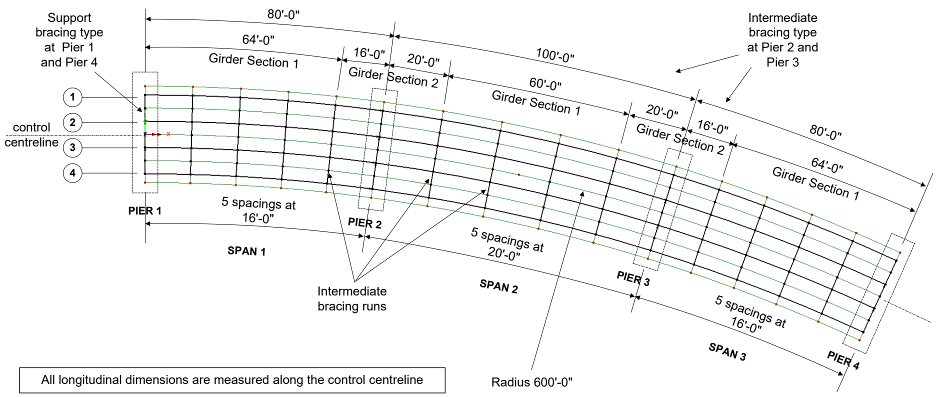
Composite Bridge Deck Design to AASHTO 9th Edition

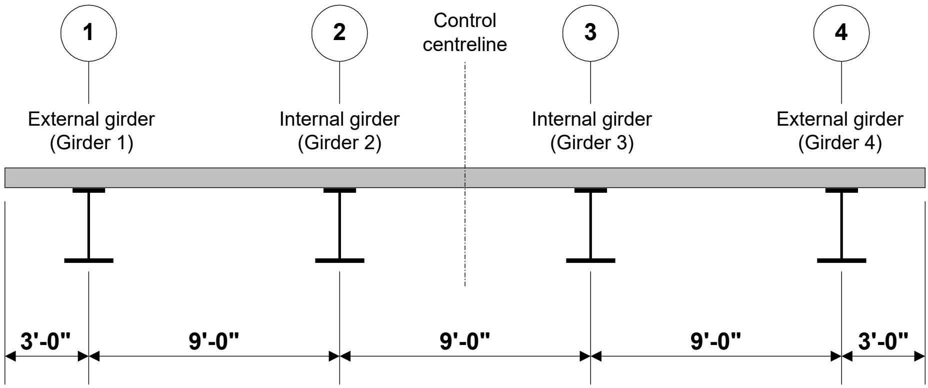
For LUSAS version:	22.0
For software product(s):	LUSAS *Civil & Structural* or LUSAS *Bridge.*
With product option(s):	Composite Bridge Deck Design, Nonlinear.
Note: This example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

This example shows how to carry out design of composite bridge deck members in accordance with the AASHTO LRFD 9th Ed. (2017) design code. The example uses a supplied model, initially created by the Steel Composite Bridge Wizard, that also includes staged construction modelling and vehicle loading in preparation for a design check to be done.



The bridge consists of three spans of length 80 feet, 100 feet and 80 feet. The bridge follows a radius of 600 feet along the control centreline and all supports are along radials. There are four girders across the width of the deck with stiffeners and bracing provided at supports and fifth point intervals along the span lengths. The deck cross-section is shown below. This example considers the design of one of these girders – Girder 2 along with the design of stiffeners.



Girder and slab thickness are as follows:

	External girder section 1	Internal girder section 1	External girder section 2	Internal girder section 2
Slab width	7’6”	9’0”	7’6”	9’0”
Slab thickness	8”	8”	8”	8”
Slab height of haunch	1.5”	1.5”	1.5”	1.5”
Slab offset to web	-9”	0”	-9”	0”
Top flange width	18”	18”	18”	18”
Top flange thickness	0.75”	0.75”	1.0”	1.0”
Web depth	4’0”	4’0”	4’0”	4’0”
Web thickness	0.5”	0.5”	0.5”	0.5”
Bottom flange width	24”	24”	24”	24”
Bottom flange thickness	1”	1”	1.5”	1.5”

Loading

For reasons of brevity and simplicity, in the in-service condition this example is limited to consideration of dead loads and traffic loads only and in the case of limit states is limited to Strength I, Service II and Fatigue I. Traffic loads for a 30'-0" wide carriageway are generated using the Vehicle Load Optimisation facility. Construction stages are considered, with a uniform construction load applied to the wet and hardened areas of concrete.

The following loads are considered in addition to the self-weight of the steel beams:

Item	Loading
Wet concrete slab	**0.10 kip/ft^2**
Construction load	**0.02 kip/ft^2**
Parapet	**0.225 kip/ft**
Surfacing	**0.023 kip/ft^2**

Note due to the modelling method used, the wet concrete slab and construction load are assigned to the model as appropriate line loads per unit length, and not as a surface load per unit area.

Load combinations considered:

- Constructability 1.4 (DC + C)
- Strength I 1.25 (DC) + 1.5 (DW) + 1.75 (LL)
- Service II DC + DW + 1.3 (LL)
- Fatigue I 1.75 (LL)

where DC = dead load: components and attachments

DW = dead load: wearing surface and utilities

C = construction load

LL = vehicle live load

Units used are kip,ft,kslug,s,F throughout.

Objective

To carry out design checks to AASHTO 9th edition and confirm the suitability of the member sizes.

Keywords

3D, Steel, Concrete, Composite, Bridge, Structural, Design code checks, AASHTO 9th, Bridge Deck Grillage, Design Member, Design Attribute, VLO, Staged Construction, Load Combinations, Branched Analysis, Member Utilisation Ratio, Member Report, Reporting, Steel Composite Bridge Wizard

Associated Files

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

- **composite_deck_design.mdl** includes the definition of design members, staged construction modelling and loading. To be used as a starting model for this example.

Associated Examples

- The worked example *'Steel Composite Bridge Wizard'* shows how to create a 3-span bridge model using the Steel Composite Bridge Wizard.
- The worked example *'Staged construction modelling of a 3-span bridge deck'* shows how to model the staged construction process included in this Composite Bridge Deck Design example.

Design Code Checking

Design code checking is carried out in LUSAS as a results processing operation following the solving of an analysis model. To enable a design check to take place, design attributes are assigned to members to provide design related information (as provided in the supplied model). Since the assigned design attributes have no impact on the structural analysis, they can be assigned either before or after the analysis has been run. Changes can be made to a design attribute (for example, changing a grade of steel, or a plate thickness) without re-running the structural analysis.

Modelling

Running LUSAS Modeller

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

Loading the supplied starting model

The supplied starting model is based on the geometry used in the worked example *'Steel Composite Bridge Wizard'* and includes staged construction modelling and loading as described in the worked example *'Staged Construction of 3-Span Bridge Deck'*.

File
Open...

- To load the model, open the file **composite_deck_design.mdl** that was downloaded and placed in a folder of your choosing.

Once opened:

- If necessary, select the isometric button.
- Toggling the Fleshing button on and off will show the steelwork and slab arrangement.
- In the treeview turn off the display of the mesh layer.

Construction and loading phases considered

The following modelling / analysis phases are included in the supplied model:

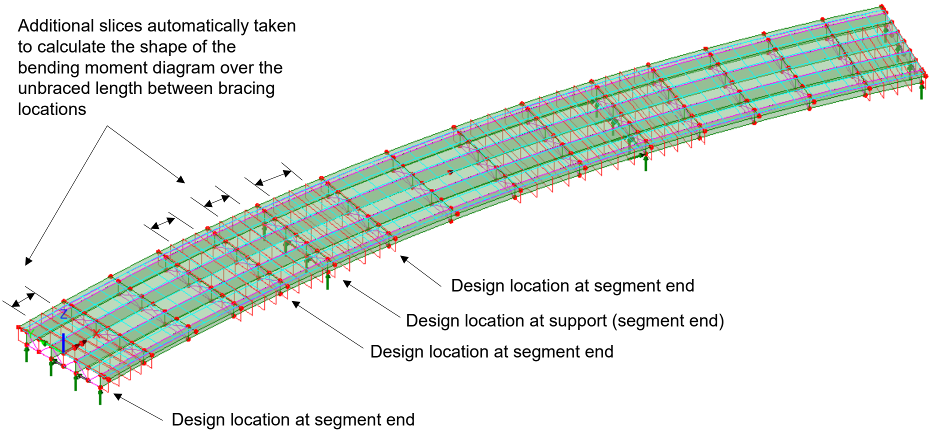
- **Slab construction stages**
- **Phase 1 - Steel only**
- **Phase 2 - Permanent and long-term loading (Parapet and surfacing loads)**
- **Phase 3 - Variable or short-term loading (Vehicle loading)**

Composite design members

When composite design members are created using the Steel Composite Bridge Wizard (as done for the supplied model), design locations as identified by the red slice planes seen on the following image are automatically created at segment ends. Note that segments will always occur within spans and that each support forms a segment end.

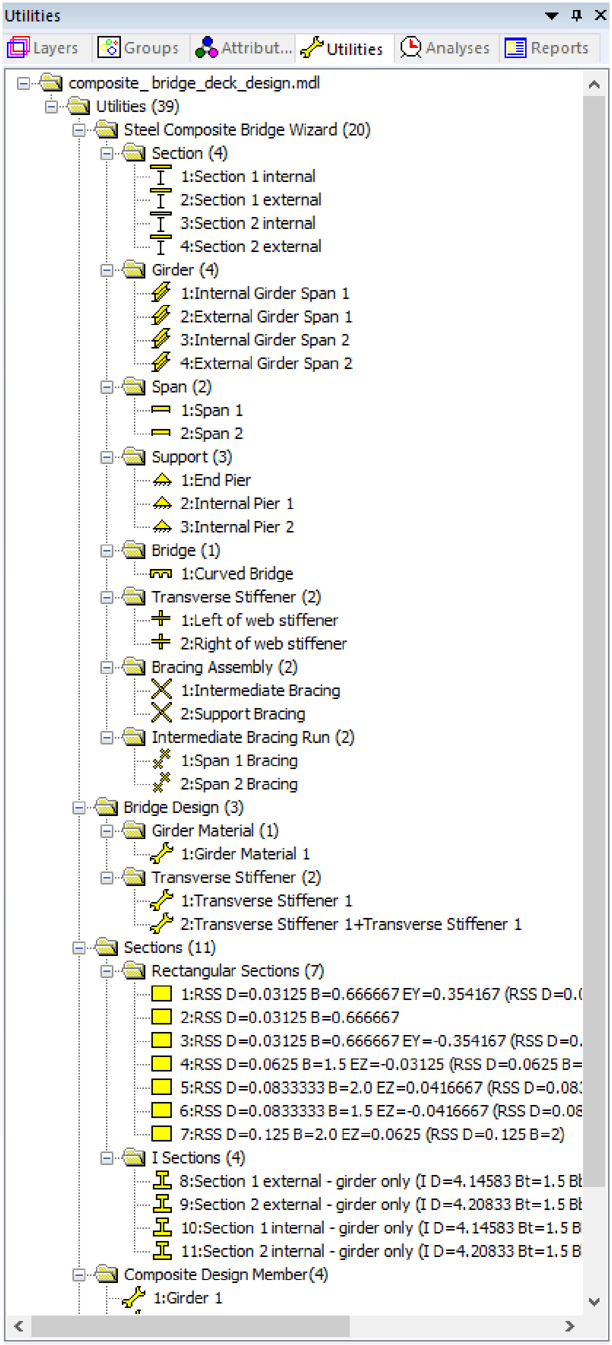
For each design location other slice locations are also defined with reference to the positions of the nearest bracing locations and the quarter points between them. These additional slice locations are used to determine the shape of the bending moment diagram over the unbraced length as well to compute the maximum lateral bending in the flanges.

Other design locations can be specified, but for the model that is loaded these are the initial design locations and additional slices created:



Viewing the utilities created by the Bridge wizard

- Select the Utilities tab to see the utilities created and used by the steel composite bridge design wizard to create the model geometry, the attributes seen in the Attributes treeview and their assignments to relevant features in the model.


Utilities
Layers
Groups
Attribut...
Utilities
Analyses
Reports
composite_ bridge_deck_design.mdl
Utilities (39)
Steel Composite Bridge Wizard (20)
Section (4)
1:Section 1 internal
2:Section 1 external
3:Section 2 internal
4:Section 2 external
Girder (4)
1:Internal Girder Span 1
2:External Girder Span 1
3:Internal Girder Span 2
4:External Girder Span 2
Span (2)
1:Span 1
2:Span 2
Support (3)
1:End Pier
2:Internal Pier 1
3:Internal Pier 2
Bridge (1)
1:Curved Bridge
Transverse Stiffener (2)
1:Left of web stiffener
2:Right of web stiffener
Bracing Assembly (2)
1:Intermediate Bracing
2:Support Bracing
Intermediate Bracing Run (2)
1:Span 1 Bracing
2:Span 2 Bracing
Bridge Design (3)
Girder Material (1)
1:Girder Material 1
Transverse Stiffener (2)
1:Transverse Stiffener 1
2:Transverse Stiffener 1+Transverse Stiffener 1
Sections (11)
Rectangular Sections (7)
1:RSS D=0.03125 B=0.666667 EY=0.354167 (RSS D=0.
2:RSS D=0.03125 B=0.666667
3:RSS D=0.03125 B=0.666667 EY=-0.354167 (RSS D=0.
4:RSS D=0.0625 B=1.5 EZ=-0.03125 (RSS D=0.0625 B=
5:RSS D=0.0833333 B=2.0 EZ=0.0416667 (RSS D=0.08
6:RSS D=0.0833333 B=1.5 EZ=-0.0416667 (RSS D=0.08
7:RSS D=0.125 B=2.0 EZ=0.0625 (RSS D=0.125 B=2)
I Sections (4)
8:Section 1 external - girder only (I D=4.14583 Bt=1.5 B
9:Section 2 external - girder only (I D=4.20833 Bt=1.5 B
10:Section 1 internal - girder only (I D=4.14583 Bt=1.5 B
11:Section 2 internal - girder only (I D=4.20833 Bt=1.5 B
Composite Design Member(4)
1:Girder 1

Save the model with a new name

File
Save As

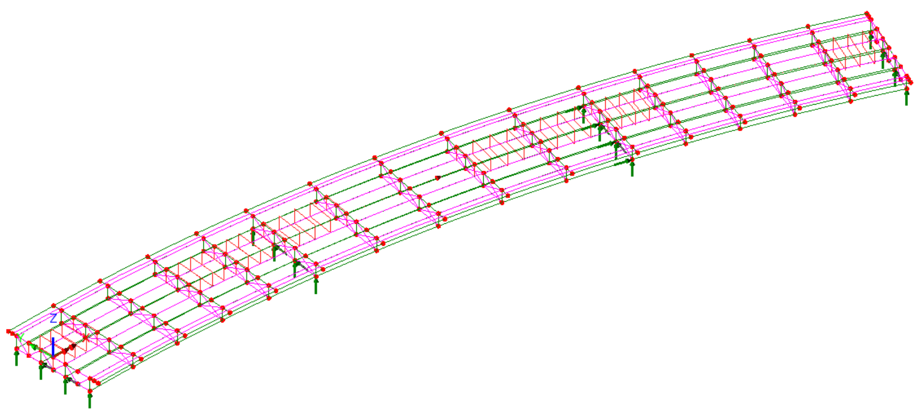
- Save the model file to a folder on your computer that has read and write permission, with the name of **composite_bridge_deck_design.mdl**

Simplifying the model

Design members are created automatically if models are created using the Steel Composite Bridge Wizard. Because this model was created in this way, the design members for the bridge are present in the Composite Design Member folder in the treeview.

Carrying out a full bridge design check for all bridge members at all locations in a model can be computationally and memory intensive and is not required for training purposes. For this example, the design of a single bridge girder (Girder 2) will be considered at a number of selected locations to demonstrate the methodology.

- In the treeview delete the Composite Design Members named **Girder 1**, **Girder 3** and **Girder 4**.
- In the treeview, turnoff the display of the mesh layer.
- In the treeview, double click the **Geometry** entry, and ensure the **Solid** option is un-checked, and click **OK**.
- In the treeview, right-click on the **DMI_1** entry, and ensure that the **Show grid** option is unchecked.



- In the treeview turn-off the display of the **Utilities** layer.

Allowing editing of the Bridge Wizard-created model

Attributes automatically created from the bridge wizard utilities (that were created from running Steel Composite Bridge Wizard in order to create this model) are protected from being assigned, de-assigned, edited or deleted unless the protection is overridden as follows:

File
Model properties

- On the Model Properties dialog, select the **Attributes** tab and check (tick) the **Allow modification of LUSAS-generated attributes** option. Click **OK**.

Editing a Design Member

Much of the information required to carry out the design of a girder is already populated in the design member utility. However, details of shear connectors must be defined. Other details such as materials, bracing or stiffeners for design can also be modified.

- In the treeview double-click on **Girder 2**.
- On the **General Arrangement** tab the actual span length of each girder can be seen. These lengths will be used when defining shear connectors.

Note. If a model has not been created using the bridge wizard, design members can be created using the 'Utilities> Bridge Design> Composite Design Member' menu option.

- Click on the **Segments**, **Bracing** and **Web Stiffeners** tabs in turn to see the values created by running the Bridge Wizard.

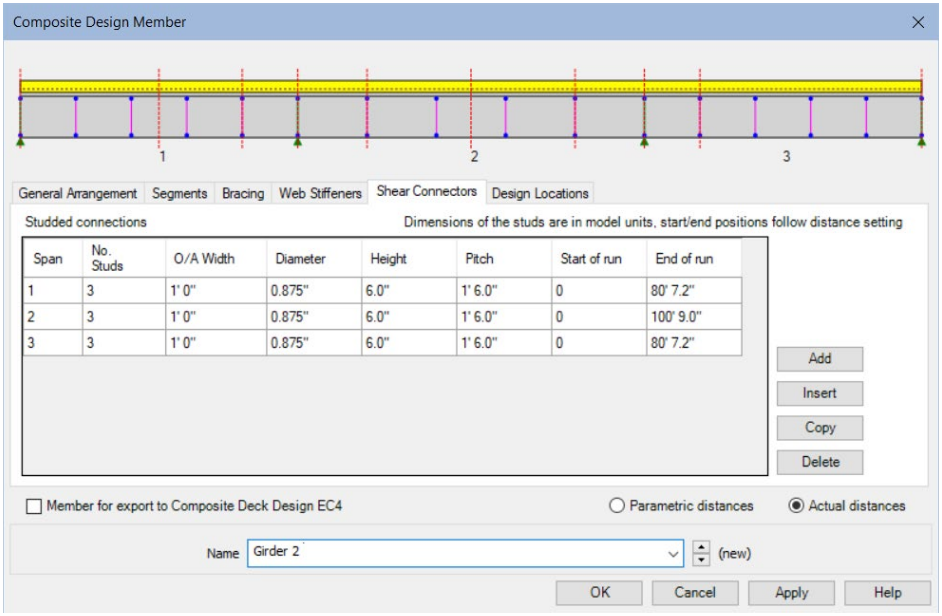
Shear connectors

Shear connectors will be added over the full length of the girder.

Note. In this example, model units were specified as feet, but some values (such as stud diameter) are best entered in inches. To signify inches, " needs to be appended to the value entered. Alternatively, the Unit convertor could be used.

- Select the **Shear Connectors** tab and click the **Add** button.
- For Span **1**, enter **3** for the number of studs.
- Click in the O/A Width cell and enter a value of **1** (see note below).
- In the Diameter cell enter a value of **0.875"**
- In the Height cell enter a value of **6"**
- In the Pitch cell enter a value of **18"**
- In the Start of run cell enter a value of **0**

- In the End of run cell enter a value of **80'7.2"** (the value that is stated for Span 1 on the General Arrangement tab)
- Click the **Copy** button twice.
- On the second row change the Span to **2**. Change the End of run value to **100'9"**
- On the third row change the Span to **3**
- Click in any cell to update the schematic diagram.



Shear connectors are represented on the schematic design member by dotted horizontal purple lines within the slab showing the length over which they apply.

Note. The O/A Width value is required for other supported design codes and is ignored and not used in the AASHTO design checks.

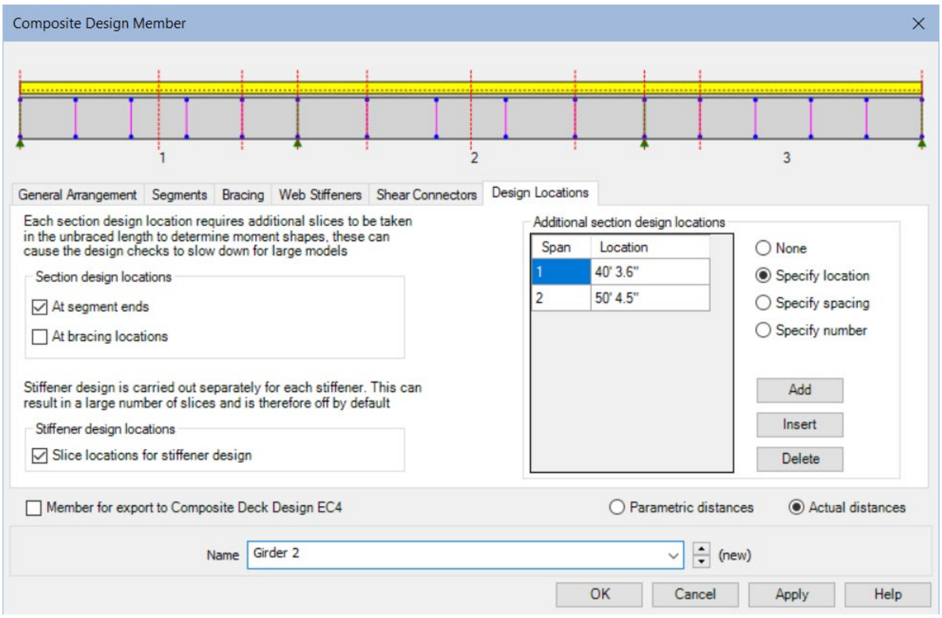
Design locations

For this example, design checks are required at segment ends (these are automatically created), and also at the mid-span of span 1 and span 2, and at all stiffener locations. To define these:

- Click on the **Design locations** tab.
- In the 'Section design locations' panel, ensure that the **At segment ends** option is selected.

- In the 'Additional section design locations' panel, select the **Specify location** option.
- Click **Add** twice.
- On the first row in the Location cell for Span **1**, enter a value of **40'3.6"**
- On the second row in the Location cell, change the span number to be Span **2** and in the Location cell enter a value of **50'4.5"**
- In the 'Stiffener design locations' panel, select the **Slice locations for stiffener design** check box.
- Click **OK**.
- If shown, click **OK** to accept the message(s) relating to Beam / Shell slice results.

Note. When deactivation attributes have been assigned to a model, beam/shell slice results may be incorrect for an analysis unless the model features and mesh have been set to exactly match the activation / deactivation stage of the model for the stage being sliced and the extent set to the visible model (not the full model).



Design locations for segment ends and bracing locations are represented on the schematic design member by vertical red dotted lines. Bracing locations are shown by blue circles. Stiffener locations are shown by magenta lines.

Note. When additional section or stiffener design locations are specified or requested, additional influence locations are automatically added to the treeview.

Geometric Attributes explained

- Click on the treeview tab.

The geometric line folder in the treeview contains assigned and unassigned attributes that were created by running the Steel Composite Bridge design wizard.

- ❑ The assigned attributes (denoted by a coloured icon) are automatically assigned to features in the model by the wizard and are used in an analysis.
- ❑ The unassigned bridge deck (grillage) geometric attributes (denoted by a grey coloured icon) are not used in an analysis. <u>However, these unassigned attributes are used as part of a design check and require slab reinforcement to be specified</u>.

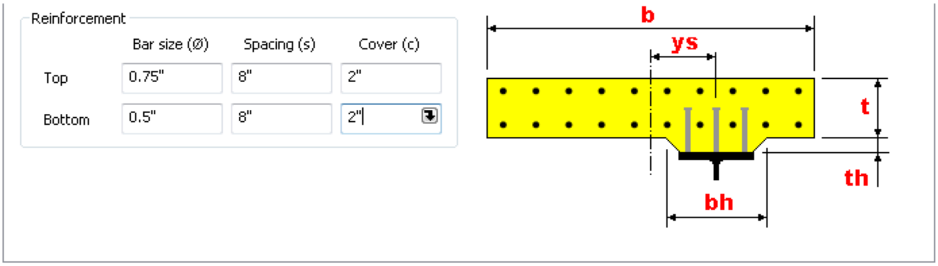
Caution. If a bridge model has been generated by the Steel Composite Bridge Wizard, unassigned bridge deck (grillage) geometric attributes created by the wizard should never be assigned to features in the model otherwise incorrect properties will be used. Unassigned bridge deck (grillage) geometric attributes are created for and used only by a code-specific design check.

Editing the Unassigned Geometric Attributes

For this example, because design checks are only to be carried out for an internal girder (Girder 2), only the unassigned geometric attributes for this internal beam need to be edited to include rebar details for a design check.

Section 1 internal

- In the treeview double click on the geometric attribute name of **Section 1 internal (Bridge Design Section)**.
- Select the **Slab** tab. Slab details for this section can be seen.
- In the reinforcement panel for the top reinforcement set the Bar size (ϕ) as **0.75”**, the Spacing (s) as **8”** and the Cover (c) as **2”**.
- In the reinforcement panel for the bottom reinforcement set the Bar size (ϕ) as **0.5”**, the Spacing (s) as **8”** and the Cover (c) as **2”**.



- Click **OK**.

Section 2 internal

- In the treeview double click on the geometric attribute name of **Section 2 internal (Bridge Design Section)** and enter the same reinforcement values for the slab as for Section 1 internal and click **OK**.

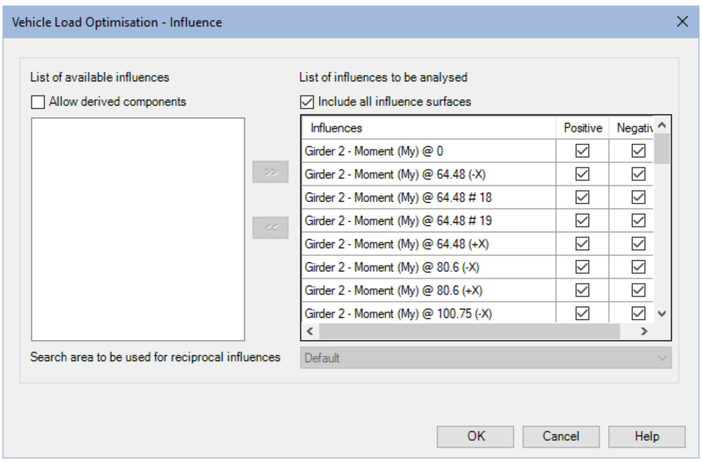
This completes the entry of the initial design settings required.

Editing of design member details

Note. When modifying a design member that has some Direct Method Influence assignments which are used in a Vehicle Load Optimisation analysis, the DMI assignments are deleted from the model and from the VLO run that is using them. The DMI attribute is then automatically re-assigned to that design member, creating new assignments. However, the list of influences previously specified in the Vehicle Load Optimisation dialog are not automatically included again and will need to be re-included to re-create the vehicle loading patterns before another analysis can be solved.

Re-define the influences to be analysed

- In the Treeview double click on the **VLO Girder 2** entry
- On the main VLO dialog, click the **Set influence surfaces** button
- On the Influence dialog select **Include all influence surfaces**. Ensure that the **Positive** and **Negative** checkboxes are selected for all included influences. Note that clicking in the header cell for a column will select all entries and allow all rows to be checked (ticked) with one click.



- ❑ Click **OK** to return to the main VLO dialog, then click **OK** again. This will then run the VLO analysis.

Running the Analysis

With the VLO loading now re-created:

Select the **Solve Now** button from the toolbar and click **OK** to run the analyses.

- Click **OK** to any warnings about deactivated elements.

A LUSAS datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

If the analysis is successful...

Analysis loadcase results are added to the Treeview

In addition, these files will be created in the LUSASFiles\<model_name> folder:

- ❑ **composite_bridge_deck_design.out** This output file contains details of model data, assigned attributes and selected statistics of the analysis.
- ❑ **composite_bridge_deck_design.mys** This is the LUSAS results file which is loaded automatically into the Treeview to allow results processing to take place.

If the analysis fails...

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

Viewing the Results

Analysis loadcase results are present in the Treeview.

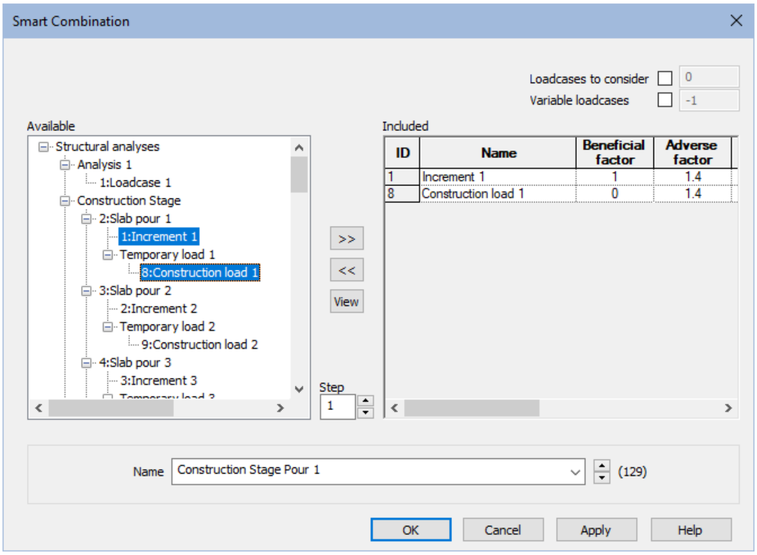
Appropriate load combinations must now be defined in preparation for a design check.

Defining load combinations

First define the Construction Stage combinations.

Analyses
Smart Combination…

- In the ‘Available’ loadcase panel, in the Construction stage analysis, select **Increment 1** and **Construction load 1** and click the >> button to add the loadcases to the combination dataset.
- Change the beneficial factor for ‘Construction load 1’ to **0** and the adverse factor for both to **1.4**.



- Change the combination name to be **Construction Stage Pour 1** and click **Apply**. A ‘Max’ and ‘Min’ loadcase entry will be added to the treeview.
- In the ‘Included’ panel select **Increment 1** and **Construction load 1** and click the << button to remove them from the panel.
- In the ‘Available’ loadcase panel select **Increment 2** and **Construction load 2** and click the >> button to add the loadcases to the combination dataset.
- Change the beneficial factor for ‘Construction load 2’ to **0** and the adverse factor for both to **1.4**.
- Change the combination name to be **Construction Stage Pour 2** and click **Apply**.
- Repeat, separately, removing then adding the remaining construction stage loadcases and factors as shown in the following table.

Loadcase Name	Beneficial factor	Adverse factor	Combination name
Increment 3	**1**	**1.4**	**Construction Stage Pour 3**
Construction load 3	**0**	**1.4**	
Increment 4	**1**	**1.4**	**Construction Stage Pour 4**
Construction load 4	**0**	**1.4**	
Increment 5	**1**	**1.4**	**Construction Stage Pour 5**
Construction load 5	**0**	**1.4**	

Next, define the In Service Loadcase combinations.

Analyses
Smart Combination…

- In the 'Available' loadcase panel, in the Phase 2 analysis, select **Parapet** and **Surfacing** and click the >> button to add the loadcases to the combination dataset, and leave all factors as their default values of **1**.
- Change the combination name to be **(DC + DW) (Service)** and click **Apply**.
- Change the Adverse factor for the Parapet loadcase to **1.25**
- Change the Adverse factor for the Surfacing loadcase to **1.5**
- Change the combination name to **(1.25DC + 1.5DW) (Strength)** and click **Apply**.
- In the 'Included' panel select **Parapet** and **Surfacing** and click the << button to remove them from the panel.
- In the 'Available' loadcase panel, in the Phase 1 (Steel only) analysis, select **Steel SW** and **Wet Slab** and click the >> button to add the loadcases to the combination, and leave all factors as their default values of **1**.
- Change the combination name to **Steel (Service)** and click **Apply**.
- Change the Adverse factor for both loads to **1.25**.
- Change the name to **Steel (Strength)** and click **OK**.

Initializing Composite Deck Design

Design
Composite Bridge Deck Design…

- On the Design Code droplist select **AASHTO LRFD 9th Edition (2017).**
- Check (tick) the option **Supports are skewed less than 20 degrees**.
- Accept the default settings for all other parameters and click **OK**.

Note. The 'Cross bracing is contiguous and parallel to supports' option is left unchecked even though the bracing used in this example meets these criteria. This will provide conservative results.

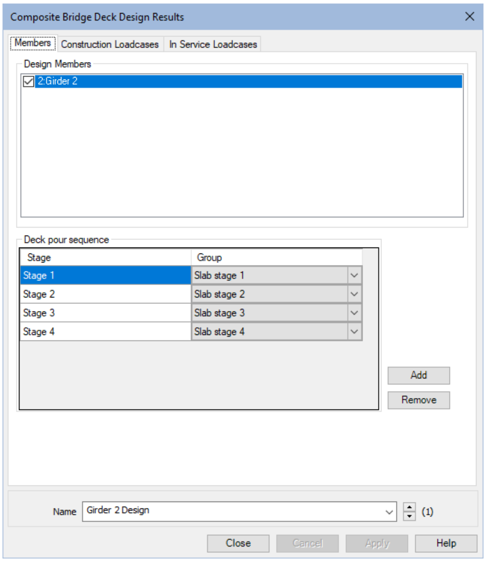
Defining a Composite Deck Design Results Utility

Prior to carrying out a design check, design members need to be associated with particular stages of construction for selected deck pours, and construction and in-service loadcases. This is done by defining a Composite Bridge Deck Design results utility.

Specifying members of interest

Design
Composite Bridge Deck Design Results…

- With the **Members** tab active, in the Design Members panel select the check box for **Girder 2**.
- Click the **Add** button four times to add four stages to the Deck pour sequence panel.
- In the 'Deck pour sequence' panel, from the Group droplist select **Slab stage 1** for Stage 1.
- Similarly, select **Slab stage 2** for Stage 2, **Slab stage 3** for Stage 3 and **Slab stage 4** for stage 4.



Note that there is no need to add Stage 5 because design checks in the construction stage are only carried out at locations where the slab is not present.

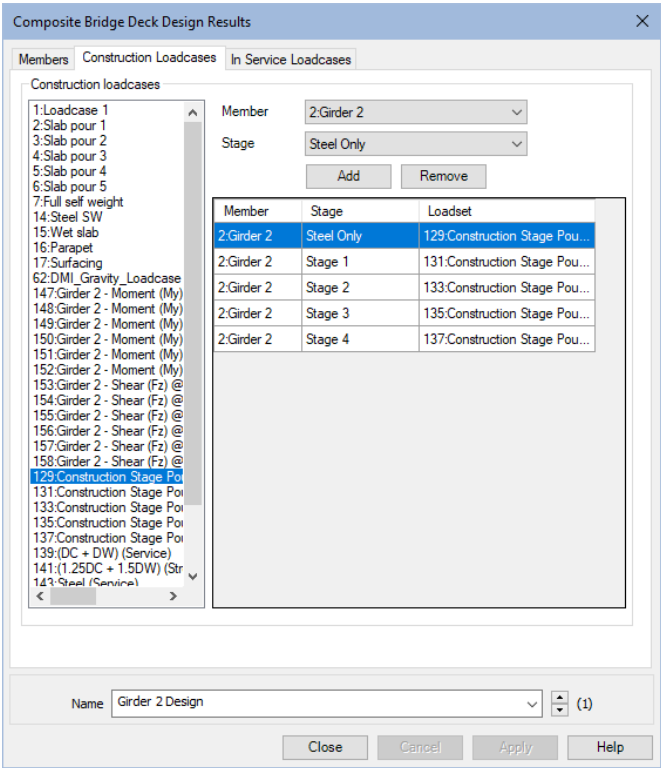
Specifying construction loadcases

- Click on the **Construction Loadcases** tab.

By default, the Member droplist will be set to **Girder 2** and the Stage will be set to **Steel Only.**

- From the 'Construction loadcases' panel select **Construction Stage Pour 1.** (It will be the lowest numbered 'Construction Stage' entry in the set available.)
- Click the **Add** button to add a row to the grid that references these 3 selections of Member, Stage and Loadset (as shown on the following image).

- Next, from the 'Construction loadcases' panel select **Construction Stage Pour 2.**
- Change the Stage droplist to be **Stage 1** and click the **Add** button to add these selections to a row in the grid.
- Repeat for the remaining concrete stages (first selecting a stage each time) to create rows for Member, Stage and Loadset as shown on the following image.



Note, again, that there is no need to add Stage 5 because design checks in the construction stage are only carried out at locations where the slab is not present.

Note. Loadcase ids may differ from those shown in this and following dialogs according to how many times the model has been previously solved. The loadcase names will not change however and should be used to confirm the correct assignments have been made.

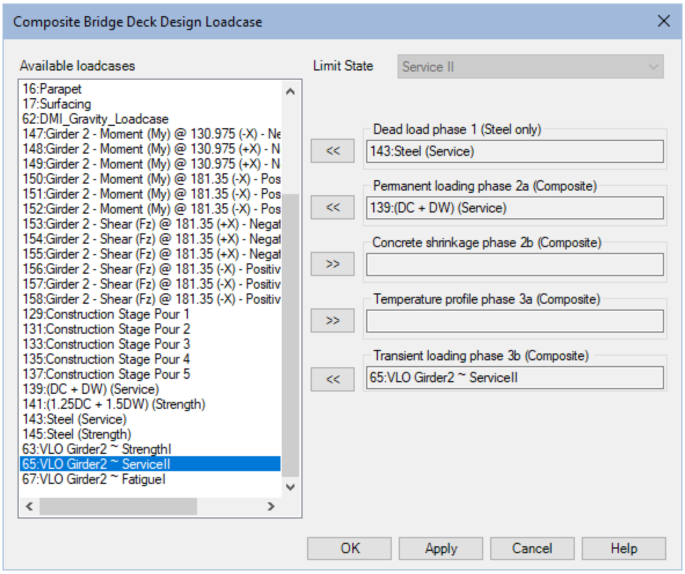
Specifying in service loadcases

- Click on the **In Service Loadcases** tab.

Three limit states are considered - Service, Fatigue and Strength, and appropriate loadcases (with the correct factors applied to those loadcases) must be assigned to each limit state. Press the **Help** button for this dialog for more details.

Service loadcases

- In the 'Service loadcases' panel, click the **Add** button.
- From the list of available loadcases select **Steel (Service)** and click the 'Add to' >> button to add the loadcase to **Dead load phase 1 (Steel only).**
- Then select loadcase **(DC + DW) (Service)** and click the 'Add to' >> button to add the loadcase to **Permanent loading phase 2a (Composite).**
- Lastly select loadcase **VLO run Girder 2 ~ Service II** and click the 'Add to' >> button to add the loadcase to **Transient loading phase 3b (Composite).**

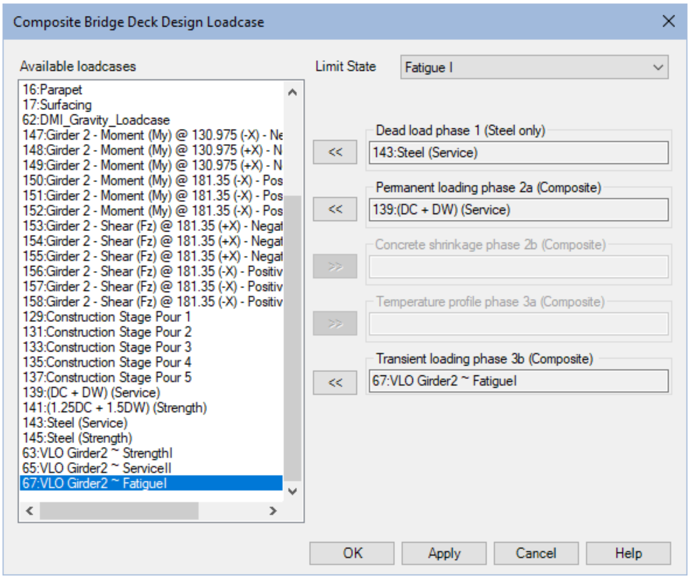


- Click **OK** to return to the parent dialog.

Fatigue loadcases

- In the 'Fatigue loadcases' panel, click the **Add** button.
- From the list of available loadcases select **Steel (Service)** and click the 'Add to' >> button to add the loadcase to **Dead load phase 1 (Steel only).**
- Then select loadcase **(DC + DW) (Service)** and click the 'Add to' >> button to add the loadcase to **Permanent loading phase 2a (Composite).**

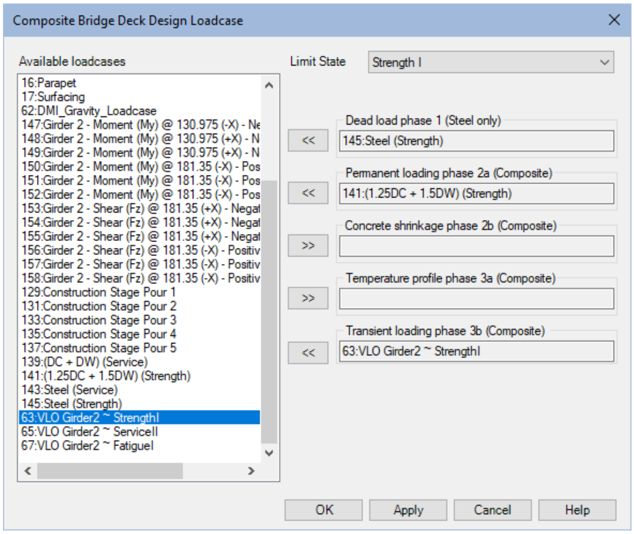
- Lastly select loadcase **VLO run Girder 2 ~ Fatigue I** and click the 'Add to' button to add the loadcase to **Transient loading phase 3b (Composite)**.



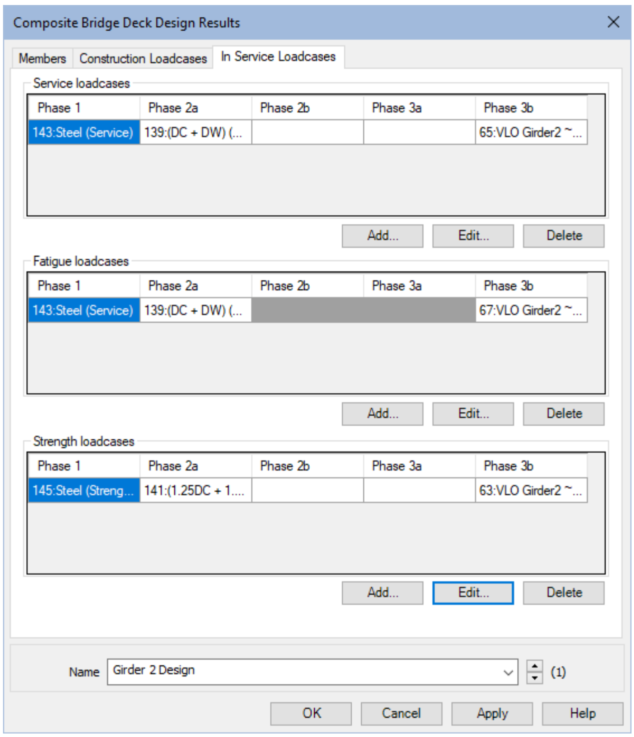
- Click **OK** to return to the parent dialog.

Strength loadcases

- In the 'Strength loadcases' panel, click the **Add** button.
- From the list of available loadcases select **Steel (Strength)** and click the 'Add to' button to add the loadcase to **Dead load phase 1(Steel only).**
- Then select loadcase **(1.25DC + 1.5DW) (Strength)** and click the 'Add to' button to add the loadcase to **Permanent loading phase 2a (Composite).**
- Lastly select loadcase **VLO run Girder 2 ~ Strength I** and click the 'Add to' button to add the loadcase to **Transient loading phase 3b (Composite).**



- Click **OK** to return to the parent dialog, where the settings made should look like this:



- Lastly change the design results name to be **Girder 2 Design** and click **OK**.

A Composite Deck Design Results entry is then added to the treeview.

Comparison of design code and LUSAS loading phases

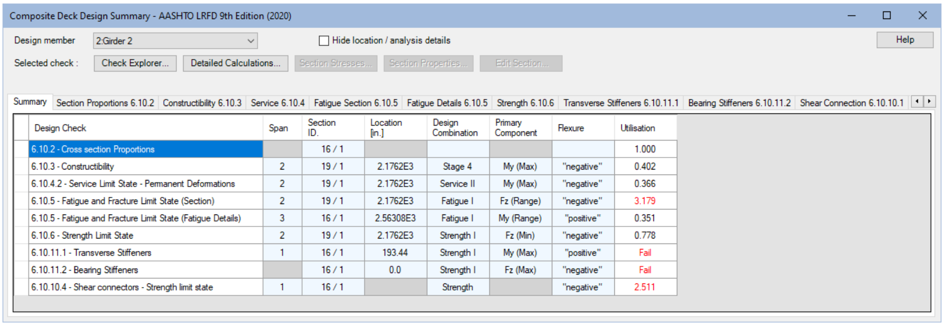
AASHTO 9th Edition (6.10.1.1.1a)	Equivalent LUSAS loading phases
Steel section	**Phase 1** is the loading applied to the steel structure before it acts compositely with the deck slab. Typically, this is the weight of the steel and the weight of the wet concrete. These forces are carried by the steelwork alone.
Long-term composite section	**Phase 2** is the permanent or long-term loading that is carried by the composite section. It is split into two sub phases: • **Phase 2a** represents the permanent loading acting on the composite section. • **Phase 2b** represents the permanent effects induced by shrinkage of the concrete
Short term composite section	**Phase 3** is the variable or short-term loading that is carried by the composite section. It is split into two sub phases: • **Phase 3a** represents temperature effects. • **Phase 3b** represents transient short-term loading, such as traffic and wind.

Viewing results

From the Composite Deck Design Results entry in the treeview, the design check results can be viewed, and tables of results can be obtained.

- In the treeview right-click on the **Girder 2 Design** results entry and select **Show Results**. It will take several minutes to compute the design checks when using beam/shell slicing.
- On completion and with the **Summary** tab selected, it can be seen that utilisation values of greater than 1 are present for the Fatigue and Fracture Limit State

(Section) and that both the Transverse Stiffeners and Bearing Stiffeners fail the design checks:



Design Check	Span	Section ID.	Location [in.]	Design Combination	Primary Component	Flexure	Utilisation
6.10.2 - Cross section Proportions		16 / 1					1.000
6.10.3 - Constructibility	2	19 / 1	2.1762E3	Stage 4	My (Max)	"negative"	0.402
6.10.4.2 - Service Limit State - Permanent Deformations	2	19 / 1	2.1762E3	Service II	My (Max)	"negative"	0.366
6.10.5 - Fatigue and Fracture Limit State (Section)	2	19 / 1	2.1762E3	Fatigue I	Fz (Range)	"negative"	3.179
6.10.5 - Fatigue and Fracture Limit State (Fatigue Details)	3	16 / 1	2.56308E3	Fatigue I	My (Range)	"positive"	0.351
6.10.6 - Strength Limit State	2	19 / 1	2.1762E3	Strength I	Fz (Min)	"negative"	0.778
6.10.11.1 - Transverse Stiffeners	1	16 / 1	193.44	Strength I	My (Max)	"positive"	Fail
6.10.11.2 - Bearing Stiffeners		16 / 1	0.0	Strength I	Fz (Max)	"negative"	Fail
6.10.10.4 - Shear connectors - Strength limit state	1	16 / 1		Strength		"negative"	2.511

Note that the 'Messages' tab provides potentially useful information about errors and warnings.

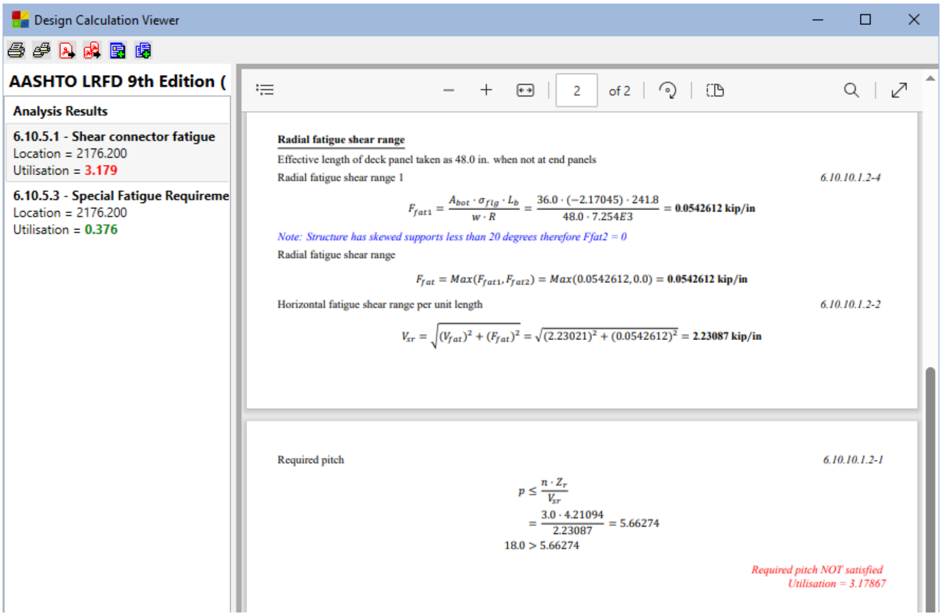
More information on the utilisation values can be obtained by right-clicking on a previously selected row in the table and selecting the appropriate check button.

- ❑ **Check Explorer** shows basic details of calculations carried out.
- ❑ **Detailed Calculations** provides full details of all calculations carried out.

Note that a right-click on an entry in the Utilisation column provides the same options.

Fatigue check

- In the Design Summary table, left-click on the Utilisation value for the Fatigue and Fracture Limit State (Section), then right-click and choose **Detailed Calculations…**
- In the left-hand panel, click on the **6.10.5.1 – Shear connector fatigue** results entry for the reported exceeded utilisation.

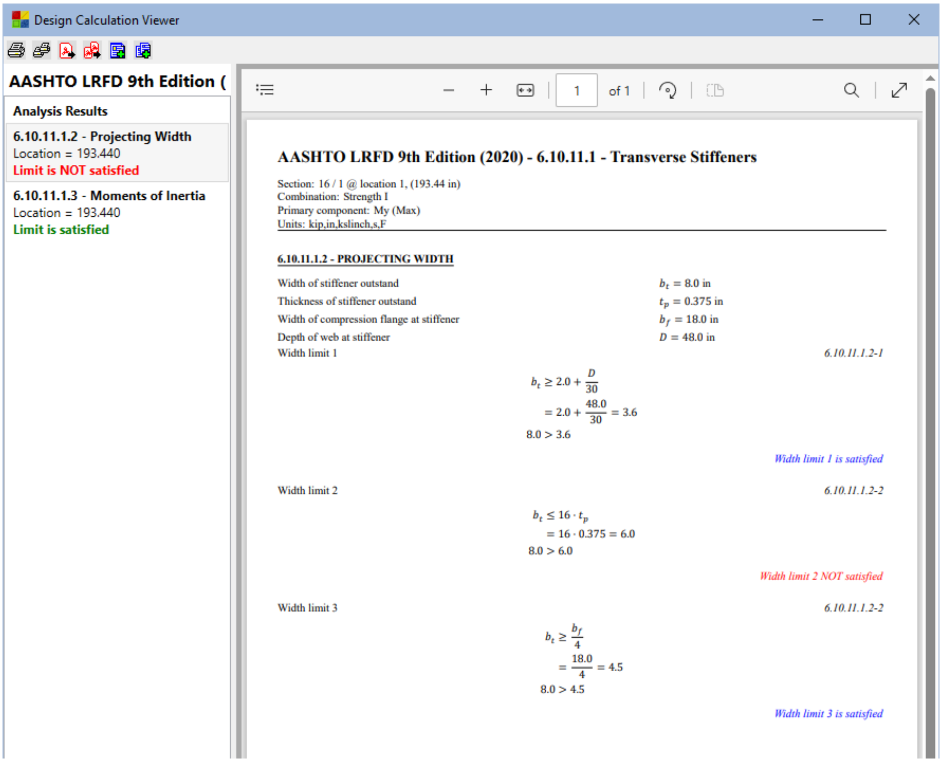


It can be seen that the required pitch of stiffeners is not satisfied. The use of a larger stud and/or a closer pitch locally by the failure zone (by the internal supports) would give the required capacity. This is covered at the end of the example.

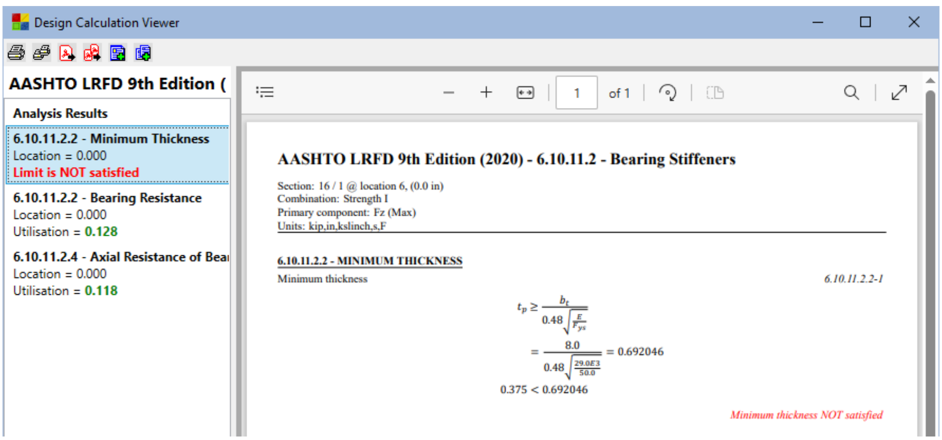
- Close the Design Calculation Viewer window.

Transverse and Bearing stiffener check

- In the Design Summary table, left-click on the ‘**Fail**’ cell for the Transverse Stiffeners row, then right-click and choose **Detailed Calculations…**
- In the left-hand panel, click on the **6.10.11.1.2 – Projecting Width** entry.



- Close the Design Calculation Viewer window.
- In the Design Summary table, left-click on the '**Fail**' cell for the Bearing Stiffeners row, then right-click and choose **Detailed Calculations…**
- In the left-hand panel, click on the **6.10.11.2.2 – Minimum Thickness** entry.

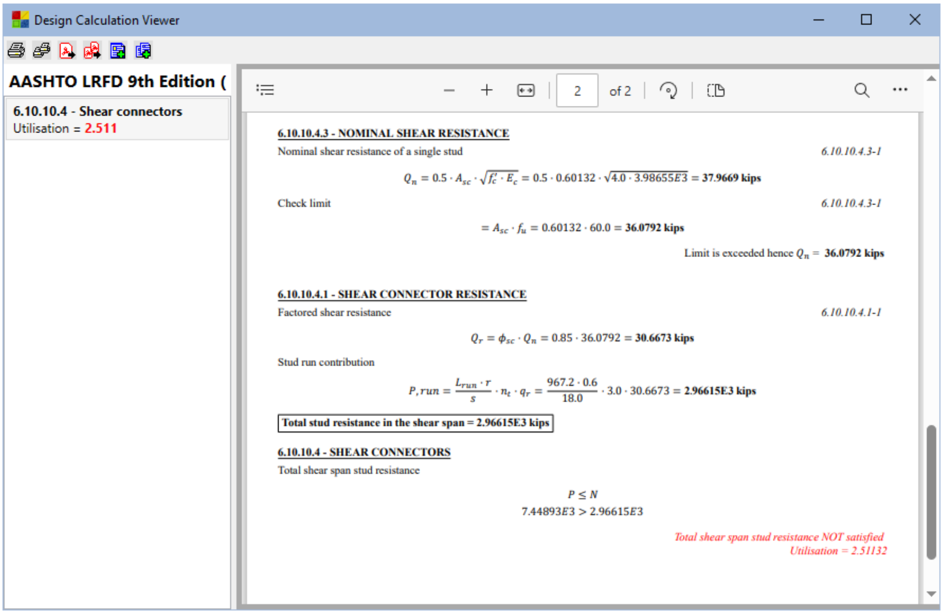


It can be seen that, for both stiffener types, the Minimum Thickness check is not satisfied. This can be resolved by increasing the thickness of the stiffeners. This is covered at the end of the example.

- Close the Design Calculation Viewer window.

Shear Connector check

- In the Design Summary table, left-click on the Utilisation value for the Shear Connectors - Strength Limit State, then right-click and choose **Detailed Calculations…**



A larger stud and/or a closer pitch locally by the failure zone (by the internal supports) will be defined to try and obtain the required capacity.

- Close the Design Calculation Viewer window.

Save the model

File
Save

Save the model file.

Changes required to pass the design check

Increase the diameter and pitch of shear studs

To pass the fatigue and shear connector checks the diameter and /or the pitch of the shear connectors need to be modified. This is done by revisiting the Composite Design Member.

- In the treeview, inside the Composite Design Member folder, double-click on **Girder 2**.
- Select the **Shear Connectors** tab.
- For each span row, in the **Diameter** cell change the value from 0.875" to **1"**
- For each span row, in the **Pitch** cell change the value from 18" to **6"**and click **OK**.

To save time, the design check will only be carried out once all changes are made.

Increase the stiffener thickness

To overcome the "Projecting width not satisfied" design check, the stiffener thickness must be increased. In this example the same stiffener types are used as web stiffeners and as bearing stiffeners, as stated on the Composite Design Member dialog on the Web stiffeners tab.

- In the treeview, inside the Transverse Stiffener folder, double-click on **Transverse Stiffener 1**
- Change the value for **Thickness, t** from 0.375" to **0.75"** and click **OK**
- In the treeview, inside the Transverse Stiffener folder double-click on **Transverse Stiffener 1 + Transverse Stiffener 1**
- Change the value for **Thickness, t** from 0.375" to **0.75"** and click **OK**

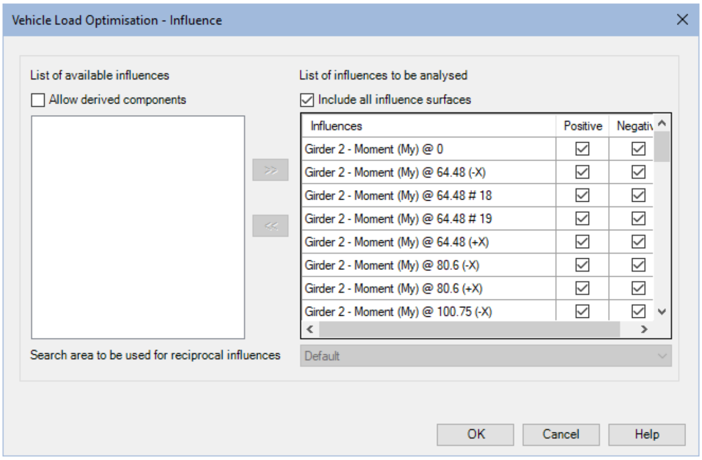
Editing of design member details

As stated previously, when modifying a design member that has some Direct Method Influence assignments which are used in a Vehicle Load Optimisation analysis, the list of influences previously specified in the Vehicle Load Optimisation dialog of the supplied model for this example will need to be re-included before running the analysis.

Re-define the influences to be analysed

- In the Treeview double click on the **VLO Girder 2** entry
- On the main VLO dialog, click the **Set influence surfaces** button

- On the Influence dialog select **Include all influence surfaces**. Ensure that the **Positive** and **Negative** checkboxes are selected for all included influences. Note that clicking in the header cell for a column will select all entries and allow all rows to be checked (ticked) with one click.



- ❑ Click **OK** to return to the main VLO dialog, then click **OK** again to run the VLO analysis.

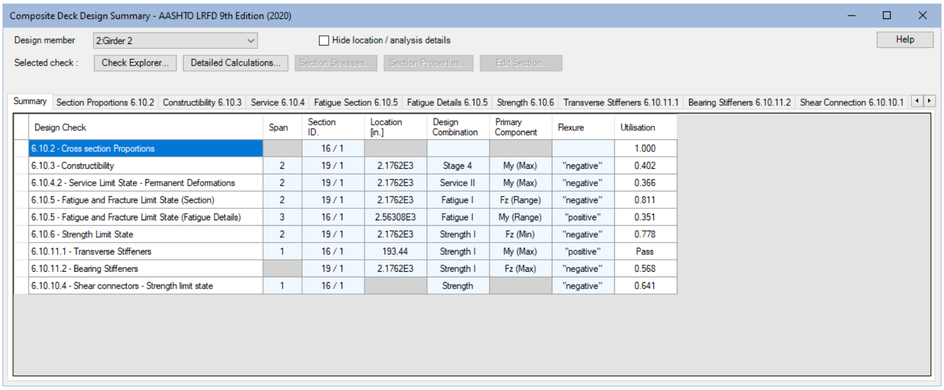
Solving for the changes made

Select the **Solve Now** button from the toolbar and click **OK** to run the analyses.

- Click **OK** to any warnings about deactivated elements.

Viewing updated design results

- In the treeview right-click on the **Girder 2 Design** results entry and select **Show Results**. This may take several minutes to compute design checks.
- With the **Summary** tab selected, it can be seen that utilisation values of less than 1.0 are present for all entries, and the Transverse stiffeners are now shown to have passed.



Design Check	Span	Section ID.	Location [in.]	Design Combination	Primary Component	Flexure	Utilisation
6.10.2 - Cross section Proportions		16 / 1					1.000
6.10.3 - Constructibility	2	19 / 1	2.1762E3	Stage 4	My (Max)	"negative"	0.402
6.10.4.2 - Service Limit State - Permanent Deformations	2	19 / 1	2.1762E3	Service II	My (Max)	"negative"	0.366
6.10.5 - Fatigue and Fracture Limit State (Section)	2	19 / 1	2.1762E3	Fatigue I	Fz (Range)	"negative"	0.811
6.10.5 - Fatigue and Fracture Limit State (Fatigue Details)	3	16 / 1	2.56308E3	Fatigue I	My (Range)	"positive"	0.351
6.10.6 - Strength Limit State	2	19 / 1	2.1762E3	Strength I	Fz (Min)	"negative"	0.778
6.10.11.1 - Transverse Stiffeners	1	16 / 1	193.44	Strength I	My (Max)	"positive"	Pass
6.10.11.2 - Bearing Stiffeners		19 / 1	2.1762E3	Strength I	Fz (Max)	"negative"	0.568
6.10.10.4 - Shear connectors - Strength limit state	1	16 / 1		Strength		"negative"	0.641

Save the model

File
Save

Save the model file.

This completes the example.