Assessment of a Composite Bridge Deck to Eurocodes

For software product(s):	LUSAS Bridge or LUSAS Bridge plus
With product option(s):	Composite Deck Designer (PontiEC4)
With additional software	Microsoft Excel installed

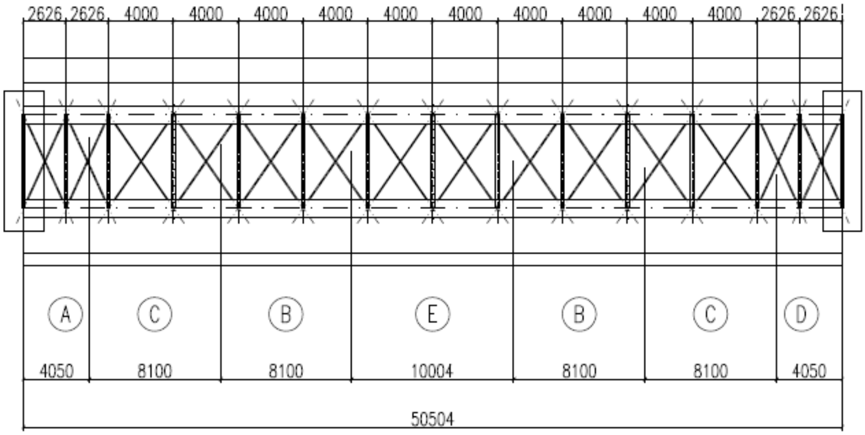
Description

The global analysis and the assessment of the main girders of a two span continuous bridge is to be carried out using the Composite Deck Designer (PontiEC4) software option and LUSAS Bridge. The example is written for use with PontiEC4 Version 3.1.

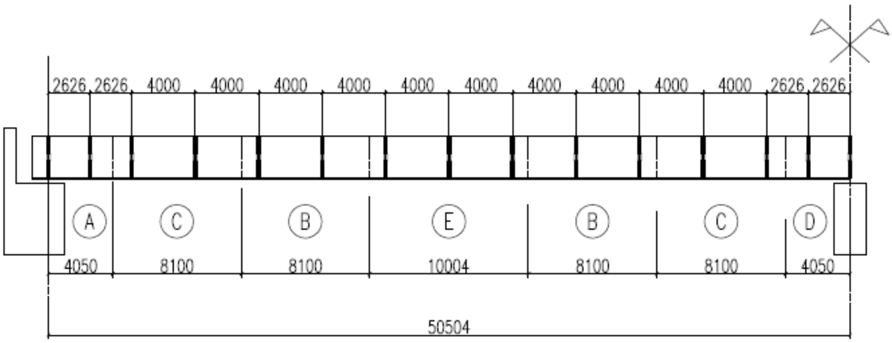
The bridge has a symmetrical composite two-girder structure with two spans of 50.504m. For the purposes of the example the following geometrical simplifications have been made:

- The horizontal alignment is straight
- The top face of the deck is flat
- The bridge is straight
- The structural steel main girders are of a constant depth of 2700 mm

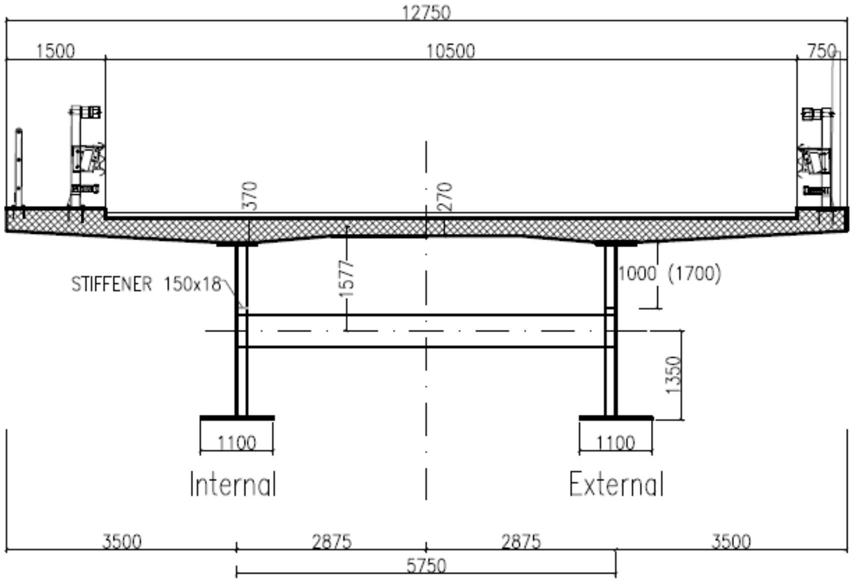
Plan, elevation and typical cross section views of the bridge deck are shown on the following pages.



Plan of the left-hand span showing location of transverse bracing.



Elevation of the left-hand span showing girder segment numbering (regions of changes in cross section). Right-hand span is similar but handed.



Typical cross section

The total slab width is 12.75 m. The centre-to-centre spacing between the main girders is 5.75 m and the slab cantilever either side is 3.5 m 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.

In the 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.

Composite Deck Designer (PontiEC4)

The 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 LUSAS 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.

Keywords

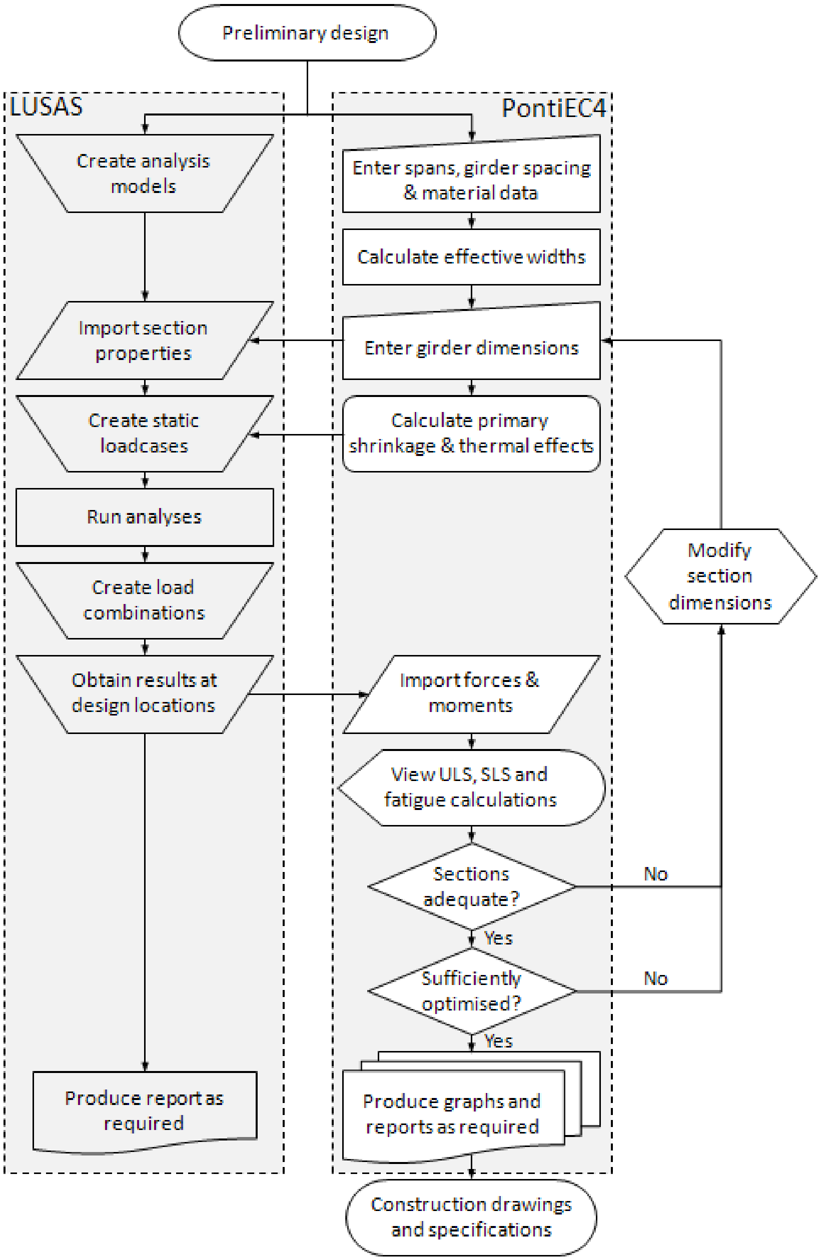
Composite Deck Designer, EC4, Design, Checking, Import, Export, ULS, SLS, Graphing

Method and Objectives

The main steps can be summarized as follows:

1. **Define geometric and material properties in the PontiEC4 Composite Deck Designer.**
2. **Export geometric and material properties from the PontiEC4 Composite Deck Designer for use in a corresponding LUSAS model.**
3. **Use LUSAS to calculate the forces and moments for all limit state combinations.**
4. **Export the force and moment data from LUSAS for use in the PontiEC4 Composite Deck Designer.**
5. **Import force and moment data into PontiEC4 Composite Deck Designer**
6. **Assess results and carry out detailed design checks in PontiEC4**
7. **Optimize the structure.**

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



Associated File

This file is supplied in case it is desired to automatically populate the material and geometry dialogs with the initial values required by the example by using the File > Open menu item. If this is done, continue at step 2 in the example entitled ‘Export geometric properties from PontiEC4 LUSAS’.

- **Composite_Bridge_Deck_Initial.csv** – Initial material and geometric data / definitions for use with PontiEC4.

Other associated files are stated at various points throughout the example.

1. Define material and geometric properties in PontiEC4

This section covers the definition of material and geometric properties for the assessment in the PontiEC4 Composite Deck Designer. At the end of this section these properties are exported to LUSAS as a VBS file to define the beam cross section properties.

Running the Composite Deck Designer (PontiEC4)

- Run the Composite Deck Designer (**PontiEC4**) and select the **File> New** menu item to create a new PontiEC4 project.

- Select **File> Save as** to save the PontiEC4 project data that will be entered to a different folder from the PontiEC4 installation folder with the name **Composite_Bridge_Deck.csv**

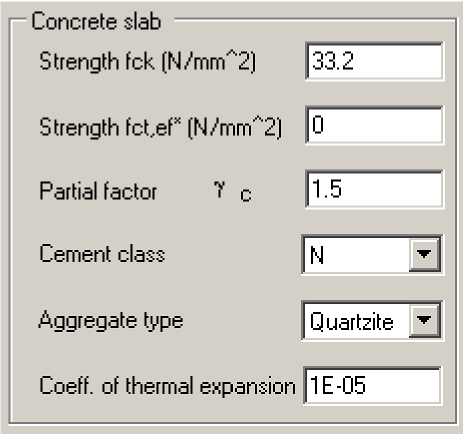
Note. All data entered in the PontEC4 dialogs will be saved to this file. Previously created and saved data, or supplied data (held in .csv files) can be loaded into PontiEC4 dialogs by using the File> Open menu item.

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.

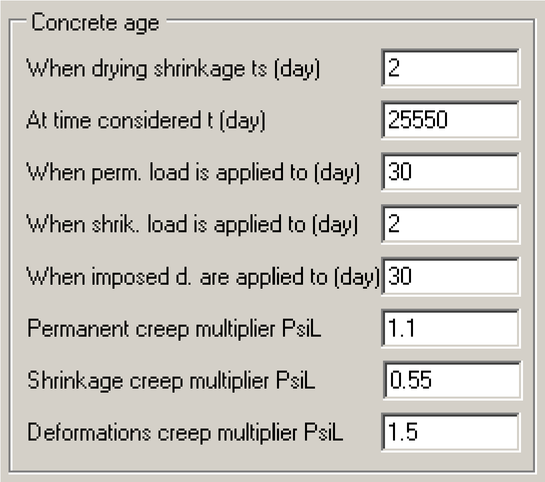
- For each panel on the dialog enter the material data shown in the following images.

Concrete slab


Concrete slab

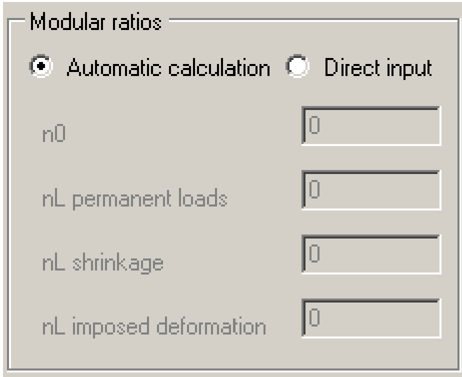
Strength fck (N/mm^2)	33.2
Strength fct,ef* (N/mm^2)	0
Partial factor γ c	1.5
Cement class	N
Aggregate type	Quartzite
Coeff. of thermal expansion	1E-05

Concrete age

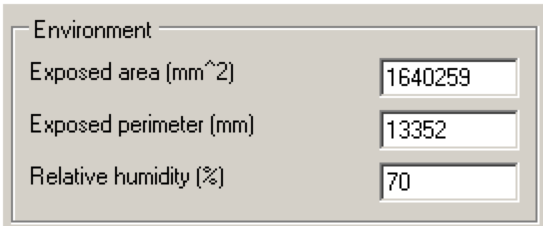

Concrete age

When drying shrinkage ts (day)	2
At time considered t (day)	25550
When perm. load is applied to (day)	30
When shrik. load is applied to (day)	2
When imposed d. are applied to (day)	30
Permanent creep multiplier PsiL	1.1
Shrinkage creep multiplier PsiL	0.55
Deformations creep multiplier PsiL	1.5

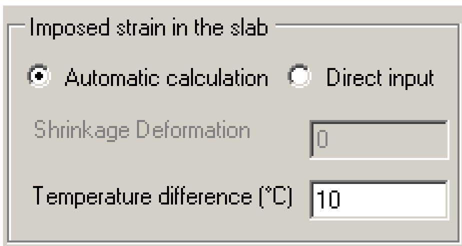
Modular ratios



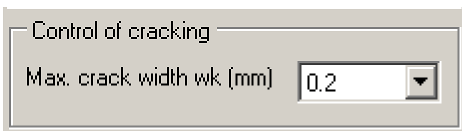
Environment



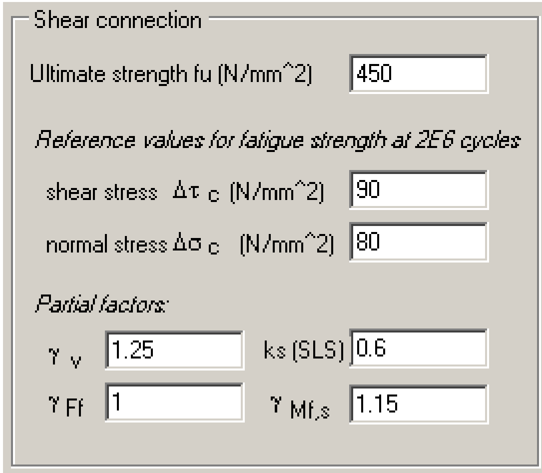
Imposed strain in the slab



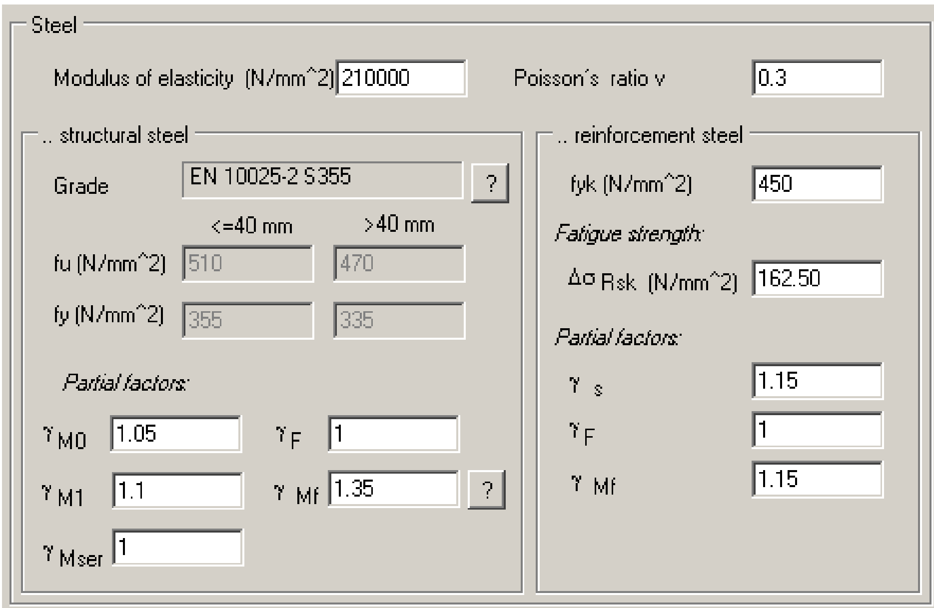
Control of cracking



Shear connection



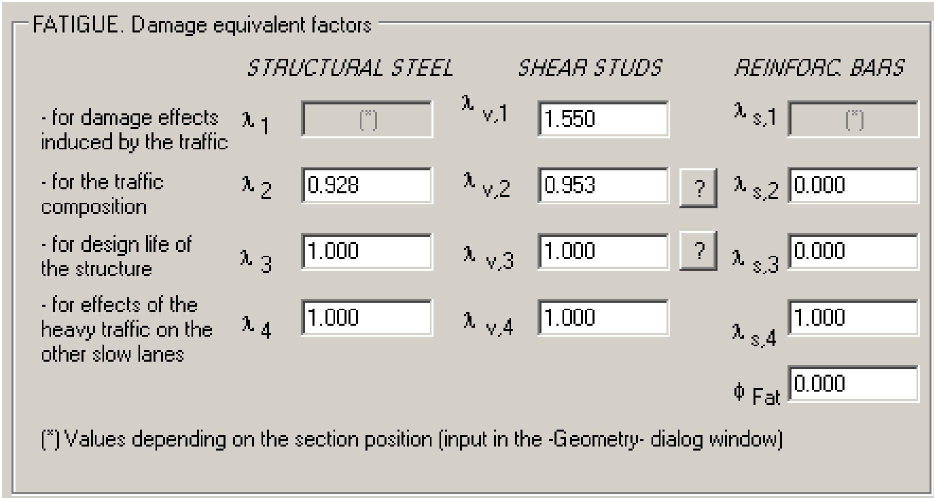
Steel



- Click on the Question mark button 

next to the Grade field, to define the Steel grade shown.

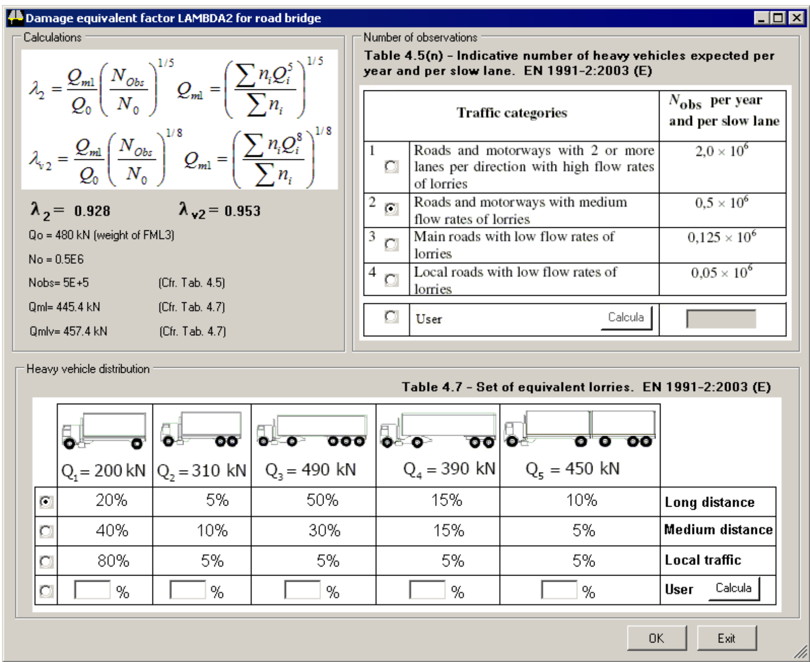
Fatigue


FATIGUE. Damage equivalent factors

		STRUCTURAL STEEL		SHEAR STUDS			REINFORC. BARS
- for damage effects induced by the traffic	λ 1	(*)	λ v,1	1.550		λ s,1	(*)
- for the traffic composition	λ 2	0.928	λ v,2	0.953	?	λ s,2	0.000
- for design life of the structure	λ 3	1.000	λ v,3	1.000	?	λ s,3	0.000
- for effects of the heavy traffic on the other slow lanes	λ 4	1.000	λ v,4	1.000		λ s,4	1.000
						Φ Fat	0.000

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

- λ_2 can be calculated with data input in the window dialog that appears when you click on the Question mark button ? next to the input field. In this case, input the data below:


Damage equivalent factor LAMBDA2 for road bridge

Calculations

$$\lambda_2 = \frac{Q_{m1}}{Q_0}\left(\frac{N_{Obs}}{N_0}\right)^{1/5} \quad Q_{m1} = \left(\frac{\sum n_i Q_i^5}{\sum n_i}\right)^{1/5}$$

$$\lambda_{v2} = \frac{Q_{m1}}{Q_0}\left(\frac{N_{Obs}}{N_0}\right)^{1/8} \quad Q_{m1} = \left(\frac{\sum n_i Q_i^8}{\sum n_i}\right)^{1/8}$$

λ 2 = 0.928 λ v2 = 0.953

Qo = 480 kN (weight of FML3)

No = 0.5E6

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

Qml= 445.4 kN (Cfr. Tab. 4.7)

Qmlv= 457.4 kN (Cfr. Tab. 4.7)

Number of observations

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

	Traffic categories	N_{obs} per year and per slow lane
1	Roads and motorways with 2 or more lanes per direction with high flow rates of lorries	$2{,}0 \times 10^6$
2 (selected)	Roads and motorways with medium flow rates of lorries	$0{,}5 \times 10^6$
3	Main roads with low flow rates of lorries	$0{,}125 \times 10^6$
4	Local roads with low flow rates of lorries	$0{,}05 \times 10^6$
	User Calcula	

Heavy vehicle distribution

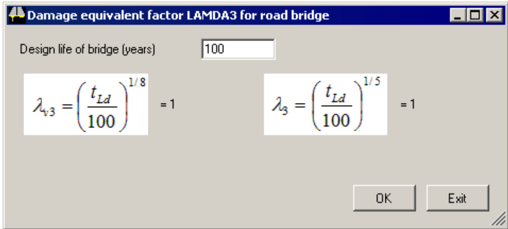
Table 4.7 - Set of equivalent lorries. EN 1991-2:2003 (E)

	Q_1 = 200 kN	Q_2 = 310 kN	Q_3 = 490 kN	Q_4 = 390 kN	Q_5 = 450 kN	
(selected)	20%	5%	50%	15%	10%	Long distance
	40%	10%	30%	15%	5%	Medium distance
	80%	5%	5%	5%	5%	Local traffic
	%	%	%	%	%	User Calcula

OK Exit

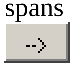
- λ_3 can be calculated with data input in the window dialog that appears when you click on the Question mark button 

next to the input field. In this case, input the data below:



Defining shear lag slab and flanges

- In order 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 in the text box in the top right corner: 50.504,50.504 and click on the 

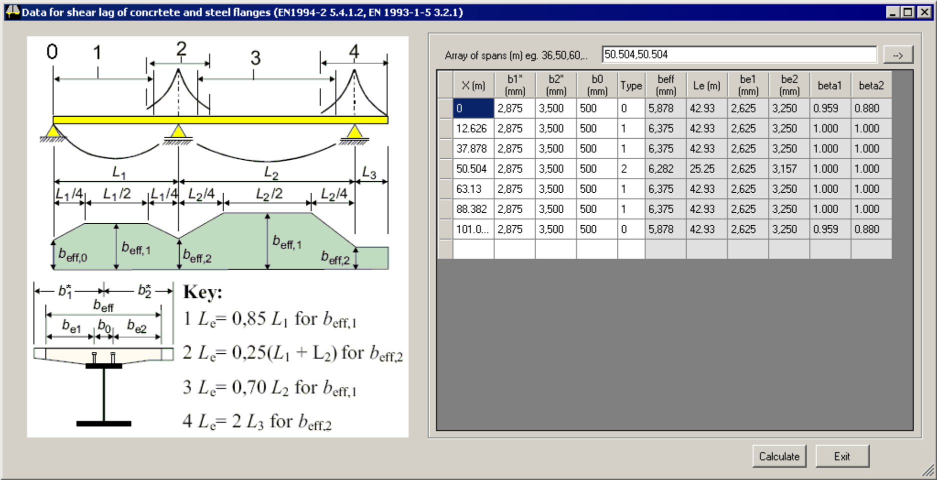
button. The X distance where the effective slab width changes will be automatically set up in the table

- 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 input the code **0** for the first and last section, **1** for the sagging regions, and **2** for the middle support.

- Click the **Calculate** button to calculate $\mathbf{b_{eff}}$**,**



- 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.

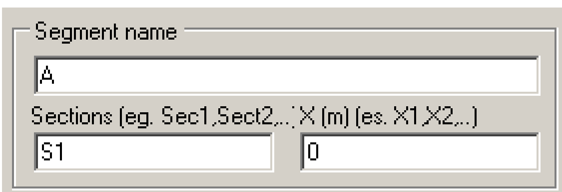
- From the main menu select **Windows> 2 Geometry** to display the Geometry dialog.

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.

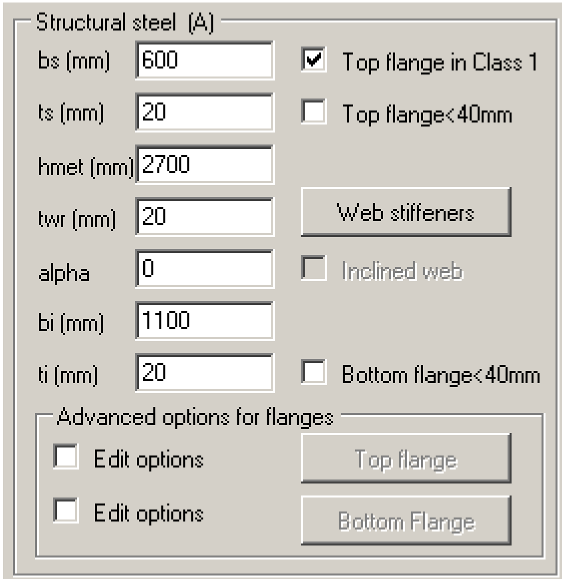
Segment name, sections, x-distances

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



Structural steel plate details

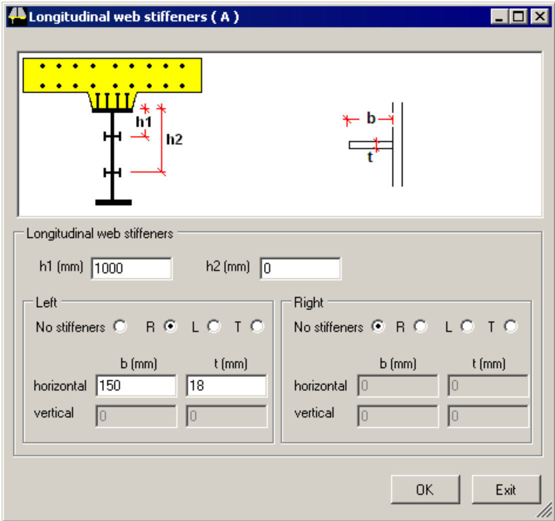
- With reference to the main table of girder dimensions, enter the following values and make sure also that **Top flange in Class 1** is checked (ticked).



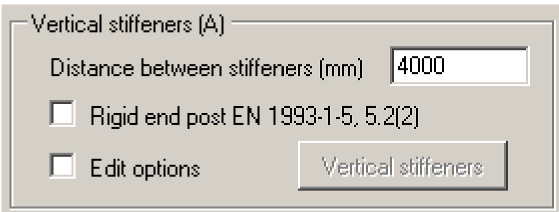
Note. The checkboxes for Top / Bottom flange <40mm should be only checked (ticked) for flanges that have a total thickness above 40mm, when they are made up from a number of plates each of a thickness less than 40mm. When done, ultimate and yielding tension for steel below 40mm thickness will be used.

Longitudinal web stiffeners details

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

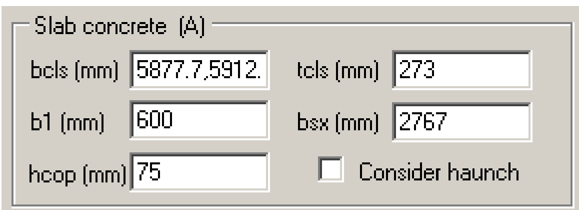


Vertical web stiffener distance



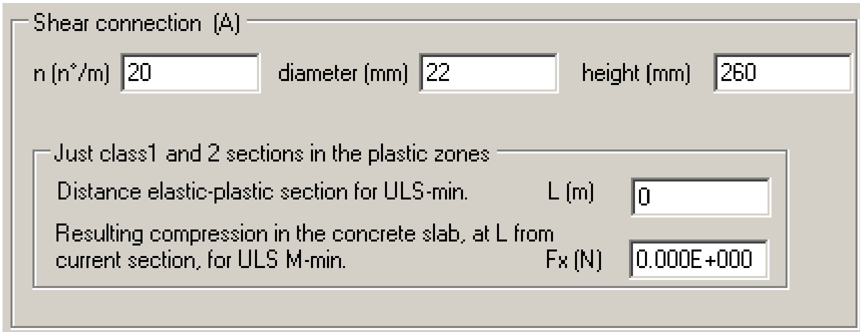
Slab concrete

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

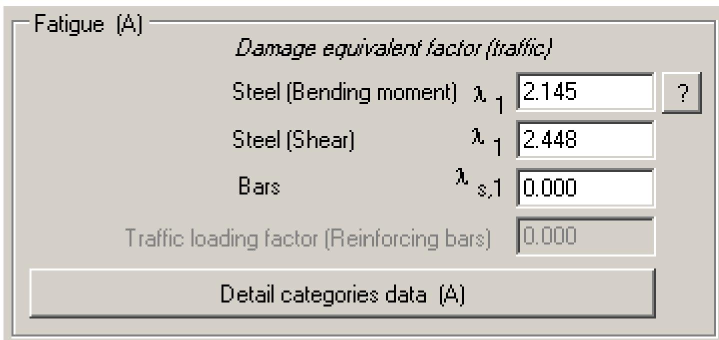


Reinforcing bars

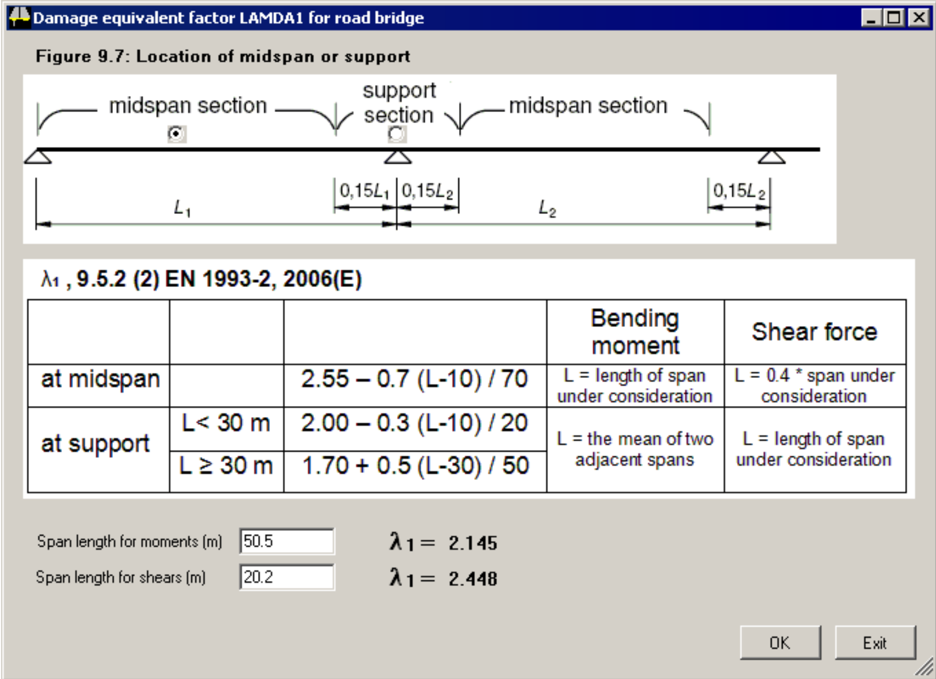
Shear connection



Fatigue data



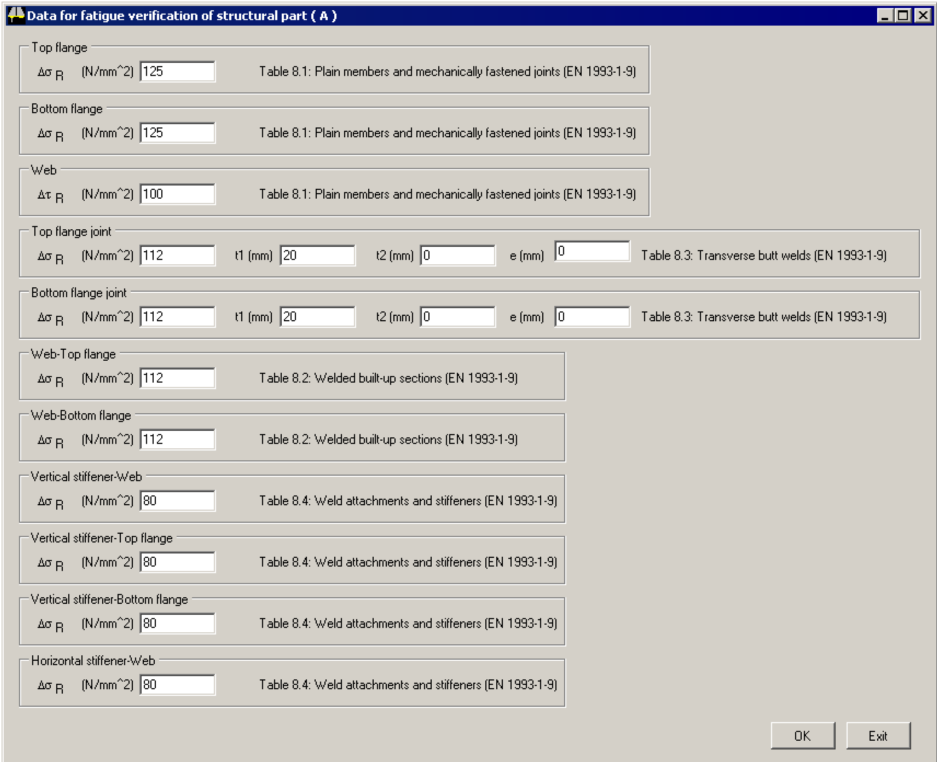
- λ_1 can be calculated automatically by clicking the Question mark button  next to the input field and then, on the Damage equivalent factor dialog, select the radio button for the **midspan section** and enter **50.5** for the Span length for moments and **20.2** for the Span length for shears.



			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		

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.**



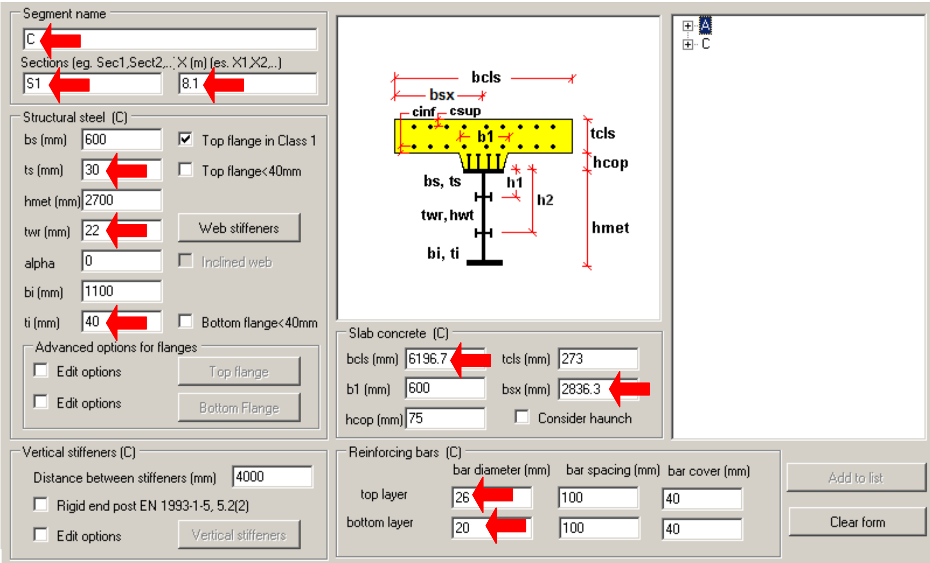
Adding data for segment A to the segment treeview

- 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

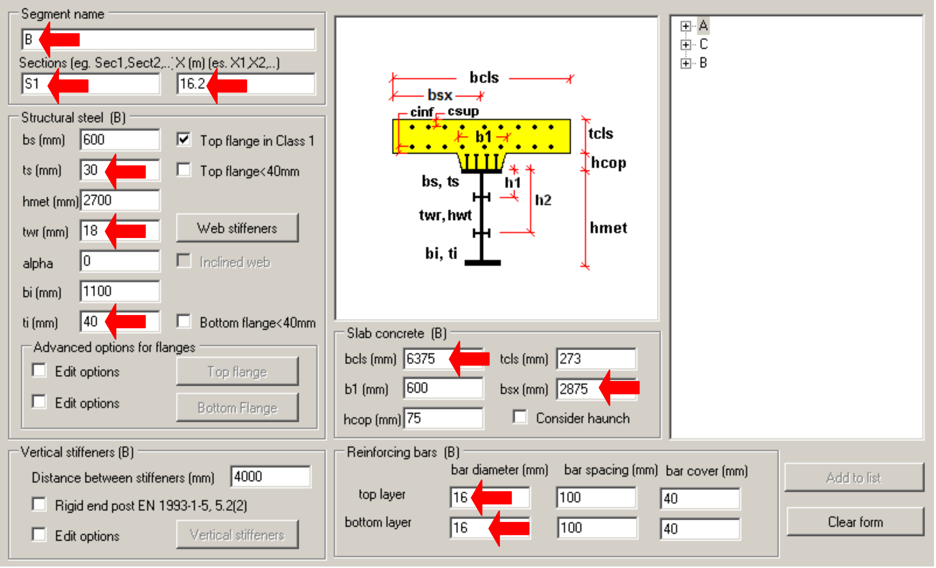
- In the Segment name field delete **A** and type **C**. Then, with reference to the following image, change or ensure that the dialog fields marked have entries as shown. Remember that the values of bcls and bsx can be calculated automatically by selecting **Calculate beff** from the context menu for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



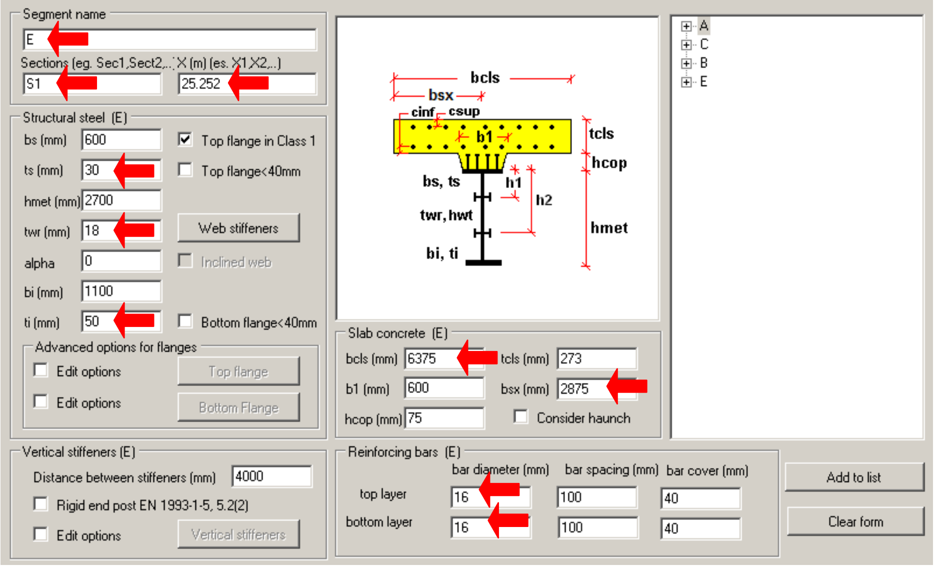
- When complete, click the **Add to list** button to save the data entered.

Defining Segment B

- Ensure that the dialog fields marked have the entries shown. When complete click the **Add to list** button to save the segment data entered. Then define Segment E in the same way, again clicking the **Add to list** button when complete.

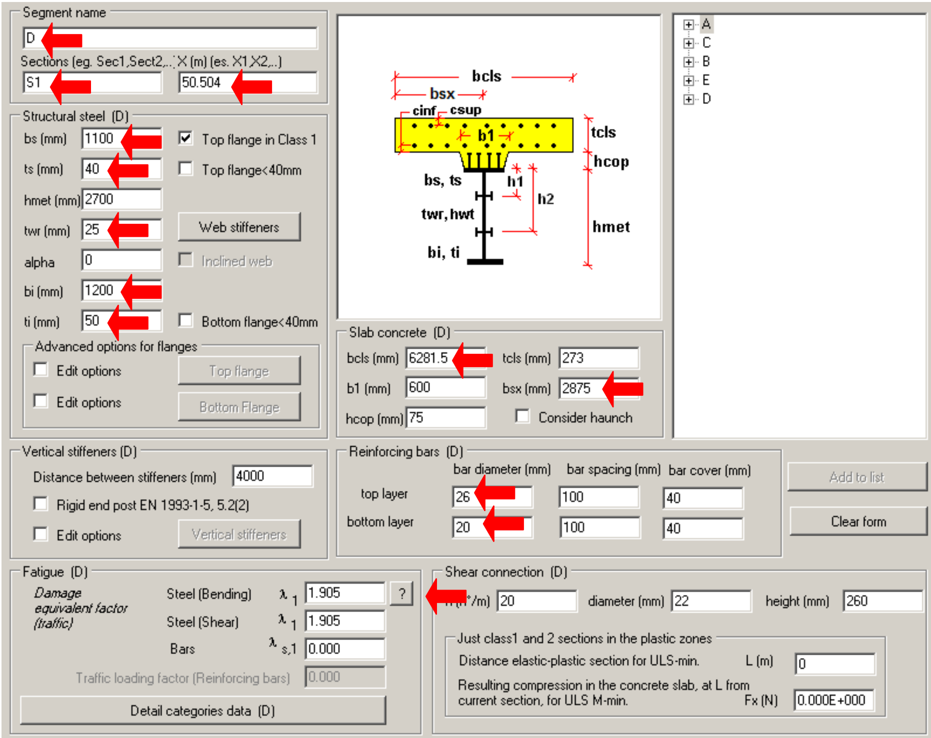


Segment E

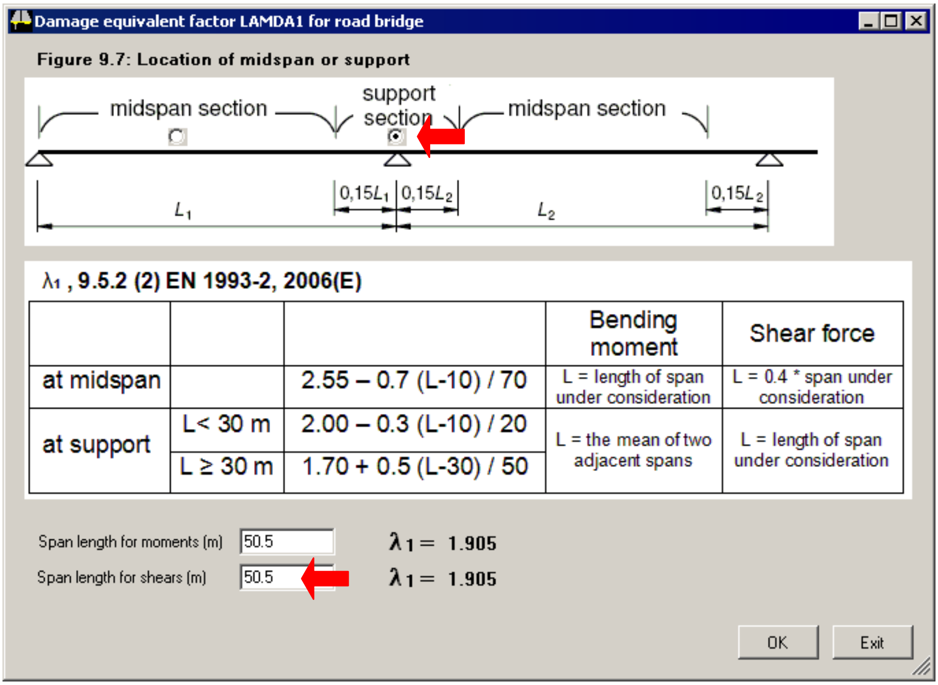


Segment D

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



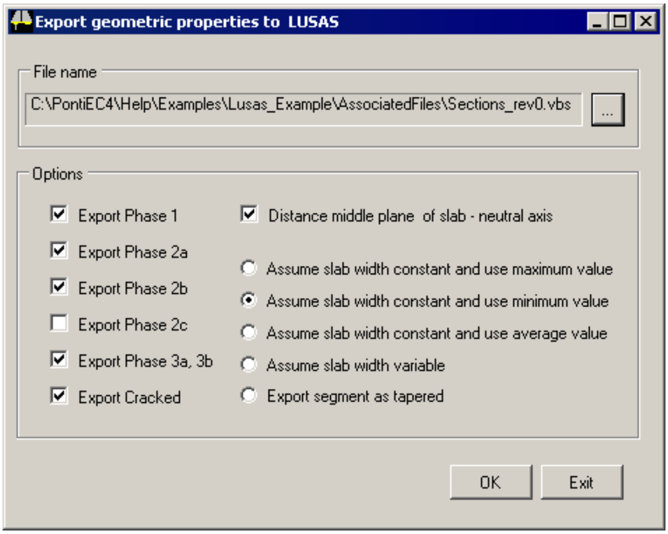
- Click on the Question mark button  next to the Steel (Bending moment) field and specify a **support section** and a Span length for shears of **50.5** as shown in the following image.



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

2. Export geometric properties from PontiEC4 to LUSAS

- On the main menu select **Window> 4 Results** to set the **Results** dialog active. PontiEC4 will check all the data that has been input and run part of the EC4 code check to calculate the required geometric properties of each cross section.
- Select **File> Export geometric properties to LUSAS**
- Browse to a suitable destination directory and enter a filename of **"Sections_Rev0.vbs".**
- Select (check) all the option boxes (apart from Export Phase 2c), and select the option **Assume slab width constant and use minimum value**. Click **OK** to finish.



The file created can be imported into LUSAS to create 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 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 LUSAS model. 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.

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. 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. The edge longitudinal lines have a dummy cross section and are included in the model to enable processing of the discrete loads.

Geometric and material properties are imported from PontiEC4 for each of the design phases considered. The multiple analysis facility is used within LUSAS, using four analyses to represent the design phases as follows:

1. Phase 1 - **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.

Note: 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.

Loading and load cases

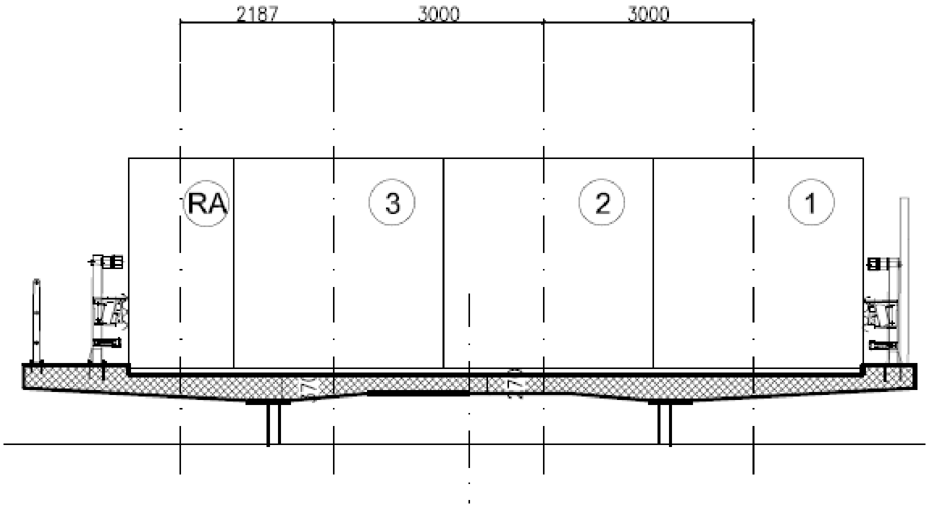
The following table summarizes the loadings and load cases used for each analysis.

Analysis	Loadings	Loadcases
Phase1	PHASE 1 Longitudinal and transv. beams Weight (m/s^2) PHASE 1 Stud Weight (N/m) __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
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^2)	From ID=2 Pos=1 to ID=36 Pos=35
	Udl23 (Eurocode Lane Load 3mx3m Load=2500N/m^2)	From ID=37 Pos=1 to ID=71 Pos=35
	Udl23 (Eurocode Lane Load 3mx3m Load=2500N/m^2)	From ID=72 Pos=1 to ID=106 Pos=35
	UdlRA (Eurocode Lane Load 3mx1.5m Load=2500N/m^2)	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

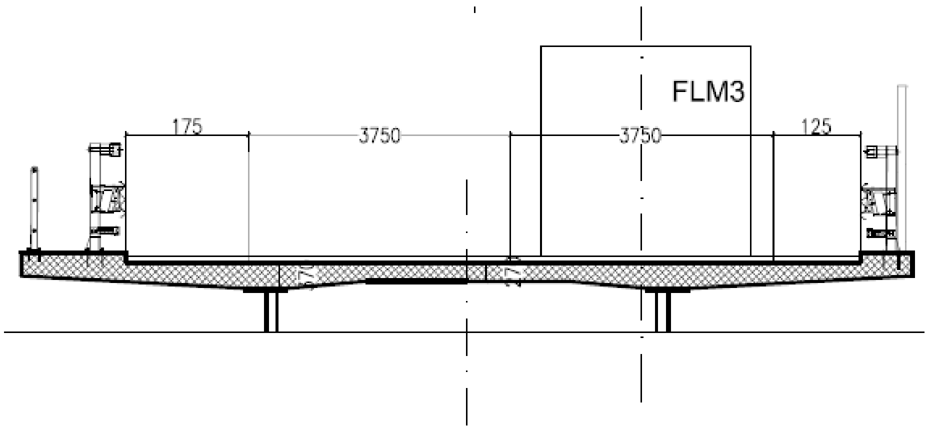
Traffic loading

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)

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_Charactersitic_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 File	Beneficial Factor	Adverse Factor	Title	Type
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)

Associated Files

These files are supplied in case of difficulties in manually creating the appropriate data.

- **Sections_Rev0.vbs** - A file created by PontiEC4 containing section data to import into LUSAS.
- **Composite_Bridge_Deck.mdl** – The 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.

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

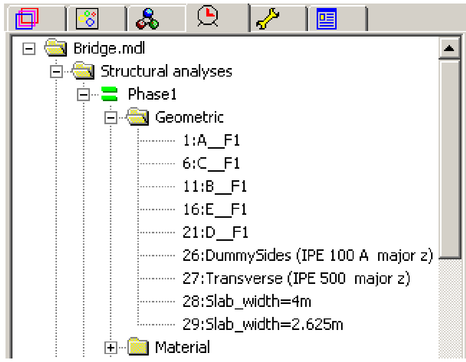
Open the associated LUSAS model

- 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 **\<LUSAS Installation Folder>\Examples\Modeller** directory.

Importing the section properties from Ponti EC4

- Select **File> Script> Run Script** and select the PontiEC4 script file "**Sections_Rev0.vbs**" that was created earlier. This imports the geometric section and material data from PontiEC4.

Note. The attributes have already been imported and assigned in the model and re-importing them simply overwrites the existing definitions.



- 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.

4. Export the force and moment data from LUSAS

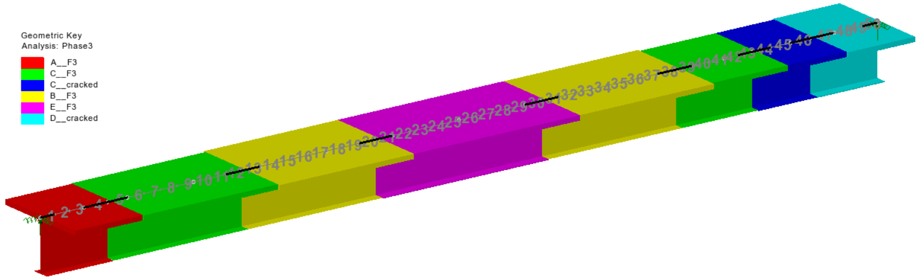
Selection of elements for checking

To enable design checks to be carried out the start and end elements of each main girder segment need to be selected. For this example, various deck members as well as the elements at the start and end of each main girder segment have been pre-selected and stored as separate Groups.

The model has been saved with group name **"Half_External_Beam"** (as seen in the Groups treeview) **Set as Only Visible**.

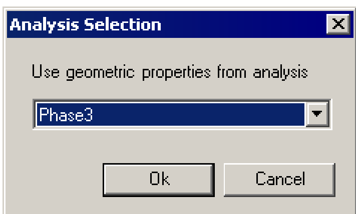
- In the Groups treeview right-click on the group name **"Selected_Elements"** and select the **Select Members** menu item. These elements selected are those that occur at the changes of geometric section, and they represent the sections at which design checks will be undertaken in PontiEC4. Note that re-meshing of the model may renumber elements and prevent the automatic selection of required elements by use of this Group name. In this was to occur a manual selection of these elements would have be done.

If fleshing of assigned geometric properties was turned on prior to the above, the elements selected would appear like this showing that they occur at the start and end of each main girder segment:

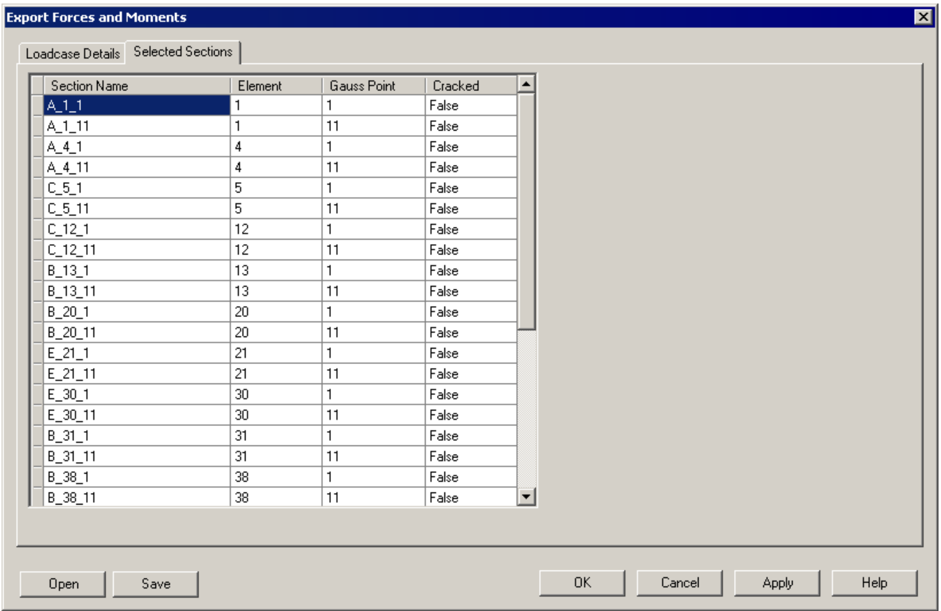


Note. The required element selections shown above can otherwise be achieved on a fleshed model by selecting the Geometric Properties dialog option **Colour by> assignment** and then selecting the **Geometry** menu item to show the geometric sections and using the Select Elements toolbar button to only select the elements required.

- To export the information for these sections, select the **Bridge> Composite Deck Designer> Export…**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.



Section Name	Element	Gauss Point	Cracked
A_1_1	1	1	False
A_1_11	1	11	False
A_4_1	4	1	False
A_4_11	4	11	False
C_5_1	5	1	False
C_5_11	5	11	False
C_12_1	12	1	False
C_12_11	12	11	False
B_13_1	13	1	False
B_13_11	13	11	False
B_20_1	20	1	False
B_20_11	20	11	False
E_21_1	21	1	False
E_21_11	21	11	False
E_30_1	30	1	False
E_30_11	30	11	False
B_31_1	31	1	False
B_31_11	31	11	False
B_38_1	38	1	False
B_38_11	38	11	False

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]

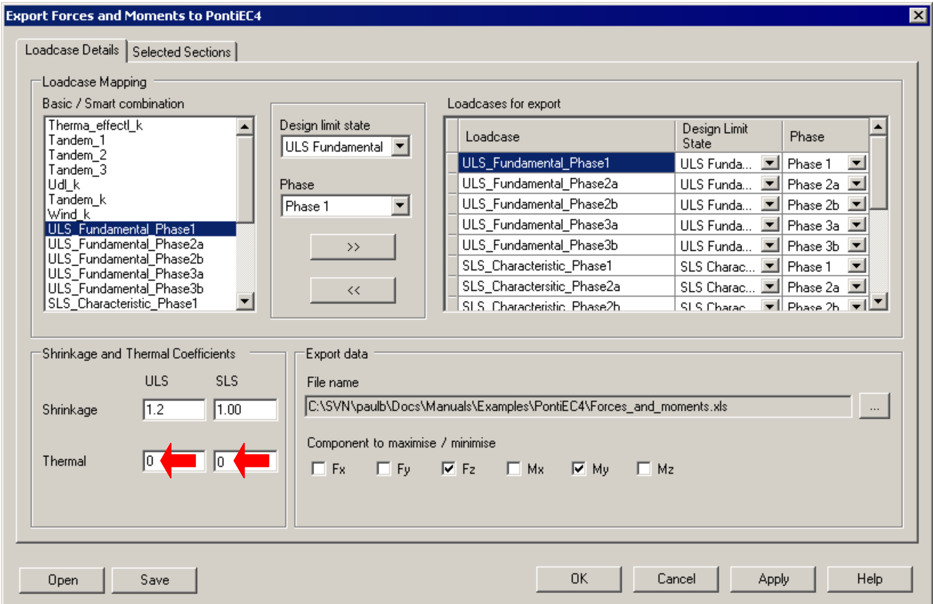
Important. The Export Forces and Moments facility obtains the segment and the phase name from the name of the assigned geometric attribute, by splitting the attribute name on the double underscore separator “__”. For this reason the cross-section geometric properties in the LUSAS model must have a name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

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_Charactersitic_Phase2a	SLS Characteristic	Phase 2a
SLS_Characteristic_Phase2b	SLS Characteristic	Phase 2b
SLS_Characteristic_Phase3a	SLS Characteristic	Phase 3a
SLS_Characteristic_Phase3b	SLS Characteristic	Phase 3b

Loadcase	Design limit state	Phase
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 (γ∗ψ=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.

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 (γ∗ψ=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.

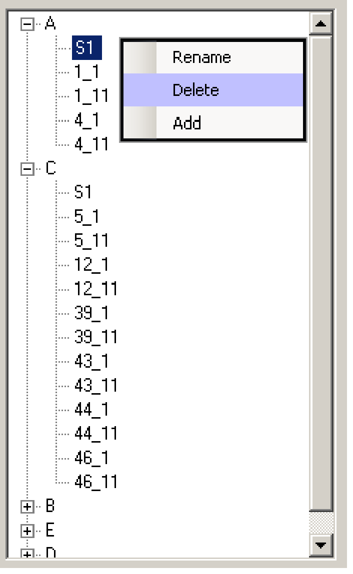
- In the Component to maximize/minimize panel ensure **Fz** and **My** are checked. (These are the components that will be considered in the smart combinations)
- Click the ellipsis button [...] to choose the folder and name of the xls file to create (e.g.: <current work directory>**Forces_and_moments.xls**), 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 for use in PontiEC4.

5. Import 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, 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 supplied file **“Composite_Bridge_Deck.csv”** loaded if no data is currently present), select the **File> Import forces/moments** menu item and load the file **Forces_and_moments.xls** exported from LUSAS.

PontiEC4 will detect that additional sections are present and 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 segments, a re-import of the forces_and_moments.xls file will re-insert any deleted segments.

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

This concludes the data input.

Associated Files

These are supplied in case of problems in manually creating the appropriate data.

- **Export_Forces_to_PontiEC4.inp** a file that can be used to re-populate the pages of the LUSAS Export Forces and Moments dialog.
- **Forces_and_moments.xls** File created by the LUSAS Export Forces and Moments dialog containing data to import into PontiEC4.

6. Assess 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 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, where code and theory details are also supplied.

Viewing results in PontiEC4

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

A check of the utilisation factors obtained should be made for each limit state. Only the Fundamental ULS combination will be demonstrated here.

Note. By clicking on a particular header the values in a column can be sorted in increasing or decreasing value. Any utilisation factor that is greater than 1.0 will be displayed in red.

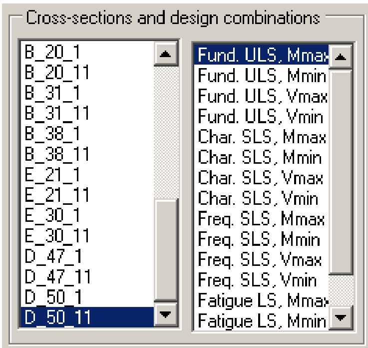
- Sort the **SigEd/fy** column. It can be seen that Segments D and C fail the check for normal stresses at ULS, as shown below.

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/(n*PRd)
D_50_11	50.494	Fund. ULS, Mmax	4	4	(.84)	1.186	0.705	1.33	0.705	1.243	0.507
C_46_11	46.441	Fund. ULS, Mmax	4	4	(.73)	1.179	0.645	1.23	0.645	1.208	0.339
D_50_11	50.494	Fund. ULS, Vmax	4	4	(.81)	1.151	0.749	1.29	0.749	1.234	0.563
D_50_1	49.638	Fund. ULS, Mmax	4	4	(.78)	1.107	0.613	1.24	0.613	1.124	0.394
C_46_1	45.265	Fund. ULS, Mmax	4	4	(.66)	1.056	0.603	1.1	0.603	1.071	0.298
D_50_1	49.638	Fund. ULS, Vmax	4	4	(.74)	1.056	0.748	1.19	0.748	1.138	0.563
C_46_11	46.441	Fund. ULS, Vmax	4	4	(.63)	1.02	0.86	1.07	0.86	1.202	0.569
D_47_11	47.862	Fund. ULS, Mmax	4	4	(.68)	0.957	0.54	1.07	0.54	0.959	0.335
C_44_11	44.078	Fund. ULS, Mmax	4	4	(.59)	0.947	0.536	0.99	0.537	No int.	0.254
B_20_11	20.239	Fund. ULS, Mmin	4	1	0.73	(.882)	0.1	1.03	0.1	No int.	0.088

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> Results** menu item in order to find out more about the failed checks.
- In the Cross-sections and design combinations panel select section **D_50_11** and **Fund. ULS, Mmax**



- Then, in the lower part of the Results dialog, select the **Stresses** tab.

Plastic check | Stresses | Shear | Geometric properties 0 | Geometric properties 1 | Geometric properties 2 | Domains Mpl-N | Studs. ULS, SLS | SLS. Web Breathing | Fatigue LS 1 | Fatigue LS 2

○ 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
σ8	0.0	1.0	0.0	1.1	0.0	0.0	0.0	0.0	1.7	0.0	4.3	0.0	0.0	0.00	σ8
σ7	0.0	17.0	31.1	16.4	31.6	0.0	0.0	62.7	10.0	32.9	25.0	82.8	178.3	0.46	σ7
σ6	0.0	13.5	26.6	13.0	27.1	0.0	0.0	53.7	7.0	28.2	17.4	70.7	152.6	0.39	σ6
σ5	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	0.00	σ5
σ4	180.5	11.5	23.9	11.0	24.4	0.0	0.0	228.8	5.2	25.3	12.9	63.5	317.7	0.94	σ4
σ3	175.5	10.8	23.0	10.3	23.5	0.0	0.0	222.0	4.6	24.4	11.3	61.0	307.4	0.91	σ3
σ2	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	σ2
σ1	-14...	-35.2	-37.5	-35.4	-37.9	0.0	0.0	-224.3	-35.7	-39.5	-91.7	-101.9	-365.6	1.15	σ1
σ0	-15...	-36.1	-38.7	-36.3	-39.1	0.0	0.0	-232.8	-36.5	-40.7	-93.7	-105.0	-378.5	1.19	σ0

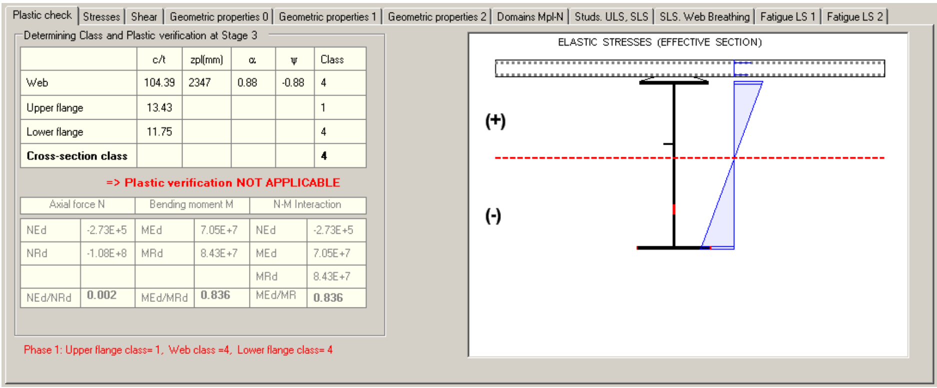
Slab stresses at Phase 2 (N/mm^2):
Total top stress = 2.03
Total bottom stress = 1.47
=> Section at the end of Phase 2: CRACKED (m.)

Slab stresses at Phase 3 (N/mm^2):
Total top stress = 8
Total bottom stress = 5.03
=> Section at the end of Phase 3: CRACKED (m.)

=> El. check Phase 3 FAILED
eta1= 1.186

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.



7. Optimize the structure

Associated Files

These are supplied in case of problems in manually creating the appropriate data.

- ❑ **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.

- In PontiEC4, select the **Window> Geometry** menu item, and then select Segment **D** in the segment treeview. The status bar should state : “Edit GEOMETRY of Segment D”
- Increase the top flange thickness (ts) from 40 to **50**

- Increase the bottom flange thickness (ti) from 50 to **80**
- Select the **Window> Results** menu item and view the updated results for Section **D_50_11**. The flange thickness changes for Segment D now allow it to pass the design check

Plastic check | Stresses | Shear | Geometric properties 0 | Geometric properties 1 | Geometric properties 2 | Domains Mpl-N | Studs. ULS, SLS | SLS. Web Breathing | Fatigue LS 1 | Fatigue LS 2

○ 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
σ8	**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**	σ8
σ7	**0.0**	15.4	27.4	14.9	27.9	0.0	0.0	**55.2**	9.1	29.0	22.9	73.0	**157.2**	**0.40**	σ7
σ6	**0.0**	12.8	23.9	12.4	24.4	0.0	0.0	**48.3**	7.0	25.3	17.3	63.7	**137.3**	**0.35**	σ6
σ5	**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**	σ5
σ4	**149.6**	11.3	21.9	10.8	22.3	0.0	0.0	**193.7**	5.7	23.2	14.0	58.1	**275.0**	**0.86**	σ4
σ3	**144.9**	10.6	21.0	10.2	21.4	0.0	0.0	**187.2**	5.1	22.2	12.5	55.7	**265.2**	**0.83**	σ3
σ2	**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**	σ2
σ1	**-96.4**	-23.6	-25.0	-23.8	-25.2	0.0	0.0	**-146.6**	-24.0	-26.2	-61.8	-67.9	**-240.7**	**0.75**	σ1
σ0	**-10...**	-24.7	-26.4	-24.8	-26.6	0.0	0.0	**-156.9**	-24.9	-27.8	-64.1	-71.8	**-256.5**	**0.80**	σ0

Slab stresses at Phase 2 (N/mm^2):
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^2):
Total top stress = 7.22
Total bottom stress = 5.04
=> Section at the end of Phase 3: CRACKED (m.)

=> El. check Phase 3 PASSED
eta1= 0.862

- Select the **Window> Geometry** menu item, and then 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 (tw) from 22 to **25**
- Increase the bottom flange thickness (ti) from 40 to **60**
- Select the **Window> Results** menu item and view the updated results for Section **C_46_11**. The flange thickness changes for Segment C now allow it to pass the design check.

Plastic check | Stresses | Shear | Geometric properties 0 | Geometric properties 1 | Geometric properties 2 | Domains Mpl-N | Studs. ULS, SLS | SLS. Web Breathing | Fatigue LS 1 | Fatigue LS 2

○ 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
σ8	**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**	σ8
σ7	**0.0**	10.6	20.6	15.5	31.8	0.0	0.0	**52.4**	9.2	33.0	17.2	62.3	**147.7**	**0.38**	σ7
σ6	**0.0**	8.6	17.9	12.6	27.7	0.0	0.0	**45.6**	6.6	28.7	12.3	54.1	**128.3**	**0.33**	σ6
σ5	**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**	σ5
σ4	**148.7**	7.5	16.3	10.8	25.2	0.0	0.0	**190.2**	5.1	26.1	9.4	49.2	**265.5**	**0.83**	σ4
σ3	**144.2**	7.0	15.6	10.1	24.1	0.0	0.0	**183.9**	4.5	25.0	8.1	47.0	**256.0**	**0.80**	σ3
σ2	**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**	σ2
σ1	**-89.9**	-19.2	-20.6	-29.2	-31.4	0.0	0.0	**-141.8**	-29.4	-32.7	-57.0	-63.9	**-238.5**	**0.75**	σ1
σ0	**-95.3**	-19.8	-21.4	-30.1	-32.7	0.0	0.0	**-149.4**	-30.2	-34.1	-58.5	-66.5	**-249.9**	**0.78**	σ0

Slab stresses at Phase 2 (N/mm^2):
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^2):
Total top stress = 6.07
Total bottom stress = 3.99
=> Section at the end of Phase 3: CRACKED (m.)

=> El. check Phase 3 PASSED
eta1= 0.832

- Select the **Window> 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 The stresses for other combinations are all satisfactory.

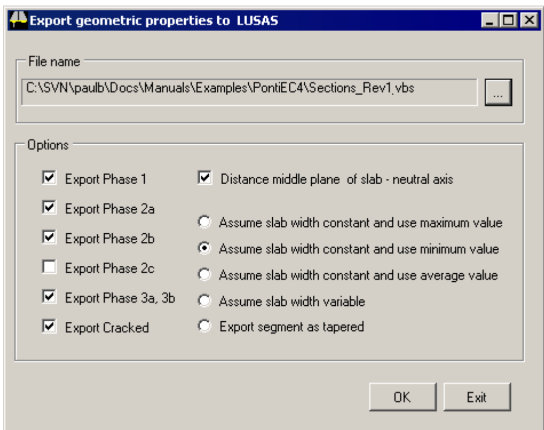
Fundamental ULS combination

Section	X (m)	Combination	Class Ph.1	Class Ph.3b	MEd/MR	SigEd /fy	VEd/VRd	MEd/Mf,Rd	VEd/Vbw,Rd	V/M/N	vEd/(n*PRd)
B_20_11	20.239	Fund. ULS, Mmin	4	1	0.73	(.882)	0.1	1.03	0.1	No int.	0.088
B_20_1	19.262	Fund. ULS, Mmin	4	1	0.72	(.874)	0.102	1.02	0.102	No int.	0.088
D_50_11	50.494	Fund. ULS, Mmax	1	3	(.65)	0.862	0.708	0.82	0.708	0.898	0.476
D_50_11	50.494	Fund. ULS, Vmax	1	3	(.63)	0.843	0.753	0.79	0.753	0.896	0.528
C_46_11	46.441	Fund. ULS, Mmax	3	3	(.59)	0.832	0.508	0.83	0.508	0.832	0.317
D_50_1	49.638	Fund. ULS, Mmax	1	3	(.61)	0.801	0.616	0.77	0.616	0.812	0.369
E_21_11	21.241	Fund. ULS, Mmin	4	1	0.67	(.798)	0.048	0.9	0.049	No int.	0.063
E_21_1	20.261	Fund. ULS, Mmin	4	1	0.67	(.795)	0.05	0.9	0.052	No int.	0.063
B_13_11	13.240	Fund. ULS, Mmin	4	1	0.66	(.786)	0.428	0.92	0.434	No int.	0.239
D_50_1	49.638	Fund. ULS, Vmax	1	3	(.58)	0.773	0.749	0.73	0.751	No int.	0.528

- 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. Browse for the destination directory and input a filename of **Sections_Rev1.vbs**. Then select (check) all the option boxes (apart from Export Phase 2c), and select the option **Assume slab width constant and use minimum value.** Click **OK** to finish.



Calculate revised forces and moments in LUSAS

- In LUSAS, and with the **Composite_Bridge_Deck.mdl** file open, select **File> Script > Run Script** and 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 as **Composite_Bridge_Deck_Optimized.mdl** and click **Yes** to close all open results files.

- 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 repeat the operations from Step 4 in this example. In summary, this involves:

- In LUSAS, select those elements of interest at the start and end of each main girder segment, and export the results for those elements by selecting the **Bridge> Composite Deck Designer> Export…** menu item (they may still be selected).

- 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.

- 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.

Assess the revised results in PontiEC4

- Select the **Window> 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.

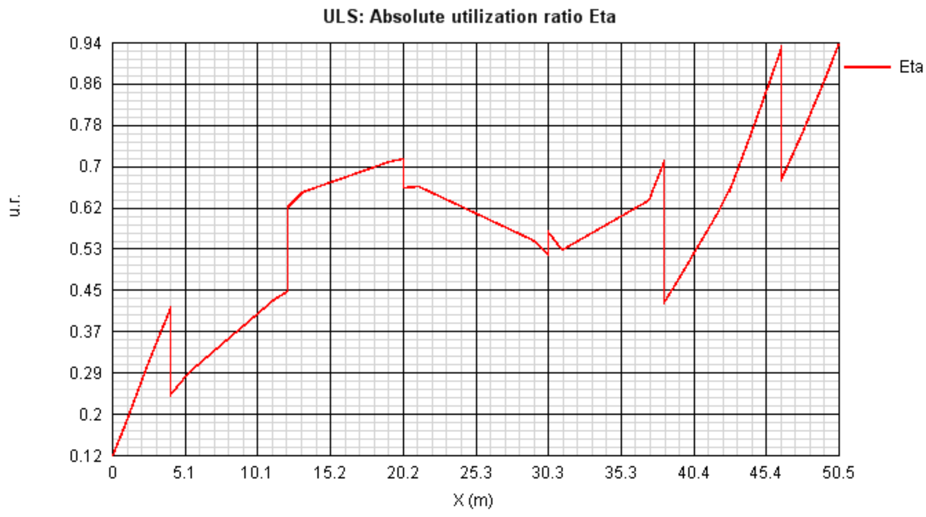
Fundamental ULS combination

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)	Longitudinal Stiffeners LTB
D_50_11	50.494	Fund. ULS, Mmax	1	3	(.71)	0.945	0.73	0.9	0.73	0.988	0.487	0.624
C_46_11	46.441	Fund. ULS, Mmax	3	3	(.67)	0.933	0.527	0.93	0.527	0.934	0.329	0.624
D_50_11	50.494	Fund. ULS, Vmax	1	3	(.69)	0.923	0.774	0.87	0.774	0.984	0.538	0.624
D_50_1	49.638	Fund. ULS, Mmax	1	3	(.67)	0.881	0.638	0.84	0.638	0.897	0.38	0.624
B_20_11	20.239	Fund. ULS, Mmin	4	1	0.71	(.855)	0.083	1	0.083	No int.	0.084	0.624
C_46_11	46.441	Fund. ULS, Vmax	3	3	(.58)	0.852	0.696	0.8	0.696	0.895	0.544	0.624
D_50_1	49.638	Fund. ULS, Vmax	1	3	(.64)	0.851	0.771	0.8	0.771	0.911	0.539	0.624
B_20_1	19.262	Fund. ULS, Mmin	4	1	0.71	(.849)	0.085	0.99	0.085	No int.	0.084	0.624
C_46_1	45.265	Fund. ULS, Mmax	3	3	(.6)	0.831	0.509	0.84	0.509	0.831	0.308	0.624
F_21_11	21.241	Fund. ULS, Mmin	4	1	0.66	(.772)	0.031	0.88	0.032	No int.	0.059	0.624

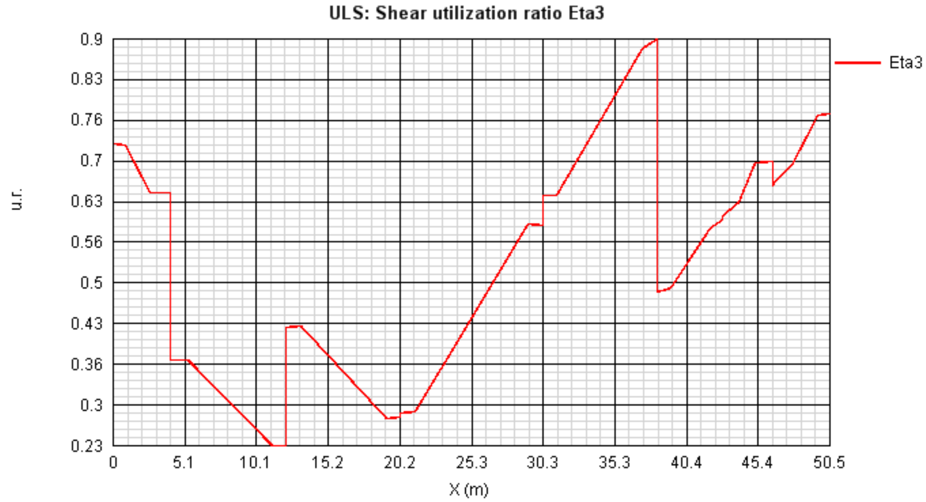
Methods for graphing data and producing detailed results will now be explained.

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:

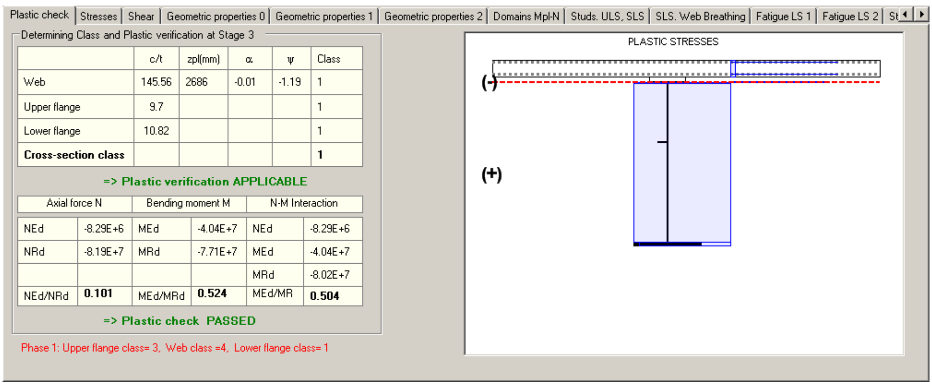


- From the first drop-down list choose **ULS: Shear utilization ratio Eta3** to plot the following graph:



Detailed results

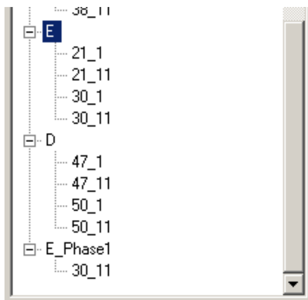
- Select the **Window> Results** menu item.
- In the Cross-sections and design combinations panel, select section **E_30_11** and **Fund. ULS, Mmin**.
- Select the **Stresses** tab to have a look at the elastic stresses on the gross section.
- Then, select the **Plastic check** tab. From this, all details 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.



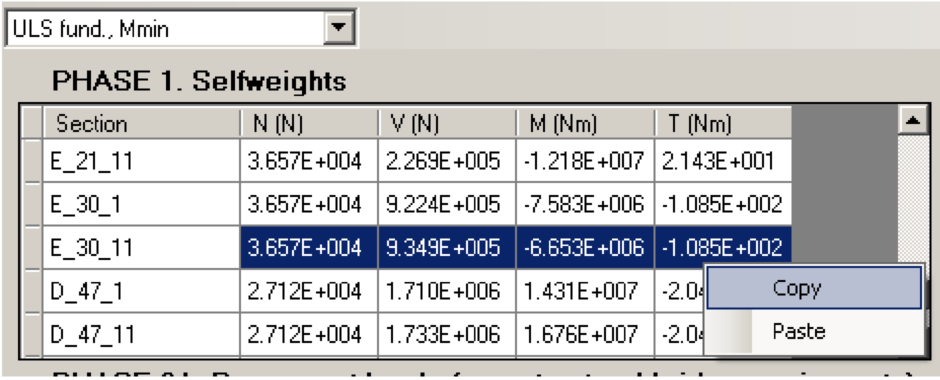
Note. At the bottom-left of the Plastic check page the section classification is also provided for Phase 1, where the top (upper) flange cannot be considered to be restrained by the slab. For this case, the section is in Class 4 as highlighted in the red message. For this situation a check of stresses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

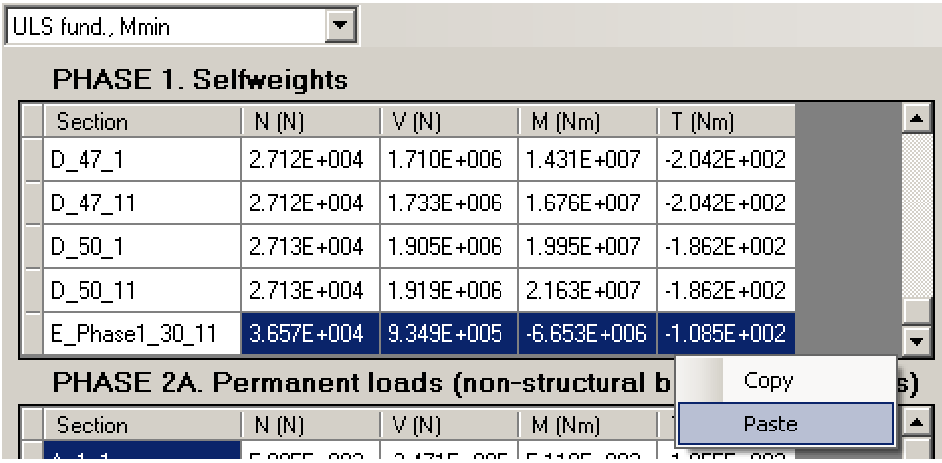
To check stresses on the effective section in Phase1

- Select the **Window> 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 30_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 **30_11** remaining. Click **OK.**
- In the Structural steel panel **uncheck** 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.

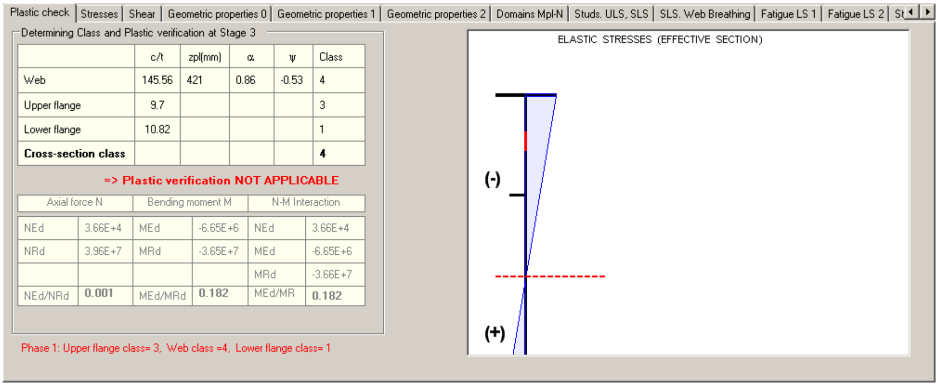


- Select the **Window> Forces and moments** menu item.
- On the Results dialog, select **ULS fund., Mmin** in the dropdown list seen in the Cross-sections and design combinations panel in the top-left corner.
- Copy the four cells of forces and moment data for Phase 1 from section **E_30_11**, (click **No** to not copy the header text) and paste the copied data into the equivalent cells for section **E_Phase1_30_11**, as shown on the following images.





- Select the **Window> Results** menu item
- Select Section **E_Phase1_30_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

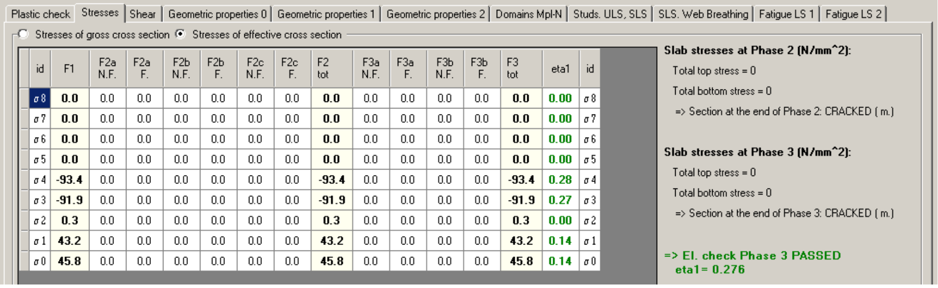


	c/t	zpl(mm)	α	ψ	Class
Web	145.56	421	0.86	-0.53	4
Upper flange	9.7				3
Lower flange	10.82				1
Cross-section class					**4**

=> Plastic verification NOT APPLICABLE

Axial force N		Bending moment M		N-M Interaction	
NEd	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

- Lastly, select the **Stresse**s tab to see the stresses calculated on an effective 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	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.00	σ 8
σ 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	σ 7
σ 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	σ 6
σ 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	σ 5
σ 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	σ 4
σ 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	σ 3
σ 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	σ 2
σ 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	σ 1
σ 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	σ 0

This completes the example.