composite design member**.
6. **VLO Analysis** - The composite section has been calculated using the **short term** properties of the concrete. A series of loadcases has been created to represent traffic loading effects (Group 1a - LM1) in characteristic and frequent combination.

Notes

- ☐ Phase 2c in PontiEC4 represents the actions arising from prestressing and is therefore not required in this example.
- ☐ Transverse slab sections have been calculated based on the appropriate width of slab with appropriate time-dependant concrete properties applied in each phase. For the purposes of this example any possible reduction in torsional stiffness for these sections is not considered.

Loading and load cases

The following table summarizes the loadings and load cases used for each analysis.

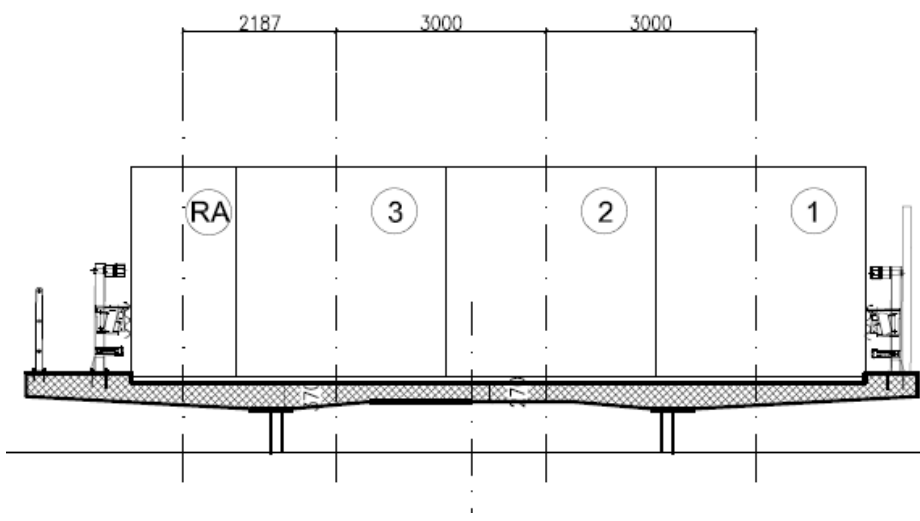
Analysis	Loadings	Loadcases
Phase1	PHASE 1 Longitudinal and transv. beams Weight (m/s ²) PHASE 1 Stud Weight (N/m) <u> </u> for beam PHASE 1 Slab Weight (N/m) <u> </u> for beam	Self_Weight

Analysis	Loadings	Loadcases
	PHASE 1 Curb Weight $w=750\text{mm}$ (N/m) _external beam PHASE 1 Curb Weight $w=1500\text{mm}$ (N/m) _internal beam	
Phase2a	PHASE 2a Permanent loading (N/m) _x _external _beam PHASE 2a Permanent loading (N/m) _x _internal beam	Permanent
Phase2b	PHASE2b Internal support settlement	Settlement
	Shrinkage_A_S1 Shrinkage_C_S1 Shrinkage_B_S1 Shrinkage_E_S1 Shrinkage_S_S1	Shrinkage
	PHASE 3a Thermal Heat	Thermal Heat
	PHASE 3a Thermal Cold	Thermal Cold
	PHASE 3b Wind V (N/m) _x _internal _beam PHASE 3b Wind V (N/m) _x _external _beam	Wind
Phase3	FLM3 (Eurocode Fatigue Load Model 3 120000N)	From ID=520 Pos=1 to ID=629 Pos=110
DMI	Influence surfaces of sections to check (frequent and characteristic combination)	
VLO Analysis	LM1 Tandem and udl	From ID=410 to ID=477

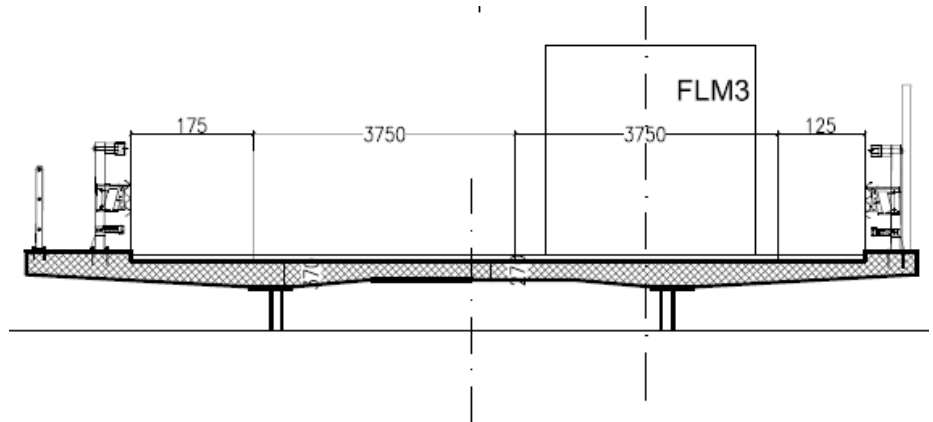
Traffic loading

The traffic loading LM1 has been generated from a VLO Analysis performed to maximize bending moments and shears in the right (external) girder beam. The influence surface related to the VLO Analysis has been defined by the Composite Design member utility.

The Fatigue model loading 3 has been generated from the vehicle libraries in LUSAS and positioned using the Moving Load Generator using an incremental distance of 1m. The lane position is the actual position of the heavy traffic lane.



Notional lanes for Load Model 1 (ULS and SLS)



Fatigue Load Model 3 positioning

Shrinkage

Shrinkage effects in analysis Phase2b have been calculated applying a thermal gradient, equivalent to an imposed curvature. The forces and moments arising from this model represent the hyperstatic effect of the shrinkage. The primary effects of shrinkage are computed directly in PontiEC4 for each section.

Smart combinations

In order to define the design envelopes for each Phase and for each Limit State, the following smart combinations and envelopes have been defined.

Preliminary combinations

Loadcase ID: 288 Title: Thermal_effect_k

Sub Type: Smart Combination

Loadcases to consider: 1

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
5	0	1.0	1.0	Thermal Heat	
6	0	1.0	1.0	Thermal Cold	

Loadcase ID: 300 Title: Wind_k

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
287	0	-1.0	1.0	Wind	

Assessment of a Composite Bridge Deck to Eurocodes

Loadcase ID: 372 Title: VLO Run 1 ~ Characteristic

Sub Type: Envelope

Loadcase	ResultsFile	Title	Type
376	0	PontIEC4 1 - M (My) @ 0 - Negative - Characteristic	
377	0	PontIEC4 1 - M (My) @ 4.05 (-X) - Negative - Characteristic	
378	0	PontIEC4 1 - M (My) @ 4.05 (+X) - Negative - Characteristic	
379	0	PontIEC4 1 - M (My) @ 12.15 (-X) - Negative - Characteristic	
380	0	PontIEC4 1 - M (My) @ 12.15 (+X) - Negative - Characteristic	
381	0	PontIEC4 1 - M (My) @ 20.25 (-X) - Negative - Characteristic	
382	0	PontIEC4 1 - M (My) @ 20.25 (+X) - Negative - Characteristic	
383	0	PontIEC4 1 - M (My) @ 25.252 (-X) - Negative - Characteristic	
384	0	PontIEC4 1 - M (My) @ 25.252 (+X) - Negative - Characteristic	
385	0	PontIEC4 1 - M (My) @ 30.254 (-X) - Negative - Characteristic	
386	0	PontIEC4 1 - M (My) @ 30.254 (+X) - Negative - Characteristic	
387	0	PontIEC4 1 - M (My) @ 38.354 (-X) - Negative - Characteristic	
388	0	PontIEC4 1 - M (My) @ 38.354 (+X) - Negative - Characteristic	
389	0	PontIEC4 1 - M (My) @ 43.198 - Negative - Characteristic	
390	0	PontIEC4 1 - M (My) @ 46.454 (-X) - Negative - Characteristic	
391	0	PontIEC4 1 - M (My) @ 46.454 (+X) - Negative - Characteristic	
392	0	PontIEC4 1 - M (My) @ 50.504 - Negative - Characteristic	
410	0	PontIEC4 1 - M (My) @ 0 - Positive - Characteristic	
411	0	PontIEC4 1 - M (My) @ 4.05 (-X) - Positive - Characteristic	
412	0	PontIEC4 1 - M (My) @ 4.05 (+X) - Positive - Characteristic	
413	0	PontIEC4 1 - M (My) @ 12.15 (-X) - Positive - Characteristic	
414	0	PontIEC4 1 - M (My) @ 12.15 (+X) - Positive - Characteristic	
415	0	PontIEC4 1 - M (My) @ 20.25 (-X) - Positive - Characteristic	
416	0	PontIEC4 1 - M (My) @ 20.25 (+X) - Positive - Characteristic	
417	0	PontIEC4 1 - M (My) @ 25.252 (-X) - Positive - Characteristic	
418	0	PontIEC4 1 - M (My) @ 25.252 (+X) - Positive - Characteristic	
419	0	PontIEC4 1 - M (My) @ 30.254 (-X) - Positive - Characteristic	
420	0	PontIEC4 1 - M (My) @ 30.254 (+X) - Positive - Characteristic	
421	0	PontIEC4 1 - M (My) @ 38.354 (-X) - Positive - Characteristic	
422	0	PontIEC4 1 - M (My) @ 38.354 (+X) - Positive - Characteristic	
423	0	PontIEC4 1 - M (My) @ 43.198 - Positive - Characteristic	
424	0	PontIEC4 1 - M (My) @ 46.454 (-X) - Positive - Characteristic	
425	0	PontIEC4 1 - M (My) @ 46.454 (+X) - Positive - Characteristic	
426	0	PontIEC4 1 - M (My) @ 50.504 - Positive - Characteristic	
444	0	PontIEC4 1 - V (Fz) @ 0 - Negative - Characteristic	
445	0	PontIEC4 1 - V (Fz) @ 4.05 (-X) - Negative - Characteristic	
446	0	PontIEC4 1 - V (Fz) @ 4.05 (+X) - Negative - Characteristic	
447	0	PontIEC4 1 - V (Fz) @ 12.15 (-X) - Negative - Characteristic	
448	0	PontIEC4 1 - V (Fz) @ 12.15 (+X) - Negative - Characteristic	
449	0	PontIEC4 1 - V (Fz) @ 20.25 (-X) - Negative - Characteristic	
450	0	PontIEC4 1 - V (Fz) @ 20.25 (+X) - Negative - Characteristic	
451	0	PontIEC4 1 - V (Fz) @ 25.252 (-X) - Negative - Characteristic	
452	0	PontIEC4 1 - V (Fz) @ 25.252 (+X) - Negative - Characteristic	
453	0	PontIEC4 1 - V (Fz) @ 30.254 (-X) - Negative - Characteristic	
454	0	PontIEC4 1 - V (Fz) @ 30.254 (+X) - Negative - Characteristic	
455	0	PontIEC4 1 - V (Fz) @ 38.354 (-X) - Negative - Characteristic	
456	0	PontIEC4 1 - V (Fz) @ 38.354 (+X) - Negative - Characteristic	
457	0	PontIEC4 1 - V (Fz) @ 43.198 - Negative - Characteristic	
458	0	PontIEC4 1 - V (Fz) @ 46.454 (-X) - Negative - Characteristic	
459	0	PontIEC4 1 - V (Fz) @ 46.454 (+X) - Negative - Characteristic	
460	0	PontIEC4 1 - V (Fz) @ 50.504 - Negative - Characteristic	
478	0	PontIEC4 1 - V (Fz) @ 0 - Positive - Characteristic	
479	0	PontIEC4 1 - V (Fz) @ 4.05 (-X) - Positive - Characteristic	
480	0	PontIEC4 1 - V (Fz) @ 4.05 (+X) - Positive - Characteristic	
481	0	PontIEC4 1 - V (Fz) @ 12.15 (-X) - Positive - Characteristic	
482	0	PontIEC4 1 - V (Fz) @ 12.15 (+X) - Positive - Characteristic	
483	0	PontIEC4 1 - V (Fz) @ 20.25 (-X) - Positive - Characteristic	
484	0	PontIEC4 1 - V (Fz) @ 20.25 (+X) - Positive - Characteristic	
485	0	PontIEC4 1 - V (Fz) @ 25.252 (-X) - Positive - Characteristic	
486	0	PontIEC4 1 - V (Fz) @ 25.252 (+X) - Positive - Characteristic	
487	0	PontIEC4 1 - V (Fz) @ 30.254 (-X) - Positive - Characteristic	
488	0	PontIEC4 1 - V (Fz) @ 30.254 (+X) - Positive - Characteristic	
489	0	PontIEC4 1 - V (Fz) @ 38.354 (-X) - Positive - Characteristic	
490	0	PontIEC4 1 - V (Fz) @ 38.354 (+X) - Positive - Characteristic	
491	0	PontIEC4 1 - V (Fz) @ 43.198 - Positive - Characteristic	
492	0	PontIEC4 1 - V (Fz) @ 46.454 (-X) - Positive - Characteristic	
493	0	PontIEC4 1 - V (Fz) @ 46.454 (+X) - Positive - Characteristic	
494	0	PontIEC4 1 - V (Fz) @ 50.504 - Positive - Characteristic	

Assessment of a Composite Bridge Deck to Eurocodes

Loadcase ID: 374 Title: VLO Run 1 ~ Frequent

Sub Type: Envelope

Loadcase	ResultsFile	Title	Type
393	0	PontiEC4 1 - M (My) @ 0 - Negative - Frequent	
394	0	PontiEC4 1 - M (My) @ 4.05 (-X) - Negative - Frequent	
395	0	PontiEC4 1 - M (My) @ 4.05 (+X) - Negative - Frequent	
396	0	PontiEC4 1 - M (My) @ 12.15 (-X) - Negative - Frequent	
397	0	PontiEC4 1 - M (My) @ 12.15 (+X) - Negative - Frequent	
398	0	PontiEC4 1 - M (My) @ 20.25 (-X) - Negative - Frequent	
399	0	PontiEC4 1 - M (My) @ 20.25 (+X) - Negative - Frequent	
400	0	PontiEC4 1 - M (My) @ 25.252 (-X) - Negative - Frequent	
401	0	PontiEC4 1 - M (My) @ 25.252 (+X) - Negative - Frequent	
402	0	PontiEC4 1 - M (My) @ 30.254 (-X) - Negative - Frequent	
403	0	PontiEC4 1 - M (My) @ 30.254 (+X) - Negative - Frequent	
404	0	PontiEC4 1 - M (My) @ 38.354 (-X) - Negative - Frequent	
405	0	PontiEC4 1 - M (My) @ 38.354 (+X) - Negative - Frequent	
406	0	PontiEC4 1 - M (My) @ 43.198 - Negative - Frequent	
407	0	PontiEC4 1 - M (My) @ 46.454 (-X) - Negative - Frequent	
408	0	PontiEC4 1 - M (My) @ 46.454 (+X) - Negative - Frequent	
409	0	PontiEC4 1 - M (My) @ 50.504 - Negative - Frequent	
427	0	PontiEC4 1 - M (My) @ 0 - Positive - Frequent	
428	0	PontiEC4 1 - M (My) @ 4.05 (-X) - Positive - Frequent	
429	0	PontiEC4 1 - M (My) @ 4.05 (+X) - Positive - Frequent	
430	0	PontiEC4 1 - M (My) @ 12.15 (-X) - Positive - Frequent	
431	0	PontiEC4 1 - M (My) @ 12.15 (+X) - Positive - Frequent	
432	0	PontiEC4 1 - M (My) @ 20.25 (-X) - Positive - Frequent	
433	0	PontiEC4 1 - M (My) @ 20.25 (+X) - Positive - Frequent	
434	0	PontiEC4 1 - M (My) @ 25.252 (-X) - Positive - Frequent	
435	0	PontiEC4 1 - M (My) @ 25.252 (+X) - Positive - Frequent	
436	0	PontiEC4 1 - M (My) @ 30.254 (-X) - Positive - Frequent	
437	0	PontiEC4 1 - M (My) @ 30.254 (+X) - Positive - Frequent	
438	0	PontiEC4 1 - M (My) @ 38.354 (-X) - Positive - Frequent	
439	0	PontiEC4 1 - M (My) @ 38.354 (+X) - Positive - Frequent	
440	0	PontiEC4 1 - M (My) @ 43.198 - Positive - Frequent	
441	0	PontiEC4 1 - M (My) @ 46.454 (-X) - Positive - Frequent	
442	0	PontiEC4 1 - M (My) @ 46.454 (+X) - Positive - Frequent	
443	0	PontiEC4 1 - M (My) @ 50.504 - Positive - Frequent	
461	0	PontiEC4 1 - V (Fz) @ 0 - Negative - Frequent	
462	0	PontiEC4 1 - V (Fz) @ 4.05 (-X) - Negative - Frequent	
463	0	PontiEC4 1 - V (Fz) @ 4.05 (+X) - Negative - Frequent	
464	0	PontiEC4 1 - V (Fz) @ 12.15 (-X) - Negative - Frequent	
465	0	PontiEC4 1 - V (Fz) @ 12.15 (+X) - Negative - Frequent	
466	0	PontiEC4 1 - V (Fz) @ 20.25 (-X) - Negative - Frequent	
467	0	PontiEC4 1 - V (Fz) @ 20.25 (+X) - Negative - Frequent	
468	0	PontiEC4 1 - V (Fz) @ 25.252 (-X) - Negative - Frequent	
469	0	PontiEC4 1 - V (Fz) @ 25.252 (+X) - Negative - Frequent	
470	0	PontiEC4 1 - V (Fz) @ 30.254 (-X) - Negative - Frequent	
471	0	PontiEC4 1 - V (Fz) @ 30.254 (+X) - Negative - Frequent	
472	0	PontiEC4 1 - V (Fz) @ 38.354 (-X) - Negative - Frequent	
473	0	PontiEC4 1 - V (Fz) @ 38.354 (+X) - Negative - Frequent	
474	0	PontiEC4 1 - V (Fz) @ 43.198 - Negative - Frequent	
475	0	PontiEC4 1 - V (Fz) @ 46.454 (-X) - Negative - Frequent	
476	0	PontiEC4 1 - V (Fz) @ 46.454 (+X) - Negative - Frequent	
477	0	PontiEC4 1 - V (Fz) @ 50.504 - Negative - Frequent	
495	0	PontiEC4 1 - V (Fz) @ 0 - Positive - Frequent	
496	0	PontiEC4 1 - V (Fz) @ 4.05 (-X) - Positive - Frequent	
497	0	PontiEC4 1 - V (Fz) @ 4.05 (+X) - Positive - Frequent	
498	0	PontiEC4 1 - V (Fz) @ 12.15 (-X) - Positive - Frequent	
499	0	PontiEC4 1 - V (Fz) @ 12.15 (+X) - Positive - Frequent	
500	0	PontiEC4 1 - V (Fz) @ 20.25 (-X) - Positive - Frequent	
501	0	PontiEC4 1 - V (Fz) @ 20.25 (+X) - Positive - Frequent	
502	0	PontiEC4 1 - V (Fz) @ 25.252 (-X) - Positive - Frequent	
503	0	PontiEC4 1 - V (Fz) @ 25.252 (+X) - Positive - Frequent	
504	0	PontiEC4 1 - V (Fz) @ 30.254 (-X) - Positive - Frequent	
505	0	PontiEC4 1 - V (Fz) @ 30.254 (+X) - Positive - Frequent	
506	0	PontiEC4 1 - V (Fz) @ 38.354 (-X) - Positive - Frequent	
507	0	PontiEC4 1 - V (Fz) @ 38.354 (+X) - Positive - Frequent	
508	0	PontiEC4 1 - V (Fz) @ 43.198 - Positive - Frequent	
509	0	PontiEC4 1 - V (Fz) @ 46.454 (-X) - Positive - Frequent	
510	0	PontiEC4 1 - V (Fz) @ 46.454 (+X) - Positive - Frequent	
511	0	PontiEC4 1 - V (Fz) @ 50.504 - Positive - Frequent	

ULS design combinations

Loadcase ID: 302 Title: ULS_Fundamental_Phase1

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
1	0	1.0	1.35	Self Weight	

Loadcase ID: 304 Title: ULS_Fundamental_Phase2a

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
2	0	1.0	1.35	Permanent	

Loadcase ID: 306 Title: ULS_Fundamental_Phase2b

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
3	0	0.0	1.2	Settlement	
4	0	1.2	1.2	Shrinkage	

Loadcase ID: 308 Title: ULS_Fundamental_Phase3a

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
288	0	1.2	1.2	Thermal effect k (Max)	
289	0	1.2	1.2	Thermal effect k (Min)	

Loadcase ID: 310 Title: ULS_Fundamental_Phase3b

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
300	0	0.9	0.9	Wind k (Max)	
301	0	0.9	0.9	Wind k (Min)	
372	0	0.0	1.35	VLO Run 1 ~ Characteristic (Max)	
373	0	0.0	1.35	VLO Run 1 ~ Characteristic (Min)	

SLS characteristic design combinations

Loadcase ID: 312 Title: SLS_Characteristic_Phase1

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
1	0	1.0	1.0	Self Weight	

Loadcase ID: 314 Title: SLS_Characteristic_Phase2a

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
2	0	1.0	1.0	Permanent	

Loadcase ID: 316 Title: SLS_Characteristic_Phase2b

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
3	0	0.0	1.0	Settlement	
4	0	1.0	1.0	Shrinkage	

Loadcase ID: 318 Title: SLS_Characteristic_Phase3a

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
288	0	1.0	2.0	Thermal effect k (Max)	
289	0	1.0	1.0	Thermal effect k (Min)	

Loadcase ID: 320 Title: SLS_Characteristic_Phase3b

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
300	0	0.6	0.6	Wind k (Max)	
301	0	0.6	0.6	Wind k (Min)	
372	0	0.0	1.0	VLO Run 1 ~ Characteristic (Max)	
373	0	0.0	1.0	VLO Run 1 ~ Characteristic (Min)	

SLS frequent design combinations

Loadcase ID: 322 Title: SLS_Frequent_Phase1

Sub Type: Smart Combination

Loadcases to consider: 1

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
1	0	1.0	1.0	Self Weight	

Loadcase ID: 324 Title: SLS_Frequent_Phase2a

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
2	0	1.0	1.0	Permanent	

Loadcase ID: 326 Title: SLS_Frequent_Phase2b

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
3	0	0.0	1.0	Settlement	
4	0	1.0	1.0	Shrinkage	

Loadcase ID: 328 Title: SLS_Frequent_Phase3a

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
5	0	1.0	1.0	Thermal Heat	
6	0	1.0	1.0	Thermal Cold	

Loadcase ID: 330 Title: SLS_Frequent_Phase3b

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
374	0	0.0	1.0	VLO Run 1 ~ Frequent (Max)	
375	0	0.0	1.0	VLO Run 1 ~ Frequent (Min)	

Fatigue design combinations

Loadcase ID: 332 Title: Fatigue_Phase3b

Sub Type: Smart Combination

Loadcases to consider: 1

Variable Loadcases: All

Loadcase	ResultsFile	BeneficialFactor	AdverseFactor	Title	Type
520	0	1.0	1.0	LoadID=23 Line=387 Dir=Fwd Pos=1	
521	0	1.0	1.0	LoadID=23 Line=387 Dir=Fwd Pos=2	
627	0	1.0	1.0	LoadID=23 Line=387 Dir=Fwd Pos=108	
628	0	1.0	1.0	LoadID=23 Line=387 Dir=Fwd Pos=109	
629	0	1.0	1.0	LoadID=23 Line=387 Dir=Fwd Pos=110	

Step 3: Use LUSAS to calculate the forces and moments for all limit state combinations

Associated Files



- ☐ **Materials_Sections_Rev0.vbs** - A file created by PontiEC4 containing materials and geometric section data to import into LUSAS. A version of this file is supplied also, but is only for use in case of difficulties in manually creating the appropriate data
- ☐ **Composite_Bridge_Deck.mdl** – A supplied LUSAS model file that is to be used to analyse the structure. After solving this model, analysis results are exported from LUSAS or use in PontiEC4 to undertake design checks.

Open the associated LUSAS model

- Run LUSAS Modeller and select the **File> Open** menu item, and open the supplied model named **Composite_Bridge_Deck.mdl** which is located in the **\<PontiEC4 Installation Folder>\Help\Examples\LUSAS_Example** directory.
- Select the **File> Save as** menu item and save the LUSAS model data to the same working project folder as that used to save the PontiEC4 dialog data earlier) keeping the same file name **Composite_Bridge_Deck.mdl**

Importing the section properties from Ponti EC4

- Select **File> Script> Run Script** and select the PontiEC4 script file “**Materials_Sections_Rev0.vbs**” that was created in the working project folder earlier. This imports the geometric section data from PontiEC4.



Note. In this worked example, the geometric beam properties calculated by Ponti EC4 have already been assigned to the relevant lines in the supplied LUSAS model, so re-importing properties of the same name that have already been assigned simply overwrites the existing definitions. Click **Yes** to overwrite each existing definition.

When importing geometric properties for the first time on a real project, the attributes created would need to be assigned to relevant lines of your model. Cracked section properties would be assigned to appropriate lines in the LUSAS model in accordance with the requirements of the code.

Looking at the Analysis treeview

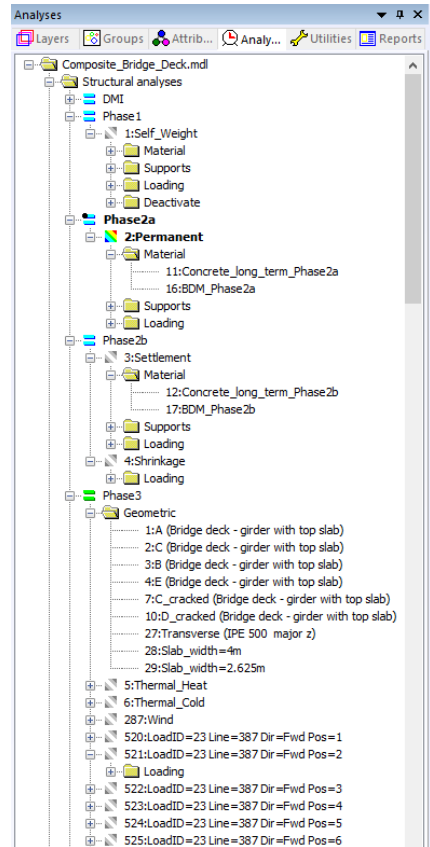
In the Analyses  treeview, notice that Phase 3 has been defined as the 'base' analysis and this shows the geometric assignments made for this, as well as the material properties for this phase.

Material properties for other phases can be seen within each Analysis Phase entry.

The stiffness of the composite section for each phase is determined from the stiffness of the girder and slab, together with the corresponding girder and slab materials assigned to the same line.

- Press the **Solve** button  to compute all loadcases in all the analyses.

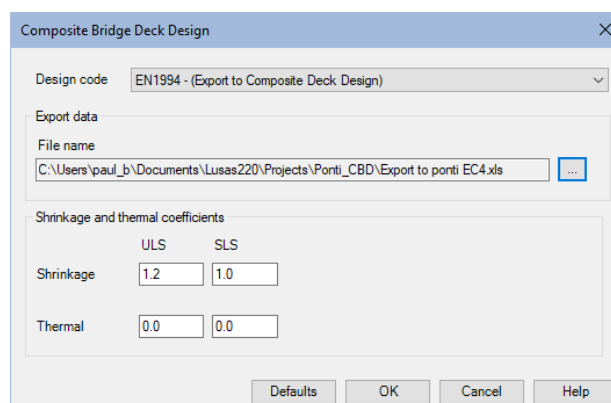
The model contains a large number of loadcases. After a short wait whilst they are all evaluated, results will be obtained.



Step 4: Export the force and moment data from LUSAS

Selection of composite bridge deck design Code

The composite deck design code is set via the **Design > Composite Bridge Deck Design** menu item, along with options for the export path and the factors for primary effects of shrinkage and thermal effects.



- Select **EN1994 – (Export to Composite Deck Design)** in the drop down list.
- Click the ellipsis button  to choose the folder and name of the xls file to create (e.g.: <current work directory>**Export to ponti EC4.xls**), pressing the **Save** button to return to the main dialog
- Input a Shrinkage coefficient of **1.2** for the ULS, and **1.0** for the SLS.
- Ensure that Thermal coefficient values of **0** are used for both the ULS and SLS.

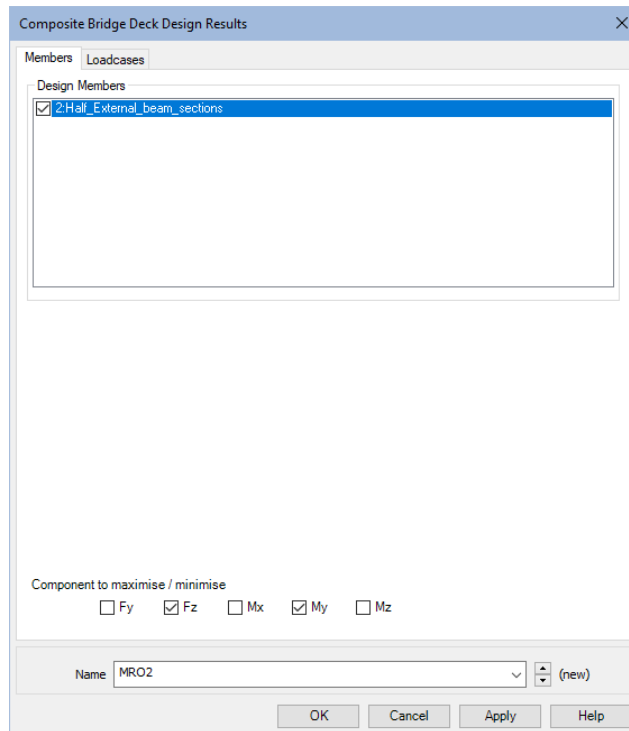


Note. These coefficients will be applied to the characteristic primary effects of the shrinkage as directly calculated in PontiEC4. The isostatic effect will be neglected ($\gamma \cdot \psi = 0$) in the cracked section (where Slab treatment = Cracked in Tension). The hyperstatic effect of the shrinkage comes from the LUSAS model and the combination factors have been already applied in the LUSAS smart combinations.

- Then, finally, click **OK** to return to LUSAS Modeller.

Export to Composite Deck Design

- Select the **Design > Export to Eurocode Composite Deck Design** menu item. This utility is for specifying the loadcases from which the forces and moments will be exported to the PontiEC4 spreadsheet.



- In the **Members** tab, select the Design Member of interest.
- In the ‘Component to maximize/minimize’ panel ensure **Fz** and **My** are checked. (These are the components that will be considered in the smart combinations)
- Select the **Loadcases** tab where input (or editing) of Design combinations for each Limit States and Phases is done. The dialog will appear as shown below.

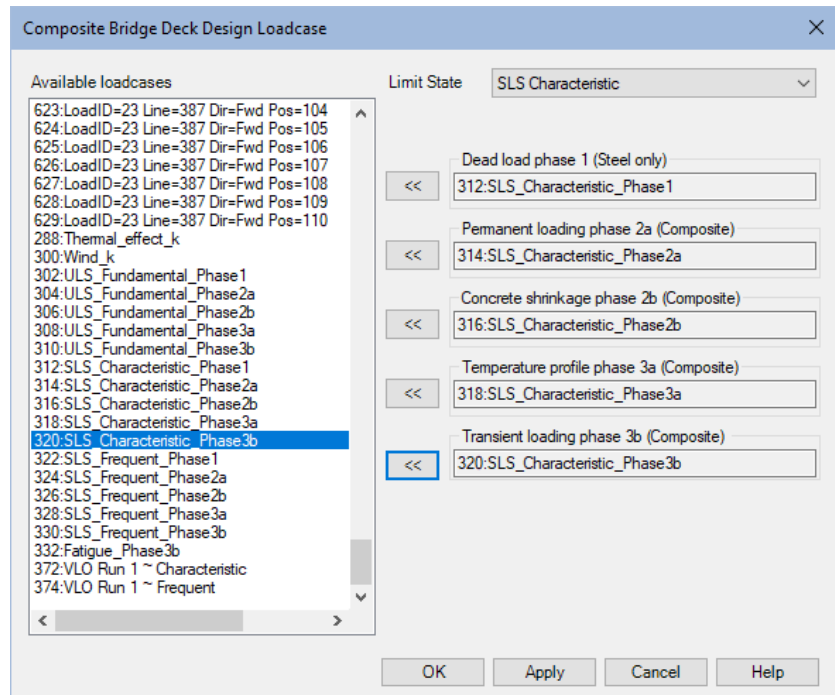
The screenshot shows a software dialog box titled "Composite Bridge Deck Design Results". It has a "Members" tab and a "Loadcases" tab, with "Loadcases" currently selected. There are three main sections for different limit states: "Serviceability loadcases", "Fatigue loadcases", and "ULS loadcases". Each section contains a table with five columns labeled "Phase 1", "Phase 2a", "Phase 2b", "Phase 3a", and "Phase 3b". Below each table are three buttons: "Add...", "Edit...", and "Delete". At the bottom of the dialog, there is a "Name" field containing the text "MRO2", a dropdown arrow, and a "(new)" button. Below the name field are four buttons: "OK", "Cancel", "Apply", and "Help".

Three limit states are considered - Serviceability, Fatigue and Strength, and appropriate loadcases (with the correct factors applied to those loadcases) must be assigned to each limit state.

- In the 'Serviceability loadcases' panel, click the **Add** button.
- From the list of available loadcases, select **SLS_Characteristic_Phase1** and click the 'Add to' button  to add the design combination to the right-hand panel, in the **Dead load phase 1** box.
- Proceed by selecting each of the following loadcases, one at a time and assigning to the appropriate phase:

Loadcase	Phase
SLS_Characteristic_Phase2a	Permanent loading phase 2a
SLS_Characteristic_Phase2b	Concrete shrinkage phase 2b
SLS_Characteristic_Phase3a	Temperature profile phase 3a
SLS_Characteristic_Phase3b	Transient loading phase 3b

Once done, the dialog should look as follows:



- Once correctly assigned, click **OK** to return to the main dialog.
- In the 'Serviceability loadcases' panel, click the **Add** button again.
- Select **SLS Frequent** in the drop down list on the top right of the dialog and proceed by selecting each of the following loadcases, one at a time and assigning to the appropriate phase:

Loadcase	Phase
SLS_Frequent_Phase1	Dead load phase 1
SLS_Frequent_Phase2a	Permanent loading phase 2a
SLS_Frequent_Phase2b	Concrete shrinkage phase 2b
SLS_Frequent_Phase3a	Temperature profile phase 3a
SLS_Frequent_Phase3b	Transient loading phase 3b

When all added, click **OK**.

- Repeat the above procedure for the other Limit States (Fatigue and Strength).
- Note that loadcase SLS_Frequent_Phase3b is not mapped as a Fatigue design limit state; the Fatigue_Phase3b loadcase must be mapped instead.

On completion, all **Loadcases**, and the corresponding **Design limit state** for each **Phase** should be as shown in the following dialog.

Composite Bridge Deck Design Results

Members Loadcases

Serviceability loadcases

Phase 1	Phase 2a	Phase 2b	Phase 3a	Phase 3b
312:SLS_Charac...	314:SLS_Charac...	316:SLS_Charac...	318:SLS_Charac...	320:SLS_Charac...
322:SLS_Freque...	324:SLS_Freque...	326:SLS_Freque...	328:SLS_Freque...	330:SLS_Freque...

Add... Edit... Delete

Fatigue loadcases

Phase 1	Phase 2a	Phase 2b	Phase 3a	Phase 3b
322:SLS_Freque...	324:SLS_Freque...	326:SLS_Freque...	328:SLS_Freque...	332:Fatigue_Pha...

Add... Edit... Delete

ULS loadcases

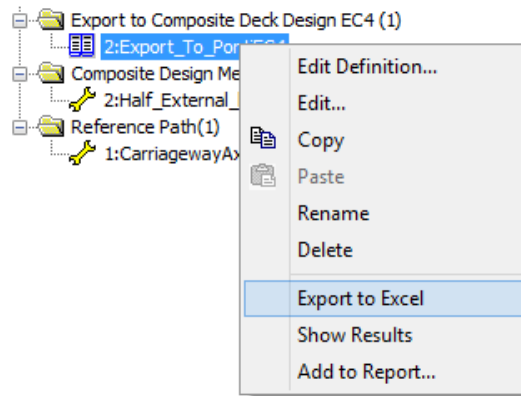
Phase 1	Phase 2a	Phase 2b	Phase 3a	Phase 3b
302:ULS_Funda...	304:ULS_Funda...	306:ULS_Funda...	308:ULS_Funda...	310:ULS_Funda...

Add... Edit... Delete

Name Export_To_PontiEC4 (1)

OK Cancel Apply Help

- Name the utility **Export_To_PontiEC4** and click **OK** to add these entry to the Utilities  treeview.
- Then, in the Utilities  treeview click the right-hand mouse button on the **Export_to_PontiEC4** item, and choose **Export to Excel** from the context menu as shown:



Step 5: Import force and moment data into PontiEC4

Associated File

This supplied file is only for use in case of problems in manually creating the appropriate data in the previous section of the example.



- ☐ **Export to ponti EC4.xls** File created by the LUSAS Export Forces and Moments dialog containing data to import into PontiEC4.

Modify the X-distance of the coincident sections



Note. A note It is necessary to modify the X-distance of coincident sections, in order to prevent sections with the same X-distance from being ordered randomly, when the graphs are created from the Utilities > Graphs menu.

This can be done either directly in PontiEC4, in the *Segment region* of the *Geometry dialog* or more simply in the “Export_to_ponti_EC4.xls” file just created.

- Open the “Model Info” sheet of the “Export_to_ponti_EC4.xls” file and increase the X value of the first section of each segment, as shown in red on the following image.

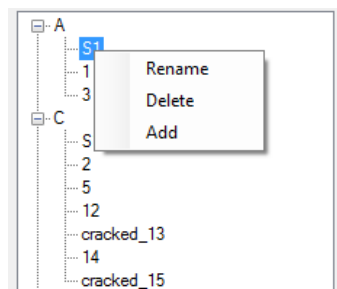
Section	Cracked	X(m)	X(m)
A_1	FALSE	0.000	0.000
A_2	FALSE	4.050	4.050
C_3	FALSE	4.050	4.051
C_4	FALSE	12.150	12.150
B_5	FALSE	12.150	12.151
B_6	FALSE	20.250	20.250
E_7	FALSE	20.250	20.251
E_8	FALSE	25.252	25.252
E_9	FALSE	30.254	30.254
B_10	FALSE	30.254	30.255
B_11	FALSE	38.354	38.354
C_12	FALSE	38.354	38.355
C_13	FALSE	43.198	43.198
C_cracked_14	TRUE	43.198	43.199
C_cracked_15	TRUE	46.454	46.454
D_cracked_16	TRUE	46.454	46.455
D_cracked_17	TRUE	50.504	50.504



Importing force and moment data into PontiEC4

This section covers the import of section force and moment data from LUSAS into PontiEC4. Refer to the 'Forces and moments' section in the PontiEC4 online help file for more information.

Whilst it is possible to manually enter forces and moments into PontiEC4 in the 'Forces and moments' dialog, and it is also possible to copy and paste data into each section, but when used with LUSAS any importing of data is far better achieved by using the automated procedure as follows.



- In PontiEC4, with the current data still loaded (or with the data from the previously saved file '**Composite_Bridge_Deck.csv**' loaded if the example had been saved and closed for some reason at that point), select the **File> Import forces/moments** menu item and load the file **Export to ponti EC4.xls** from working project folder.

PontiEC4 will detect that 17 additional sections are present (these are the sections at the beginning and end of each segment and at midspan) and ask if they are to be added to their corresponding segments.

- Click **Yes** to import the new sections. This operation may take a few seconds to complete. The status bar will show the progress.

- From the main menu select **Windows> 2 Geometry** to display the Geometry dialog. In the Segment treeview the new sections added to each segment can be seen.
- If any previously defined initial sections for each segment (named “S1”) are present these are no longer necessary and should be deleted. To do this, select each section named S1 in turn, click the right-hand mouse button, and choose **Delete**.



Note. If any mistakes are made in deleting the unwanted sections, re-importing the forces_and_moments.xls file will re-insert any deleted sections.

- Select **File> Save As** and enter a file name of **Composite_Bridge_Deck.csv** to save the PontiEC4 dialog data to the working project folder before proceeding to view the results.

This concludes the data input.

Step 6: Assess the results and detailed design checks in PontiEC4

This section focuses on the procedure required to assess the main girder beams for Ultimate, Serviceability, and Fatigue limit states. It involves:

1. **Viewing utilisation factors for all checks performed**
2. **Updating section sizes for any over-utilised sections**
3. **Viewing updated results**
4. **Updating the LUSAS model and re-analysing the revised sections**

It is usually enough to cycle through steps 1 to 4 once, or perhaps twice, to obtain an optimized structure.

Limited details have been given about all the checks that are carried out. To find out more about the checks performed refer to the PontiEC4 online Help (accessed via the question mark (?) button on the menu banner, where design code and theory details are also supplied.

Viewing results in PontiEC4

- With the results loaded in **PontiEC4** select the **Window> 5 Summary of results** menu item. This carries out the design computations in PontiEC4 and displays the results for deck sections in a summary form.

The meaning of the coloured utilization factor entries is explained in the following table:

Colour of value / text	Utilization factor	Meaning
------------------------	--------------------	---------

Assessment of a Composite Bridge Deck to Eurocodes

Red	Greater than 1.0	Value has failed the check carried out.
Green	Less than 1.0	Value has passed the check carried out.
Grey - without parentheses	Any	Value of potential interest but the utilisation factor is not relevant to the code.
(Grey) - with parentheses	Any	Value of potential interest but the utilisation factor is not relevant to the code in this particular context.
Grey stating 'No int.'	Not applicable	No interaction between bending and shear.

A check of the utilization factors obtained should be made for each limit state. Only a check for the **Fundamental ULS combination** will be shown here.



Note. By clicking on a particular header the values in a column can be sorted in increasing or decreasing value.

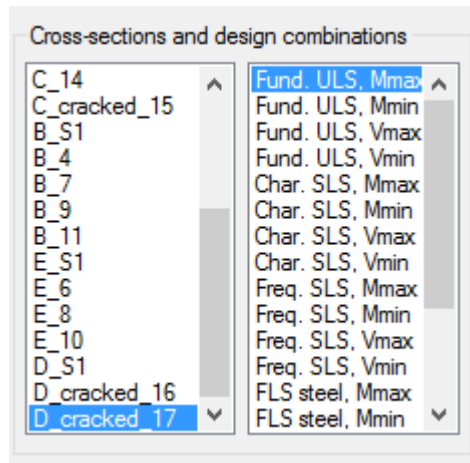
- Click twice to sort the **SigEd/fy** column in descending order. It can be seen that some of Segments D and C fail the check for normal stresses at ULS, as shown below.

Fundamental ULS combination												
Section	X (m)	Combination	Class Ph.1	Class Ph.3b	MEd/MR	SigEd /fy	VEd/VRd	MEd/Mf,Rd	VEd/Vbw,Rd	V/M/N	vEd/(h*PRd)	
D_cracked_17	50.504	Fund. ULS, Mmax	4	4	(.83)	1.171	0.691	1.31	0.691	1.219	0.509	
C_cracked_15	46.454	Fund. ULS, Mmax	4	4	(.73)	1.164	0.651	1.21	0.651	1.196	0.364	
D_cracked_17	50.504	Fund. ULS, Vmax	4	4	(.79)	1.119	0.742	1.26	0.742	1.197	0.573	
C_cracked_15	46.454	Fund. ULS, Vmax	4	4	(.63)	1.011	0.85	1.06	0.85	1.181	0.576	
C_14	43.198	Fund. ULS, Mmax	4	4	(.44)	0.859	0.524	0.97	0.527	No int.	0.27	
C_cracked_13	43.198	Fund. ULS, Mmax	4	4	(.55)	0.859	0.519	0.89	0.527	No int.	0.27	
B_7	20.250	Fund. ULS, Mmin	4	1		0.72	(.856)	0.041	1	0.041	No int.	0.05
D_cracked_16	46.454	Fund. ULS, Mmax	4	4	(.6)	0.837	0.506	0.94	0.511	No int.	0.332	
E_6	20.250	Fund. ULS, Mmin	4	1		0.66	(.772)	0.04	0.88	0.041	No int.	0.049
B_4	10.125	Fund. ULS, Mmin	4	1		0.63	(.772)	0.04	0.87	0.041	No int.	0.039



Note. Changing from the Geometry, Materials or Forces and moments dialogs to the Results, Summary of results, Report, or Cracking dialogs causes a re-analysis of results.

- Select the **Window> 4 Results** menu item in order to find out more about the failed checks.
- In the Cross-sections and design combinations panel select section **D_cracked_17** and **Fund. ULS, Mmax**



- Then, in the lower part of the Results dialog, select the **Stresses** tab and look at **Stresses of effective cross section**

Plastic check Stresses Shear Geometric properties 0 Geometric properties 1 Geometric properties 2 Domains Mpl-N Studs. ULS, SLS SLS, Web Breathing FLS steel FLS bars Stiffeners																	
Stresses of gross cross section Stresses of effective cross section																	
id	Ph.1	Ph.2a N.C.	Ph.2a C.	Ph.2b N.C.	Ph.2b C.	Ph.2c N.C.	Ph.2c C.	Ph.2 Tot.	Ph.3a N.C.	Ph.3a C.	Ph.3b N.C.	Ph.3b C.	Ph.3 Tot.	Eta1	id		
σ 8	0.0	1.2	0.0	1.1	0.0	0.0	0.0	0.0	1.7	0.0	4.3	0.0	0.0	0.00	σ 8		
σ 7	0.0	20.3	37.2	16.7	32.2	0.0	0.0	69.5	10.2	33.5	25.4	83.4	186.3	0.48	σ 7		
σ 6	0.0	16.3	31.9	13.3	27.6	0.0	0.0	59.5	7.2	28.7	17.8	71.4	159.6	0.41	σ 6		
σ 5	0.0	0.9	0.0	0.8	0.0	0.0	0.0	0.0	1.0	0.0	2.6	0.0	0.0	0.00	σ 5		
σ 4	166.6	13.9	28.7	11.2	24.9	0.0	0.0	220.2	5.4	25.8	13.3	64.3	310.3	0.92	σ 4		
σ 3	162.0	13.0	27.6	10.5	23.9	0.0	0.0	213.6	4.7	24.8	11.8	61.8	300.2	0.89	σ 3		
σ 2	0.0	0.0	0.0	0.1	0.1	0.0	0.0	0.1	0.0	0.0	0.0	0.0	0.1	0.00	σ 2		
σ 1	-13...	-41.9	-44.6	-36.0	-38.4	0.0	0.0	-220.9	-36.3	-40.1	-90.6	-99.9	-360.9	1.13	σ 1		
σ 0	-14...	-42.9	-46.0	-36.8	-39.6	0.0	0.0	-229.3	-37.1	-41.3	-92.5	-103.0	-373.6	1.17	σ 0		

Slab stresses at Phase 2 (N/mm²):

Total top stress = 2.25

Total bottom stress = 1.63

=> Section at the end of Phase 2: CRACKED (m.)

Slab stresses at Phase 3 (N/mm²):

Total top stress = 8.31

Total bottom stress = 5.29

=> Section at the end of Phase 3: CRACKED (m.)

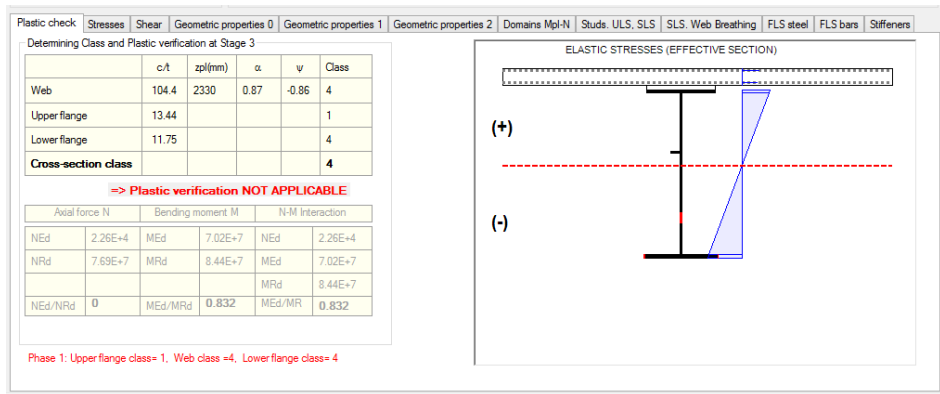
=> EI check Phase 3 FAILED
eta1= 1.171



Note. Phase totals listed are a summation of all the Cracked values for each Phase.

The normal stresses are very large in the bottom flange where they exceed f_y . They are also quite large in the top flange.

- Select the **Plastic check** tab to view the classification of the section (Class 4).



Step 7: Optimize the structure

Associated Files

These supplied files are only for use in case of problems in manually creating the appropriate data in this section of the example.



- ☐ **Composite_Bridge_Deck_Optimized.csv** - the input file for PontieC4 with updated steel flange values for segments D and C.
- ☐ **Sections_rev1.vbs** - A file created by PontieC4 containing revised section data to import into LUSAS to update the geometric properties in the LUSAS model.

For this example, optimizing the structure involves making changes to flange plate thicknesses in the PontieC4 Composite Deck Designer for those segments that failed the design check, then using LUSAS to calculate revised forces and moments, and then re-assessing the revised results and design checks in PontieC4.

Revise selected flange thicknesses in PontieC4:

Flange thicknesses will be increased on those segments that currently fail the check for normal stresses at ULS.

Segment D

- In PontieC4, select the **Window> 2 Geometry** menu item, and then click ONCE to select Segment **D** in the segment treeview. Note that the status bar at the bottom of the interface should state: 'Edit GEOMETRY of Segment D' showing that you are only editing an existing segment and not inputting data for a new one.
- Increase the top flange thickness (ts) from 40 to **50**

- Increase the bottom flange thickness (t_i) from 50 to **80**
- Select the **Window> 4 Results** menu item. This causes the results for Section **D_cracked_17** and **Fund. ULS Mmax** (and all other segments) to be updated.

Because of the modification to the flanges for Segment D, the **Plastic check** dialog will show that it has now moved from Class 4 to Class 3. As a result, the ‘Stresses of effective cross section’ radio button will no longer be available on the ‘Stresses’ results page of the Results dialog.



Note. Effective stresses are available only for the Class 4 sections.

- Select the **Stresses** tab and look at **Stresses of gross cross section**.

Plastic check Stresses Shear Geometric properties 0 Geometric properties 1 Geometric properties 2 Domains Mpl-N Studs. ULS, SLS SLS Web Breathing FLS steel FLS bars Stiffeners

☒ Stresses of gross cross section ☐ Stresses of effective cross section

id	Ph.1	Ph.2a N.C.	Ph.2a C.	Ph.2b N.C.	Ph.2b C.	Ph.2c N.C.	Ph.2c C.	Ph.2 Tot.	Ph.3a N.C.	Ph.3a C.	Ph.3b N.C.	Ph.3b C.	Ph.3 Tot.	Eta1	id
σ 5	0.0	1.1	0.0	1.0	0.0	0.0	0.0	0.0	1.6	0.0	3.9	0.0	0.0	0.00	σ 8
σ 7	0.0	18.5	32.8	15.2	28.4	0.0	0.0	61.2	9.3	29.5	23.2	73.5	164.1	0.42	σ 7
σ 6	0.0	15.4	28.7	12.6	24.8	0.0	0.0	53.5	7.1	25.8	17.7	64.2	143.5	0.37	σ 6
σ 5	0.0	0.8	0.0	0.7	0.0	0.0	0.0	0.0	1.1	0.0	2.6	0.0	0.0	0.00	σ 5
σ 4	138.1	13.6	26.2	11.1	22.7	0.0	0.0	187.1	5.8	23.6	14.4	58.7	269.4	0.84	σ 4
σ 3	133.8	12.8	25.2	10.4	21.8	0.0	0.0	180.7	5.2	22.6	12.9	56.4	259.7	0.81	σ 3
σ 2	0.0	0.0	0.0	0.1	0.1	0.0	0.0	0.1	0.0	0.0	0.0	0.0	0.1	0.00	σ 2
σ 1	-89.3	-28.1	-29.6	-24.1	-25.5	0.0	0.0	-144.5	-24.4	-26.6	-60.8	-66.4	-237.5	0.74	σ 1
σ 0	-96.3	-29.3	-31.3	-25.2	-27.0	0.0	0.0	-154.6	-25.3	-28.2	-63.1	-70.2	-253.0	0.79	σ 0

Slab stresses at Phase 2 (N/mm²):
Total top stress = 2.03
Total bottom stress = 1.56
=> Section at the end of Phase 2: CRACKED (m.)

Slab stresses at Phase 3 (N/mm²):
Total top stress = 7.5
Total bottom stress = 5.27
=> Section at the end of Phase 3: CRACKED (m.)

=> El. check Phase 3 PASSED
eta1= 0.844

The flange thickness changes for Segment D now allow it to pass the design check.

Segment C

- Select the **Window> 2 Geometry** menu item, and click once to select Segment C in the segment treeview. The status bar should state : ‘Edit GEOMETRY of Segment C’
- Increase the top flange thickness (t_s) from 30 to **50**
- Increase the web thickness (t_{wr}) from 22 to **25**
- Increase the bottom flange thickness (t_i) from 40 to **60**
- Select the **Window> 4 Results** menu item and view the updated results for Section **C_cracked_15** and **Fund. ULS Mmax**. The flange thickness changes for Segment C now allow it to pass the design check. (And selecting the Plastic check tab would show that it, too, has moved from Class 4 to Class 3).

Assessment of a Composite Bridge Deck to Eurocodes

Plastic check Stresses Shear Geometric properties 0 Geometric properties 1 Geometric properties 2 Domains Mpl-N Studs ULS, SLS SLS: Web Breathing FLS steel FLS bars Stiffeners																			
Stresses of gross cross section Stresses of effective cross section																			
id	Ph.1	Ph.2a N.C.	Ph.2a C.	Ph.2b N.C.	Ph.2b C.	Ph.2c N.C.	Ph.2c C.	Ph.2 Tot.	Ph.3a N.C.	Ph.3a C.	Ph.3b N.C.	Ph.3b C.	Ph.3 Tot.	Ela1	id	Slab stresses at Phase 2 (N/mm²): Total top stress = 1.75 Total bottom stress = 1.31 => Section at the end of Phase 2: CRACKED (m.)			
σ 8	0.0	0.7	0.0	1.0	0.0	0.0	0.0	0.0	1.6	0.0	3.0	0.0	0.0	0.0	σ 8				
σ 7	0.0	12.8	24.9	15.8	32.4	0.0	0.0	57.3	9.3	33.6	17.6	63.2	154.1	0.39	σ 7				
σ 6	0.0	10.5	21.6	12.8	28.2	0.0	0.0	49.8	6.8	29.2	12.7	55.0	134.0	0.34	σ 6				
σ 5	0.0	0.6	0.0	0.8	0.0	0.0	0.0	0.0	1.0	0.0	1.9	0.0	0.0	0.00	σ 5				
σ 4	137.3	9.1	19.7	11.0	25.7	0.0	0.0	182.7	5.2	26.6	9.9	50.1	259.4	0.81	σ 4				
σ 3	133.2	8.5	18.9	10.3	24.6	0.0	0.0	176.6	4.6	25.5	8.6	47.9	250.0	0.78	σ 3	Slab stresses at Phase 3 (N/mm²): Total top stress = 6.31 Total bottom stress = 4.19 => Section at the end of Phase 3: CRACKED (m.) => EI check Phase 3 PASSED eta1= 0.813			
σ 2	0.0	0.0	0.0	0.1	0.1	0.0	0.0	0.1	0.0	0.0	0.0	0.0	0.1	0.00	σ 2				
σ 1	-83.4	-23.0	-24.6	-29.6	-31.8	0.0	0.0	-139.8	-29.9	-33.2	-56.2	-62.4	-235.5	0.74	σ 1				
σ 0	-88.4	-23.7	-25.6	-30.5	-33.1	0.0	0.0	-147.1	-30.7	-34.6	-57.7	-65.0	-246.7	0.77	σ 0				

- Select the **Window> 5 Summary of results** menu item. Sort the **SigEd/fy** column. It can be seen that Segments D and C now both pass the check for normal stresses at ULS, as shown below. Stresses for other combinations are all satisfactory.

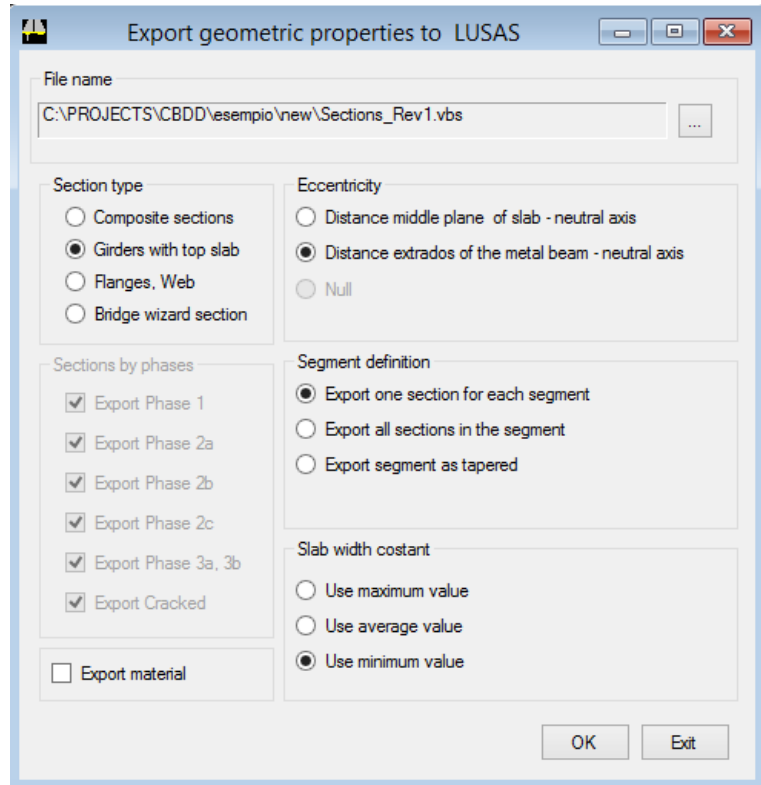
Fundamental ULS combination												
Section	X (m)	Combination	Class Ph.1	Class Ph.3b	MEd/MR	SigEd fy	VEd/VRd	MEd/Mf,Rd	VEd/Vbw,Rd	V/M/N	vEd/n*PR	
B_6	20.250	Fund. ULS, Mmin	4	1	0.72	(.856)	0.041	1	0.041	No int.	0.05	
D_cracked_17	50.504	Fund. ULS, Mmax	1	3	0.844	(.65)	0.694	0.81	0.694	0.875	0.478	
D_cracked_17	50.504	Fund. ULS, Vmax	1	3	0.815	(.62)	0.745	0.78	0.745	0.864	0.538	
C_cracked_15	46.454	Fund. ULS, Mmax	3	3	0.813	(.59)	0.513	0.82	0.513	0.813	0.34	
E_7	20.251	Fund. ULS, Mmin	4	1	0.66	(.772)	0.04	0.88	0.041	No int.	0.049	
C_cracked_15	46.454	Fund. ULS, Vmax	3	3	0.735	(.51)	0.669	0.71	0.669	0.767	0.54	
B_5	12.151	Fund. ULS, Mmin	4	1	0.62	(.727)	0.413	0.87	0.424	No int.	0.229	
E_8	25.252	Fund. ULS, Mmin	4	1	0.64	(.727)	0.275	0.84	0.288	No int.	0.122	
B_5	12.151	Fund. ULS, Vmin	4	1	0.59	(.688)	0.445	0.82	0.46	No int.	0.254	
B_6	20.250	Fund. ULS, Vmax	4	1	0.45	(.661)	0.27	0.63	0.286	No int.	0.181	
B_6	20.250	Fund. ULS, Vmin	4	1	0.58	(.657)	0.174	0.8	0.18	No int.	0.144	
B_10	30.255	Fund. ULS, Mmin	4	1	0.58	(.654)	0.474	0.81	0.49	No int.	0.21	
E_7	20.251	Fund. ULS, Vmax	4	1	0.42	(.653)	0.258	0.56	0.286	No int.	0.178	
E_8	25.252	Fund. ULS, Vmax	4	1	0.42	(.594)	0.439	0.56	0.486	No int.	0.259	
E_7	20.251	Fund. ULS, Vmin	4	1	0.53	(.593)	0.167	0.71	0.18	No int.	0.141	
E_9	30.254	Fund. ULS, Mmin	4	1	0.54	(.591)	0.455	0.72	0.49	No int.	0.206	
D cracked 16	46.455	Fund. ULS, Mmax	1	3	0.59	(.46)	0.471	0.58	0.514	No int.	0.311	

- Select **File> Save As** to save the current PontiEC4 project data as **Composite_Bridge_Deck_Optimized.csv**.

Export the revised geometric data to LUSAS

- Select the **File> Export geometric properties to LUSAS** menu item.
- Ensure that the destination directory is set to your working project folder and input a filename of **Sections_Rev1.vbs**.
- Then for section type ensure that the option **Girders with top slab** is selected.

- For eccentricity select **Distance extrados of the metal beam – neutral axis**
- For segment definition select **Export one section for each segment**
- For slab width constant select **Use minimum value**
- Click **OK** to finish.



Calculate revised forces and moments in LUSAS

- In LUSAS, and with the **Composite_Bridge_Deck.mdl** file open, select **File> Script > Run Script** and from the working project folder select the PontiEC4 script file **Sections_Rev1.vbs** to import the optimized sections into the model.
- Select the **File> Save As** menu item to save the model to the working project folder as **Composite_Bridge_Deck_Optimized.mdl**. Any open results files will be closed.
- Press the **Solve** button  to compute all loadcases in all the analyses. After a short wait the solving will be completed.

Export the revised force and moment data from LUSAS

This involves repeating the operations from Step 4 in this example. In summary, this involves:

- Select the **Design > Composite Bridge Deck Design** menu item.
- Click the ellipsis button  to choose the folder and name of the xls file to create (e.g.: <current work directory>**Export to Ponti EC4 rev1.xls**), pressing the **Save** button to return to the main dialog.
- Leave all other input data as before and click **OK** to save data for the revised XLS file for use in PontiEC4.
- Right-click the utilities item “**Export_to_PontiEC4**” and choose “**Export to Excel**”.

Import revised forces and moments into PontiEC4

- In PontiEC4, with the current data still loaded, select the **File> Import forces/moments** menu item and load the file **Export to Ponti EC4 rev1.xls** that was created by LUSAS from the working project folder.

Assess the revised results in PontiEC4

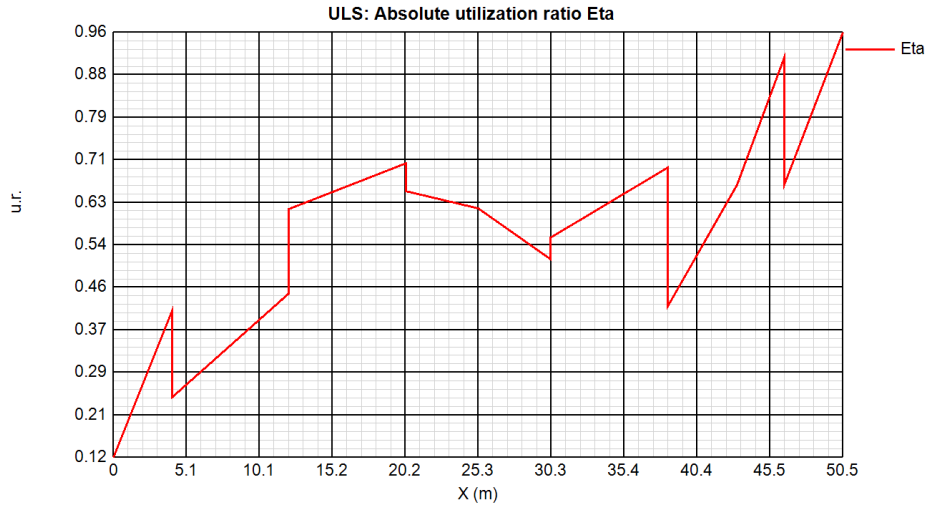
- Select the **Window> 5 Summary of results** menu item.
- Sort the **SigEd/fy** column. It can be seen that values of SigEd/fy have changed slightly from the values previously calculated in PontiEC4 that used the initial forces and moments calculated by LUSAS, but values for all segments can be seen to pass the check for normal stresses at ULS.

Section	X (m)	Combination	Class Ph.1	Class Ph.3b	MEd/MR	SigEd /fy	VEd/VRd	MEd/ME,Rd	VEd/Vbw,Rd	V/M/N	vEd/(n*PRd)	End Studs	L
D_cracked_17	50.504	Fund. ULS, Mmax	1	3	(.71)	0.924	0.714	0.89	0.714	0.962	0.49	0	0.
C_cracked_15	46.454	Fund. ULS, Mmax	3	3	(.66)	0.911	0.531	0.92	0.531	0.912	0.353	0	0.
D_cracked_17	50.504	Fund. ULS, Vmax	1	3	(.68)	0.891	0.764	0.85	0.764	0.948	0.549	0	0.
B_7	20.250	Fund. ULS, Mmin	4	1	0.7	(.829)	0.025	0.98	0.025	No int.	0.046	0	0.
C_cracked_15	46.454	Fund. ULS, Vmax	3	3	(.58)	0.829	0.686	0.8	0.686	0.868	0.551	0	0.
E_6	20.250	Fund. ULS, Mmin	4	1	0.65	(.748)	0.024	0.86	0.025	No int.	0.045	0	0.
B_4	12.150	Fund. ULS, Mmin	4	1	0.61	(.713)	0.398	0.85	0.409	No int.	0.225	0	0.
E_8	25.252	Fund. ULS, Mmin	4	1	0.61	(.695)	0.288	0.81	0.304	No int.	0.126	0	0.
B_11	38.354	Fund. ULS, Mmax	4	4	(.38)	0.695	0.554	0.72	0.581	No int.	0.176	0	0.

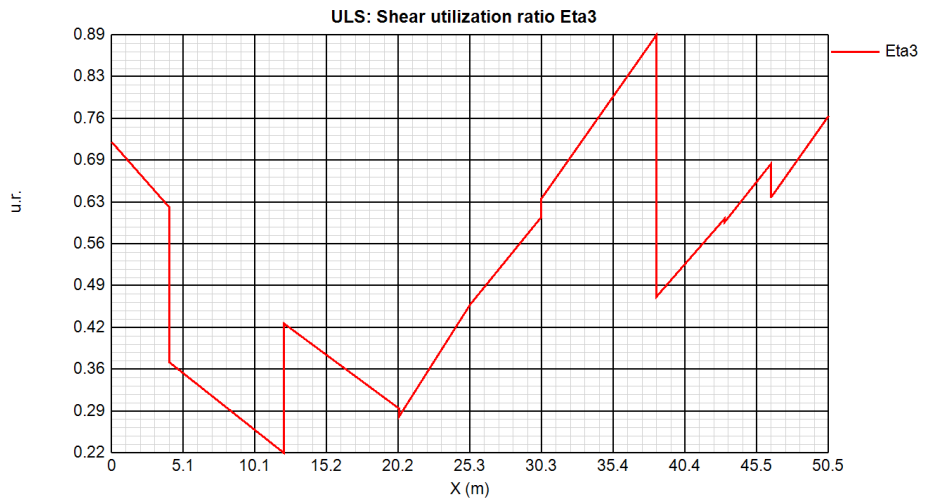
Methods for graphing data and producing detailed results will now be explained.

Creating Graphs

- In PontiEC4, with the optimized data still loaded, select the **Utilities> Graphs** menu item. (If the optimized data is no longer loaded open the file Composite_Bridge_Deck_Optimized.csv to populate the dialogs)
- From the first drop-down list choose **ULS: Absolute utilization ratio Eta** to plot the following graph:



- From the first drop-down list choose **ULS: Shear utilization ratio Eta3** to plot the following graph:

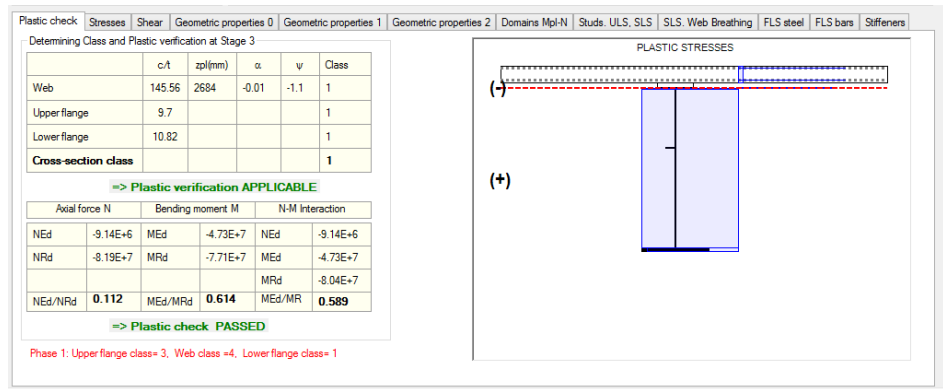




Note. Graphs can also be viewed by scrolling the mouse wheel up and down in the graph window.

Obtaining more detailed results

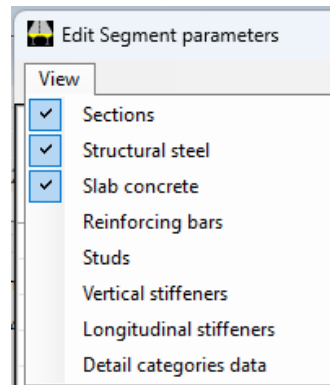
- With the optimized data still loaded, select the **Window> 4 Results** menu item.
- In the Cross-sections and design combinations panel, select the centre section **E_8** and **Fund. ULS, Mmin.**
- Select the **Stresses** tab to have a look at the elastic stresses on the gross section.
- Then, select the **Plastic check** tab. From this, all details about the classification and plastic check are supplied, and it can be seen that for Stage 3, this section, that is part of the segment in the middle of the span, is in Class 1.



Note. At the bottom-left of the Plastic check page the section classification is also provided for Phase 1, where the top (upper) flange cannot be considered to be restrained by the slab. For this case, the section is in Class 4 as highlighted in the red message. For this situation a check of stresses in Phase 1 should be performed, using an effective section rather than a gross section.

To check stresses on the effective section in Phase1

- Save then file as “Composite_Bridge_Deck_Optimized_Ph1.csv”.
- Select the **Window> 2 Geometry** menu item.
- Right-click inside the dialog and select **Edit segments by data grid**
- Check **Sections**, **Structural steel** and **Slab concrete** in the **View** menu



- For all segments uncheck the option **Top flange in Class 1**, input **0** the column **bcl**, **b1** and **bsx**

Segment name	Sections	X(m)	b _{sup} (mm)	t _{sup} (mm)	Top flange in Class 1	Top flange ≈40mm	h _{net} (mm)	t _{wr} (mm)	alpha	b _{inf} (mm)	t _{inf} (mm)	Bottom flange ≈40mm	b _{cl} (mm)	t _{cl} (mm)	b1 (mm)	b _{ax} (mm)	h _{cop} (mm)	Consider haunch
A	1 2	0 4.05	600	20	☐	☐	2700	20	0	1100	20	☐	0	273	0	0	75	☐
C	3 4 12 13 cracke...	4.051 12.15 ...	600	50	☐	☐	2700	25	0	1100	60	☐	0	273	0	0	75	☐
B	5 6 10 11	12.151 20.25 ...	600	30	☐	☐	2700	18	0	1100	40	☐	0	273	0	0	75	☐
E	7 8 9	20.251 25.25 ...	600	30	☐	☐	2700	18	0	1100	50	☐	0	273	0	0	75	☐
D	cracked_16 crac...	46.455 50.504	1100	50	☐	☐	2700	25	0	1200	80	☐	0	273	0	0	75	☐

- Click OK
- Select the **Window> 3 Forces and moments** menu item, Right-click inside the dialog and select **Reset>Phase 2a**. Click Yes in the information message box. All forces and moments in Phase 2a will be deleted.
- Repeat the same operations above to reset Phase 2b, Phase 3a, and Phase 3b
- Select the **Window> 4 Results** menu item
- Select Section **E_8** and **Fund. ULS, Mmin** from the Cross-sections and design combinations panel.
- Select the **Plastic check** tab to see a coherent classification with an only steel structural section

Plastic check | **Stresses** | Shear | Geometric properties 0 | Geometric properties 1 | Geometric properties 2 | Domains Mpl-N | Studs: ULS, SLS | SLS: Web Breathing | FLS steel | FLS bars | Stiffeners

Determining Class and Plastic verification at Stage 3

	c/t	zpl(mm)	α	ψ	Class
Web	145.56	418	0.86	-0.52	4
Upper flange	9.7				3
Lower flange	10.82				1
Cross-section class					4

=> Plastic verification NOT APPLICABLE

	Axial force N	Bending moment M	N-M interaction
NEd	2.46E-9	MEd	-9.7E+6
NRd	3.96E+7	MRd	-3.65E+7
			MRd
NEd/NRd	0	MEd/MRd	0.265
			MEd/MR
			0.265

Phase 1: Upper flange class= 3, Web class =4, Lower flange class= 1

ELASTIC STRESSES (EFFECTIVE SECTION)

- Lastly, select the **Stresses** tab to see the stresses calculated on an effective cross section

Plastic check | **Stresses** | Shear | Geometric properties 0 | Geometric properties 1 | Geometric properties 2 | Domains Mpl-N | Studs: ULS, SLS | SLS: Web Breathing | FLS steel | FLS bars | Stiffeners

☐ Stresses of gross cross section ☒ Stresses of effective cross section

id	F1	F2a N.F.	F2a F.	F2b N.F.	F2b F.	F2c N.F.	F2c F.	F2 tot	F3a N.F.	F3a F.	F3b N.F.	F3b F.	F3 tot	eta 1	id
$\sigma 8$	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.00	$\sigma 8$
$\sigma 7$	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.00	$\sigma 7$
$\sigma 6$	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.00	$\sigma 6$
$\sigma 5$	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.00	$\sigma 5$
$\sigma 4$	-13...	0.0	0.0	0.0	0.0	0.0	0.0	-136.8	0.0	0.0	0.0	0.0	-136.8	0.40	$\sigma 4$
$\sigma 3$	-13...	0.0	0.0	0.0	0.0	0.0	0.0	-134.5	0.0	0.0	0.0	0.0	-134.5	0.40	$\sigma 3$
$\sigma 2$	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.00	$\sigma 2$
$\sigma 1$	62.5	0.0	0.0	0.0	0.0	0.0	0.0	62.5	0.0	0.0	0.0	0.0	62.5	0.20	$\sigma 1$
$\sigma 0$	66.3	0.0	0.0	0.0	0.0	0.0	0.0	66.3	0.0	0.0	0.0	0.0	66.3	0.21	$\sigma 0$

Slab stresses at Phase 2 (N/mm²):
 Total top stress = 0
 Total bottom stress = 0
 => Section at the end of Phase 2: CRACKED (m.)

Slab stresses at Phase 3 (N/mm²):
 Total top stress = 0
 Total bottom stress = 0
 => Section at the end of Phase 3: CRACKED (m.)

=> EI check Phase 3 PASSED
 eta1 = 0.404

This completes the example.

Associated Files

These supplied files are only for use in case of problems in manually creating the appropriate data in this section of the example.



- ☐ **Composite Bridge Deck Optimized_Ph1.csv**- the input file for PontiEC4 without slab, with top flanges that can also be classified outside class 1, and with forces and moments of the phase 1 alone.