

# Assessment of a Composite Bridge Deck to Eurocodes

For LUSAS version:	18.0
For software product(s):	LUSAS Bridge or LUSAS Bridge plus
With product option(s):	Steel and Composite Deck Designer (PontiEC4 – V3.2.7)
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.

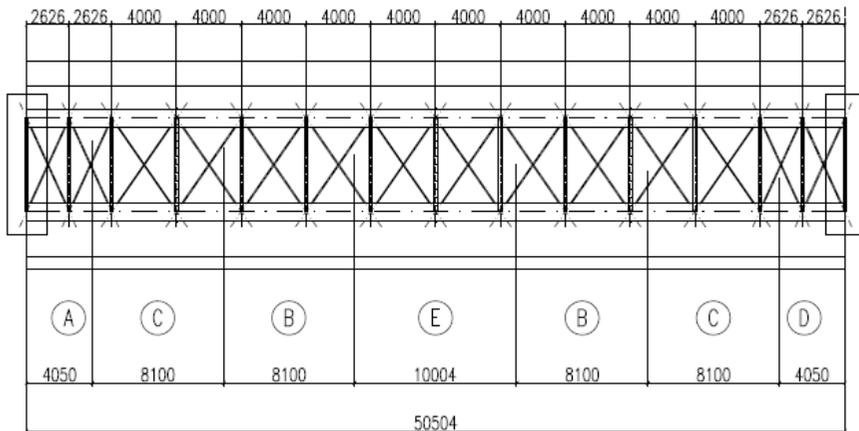
Because the primary aim of this particular example is to show the use of the PontiEC4 Composite Deck Designer and the interaction between it and LUSAS, no step-by-step instruction is provided for construction of the supplied LUSAS model that is used in this example. A separate example 'Composite Highway 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

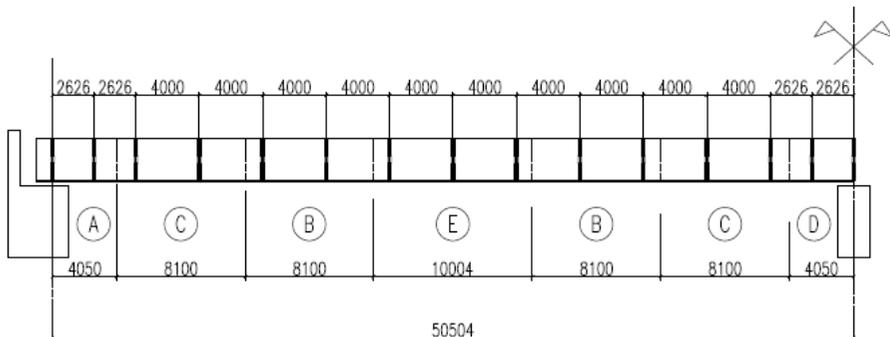
Plan, elevation and typical cross section views of the bridge deck are shown on the following pages.

## Assessment of a Composite Bridge Deck to Eurocodes



Plan of left-hand span.

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



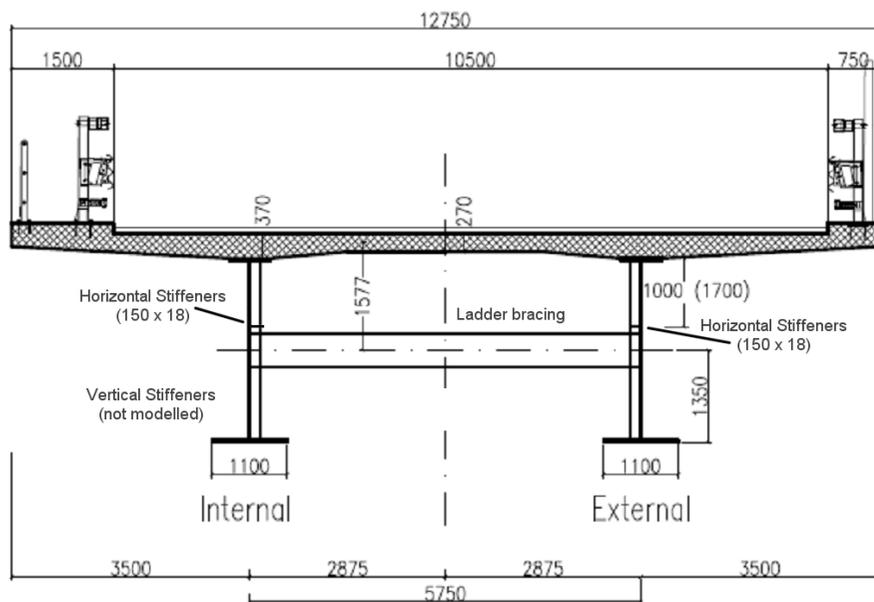
Elevation of left-hand span. Right-hand span is similar but handed.



### 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 elements 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



- ❑ **Composite\_Bridge\_Deck\_Initial.csv** – Contains initial material and geometric data / definitions for use with PontiEC4
- ❑ **Sections\_Rev0.vbs** - A file created by PontiEC4 containing geometric section data to import into LUSAS. A version of this file is supplied also, but is only 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.
- ❑ **Forces\_and\_moments.xls** File created by the LUSAS Export Forces and Moments 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.
- ❑ **Export\_Forces\_to\_PontiEC4.inp** File containing dialog selections made on the Export Forces and Moments dialog.
- ❑ **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.

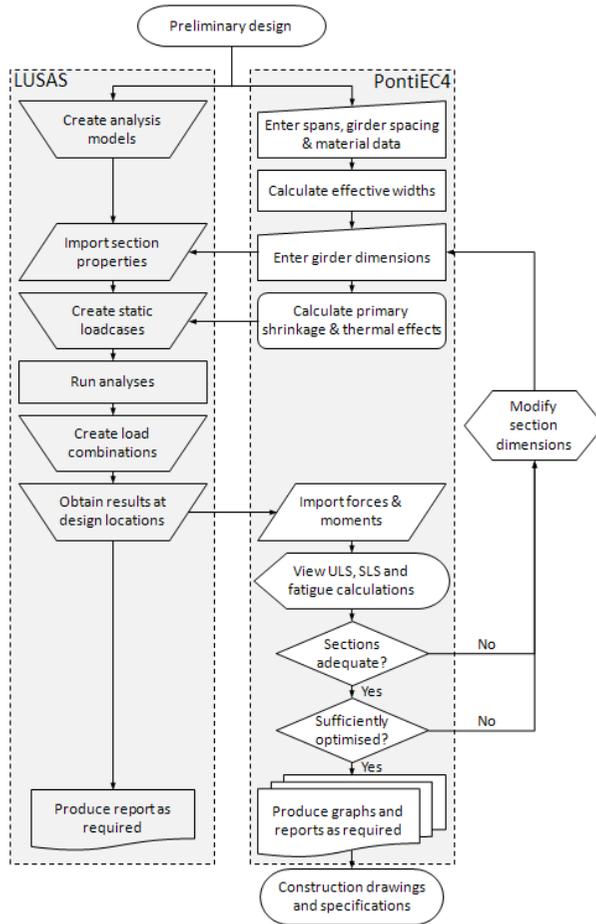
## Method and Objectives

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 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.

# Assessment of a Composite Bridge Deck to Eurocodes



## 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 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 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 '*Export geometric properties from PontiEC4 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*' 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 PontEC4 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 fck (N/mm <sup>2</sup> )	33.2
Strength fct,ef* (N/mm <sup>2</sup> )	0
Partial factor $\gamma_c$	1.5
Cement class	N
Aggregate type	Quartzite
Coeff. of thermal expansion	1E-05
Max. crack width wk (mm)	0.2

### Concrete age

Concrete age	
At onset of drying shrinkage $t_s$ (day)	<input type="text" value="2"/>
At time considered $t$ (day)	<input type="text" value="25550"/>
<b>See note 1</b> When perm. load is applied to (day)	<input type="text" value="30"/>
<b>See note 2</b> When shrink. load is applied to (day)	<input type="text" value="2"/>
<b>See note 3</b> When imposed d. are applied to (day)	<input type="text" value="30"/>
Permanent creep multiplier $\Psi_{sL}$	<input type="text" value="1.1"/>
Shrinkage creep multiplier $\Psi_{sL}$	<input type="text" value="0.55"/>
Deformations creep multiplier $\Psi_{sL}$	<input type="text" value="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	
<input checked="" type="radio"/> Automatic calculation	<input type="radio"/> Direct input
$n_0$	<input type="text" value="0"/>
$n_L$ permanent loads	<input type="text" value="0"/>
$n_L$ shrinkage	<input type="text" value="0"/>
$n_L$ imposed deformation	<input type="text" value="0"/>

### Environment

Environment	
Exposed area (mm <sup>2</sup> )	1640259
Exposed perimeter (mm)	13352
Relative humidity (%)	70

### Imposed strain in the slab

Imposed strain in the slab	
<input checked="" type="radio"/> Automatic calculation	<input type="radio"/> Direct input
Shrinkage Deformation	0
Shrinkage comb. factor	1
Temperature difference (°C)	10
Temperature comb. factor	1.5

### Shear connection

Shear connection			
Ultimate strength $f_u$ (N/mm <sup>2</sup> )	450		
<i>Reference values for fatigue strength at 2E6 cycles</i>			
shear stress $\Delta\tau_c$ (N/mm <sup>2</sup> )	90		
normal stress $\Delta\sigma_c$ (N/mm <sup>2</sup> )	80		
<i>Partial factors:</i>			
$\gamma_v$	1.25	$k_s$ (SLS)	0.6
$\gamma_{Ff}$	1	$\gamma_{Mf,s}$	1.15

### Steel

- Click on the Question mark button  next to the Grade field, to define the Steel grade required, then define all other values as shown:

Steel			
Modulus of elasticity (N/mm <sup>2</sup> )	<input type="text" value="210000"/>	Poisson's ratio $\nu$	<input type="text" value="0.3"/>
.. structural steel		.. reinforcement steel	
Grade	<input type="text" value="EN 10025-2 S355"/> <input type="button" value="?"/>	$f_{yk}$ (N/mm <sup>2</sup> )	<input type="text" value="450"/>
	<input type="text" value="≤40 mm"/> <input type="text" value="&gt;40 mm"/>	<i>Fatigue strength:</i>	
$f_u$ (N/mm <sup>2</sup> )	<input type="text" value="510"/> <input type="text" value="470"/>	$\Delta\sigma_{Rsk}$ (N/mm <sup>2</sup> )	<input type="text" value="162.50"/>
$f_y$ (N/mm <sup>2</sup> )	<input type="text" value="355"/> <input type="text" value="335"/>	<i>Partial factors:</i>	
<i>Partial factors:</i>		$\gamma_s$	<input type="text" value="1.15"/>
$\gamma_{M0}$	<input type="text" value="1.05"/>	$\gamma_F$	<input type="text" value="1"/>
$\gamma_{M1}$	<input type="text" value="1.1"/>	$\gamma_{Mf}$	<input type="text" value="1.15"/>
$\gamma_{Mser}$	<input type="text" value="1"/>		
			<input type="text" value="1.35"/> <input type="button" value="?"/>

### Fatigue

FATIGUE. Damage equivalent factors			
	STRUCTURAL STEEL	SHEAR STUDS	REINFORC. BARS
- for damage effects induced by the traffic	$\lambda_1$ <input type="text" value="(*)"/>	$\lambda_{v,1}$ <input type="text" value="1.550"/>	$\lambda_{s,1}$ <input type="text" value="(*)"/>
- for the traffic composition	$\lambda_2$ <input type="text" value="0.928"/>	$\lambda_{v,2}$ <input type="text" value="0.953"/> <input type="button" value="?"/>	$\lambda_{s,2}$ <input type="text" value="0.000"/>
- for design life of the structure	$\lambda_3$ <input type="text" value="1.000"/>	$\lambda_{v,3}$ <input type="text" value="1.000"/> <input type="button" value="?"/>	$\lambda_{s,3}$ <input type="text" value="0.000"/>
- for effects of the heavy traffic on the other slow lanes	$\lambda_4$ <input type="text" value="1.000"/>	$\lambda_{v,4}$ <input type="text" value="1.000"/>	$\lambda_{s,4}$ <input type="text" value="1.000"/>
			$\phi_{Fat}$ <input type="text" value="0.000"/>
(*) Values depending on the section position (input in the -Geometry- dialog window)			

## Step 1: Define material and geometric properties in PontiEC4

- Note that  $\lambda_2$  can be calculated from selections made in the dialog that appears when you click on the Question mark 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

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

$$\lambda_{v,2} = \frac{Q_{ml}}{Q_0} \left( \frac{N_{Obs}}{N_0} \right)^{1/8} \quad Q_{ml} = \left( \frac{\sum n_i Q_i^8}{\sum n_i} \right)^{1/8}$$

$\lambda_2 = 0.928$        $\lambda_{v,2} = 0.953$

$Q_0 = 480$  kN (weight of FML3)  
 $N_0 = 0.5E6$   
 $Nobs = 5E+5$  (Cfr. Tab. 4.5)  
 $Q_{ml} = 445.4$  kN (Cfr. Tab. 4.7)  
 $Q_{mlv} = 457.4$  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
<input type="checkbox"/> 1 Roads and motorways with 2 or more lanes per direction with high flow rates of lorries	$2,0 \times 10^6$
<input checked="" type="checkbox"/> 2 Roads and motorways with medium flow rates of lorries	$0,5 \times 10^6$
<input type="checkbox"/> 3 Main roads with low flow rates of lorries	$0,125 \times 10^6$
<input type="checkbox"/> 4 Local roads with low flow rates of lorries	$0,05 \times 10^6$
<input type="checkbox"/> User	

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	
<input checked="" type="checkbox"/> Long distance	20%	5%	50%	15%	10%	
<input type="checkbox"/> Medium distance	40%	10%	30%	15%	5%	
<input type="checkbox"/> Local traffic	80%	5%	5%	5%	5%	
<input type="checkbox"/> User	<input type="text"/> %	Calculate				

- Note that  $\lambda_3$  can be calculated from data entered in the dialog that appears when you click on the Question mark button  next to the  $\lambda_{v,3}$  input field. In this case, input the data below:

**Damage equivalent factor LAMBDA3 for road bridge**

Design life of bridge (years)

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

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

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 (beff) 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<sub>eff</sub>**.

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

Array of spans (m) eg. 36.50,60...  -->

X (m)	b1* (mm)	b2* (mm)	b0 (mm)	Type	beff (mm)	Le (m)	be1 (mm)	be2 (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 Le 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.

Segment name

A

Sections (eg. Sec1, Sect2, ... X (m) (es. X1 X2, ...))

S1 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).



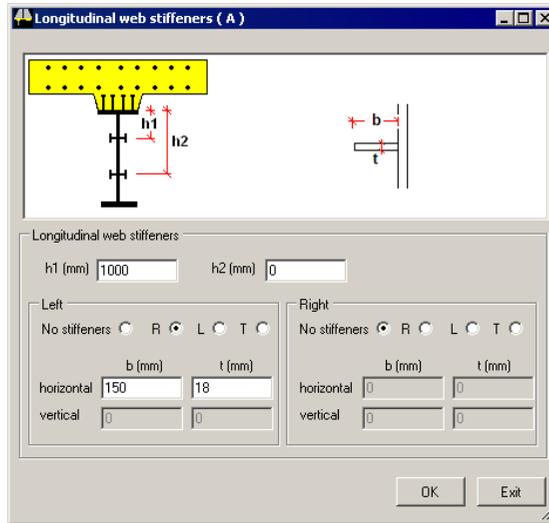
**Note.** The checkboxes for Top / Bottom flange <40mm should be only checked (ticked) for flanges that have a total thickness above 40mm, when they are made up

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.

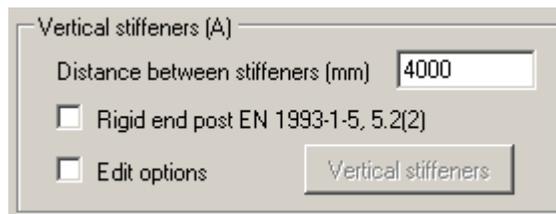
### Longitudinal web stiffeners details

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 *b<sub>cls</sub>* 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.

## Step 1: Define material and geometric properties in PontiEC4

Slab concrete (A)			
bcls (mm)	5877.7	tcls (mm)	273
b1 (mm)	600	bsx (mm)	2767
hcop (mm)	75	<input type="checkbox"/>	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 <sup>2</sup> /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)			
Damage equivalent factor (traffic)	Steel (Bending)	$\lambda_1$	2.145 ?
	Steel (Shear)	$\lambda_1$	2.448
	Bars	$\lambda_{s,1}$	0.000
Traffic loading factor (Reinforcing bars)			0.000
Detail categories data (A)			

- $\lambda_1$  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**

$\lambda_1$ , 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<sup>2</sup>) 125 Table 8.1: Plain members and mechanically fastened joints (EN 1993-1-9)

Bottom flange  
 $\Delta\sigma_R$  (N/mm<sup>2</sup>) 125 Table 8.1: Plain members and mechanically fastened joints (EN 1993-1-9)

Web  
 $\Delta\tau_R$  (N/mm<sup>2</sup>) 100 Table 8.1: Plain members and mechanically fastened joints (EN 1993-1-9)

Top flange joint  
 $\Delta\sigma_R$  (N/mm<sup>2</sup>) 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<sup>2</sup>) 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<sup>2</sup>) 112 Table 8.2: Welded built-up sections (EN 1993-1-9)

Web-Bottom flange  
 $\Delta\sigma_R$  (N/mm<sup>2</sup>) 112 Table 8.2: Welded built-up sections (EN 1993-1-9)

Vertical stiffener-Web  
 $\Delta\sigma_R$  (N/mm<sup>2</sup>) 80 Table 8.4: Weld attachments and stiffeners (EN 1993-1-9)

Vertical stiffener-Top flange  
 $\Delta\sigma_R$  (N/mm<sup>2</sup>) 80 Table 8.4: Weld attachments and stiffeners (EN 1993-1-9)

Vertical stiffener-Bottom flange  
 $\Delta\sigma_R$  (N/mm<sup>2</sup>) 80 Table 8.4: Weld attachments and stiffeners (EN 1993-1-9)

Horizontal stiffener-Web  
 $\Delta\sigma_R$  (N/mm<sup>2</sup>) 80 Table 8.4: Weld attachments and stiffeners (EN 1993-1-9)

OK Exit

### 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.

The screenshot shows a software interface for defining Segment C. The 'Segment name' field contains 'C'. The 'Structural steel (C)' section includes fields for *bs* (600), *ts* (30), *hmet* (2700), *twr* (22), *alpha* (0), *bi* (1100), and *ti* (40). The 'Slab concrete (C)' section includes fields for *bcls* (6196.7), *tcls* (273), *b1* (600), *bsx* (2836.3), and *hcop* (75). The 'Reinforcing bars (C)' section includes fields for top layer bar diameter (26), bar spacing (100), and bar cover (40), and bottom layer bar diameter (20), bar spacing (100), and bar cover (40). A technical drawing of a composite bridge deck cross-section is shown in the center, with dimensions labeled: *bcls*, *bsx*, *b1*, *tcls*, *hcop*, *hmet*, *h1*, *h2*, *bs*, *ts*, *twr*, *hwt*, *bi*, *ti*, and *cinf - csup*. Red arrows point to the *Segment name* field, the *bs* field, the *ts* field, the *twr* field, the *alpha* field, the *bi* field, the *ti* field, the *bcls* field, the *tcls* field, the *b1* field, the *bsx* field, the top layer bar diameter field, the top layer bar spacing field, the top layer bar cover field, the bottom layer bar diameter field, the bottom layer bar spacing field, and the bottom layer bar cover field.

- 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.

## Step 1: Define material and geometric properties in PontiEC4

Segment name  
  
 Sections (eg. Sec1,Sec2,...X(m) (es. X1,X2,...))  
 |

Structural steel (B)  
 bs (mm)   Top flange in Class 1  
 ts (mm)   Top flange<40mm  
 hmet (mm)   
 twr (mm)    
 alpha   Inclined web  
 bi (mm)   
 ti (mm)   Bottom flange<40mm

Advanced options for flanges  
 Edit options   
 Edit options

Vertical stiffeners (B)  
 Distance between stiffeners (mm)   
 Rigid end post EN 1993-1-5, 5.2(2)  
 Edit options

Slab concrete (B)  
 bcls (mm)  tcls (mm)   
 b1 (mm)  bsx (mm)   
 hcop (mm)   Consider haunch

Reinforcing bars (B)

	bar diameter (mm)	bar spacing (mm)	bar cover (mm)
top layer	<input type="text" value="16"/>	<input type="text" value="100"/>	<input type="text" value="40"/>
bottom layer	<input type="text" value="16"/>	<input type="text" value="100"/>	<input type="text" value="40"/>

### 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.

Segment name  
  
 Sections (eg. Sec1,Sec2,...X(m) (es. X1,X2,...))  
 |

Structural steel (E)  
 bs (mm)   Top flange in Class 1  
 ts (mm)   Top flange<40mm  
 hmet (mm)   
 twr (mm)    
 alpha   Inclined web  
 bi (mm)   
 ti (mm)   Bottom flange<40mm

Advanced options for flanges  
 Edit options   
 Edit options

Vertical stiffeners (E)  
 Distance between stiffeners (mm)   
 Rigid end post EN 1993-1-5, 5.2(2)  
 Edit options

Slab concrete (E)  
 bcls (mm)  tcls (mm)   
 b1 (mm)  bsx (mm)   
 hcop (mm)   Consider haunch

Reinforcing bars (E)

	bar diameter (mm)	bar spacing (mm)	bar cover (mm)
top layer	<input type="text" value="16"/>	<input type="text" value="100"/>	<input type="text" value="40"/>
bottom layer	<input type="text" value="16"/>	<input type="text" value="100"/>	<input type="text" value="40"/>

## Segment D

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

The dialog box contains the following sections and fields:

- Segment name:** D
- Sections (eg. Sec1.Sect2... X (m) (es. X1 X2...)):** S1, 50.504
- Structural steel (D):**
  - bs (mm): 1100
  - ts (mm): 40
  - hmet (mm): 2700
  - twr (mm): 25
  - alpha: 0
  - bi (mm): 1200
  - ti (mm): 50
- Slab concrete (D):**
  - bcls (mm): 6281.5
  - tcls (mm): 273
  - b1 (mm): 1100
  - bsx (mm): 2875
  - hcop (mm): 75
- Vertical stiffeners (D):**
  - Distance between stiffeners (mm): 4000
- Reinforcing bars (D):**
  - bar diameter (mm): 26 (top layer), 20 (bottom layer)
  - bar spacing (mm): 100
  - bar cover (mm): 40
- Fatigue (D):**
  - Steel (Bending)  $\lambda_1$ : 1.905
  - Steel (Shear)  $\lambda_1$ : 1.905
  - Bars  $\lambda_{s,1}$ : 0.000
  - Traffic loading factor (Reinforcing bars): 0.000
- Shear connection (D):**
  - height (mm): 260
  - diameter (mm): 22
  - 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

$\lambda_1$ , 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.

## Assessment of a Composite Bridge Deck to Eurocodes

The screenshot displays the 'Results' window with the following data tables:

**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
Rt.leo	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.leo	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 <sub>cr</sub>
Shrinkage	-3.366E-4	-7.04E+6	-5E+6	0
Thermal var.	1E-4	5.4E+6	2.19E+6	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

**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
zG	1196.33	2161.3	2200.73	2091.17	2504.97	1562.09
DeltazG	--	--	--	--	--	--
Jy	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

- When complete, select **File> Export geometric properties to LUSAS**
- Browse to a suitable project folder and enter a filename of “**Sections\_Rev0.vbs**”.
- Select (check) all the **Export** check boxes (apart from Export Phase 2c), select (check) **Distance middle plane of slab - neutral axis**, and select the option **Assume slab width constant and use minimum value**. Click **OK** to finish.

The dialog box 'Export geometric properties to LUSAS' contains the following options:

- Export Phase 1
- Export Phase 2a
- Export Phase 2b
- Export Phase 2c
- Export Phase 3a, 3b
- Export Cracked
- Distance middle plane of slab - neutral axis
- Assume slab width constant and use maximum value
- Assume slab width constant and use minimum value
- Assume slab width constant and use average value
- Assume slab width variable
- Export segment as tapered

The file created can be imported into LUSAS to create new, or overwrite any similarly named geometric and material properties. Note that Phase 2c relates to prestressing and is not required here.

## 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, or line beam methods. Users can choose the most suitable approach to their method of working, and obtain forces and moments as required.

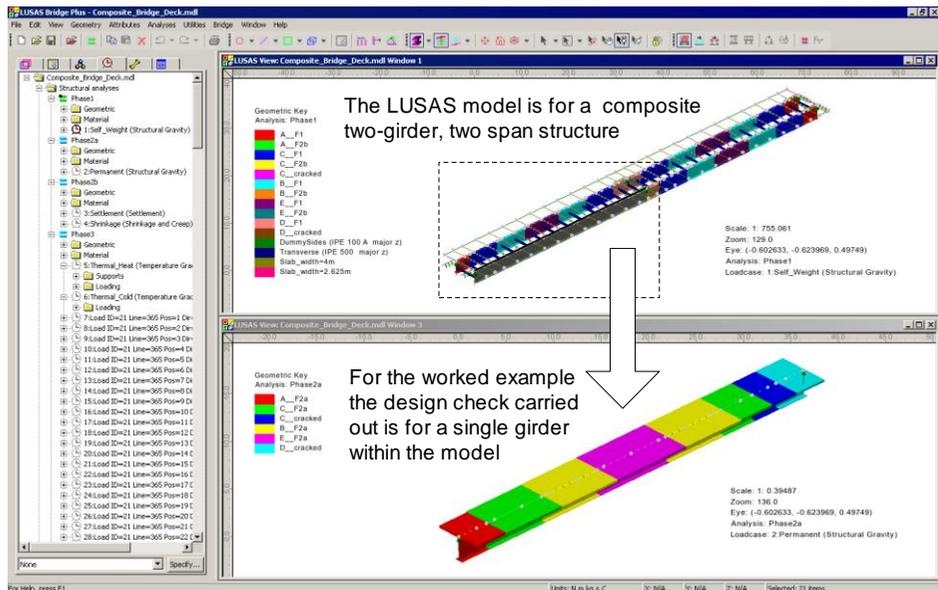
### Modelling

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

BMS3 elements have been used throughout and all mesh lies on the middle plane of the slab: the cross sections of the main girders and transverse beams are given an appropriate eccentricity within LUSAS. The edge longitudinal lines have a dummy cross section and are included in the model to enable processing of the discrete loads.

# Assessment of a Composite Bridge Deck to Eurocodes



## 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 mid-plane of the slab 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, whilst defined in PontiEC4 for effective section and other calculations, are not imported into the LUSAS model, so when starting a real-life model for the first time, these would need to be defined and assigned to relevant features. In this example, the required materials have been defined and assigned to the appropriate lines.

## 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.

## Analysis

The LUSAS multiple analysis facility is used, with four separate analyses being defined to represent the four design phases that need to be considered. 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 have been created to represent thermal action, LM1 Tandem and UDL, fatigue vehicle FLM3 and wind effects.

## 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 <sup>2</sup> ) PHASE 1 Stud Weight (N/m) __for_beam PHASE 1 Slab Weight (N/m)__for_beam PHASE 1 Curb Weight w=750mm (N/m)__external_beam PHASE 1 Curb Weight w=1500mm (N/m)__internal_beam	Self_Weight
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

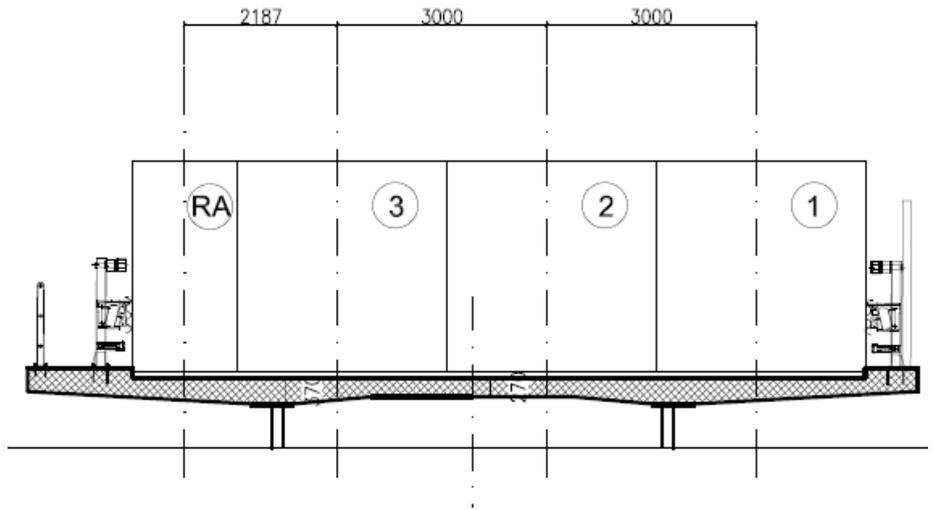
## Assessment of a Composite Bridge Deck to Eurocodes

Analysis	Loadings	Loadcases
Phase3	PHASE 3a Thermal Heat	Thermal_Heat
	PHASE 3a Thermal Cold	Thermal_Cold
	Tandem1 (Eurocode Load model 1 300000N)	From ID=2 Pos=1 to ID=36 Pos=35
	Tandem2 (Eurocode Load model 1 200000N)	From ID=37 Pos=1 to ID=71 Pos=35
	Tandem3 (Eurocode Load model 1 100000N)	From ID=72 Pos=1 to ID=106 Pos=35
	Udl1 (Eurocode Lane Load 3mx3m Load=9000N/m <sup>2</sup> )	From ID=2 Pos=1 to ID=36 Pos=35
	Udl23 (Eurocode Lane Load 3mx3m Load=2500N/m <sup>2</sup> )	From ID=37 Pos=1 to ID=71 Pos=35
	Udl23 (Eurocode Lane Load 3mx3m Load=2500N/m <sup>2</sup> )	From ID=72 Pos=1 to ID=106 Pos=35
	UdlRA (Eurocode Lane Load 3mx1.5m Load=2500N/m <sup>2</sup> )	From ID=107 Pos=1 to ID=141 Pos=35
	FLM3 (Eurocode Fatigue Load Model 3 120000N)	From ID=2 Pos=1 to ID=36 Pos=35
	PHASE 3b Wind V (N/m) __x_internal_beam PHASE 3b Wind V (N/m) __x_external_beam	Wind

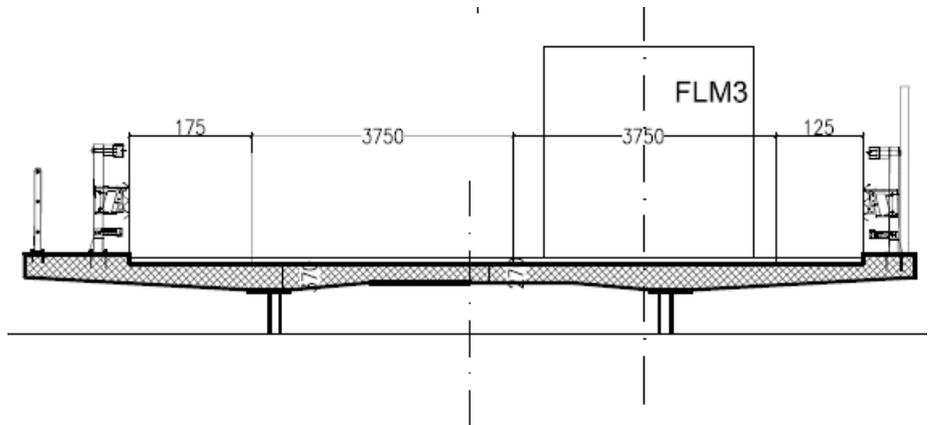
### Traffic loading

The traffic loading has been generated from the vehicle libraries in LUSAS and positioned using the Moving Load Generator. LM1 Tandem loadings have been moved along the lanes using an incremental distance of 3m. UDL loadings are defined as a segment with a length of 3m, and moved along the lanes by an incremental distance of 3m. (These patches will be combined in Smart Combinations and applied in adverse area only)

The notional lane position for the LM1 has been considered to maximize bending moments and shears in the right (external) girder beam. The lane position of the Fatigue model loading 3 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 have been defined.

### Preliminary combinations

**Loadcase ID: 288 Title: Thermal\_effect\_k**

Sub Type: Smart Combination

Loadcases to consider: 1

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
5	0	1.0	1.0	Thermal_Heat	Temperature Gradient (TG)
6	0	1.0	1.0	Thermal_Cold	Temperature Gradient (TG)

**Loadcase ID: 290 Title: Tandem\_1**

Sub Type: Smart Combination

Loadcases to consider: 1

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
7	0	1.0	1.0	Load ID=21 Line=365 Pos=1 Dir=Fwd (7)	Live Load (L)
....					
41	0	1.0	1.0	Load ID=21 Line=365 Pos=35 Dir=Fwd (4)	Live Load (L)

**Loadcase ID: 292 Title: Tandem\_2**

Sub Type: Smart Combination

Loadcases to consider: 1

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
42	0	1.0	1.0	Load ID=22 Line=368 Pos=1 Dir=Fwd (42)	Live Load (L)
....					
76	0	1.0	1.0	Load ID=22 Line=368 Pos=35 Dir=Fwd (76)	Live Load (L)

**Loadcase ID: 294 Title: Tandem\_3**

Sub Type: Smart Combination

Loadcases to consider: 1

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
77	0	1.0	1.0	Load ID=23 Line=370 Pos=1 Dir=Fwd (77)	Live Load (L)
....					
111	0	1.0	1.0	Load ID=23 Line=370 Pos=35 Dir=Fwd (111)	Live Load (L)

**Loadcase ID: 296 Title: Udl\_k**

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
112	0	0.0	1.0	Load ID=7 Line=365 Pos=1 Dir=Fwd (112)	Live Load (L)
....					
146	0	0.0	1.0	Load ID=7 Line=365 Pos=35 Dir=Fwd (146)	Live Load (L)
147	0	0.0	1.0	Load ID=10 Line=368 Pos=1 Dir=Fwd (147)	Live Load (L)
....					
216	0	0.0	1.0	Load ID=10 Line=370 Pos=35 Dir=Fwd (216)	Live Load (L)
217	0	0.0	1.0	Load ID=24 Line=372 Pos=1 Dir=Fwd (217)	Live Load (L)
....					
251	0	0.0	1.0	Load ID=24 Line=372 Pos=35 Dir=Fwd (251)	Live Load (L)

**Loadcase ID: 298 Title: Tandem\_k**

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title
290	0	0.0	1.0	Tandem_1 (Max)
291	0	0.0	1.0	Tandem_1 (Min)
292	0	0.0	1.0	Tandem_2 (Max)
293	0	0.0	1.0	Tandem_2 (Min)
294	0	0.0	1.0	Tandem_3 (Max)
295	0	0.0	1.0	Tandem_3 (Min)

**Loadcase ID: 300 Title: Wind\_k**

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
287	0	-1.0	1.0	Wind	Wind Load (W)

**ULS design combinations**

**Loadcase ID: 302 Title: ULS\_Fundamental\_Phase1**

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
1	0	1.0	1.35	Self_Weight	Structural Gravity (G)

**Loadcase ID: 304 Title: ULS\_Fundamental\_Phase2a**

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
2	0	1.0	1.35	Permanent	Structural Gravity (G)

**Loadcase ID: 306 Title: ULS\_Fundamental\_Phase2b**

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
3	0	0.0	1.2	Settlement	Settlement (STL)
4	0	1.2	1.2	Shrinkage	Shrinkage and Creep (SC)

**Loadcase ID: 308 Title: ULS\_Fundamental\_Phase3a**

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title
288	0	1.2	1.2	Thermal_effect_k (Max)
289	0	1.2	1.2	Thermal_effect_k (Min)

## Assessment of a Composite Bridge Deck to Eurocodes

---

### Loadcase ID: 310 Title: ULS\_Fundamental\_Phase3b

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title
296	0	0.0	1.35	Udl_k (Max)
297	0	0.0	1.35	Udl_k (Min)
298	0	0.0	1.35	Tandem_k (Max)
299	0	0.0	1.35	Tandem_k (Min)
300	0	0.9	0.9	Wind_k (Max)
301	0	0.9	0.9	Wind_k (Min)

### SLS characteristic design combinations

#### Loadcase ID: 312 Title: SLS\_Characteristic\_Phase1

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
1	0	1.0	1.0	Self_Weight	Structural Gravity (G)

#### Loadcase ID: 314 Title: SLS\_Characteristic\_Phase2a

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
2	0	1.0	1.0	Permanent	Structural Gravity (G)

#### Loadcase ID: 316 Title: SLS\_Characteristic\_Phase2b

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
3	0	0.0	1.0	Settlement	Settlement (STL)
4	0	1.0	1.0	Shrinkage	Shrinkage and Creep (SC)

#### Loadcase ID: 318 Title: SLS\_Characteristic\_Phase3a

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title
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	Results File	Beneficial Factor	Adverse Factor	Title
296	0	0.0	1.0	Udl_k (Max)
297	0	0.0	1.0	Udl_k (Min)
298	0	0.0	1.0	Tandem_k (Max)
299	0	0.0	1.0	Tandem_k (Min)
300	0	0.6	0.6	Wind_k (Max)
301	0	0.6	0.6	Wind_k (Min)

**SLS frequent design combinations**

**Loadcase ID: 322 Title: SLS\_Frequent\_Phase1**

Sub Type: Smart Combination

Loadcases to consider: 1

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
1	0	1.0	1.0	Self_Weight	Structural Gravity (G)

**Loadcase ID: 324 Title: SLS\_Frequent\_Phase2a**

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
2	0	1.0	1.0	Permanent	Structural Gravity (G)

**Loadcase ID: 326 Title: SLS\_Frequent\_Phase2b**

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
3	0	0.0	1.0	Settlement	Settlement (STL)
4	0	1.0	1.0	Shrinkage	Shrinkage and Creep (SC)

**Loadcase ID: 328 Title: SLS\_Frequent\_Phase3a**

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title	Type
5	0	1.0	1.0	Thermal_Heat	Temperature Gradient (TG)
6	0	1.0	1.0	Thermal_Cold	Temperature Gradient (TG)

**Loadcase ID: 330 Title: SLS\_Frequent\_Phase3b**

Sub Type: Smart Combination

Loadcases to consider: All

Variable Loadcases: All

Loadcase	Results File	Beneficial Factor	Adverse Factor	Title
296	0	0.0	0.4	Udl_k (Max)
297	0	0.0	0.4	Udl_k (Min)
298	0	0.0	0.75	Tandem_k (Max)
299	0	0.0	0.75	Tandem_k (Min)

### Fatigue design combinations

Loadcase ID: 332 Title: Fatigue\_Phase3b

Sub Type: Smart Combination

Loadcases to consider: 1

Variable Loadcases: All

Loadcase	Results	Beneficial	Adverse	Title	Type
	File	Factor	Factor		
252	0	1.0	1.0	Load ID=25 Line=374 Pos=1 Dir=Fwd (252)	Live Load (L)
....					
286	0	1.0	1.0	Load ID=25 Line=374 Pos=35 Dir=Fwd (286)	Live Load (L)

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

### Associated Files



- Sections\_Rev0.vbs** - A file created by PontiEC4 containing geometric section data to import into LUSAS. A version of this file is supplied also, but is only 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.

### 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.

The model supplied is a LUSAS Version 15 model. This is so that the same model can be opened regardless of the version of LUSAS being used with PontiEC4. As a result:

- Click **OK** to update the model to your current version
- 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

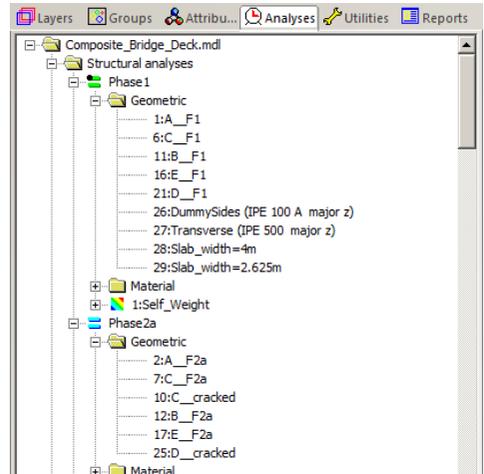
- Select **File> Script> Run Script** and select the PontiEC4 script file “**Sections\_Rev0.vbs**” that was created in the working project folder earlier. This imports the geometric section data from PontiEC4.



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

- In the Analyses treeview  notice that different geometric assignments have been made for the main girders and material assignments for the slab for each analysis to represent the various design phases.
- 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 elements for checking

To enable design checks to be carried out the start and end elements of each main girder segment need to be selected. These elements are those that occur at the changes of geometric section, and they represent the sections at which design checks will be undertaken in PontiEC4.

#### In preparation:

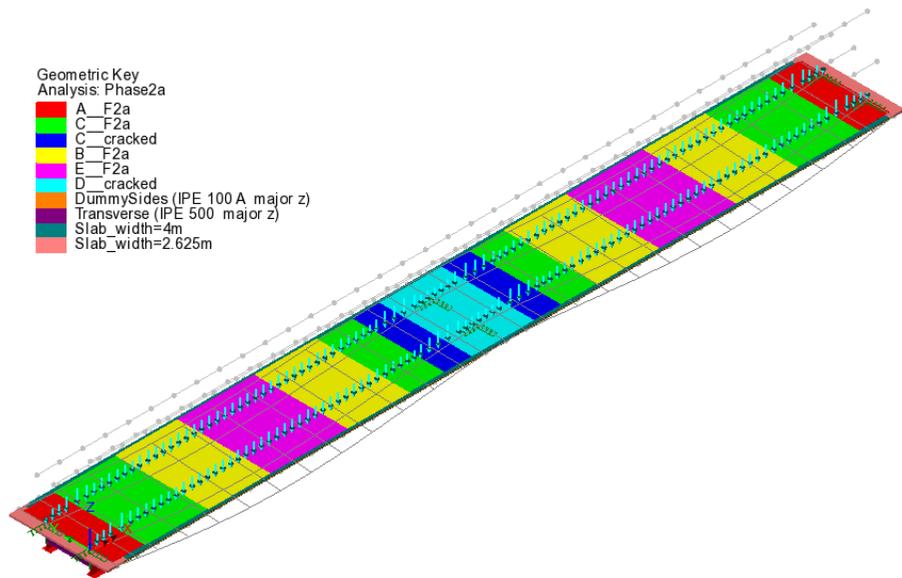


Press the Isometric toolbar button to see the full model of the bridge deck.



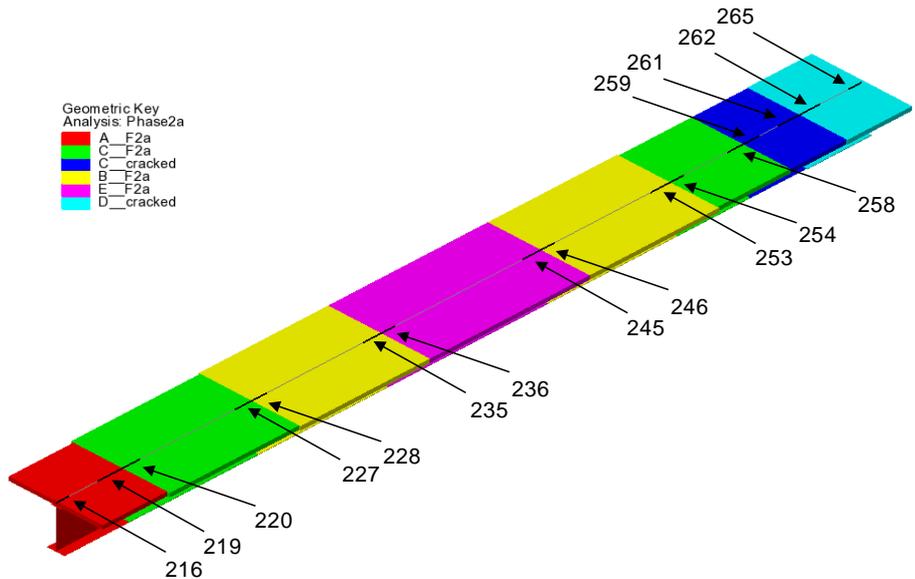
Press the fleshing button

- In the Analyses  treeview expand the **Phase 2a** analysis entry and right-click on the loadcase named **Permanent** and choose **Set active**. This is the loadcase where the concrete deck is constructed.
- In the Layers  Treeview, double-click the **Geometry** entry and on the Properties dialog choose to Colour by **Assignment**, and then press the **Set** button. Choose to colour by **Geometric** attribute type and click **OK**.

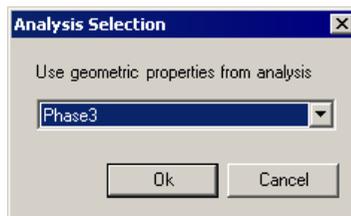


- In the Groups  treeview right-click on the group name **Half\_External\_Beam** and choose **Set as Only Visible**.
- In the Layers  Treeview turn off the display of the **Deformed mesh** layer
- In the Layers  Treeview move the **Mesh** layer to be beneath the **Attributes** layer. This ensures that the required elements can be seen and selected.
- In the Layers  Treeview, double-click on the **View properties** entry, select the **View Axes** tab and deselect **Visualise coordinate system**.
-  Turn off the display of the supports
- Change the cursor selection mode to be **Select Elements** .
- Holding-down the **Shift** key, select each element at the start and end of each segment. The element numbers selected will be as shown.

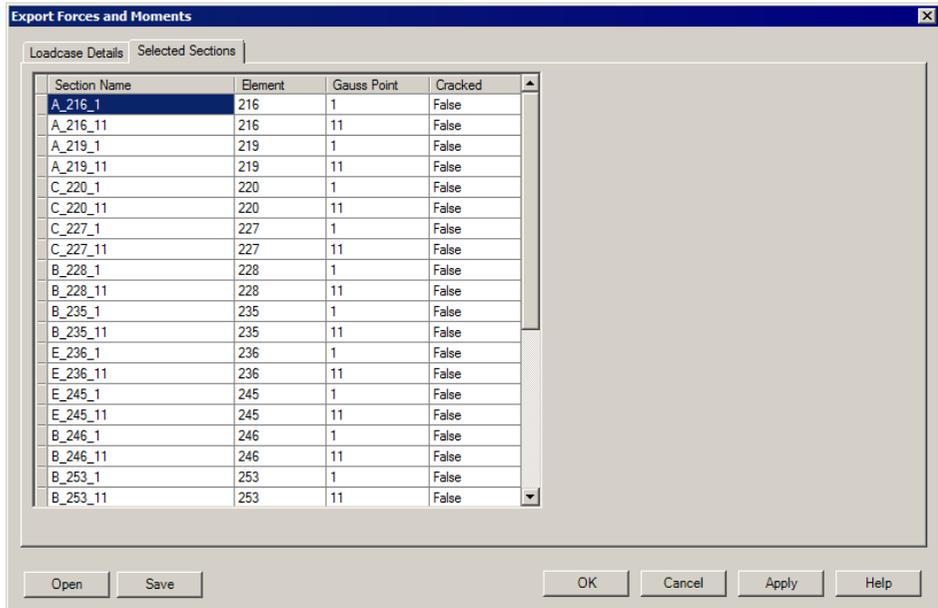
## Step 4: Export the force and moment data from LUSAS



- Once done, to export the information for these sections, select the **Design > Export to Composite Deck Design...** menu item.
- In the Analysis selection dialog that appears, select **Phase 3** from the drop-down list provided, and click **OK**. The geometric properties assigned to Phase3 will be used to determine the section details for export.



- On the Export Forces and Moments dialog, select the **Selected Sections** tab. These sections have been defined based on the elements selected prior to selecting the **Bridge > Composite Deck Designer > Export...** menu item.



For each selected element, the table defines two sections, representing the first and last Gauss points. Each Section name listed has the following information: **Element (number)**, **Gauss point (number)**, **Cracked/uncracked section status**.

The section names in this table are created from the following named objects/values in the LUSAS model:

“[Segment Name]+“\_”+ “[Element number]+“\_”+[Gauss Point Number]



**Important.** The Export Forces and Moments facility obtains the segment and the phase name from the name of the assigned geometric attribute, by splitting the attribute name on the double underscore separator “\_”. For this reason the cross-section geometric properties in any associated LUSAS model must have a name that follows this standard:

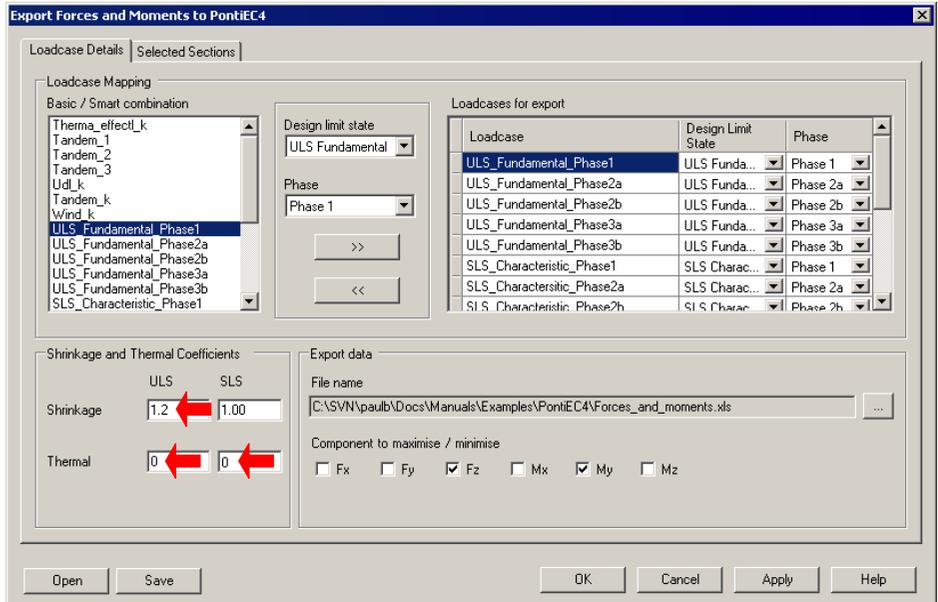
[Segment Name] +”\_”+ [Phase name]; where the phase name should be: {F1, F2a, F2b, F3a, F3b, cracked}.

The cracked status of the section is a Boolean value taken as true when the phase name is “cracked”, and false otherwise.

- Click the **Loadcase Details** tab.

## Step 4: Export the force and moment data from LUSAS

On the Loadcase Details tab the design combinations, design limit states and phases are to be stated, along with the specification of shrinkage/thermal coefficients and component values of interest.



To assemble a design combination, proceed as follows:

- In the left-hand panel select **ULS\_Fundamental\_Phase1**, select a Design limit state of **ULS\_Fundamental**, and from the Phase drop-down list select **Phase 1**. Click the Add to button  to add the design combination to the right-hand panel
- Repeat the above sequence for all combinations listed in the table that follows, **selecting** the corresponding **Design limit state** and **Phase** for each one.

Loadcase	Design limit state	Phase
ULS_Fundamental_Phase1	ULS Fundamental	Phase 1
ULS_Fundamental_Phase2a	ULS Fundamental	Phase 2a
ULS_Fundamental_Phase2b	ULS Fundamental	Phase 2b
ULS_Fundamental_Phase3a	ULS Fundamental	Phase 3a
ULS_Fundamental_Phase3b	ULS Fundamental	Phase 3b
SLS_Characteristic_Phase1	SLS Characteristic	Phase 1
SLS_Characteristic_Phase2a	SLS Characteristic	Phase 2a
SLS_Characteristic_Phase2b	SLS Characteristic	Phase 2b
SLS_Characteristic_Phase3a	SLS Characteristic	Phase 3a

## Assessment of a Composite Bridge Deck to Eurocodes

---

Loadcase	Design limit state	Phase
SLS_Characteristic_Phase3b	SLS Characteristic	Phase 3b
SLS_Frequent_Phase1	SLS Frequent	Phase 1
SLS_Frequent_Phase2a	SLS Frequent	Phase 2a
SLS_Frequent_Phase2b	SLS Frequent	Phase 2b
SLS_Frequent_Phase3a	SLS Frequent	Phase 3a
SLS_Frequent_Phase3b	SLS Frequent	Phase 3b
SLS_Frequent_Phase1	Fatigue	Phase 1
SLS_Frequent_Phase2a	Fatigue	Phase 2a
SLS_Frequent_Phase2b	Fatigue	Phase 2b
SLS_Frequent_Phase3a	Fatigue	Phase 3a
Fatigue_Phase3b	Fatigue	Phase 3b

Note that loadcase SLS\_Frequent\_Phase3b is not mapped as a Fatigue design limit state; the Fatigue\_Phase3b loadcase must be mapped instead.

- Input a Shrinkage coefficient of **1.2** for the ULS, and **1.0** for the SLS.
- **Remove the default thermal coefficient values** that are present and enter values of **0** for both the ULS and SLS Thermal coefficients



**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 ( $\gamma*\psi=0$ ) in the cracked section (True in cracked column of the Section Table column). The hyperstatic effect of the shrinkage comes from the LUSAS model and the combination factors have been already applied in the LUSAS smart combinations.

- 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)
- Click the ellipsis button  to choose the folder and name of the xls file to create (e.g.: <current work directory>\**Forces\_and\_moments.xls**), pressing the **Save** button to return to the main dialog
- Click the **Save** button on the bottom-left of the dialog to save the data entered in the dialog pages for any possible re-use. Save the input as **“Export\_Forces\_to\_PontiEC4.inp”**.
- Then, finally, click **OK** to create the XLS file containing forces and moments for use in PontiEC4.

## 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.

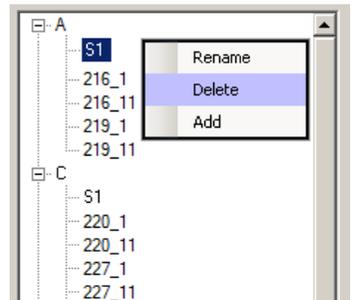


- ❑ **Forces\_and\_moments.xls** File created by the LUSAS Export Forces and Moments dialog containing data to import into PontiEC4.

### 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, 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 **Forces\_and\_moments.xls** from working project folder.

PontiEC4 will detect that 32 additional sections are present (these are the sections at the beginning and end of each beam element that was selected to signify a change of cross-section geometric property in the LUSAS model) 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 ? 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
<b>Red</b>	Greater than 1.0	Value has failed the check carried out.
<b>Green</b>	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.

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

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.



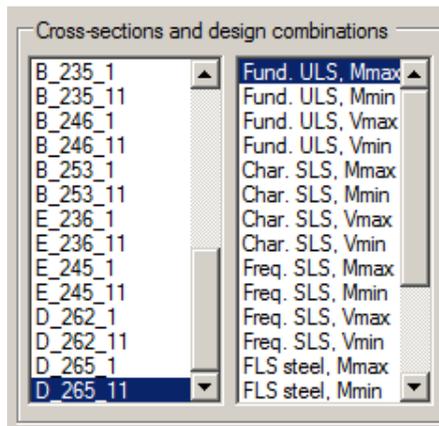
**Note.** By clicking on a particular header the values in a column can be sorted in increasing or decreasing value. Click twice to sort the **SigEd/fy** column in descending order. It can be seen that some of Segments D and C fail the check for normal stresses at ULS, as shown below.

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/( $\eta$ *PRd)	
D_265_11	50.504	Fund. ULS, Mmax	4	4	(.84)	1.185	0.705	1.33	0.705	1.241	0.506	
C_261_11	46.454	Fund. ULS, Mmax	4	4	(.73)	1.176	0.644	1.23	0.644	1.206	0.339	
D_265_11	50.504	Fund. ULS, Vmax	4	4	(.81)	1.149	0.75	1.29	0.75	1.232	0.563	
D_265_1	49.629	Fund. ULS, Mmax	4	4	(.78)	1.105	0.613	1.24	0.613	1.122	0.393	
C_261_1	45.252	Fund. ULS, Mmax	4	4	(.66)	1.054	0.603	1.1	0.603	1.069	0.298	
D_265_1	49.629	Fund. ULS, Vmax	4	4	(.74)	1.054	0.748	1.19	0.748	1.136	0.563	
C_261_11	46.454	Fund. ULS, Vmax	4	4	(.63)	1.018	0.861	1.07	0.861	1.199	0.569	
D_262_11	47.878	Fund. ULS, Mmax	4	4	(.68)	0.955	0.54	1.07	0.54	0.957	0.335	
C_259_11	44.090	Fund. ULS, Mmax	4	4	(.59)	0.945	0.535	0.99	0.535	No int.	0.254	
R_235_11	20.250	Fund. ULS, Mmin	4	1	0.73	(.879)	0.104	1.02	0.104	No int.	0.091	



**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\_265\_11** 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**

## Assessment of a Composite Bridge Deck to Eurocodes

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	eta1	id														
σ 8	0.0	1.0	0.0	1.1	0.0	0.0	0.0	0.0	1.7	0.0	4.2	0.0	0.0	0.0	σ 8														
σ 7	0.0	17.0	31.1	16.4	31.6	0.0	0.0	62.7	10.0	32.9	24.9	82.4	177.9	0.45	σ 7														
σ 6	0.0	13.5	26.6	13.0	27.1	0.0	0.0	53.7	7.0	28.2	17.3	70.4	152.2	0.39	σ 6														
σ 5	0.0	0.7	0.0	0.8	0.0	0.0	0.0	0.0	1.0	0.0	2.5	0.0	0.0	0.0	σ 5														
σ 4	180.5	11.5	23.9	11.0	24.4	0.0	0.0	228.8	5.2	25.3	12.8	63.2	317.4	0.94	σ 4														
σ 3	175.5	10.8	23.0	10.3	23.5	0.0	0.0	222.0	4.6	24.4	11.3	60.8	307.1	0.91	σ 3														
σ 2	0.3	-0.1	-0.2	0.0	0.0	0.0	0.0	0.1	0.0	0.0	-0.6	-1.3	-1.2	0.00	σ 2														
σ 1	-14.0	-35.2	-37.5	-35.4	-37.9	0.0	0.0	-224.3	-35.7	-39.4	-91.3	-101.4	-365.1	1.14	σ 1														
σ 0	-15.0	-36.1	-38.7	-36.3	-39.1	0.0	0.0	-232.8	-36.5	-40.7	-93.2	-104.5	-378.0	1.18	σ 0														

**Slab stresses at Phase 2 (N/mm<sup>2</sup>):**  
 Total top stress = 2.04  
 Total bottom stress = 1.47  
 => Section at the end of Phase 2: CRACKED (m.)

**Slab stresses at Phase 3 (N/mm<sup>2</sup>):**  
 Total top stress = 7.98  
 Total bottom stress = 5.03  
 => Section at the end of Phase 3: CRACKED (m.)

**=> El. check Phase 3 FAILED**  
 eta1 = 1.185



**Note.** Currently, the ‘Stresses tables’ contain non-English abbreviations in their header that need to be explained: ‘F1’, ‘F2a’, ‘F2b’ etc come from the Italian ‘Fase’ – meaning ‘Phase’ in English, as in Phase 1, Phase 2 etc. Additionally ‘N.F.’ is the Italian abbreviation for ‘Non Fessurata’ – meaning ‘Not Cracked’, and ‘F’ denotes ‘Fessurata’ – meaning ‘Cracked’.

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 fy. They are also quite large in the top flange.

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

Determining Class and Plastic verification at Stage 3														
	c/t	zpl(mm)	α	ψ	Class									
Web	104.4	2347	0.88	-0.87	4									
Upper flange	13.44				1									
Lower flange	11.75				4									
<b>Cross-section class</b>					<b>4</b>									

=> Plastic verification NOT APPLICABLE

Axial force N	Bending moment M	N-M Interaction
NEd	7.05E+7	NEd
MRd	8.44E+7	MEd
		MRd
		8.44E+7
NEd/NRd	0.002	MEd/MRd
		0.836
		MEd/MR
		0.836

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

ELASTIC STRESSES (EFFECTIVE SECTION)

## 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 (ti) from 50 to **80**
- Select the **Window > 4 Results** menu item. This causes the results for Section **D\_265\_11** 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 Stress 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.**

Stresses of gross 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	eta1	id	
σ 8	0.0	0.9	0.0	1.0	0.0	0.0	0.0	0.0	1.5	0.0	3.8	0.0	0.0	0.00	σ 8	
σ 7	0.0	15.4	27.4	14.9	27.9	0.0	0.0	55.2	9.1	29.0	22.8	72.6	156.8	0.40	σ 7	
σ 6	0.0	12.8	23.9	12.4	24.4	0.0	0.0	48.3	7.0	25.3	17.2	63.4	137.0	0.35	σ 6	
σ 5	0.0	0.7	0.0	0.7	0.0	0.0	0.0	0.0	1.0	0.0	2.6	0.0	0.0	0.00	σ 5	
σ 4	149.6	11.3	21.9	10.8	22.3	0.0	0.0	193.7	5.7	23.2	13.9	57.9	274.8	0.86	σ 4	
σ 3	144.9	10.6	21.0	10.2	21.4	0.0	0.0	187.2	5.1	22.2	12.5	55.5	264.9	0.83	σ 3	
σ 2	0.2	-0.1	-0.1	0.0	0.0	0.0	0.0	0.1	0.0	0.0	-0.5	-1.0	-1.0	0.00	σ 2	
σ 1	-96.4	-23.6	-25.0	-23.8	-25.2	0.0	0.0	-146.6	-24.0	-26.2	-61.5	-67.6	-240.4	0.75	σ 1	
σ 0	-10...	-24.7	-26.4	-24.8	-26.6	0.0	0.0	-156.9	-24.9	-27.8	-63.8	-71.4	-256.1	0.80	σ 0	

**Slab stresses at Phase 2 (N/mm<sup>2</sup>):**  
 Total top stress = 1.84  
 Total bottom stress = 1.41  
 => Section at the end of Phase 2: CRACKED (m.)

**Slab stresses at Phase 3 (N/mm<sup>2</sup>):**  
 Total top stress = 7.2  
 Total bottom stress = 5.03  
 => Section at the end of Phase 3: CRACKED (m.)

=> El. check Phase 3 PASSED  
 eta1= 0.861

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

## Assessment of a Composite Bridge Deck to Eurocodes

### 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 (ts) from 30 to **50**
- Increase the web thickness (twr) from 22 to **25**
- Increase the bottom flange thickness (ti) from 40 to **60**
- Select the **Window> 4 Results** menu item and view the updated results for Section **C\_261\_11** 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).

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	eta1	id														
σ 8	0.0	0.6	0.0	1.0	0.0	0.0	0.0	0.0	1.6	0.0	2.9	0.0	0.0	0.00	σ 8														
σ 7	0.0	10.6	20.6	15.5	31.8	0.0	0.0	52.4	9.2	33.0	17.0	61.9	147.3	0.38	σ 7														
σ 6	0.0	8.6	17.9	12.6	27.7	0.0	0.0	45.6	6.6	28.7	12.2	53.7	128.0	0.33	σ 6														
σ 5	0.0	0.5	0.0	0.7	0.0	0.0	0.0	0.0	1.0	0.0	1.8	0.0	0.0	0.00	σ 5														
σ 4	148.7	7.5	16.3	10.8	25.2	0.0	0.0	190.2	5.1	26.1	9.3	48.8	265.2	0.83	σ 4														
σ 3	144.2	7.0	15.6	10.1	24.1	0.0	0.0	183.9	4.5	25.0	8.1	46.7	255.7	0.80	σ 3														
σ 2	0.3	-0.1	-0.1	0.0	0.0	0.0	0.0	0.2	0.0	0.0	-0.5	-1.1	-1.0	0.00	σ 2														
σ 1	-89.9	-19.2	-20.6	-29.2	-31.4	0.0	0.0	-141.8	-29.4	-32.7	-56.6	-63.5	-238.0	0.75	σ 1														
σ 0	-95.3	-19.8	-21.4	-30.1	-32.7	0.0	0.0	-149.4	-30.2	-34.1	-58.1	-66.0	-249.5	0.78	σ 0														

**Slab stresses at Phase 2 (N/mm<sup>2</sup>):**  
 Total top stress = 1.6  
 Total bottom stress = 1.2  
 => Section at the end of Phase 2: CRACKED (m.)

**Slab stresses at Phase 3 (N/mm<sup>2</sup>):**  
 Total top stress = 6.05  
 Total bottom stress = 3.97  
 => Section at the end of Phase 3: CRACKED (m.)

-> El. check Phase 3 PASSED  
 eta1 = 0.831

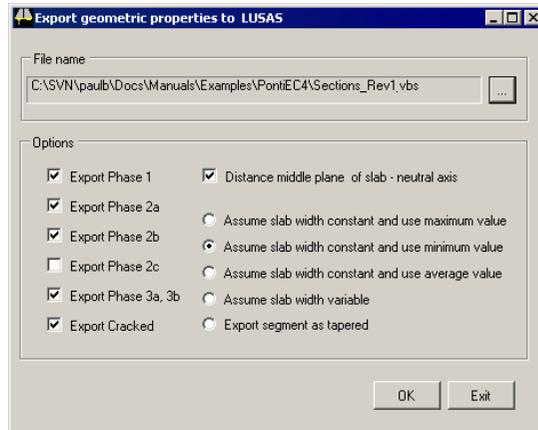
- Select the **Window> 5 Summary of results** menu item. Sort the **SigEd/fy** column. It can be seen that Segments D and C now both pass the check for normal stresses at ULS, as shown below. Stresses for other combinations are all satisfactory.

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/(η*PRd)	
B_235_11	20.250	Fund. ULS, Mmin	4	1	0.73	(.879)	0.104	1.02	0.104	No int.	0.091	
B_235_1	19.251	Fund. ULS, Mmin	4	1	0.72	(.871)	0.107	1.01	0.107	No int.	0.091	
D_265_11	50.504	Fund. ULS, Mmax	1	3	(.65)	0.861	0.708	0.82	0.708	0.897	0.475	
D_265_11	50.504	Fund. ULS, Vmax	1	3	(.63)	0.842	0.753	0.79	0.753	0.895	0.529	
C_261_11	46.454	Fund. ULS, Mmax	3	3	(.59)	0.831	0.507	0.82	0.507	0.831	0.316	
D_265_11	49.629	Fund. ULS, Mmax	1	3	(.61)	0.8	0.616	0.76	0.616	0.811	0.369	
E_236_11	21.252	Fund. ULS, Mmin	4	1	0.67	(.796)	0.053	0.9	0.054	No int.	0.067	
E_236_1	20.250	Fund. ULS, Mmin	4	1	0.67	(.793)	0.055	0.9	0.057	No int.	0.067	
B_228_11	13.252	Fund. ULS, Mmin	4	1	0.66	(.786)	0.428	0.92	0.434	No int.	0.239	
D_265_1	49.629	Fund. ULS, Vmax	1	3	(.58)	0.772	0.749	0.73	0.751	No int.	0.529	

- 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 ensure that all the **Export** option boxes (apart from Export Phase 2c), and selected (checked) and that **Distance middle plane of slab – neutral axis**, and **Assume slab width constant and use minimum value options** are also selected (checked). 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. The new sections will overwrite the geometric and material properties previously used.
- 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:

- In LUSAS, selecting those elements of interest at the start and end of each main girder segment (they may still be selected from when this was done earlier), and exporting the results for those elements by selecting the **Design> Export to Composite Deck Design...** menu item.

## Assessment of a Composite Bridge Deck to Eurocodes

- Ensure that **Phase 3** is chosen, and when the **Export Forces and Moments** dialog appears press the **Open** button to populate the dialog with previously saved entries from the **Export\_Forces\_to\_PontiEC4.inp** file that related to the loadcase mapping and other settings. Click **OK** to overwrite existing data.
- Then, on the same dialog, click the ellipsis button  to choose the folder and name of the xls file that should be created (e.g.: **Forces\_and\_moments\_rev1.xls**), and press **Save** to return to the main dialog.
- Then, finally, click **OK** to create the revised XLS file for use in PontiEC4

### Import revised forces and moments into PontEC4

- In PontiEC4, with the current data still loaded, select the **File> Import forces/moments** menu item and load the file **Forces\_and\_moments\_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.

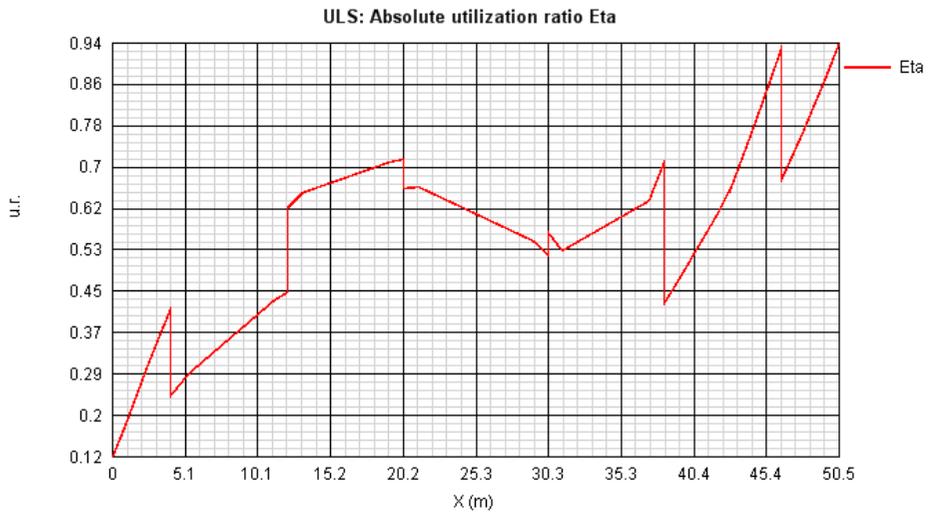
Fundamental ULS combination											
Section	X (m)	Combination	Class Ph.1	Class Ph.3b	MEd/MR	SigEd/fy	VEd/VRd	MEd/Mf,Rd	VEd/Vbw,Rd	V/M/N	vEd/(n*PRd)
D_265_11	50.504	Fund. ULS. Mmax	1	3	(.71)	0.944	0.73	0.9	0.73	0.987	0.487
C_261_11	46.454	Fund. ULS. Mmax	3	3	(.66)	0.932	0.527	0.92	0.527	0.933	0.329
D_265_11	50.504	Fund. ULS. Vmax	1	3	(.69)	0.922	0.774	0.87	0.774	0.983	0.539
D_265_1	49.629	Fund. ULS. Mmax	1	3	(.67)	0.88	0.637	0.84	0.637	0.896	0.38
B_235_11	20.250	Fund. ULS. Mmin	4	1	0.71	(.852)	0.087	1	0.087	No int.	0.087
C_261_11	46.454	Fund. ULS. Vmax	3	3	(.58)	0.85	0.696	0.8	0.696	0.894	0.544
D_265_1	49.629	Fund. ULS. Vmax	1	3	(.64)	0.85	0.772	0.8	0.772	0.91	0.539
B_235_1	19.251	Fund. ULS. Mmin	4	1	0.7	(.846)	0.09	0.99	0.09	No int.	0.087
C_261_1	45.252	Fund. ULS. Mmax	3	3	(.6)	0.83	0.508	0.84	0.508	0.83	0.308
R_228_11	13.252	Fund. ULS. Mmin	4	1	0.65	(.771)	0.411	0.91	0.418	No int.	0.236

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

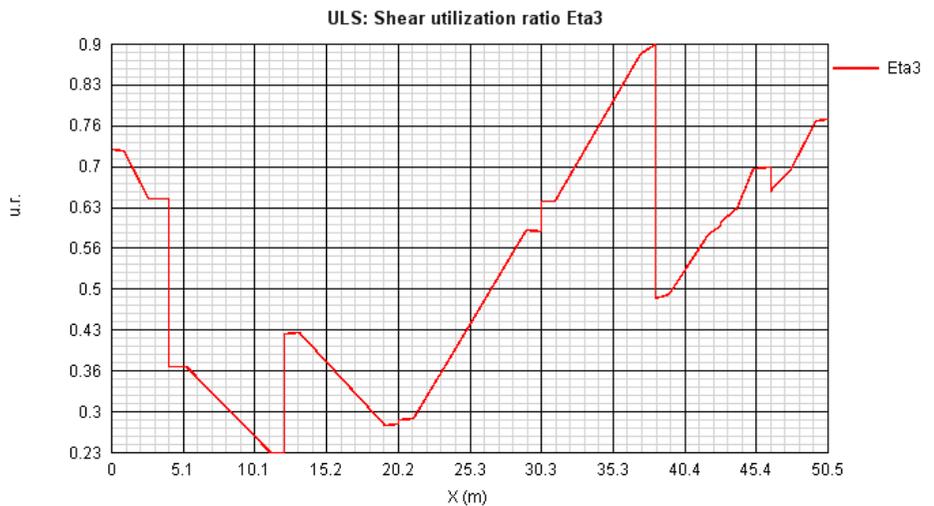
### Creating Graphs

- 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:



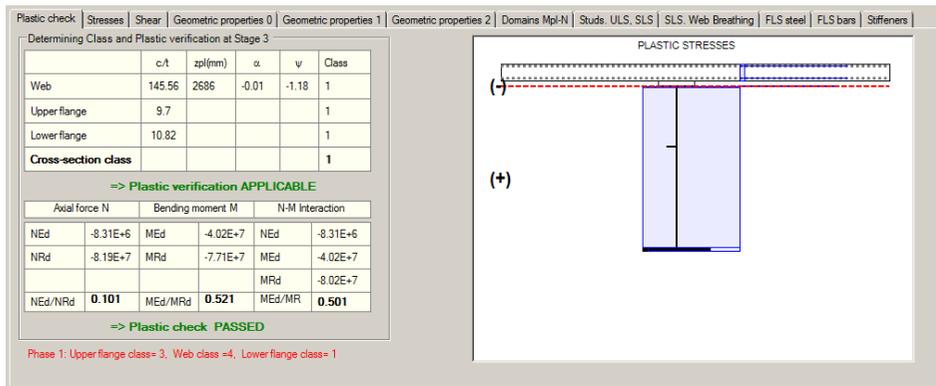
- From the first drop-down list choose **ULS: Shear utilization ratio Eta3** to plot the following graph:



**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 section **E\_245\_11** 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.



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

Determining Class and Plastic verification at Stage 3

	c/t	zpl(mm)	$\alpha$	$\psi$	Class
Web	145.56	2686	-0.01	-1.18	1
Upper flange	9.7				1
Lower flange	10.82				1
<b>Cross-section class</b>					<b>1</b>

=> Plastic verification APPLICABLE

Axial force N	Bending moment M	N-M interaction
NEd -8.31E+6	MEd -4.02E+7	NEd -8.31E+6
NRd -8.19E+7	MRd -7.71E+7	MEd -4.02E+7
		MRd -8.02E+7
NEd/NRd <b>0.101</b>	MEd/MRd <b>0.521</b>	MEd/MR <b>0.501</b>

=> Plastic check PASSED

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

PLASTIC STRESSES

(-)

(+)

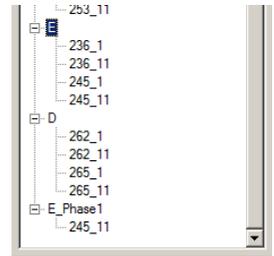


**Note.** At the bottom-left of the Plastic check page the section classification is also provided for Phase 1, where the top (upper) flange cannot be considered to be restrained by the slab. For this case, the section is in Class 4 as highlighted in the red message. For this situation a check of stresses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

### To check stresses on the effective section in Phase1

- Select the **Window> 2 Geometry** menu item.
- On the Geometry dialog, double-click on Segment **E** in the segment treeview to change from Edit mode to Input mode. The status bar should state: "Input GEOMETRY"
- In the Segment name field in the top-left of the dialog page (and not in the segment treeview panel) change the segment name from **E** to **E\_Phase1**, then click outside of the text box to be sure that the new name is saved.
- Right-click inside the Sections text box and select **Data grid input**

- Select all rows except for the one with 245\_11 in it (use the Ctrl key for a multiple selection of rows), open the context menu with the right-hand click of the mouse, and select **Remove rows** to leave just **245\_11** remaining. Click **OK**.
- In the Structural steel panel **unchecked** the option **Top Flange in Class 1**
- In the Slab concrete panel input **0** in each of the **bcls**, **b1** and **bsx** text boxes
- Click the **Add to list** button beneath the Segment treeview to add the new segment to the list of other segments in the segment treeview, as shown right.
- Select the **Window > 3 Forces and moments** menu item, and on the Results dialog, select **ULS fund., Mmin** in the dropdown list seen in the Cross-sections and design combinations panel in the top-left corner.
- Copy the four cells of forces and moment data for Phase 1 from section **E\_245\_11**, (click **No** to not copy the header text) and paste the copied data into the equivalent cells for section **E\_Phase1\_245\_11**, as shown on the following images.



ULS fund., Mmin

PHASE 1. Selfweights				
Section	N (N)	V (N)	M (Nm)	T (Nm)
E_236_11	3.657E+004	2.269E+005	-1.218E+007	2.143E+001
E_245_1	3.657E+004	9.224E+005	-7.583E+006	-1.085E+002
E_245_11	3.657E+004	9.349E+005	-6.653E+006	-1.085E+002
D_262_1	2.712E+004	1.710E+006	1.431E+007	-2.042E+002
D_262_11	2.712E+004	1.733E+006	1.676E+007	-2.042E+002

PHASE 2A. Permanent loads (non-structural bridge equipments)

ULS fund., Mmin

PHASE 1. Selfweights				
Section	N (N)	V (N)	M (Nm)	T (Nm)
D_262_1	2.712E+004	1.710E+006	1.431E+007	-2.042E+002
D_262_11	2.712E+004	1.733E+006	1.676E+007	-2.042E+002
D_265_1	2.713E+004	1.905E+006	1.995E+007	-1.862E+002
D_265_11	2.713E+004	1.919E+006	2.163E+007	-1.862E+002
E_Phase1_245_11	3.657E+004	9.349E+005	-6.653E+006	-1.085E+002

PHASE 2A. Permanent loads (non-structural bridge equipments)

- Select the **Window > 4 Results** menu item
- Select Section **E\_Phase1\_245\_11** 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)	$\alpha$	$\psi$	Class
Web	145.56	421	0.86	-0.53	4
Upper flange	9.7				3
Lower flange	10.82				1
<b>Cross-section class</b>					<b>4</b>

=> Plastic verification NOT APPLICABLE

Axial force N		Bending moment M		N-M Interaction	
NEd	3.66E+4	MEd	-6.65E+6	NEd	3.66E+4
NRd	3.96E+7	MRd	-3.65E+7	MEd	-6.65E+6
				MRd	-3.66E+7
NEd/NRd	0.001	MEd/MRd	0.182	MEd/MR	0.182

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

ELASTIC STRESSES (EFFECTIVE SECTION)

## Assessment of a Composite Bridge Deck to Eurocodes

- 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 Mp+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	eta1	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	0.0	$\sigma_8$
$\sigma_7$	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	$\sigma_5$
$\sigma_4$	-93.4	0.0	0.0	0.0	0.0	0.0	0.0	-93.4	0.0	0.0	0.0	0.0	-93.4	0.28	$\sigma_4$
$\sigma_3$	-91.9	0.0	0.0	0.0	0.0	0.0	0.0	-91.9	0.0	0.0	0.0	0.0	-91.9	0.27	$\sigma_3$
$\sigma_2$	0.3	0.0	0.0	0.0	0.0	0.0	0.0	0.3	0.0	0.0	0.0	0.0	0.3	0.00	$\sigma_2$
$\sigma_1$	43.2	0.0	0.0	0.0	0.0	0.0	0.0	43.2	0.0	0.0	0.0	0.0	43.2	0.14	$\sigma_1$
$\sigma_0$	45.8	0.0	0.0	0.0	0.0	0.0	0.0	45.8	0.0	0.0	0.0	0.0	45.8	0.14	$\sigma_0$

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

**Slab stresses at Phase 3 (N/mm<sup>2</sup>):**  
 Total top stress = 0  
 Total bottom stress = 0  
 => Section at the end of Phase 3: CRACKED (m.)

=> **EI. check Phase 3 PASSED**  
**eta1 = 0.276**

This completes the example.