AASHTOWare BrDR 7.5.1

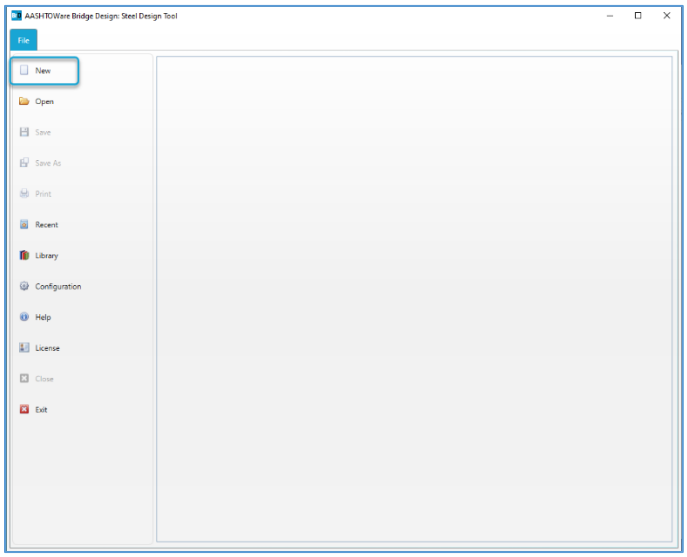
Steel Design Tool

Two Span Girder Design Example

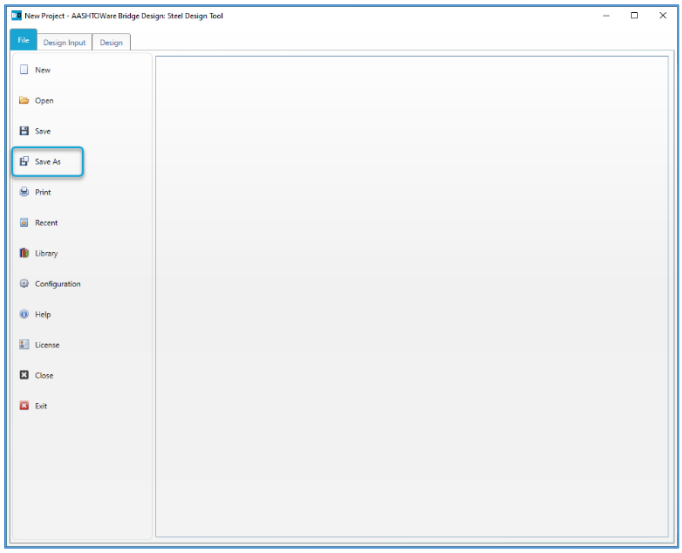
Two Span Girder Design Example

Start the **Steel Design Tool** program, create a new input file using the **File | New** command. The program will switch from the **File** tab to the **Design Input** tab.

File | New and **File | Save As**



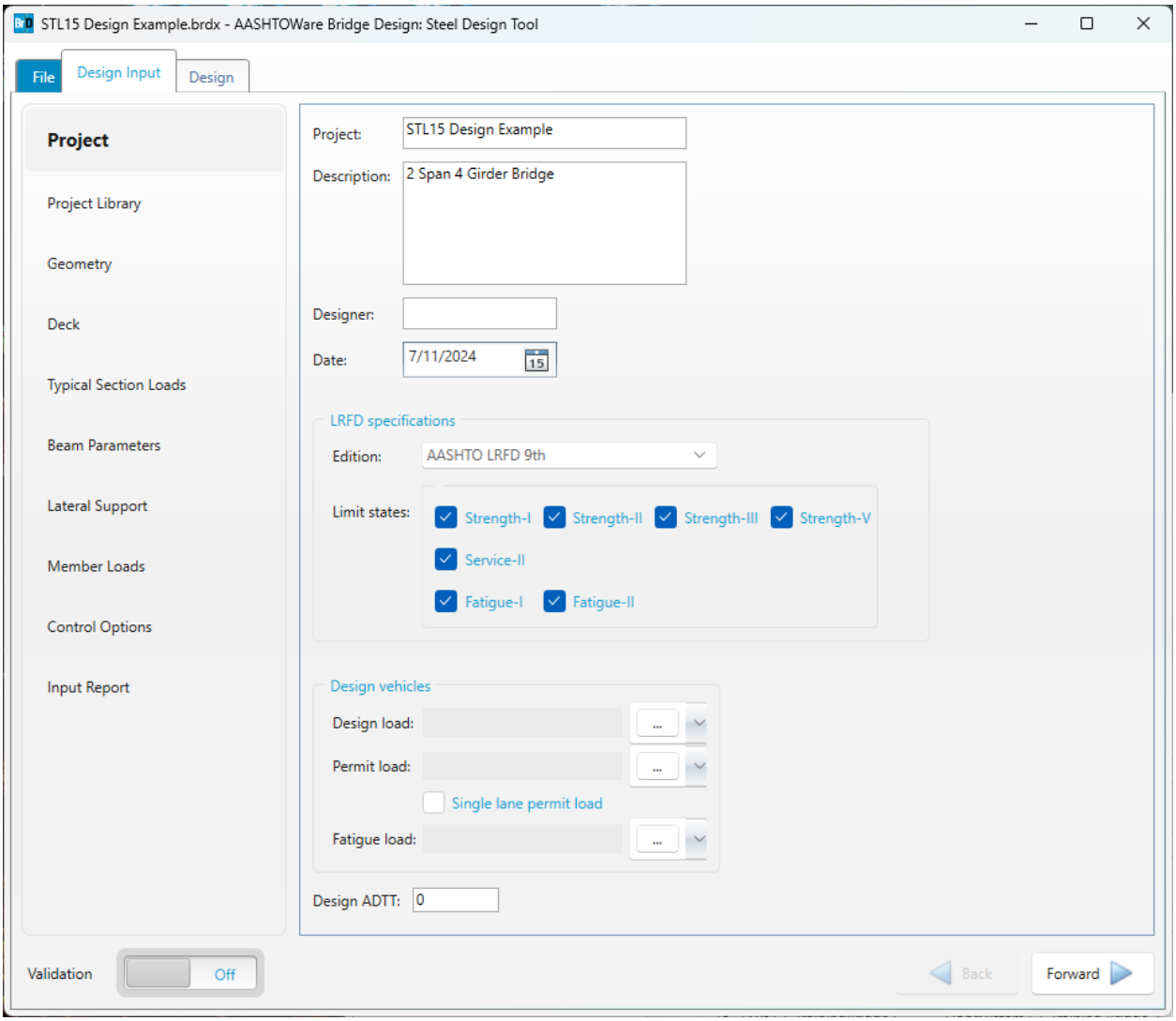
Before proceeding with **Design Input** return to the **File** tab and click **Save As** to rename the file from **New Project** to **STL15 Design Example**.



The new file name will appear in the program title bar and the program will again bring up the the **Design Input** with the **Project** input screen. The **Project** property will still say **New Project** and this will be changed in the next step. The **Project** property determines the name of the subfolder in the Documents\AASHTOWare\SteelDesign75\ folder where design run output files will be stored. In the bottom left corner of the program window, there is a **Validation** button that enables input validation. When validation is enabled, the program will mark sections and input boxes with missing or incorrect information. For the purposes of this example, the **Validation** will be disabled during input and will be enabled after all input is entered to verify that there are no validation errors.

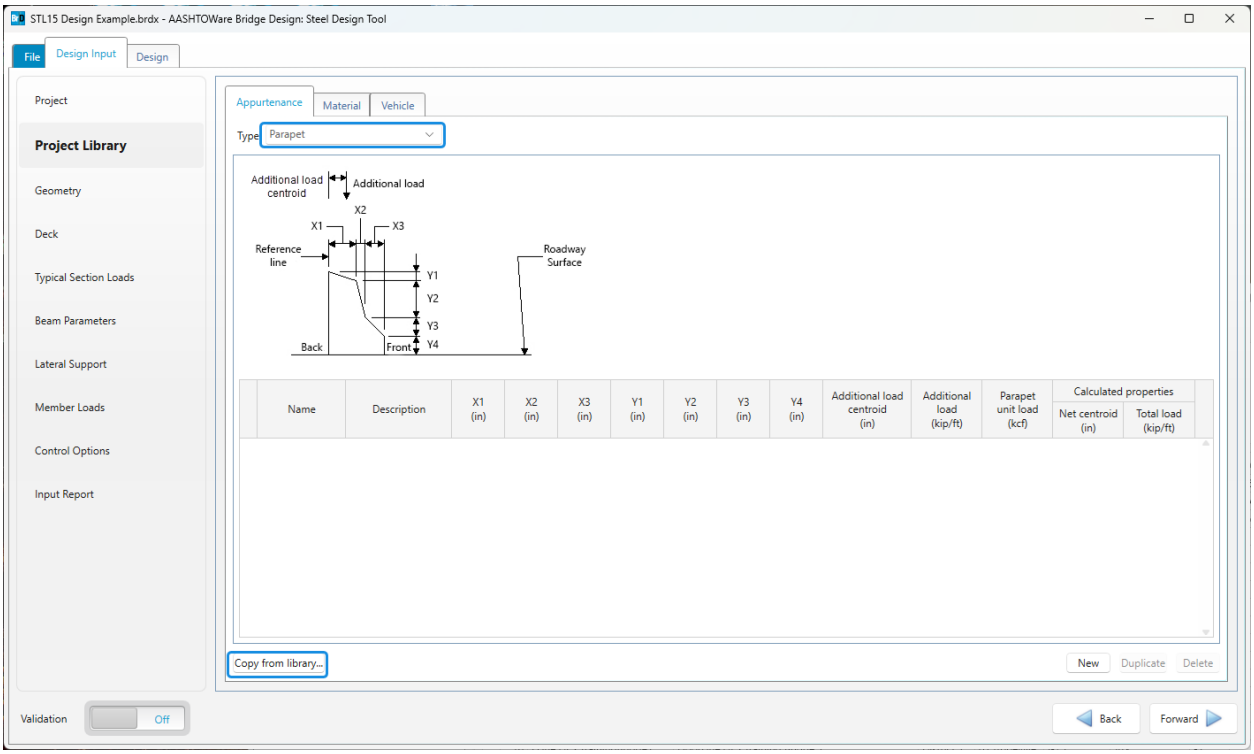
Design Input | Project

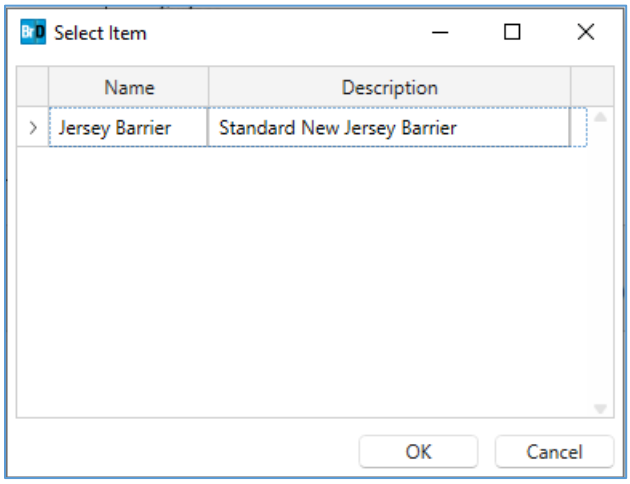
On the Design Input | **Project** input screen, enter the data as shown below.

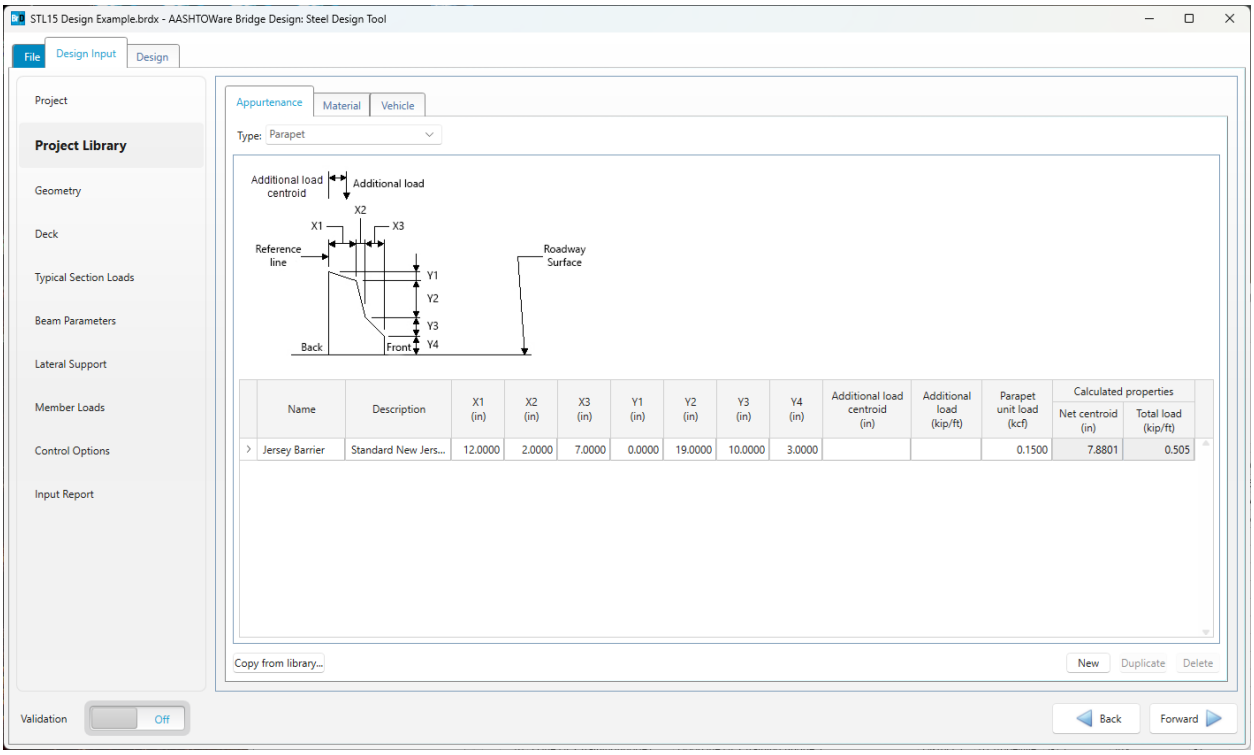


Design Input | Project Library | Appurtenance

On the **Apputenance** tab, select **Parapet** for **Type** from the drop down menu and click the **Copy from library** button to add a new parapet. Select the Jersey Barrier.

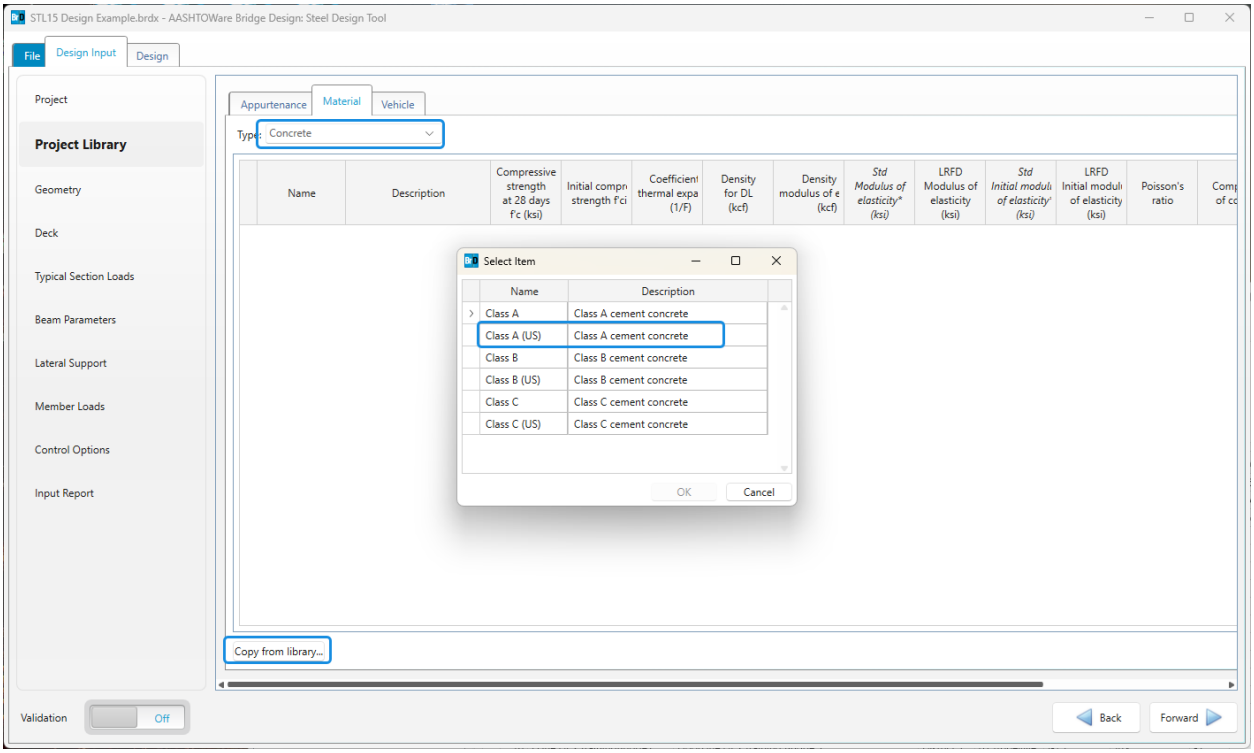




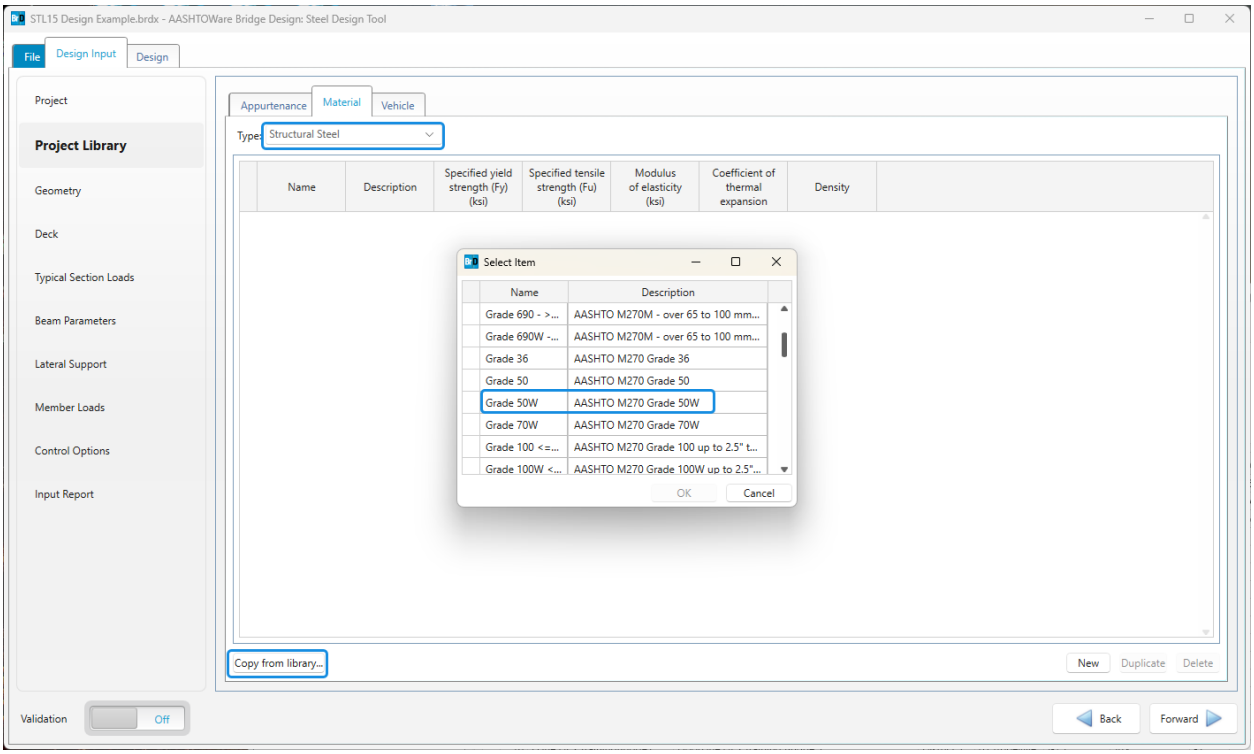

STL15 Design Example.brdx - AASHTOWare Bridge Design: Steel Design Tool
File
Design Input
Design
Project
Project Library
Geometry
Deck
Typical Section Loads
Beam Parameters
Lateral Support
Member Loads
Control Options
Input Report
Appurtenance
Material
Vehicle
Type: Parapet
Additional load centroid
Additional load
X1
X2
X3
Reference line
Roadway Surface
Y1
Y2
Y3
Y4
Back
Front
Name
Description
X1 (in)
X2 (in)
X3 (in)
Y1 (in)
Y2 (in)
Y3 (in)
Y4 (in)
Additional load centroid (in)
Additional load (kip/ft)
Parapet unit load (kcf)
Calculated properties
Net centroid (in)
Total load (kip/ft)
Jersey Barrier
Standard New Jers...
12.0000
2.0000
7.0000
0.0000
19.0000
10.0000
3.0000
0.1500
7.8801
0.505
Copy from library...
New
Duplicate
Delete
Validation
Off
Back
Forward

Design Input | Project Library | Material

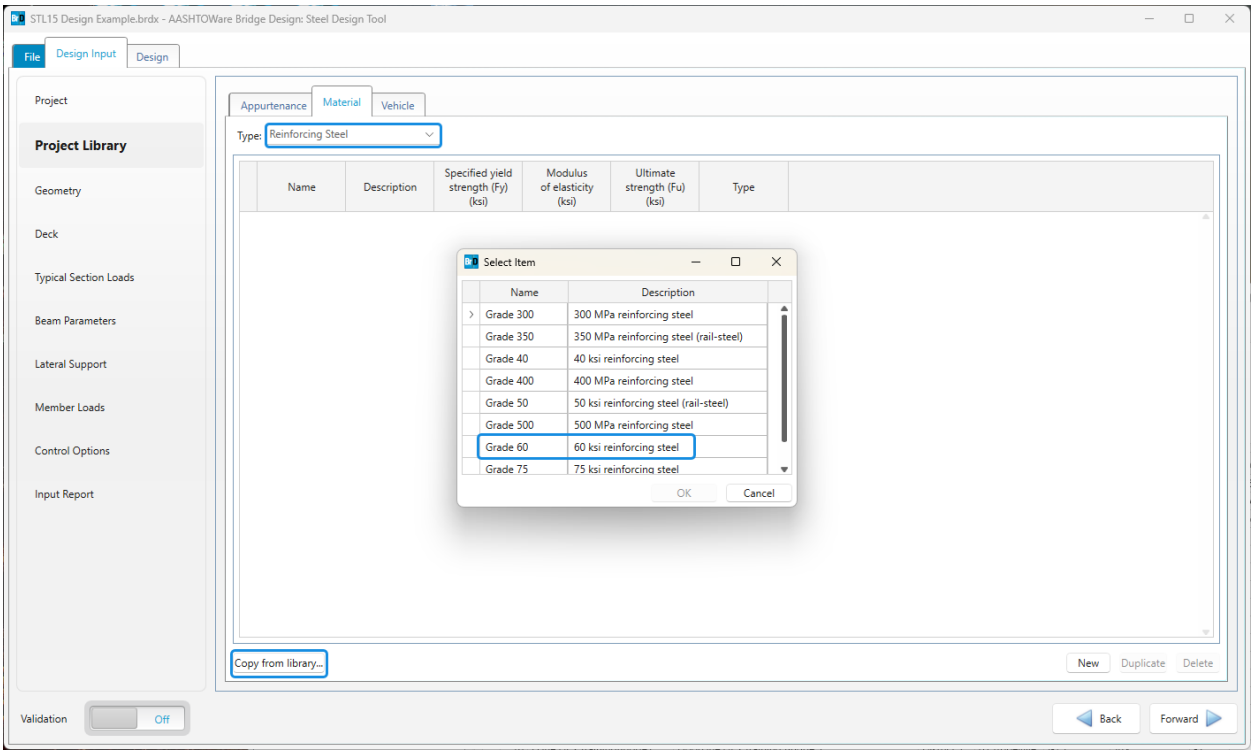
On the **Material** tab, select **Concrete** for **Type** from the drop down menu and click the **Copy from library** button to copy the **Class A (US)** concrete material definition from **File | Library** to the **Project Library**.



On the **Material** tab, select **Structural Steel** for **Type** from the drop down menu and click the **Copy from library** button to copy the **Grade 50W** steel material definition from **File | Library** to the **Project Library**.

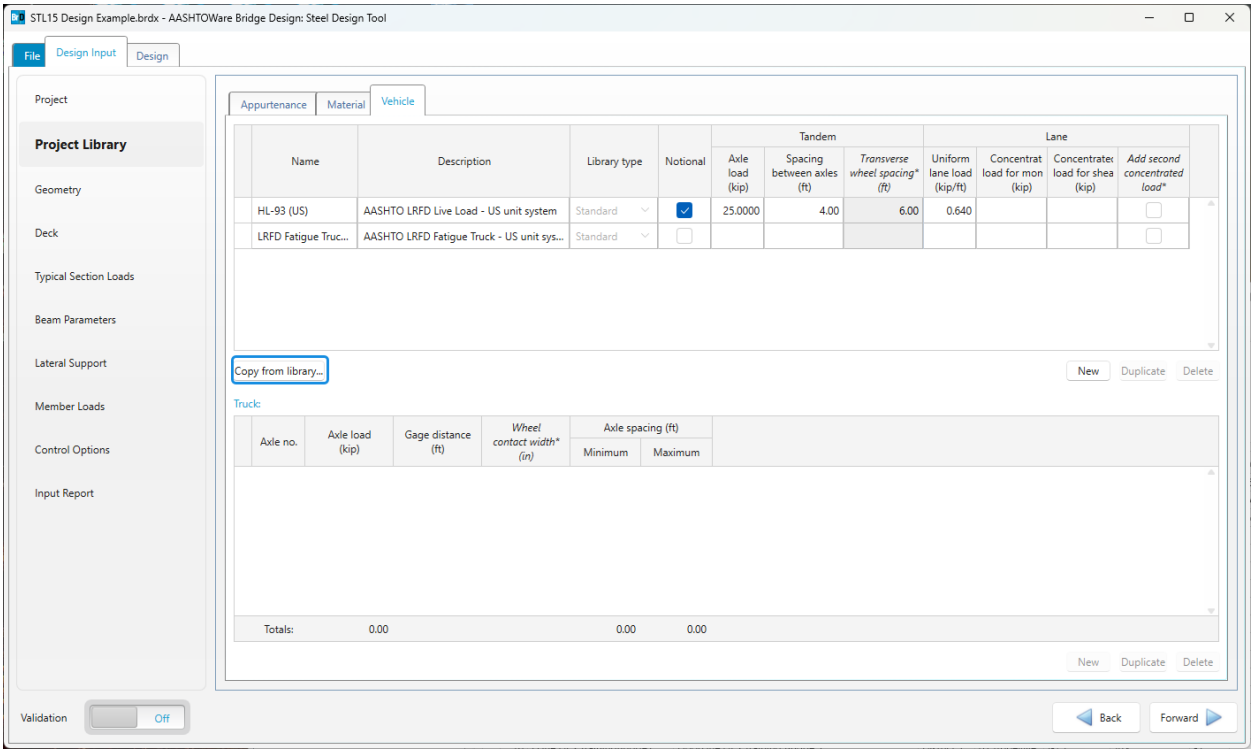


On the **Material** tab, select **Reinforcing Steel** for **Type** from the drop down menu and click the **Copy from library** button to copy the **Grade 60** reinforcing steel material definition from **File | Library** to the **Project Library**.



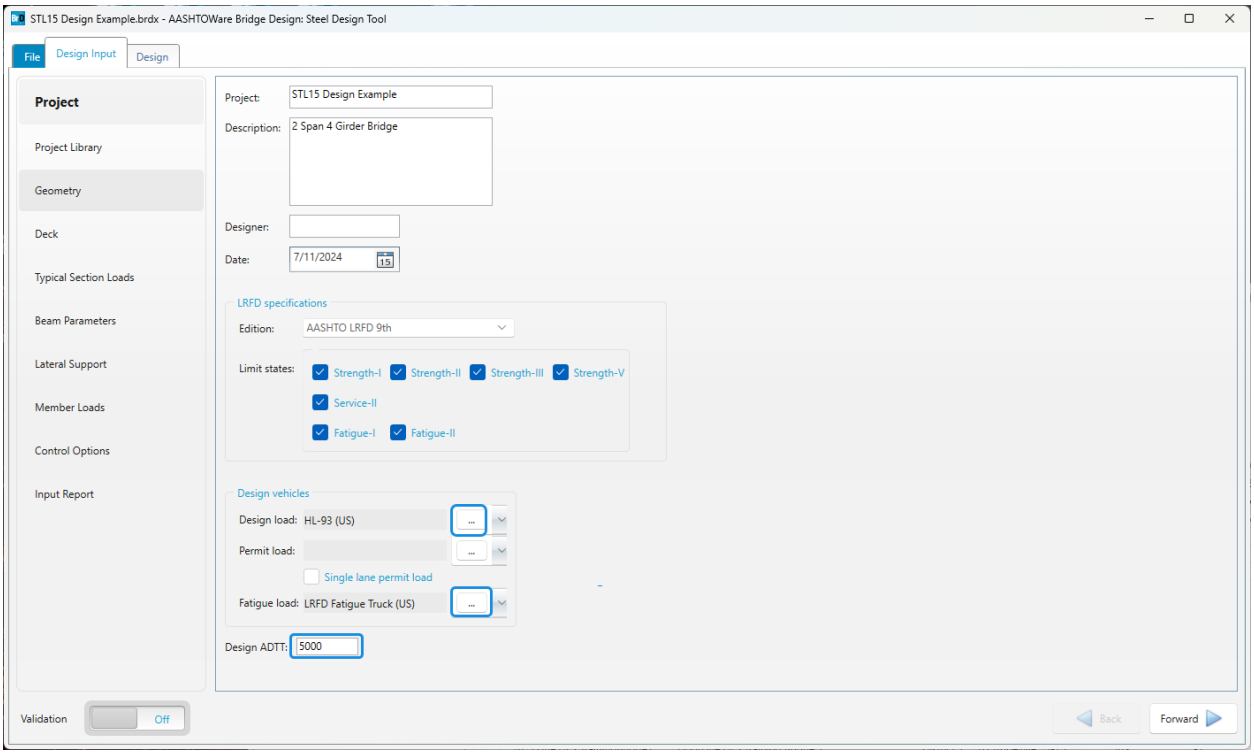
Design Input | Project Library | Vehicle

On the **Vehicle** tab, copy the HL-93 (US) and LRFD Fatigue Truck (US) vehicle definitions from library.



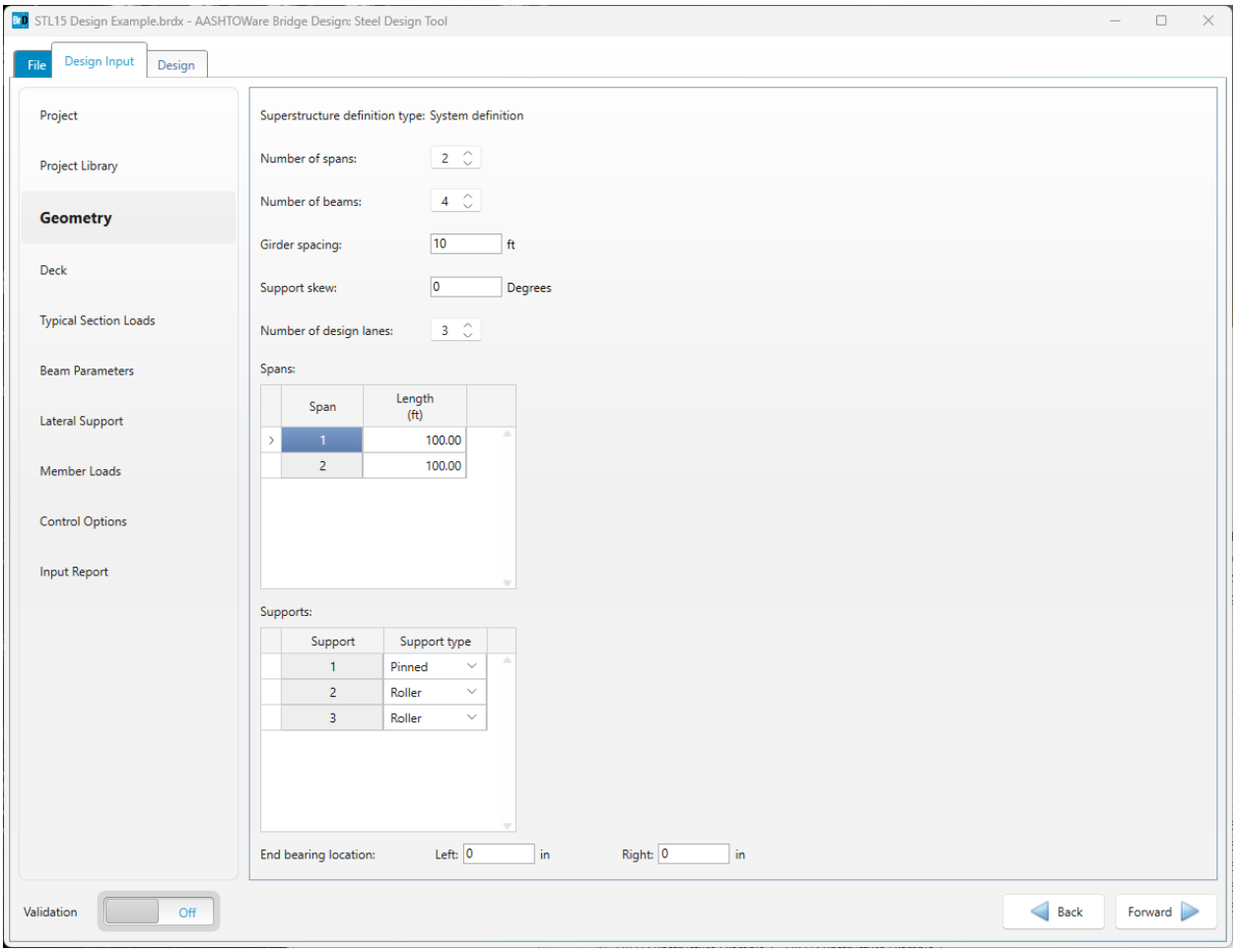
Design Input | Project

Return to the **Project** tab to define the vehicles for the girder design. Click on the ellipsis button to assign the HL-93 (US) vehicle as the design load and the LRFD Fatigue Truck (US) as the fatigue load. Leave the permit load blank. Define the design average daily truck traffic as 5000.



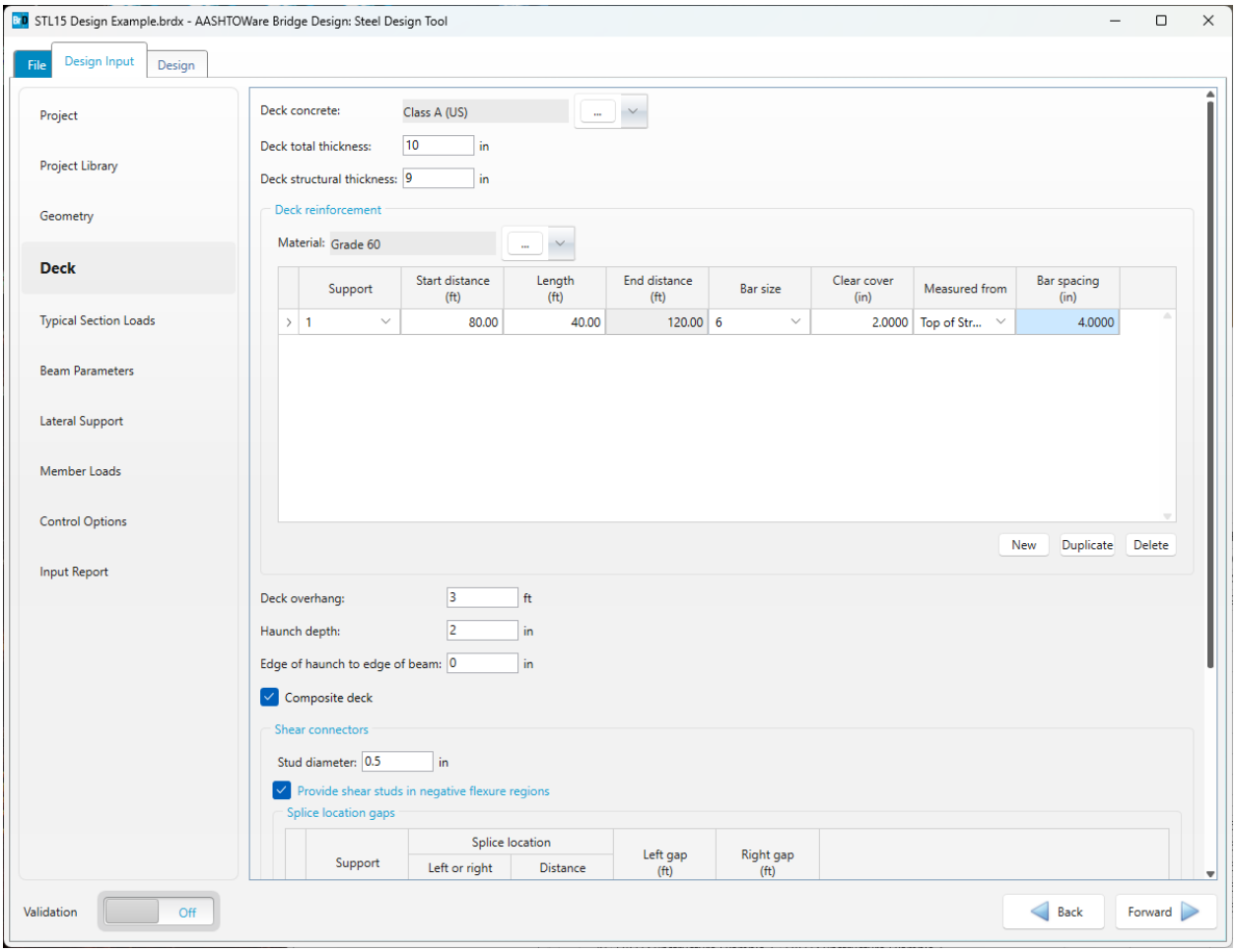
Design Input | Geometry

On the **Geometry** tab, enter the data as shown below. Depending on your screen resolution, scroll down to enter the **Support** information.



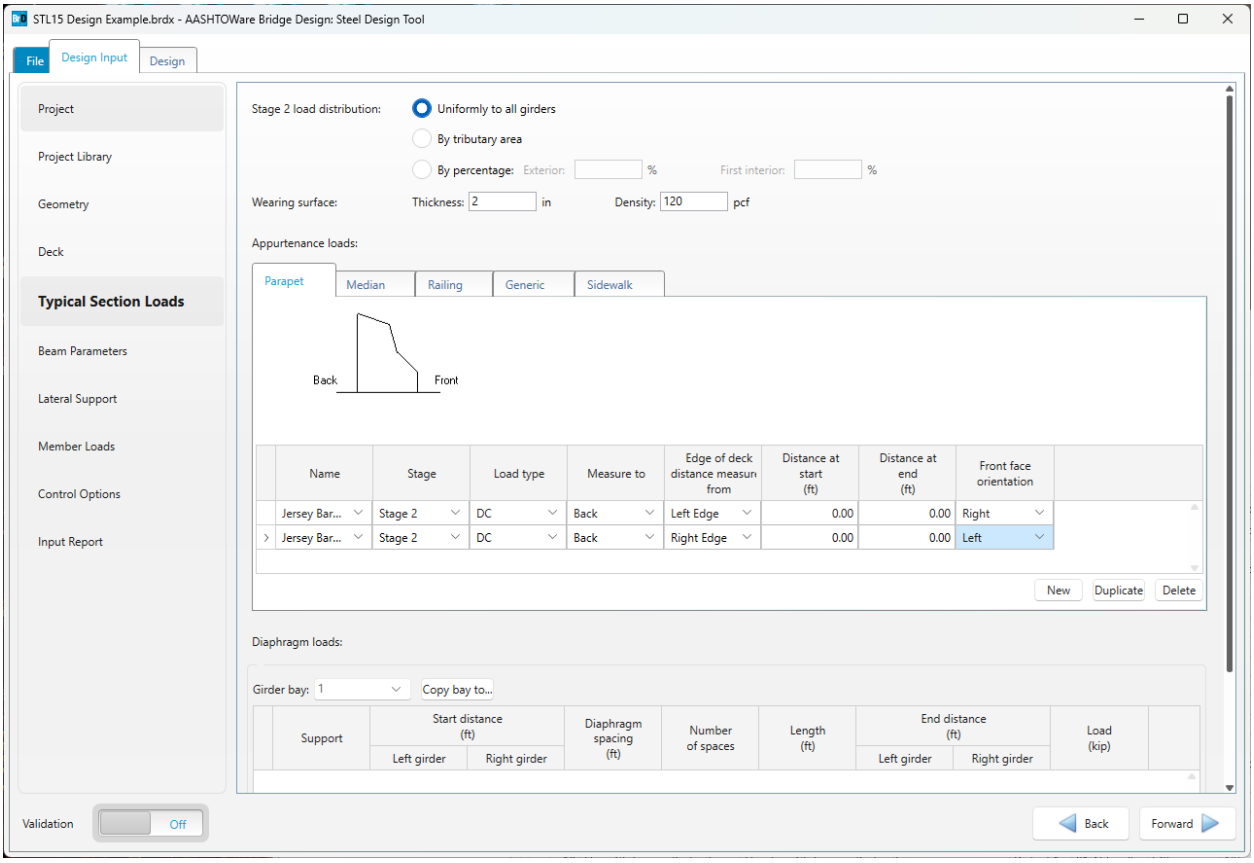
Design Input | Deck

On the **Deck** tab, enter the data as shown below. The **Splice location gaps** table can be used to input regions where the program should avoid placing shear studs. For this example, leave this table empty.

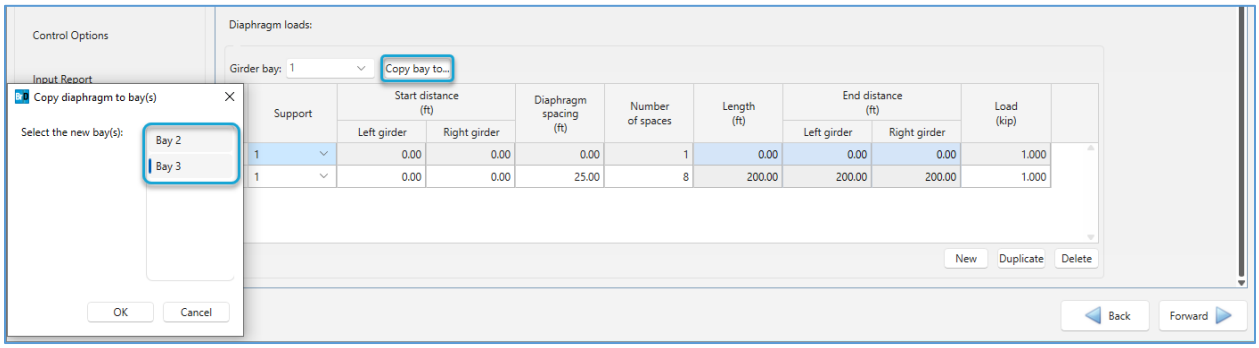


Design Input | Typical Section Loads

On the **Parapet** tab in **Typical Section Loads**, enter the data as shown below.

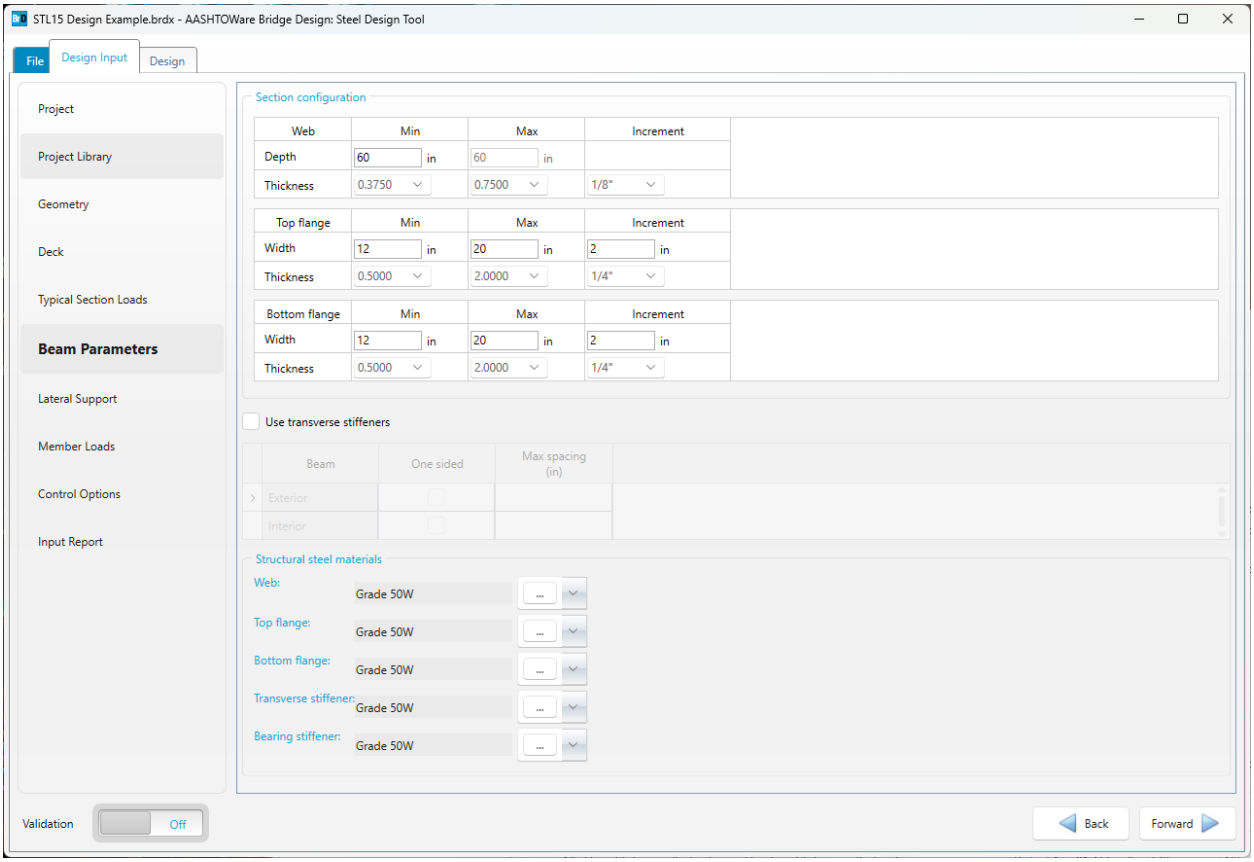


Make sure to scroll to the bottom of the page to define the diaphragm loads. Input the diaphragm loads as shown for Girder Bay 1 and use the **Copy bay to…** button to copy the loads to Girder Bay 2 and Girder Bay 3.



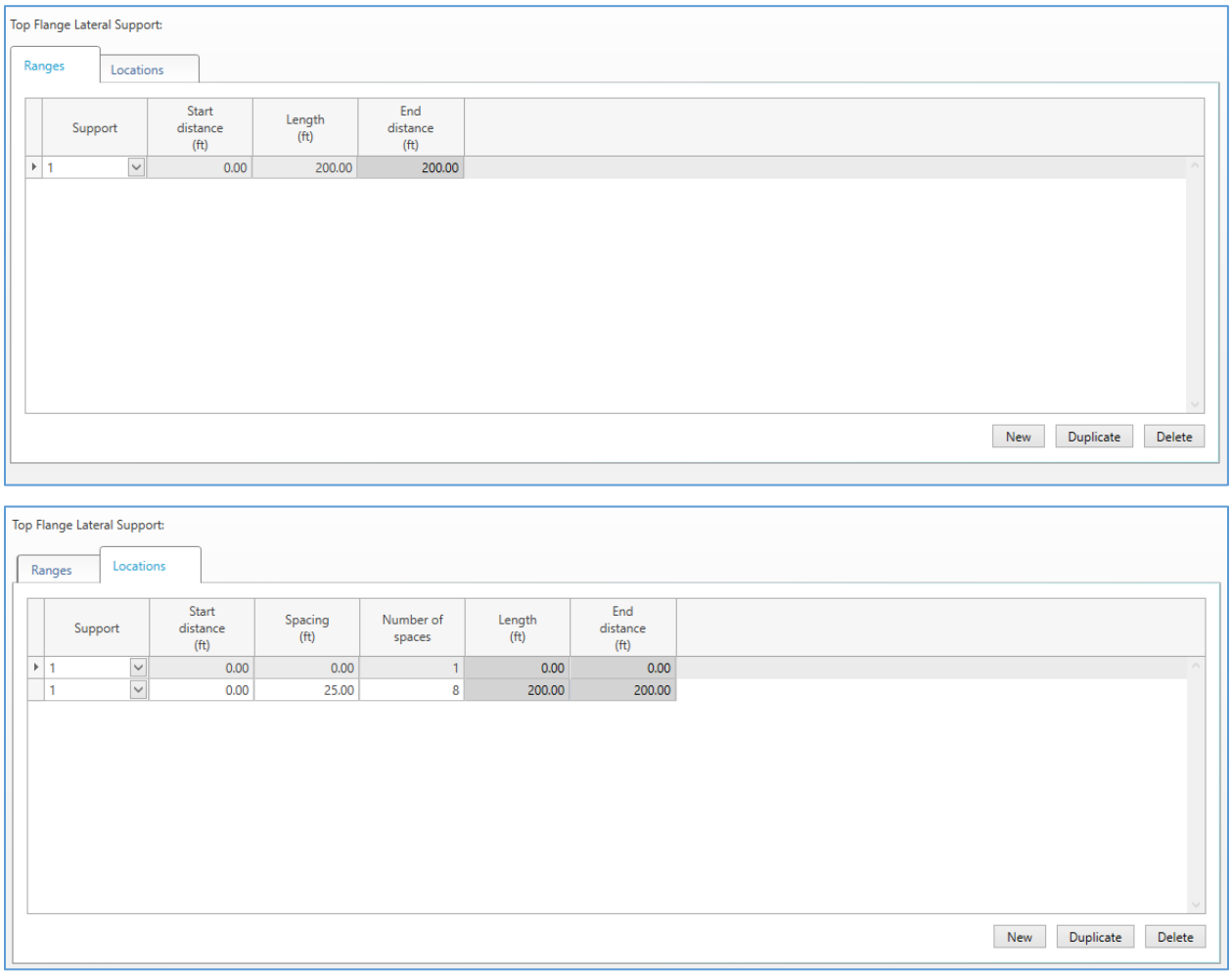
Design Input | Beam Parameters

In the **Beam Parameters** input section, enter the data as shown below. All of the plate dimensions except for the web depth can be designed. Providing a larger range of values for any given parameter can help the design tool converge on a solution. The program will design transverse stiffeners for shear resistance if the **Use transverse stiffeners** button is selected. This example does not use this option.



Design Input | Lateral Support

In the **Lateral Support** input section, enter the data as shown below. Top flange lateral support ranges are regions where the top flange is continuously laterally supported and top flange lateral support locations are discrete points of lateral support.These entries define the top flange lateral support for Stage 2 and Stage 3.

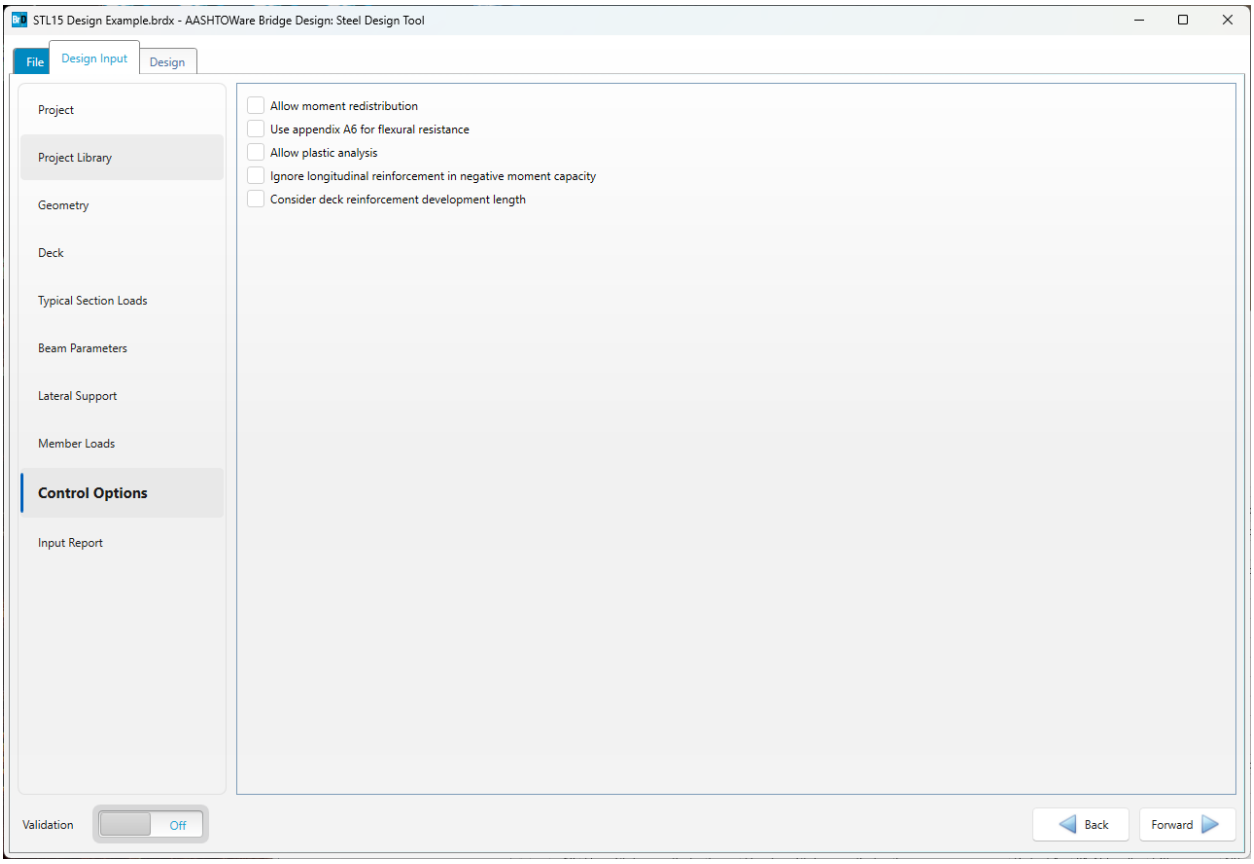


Design Input | Member Loads

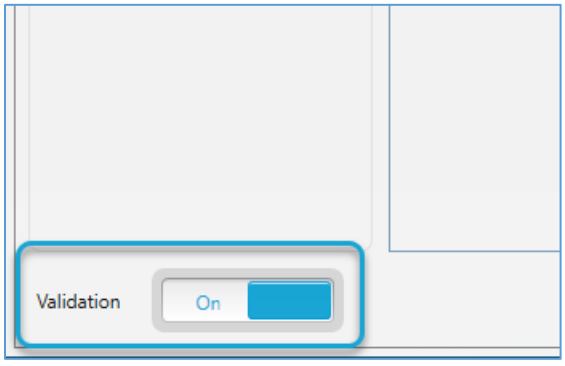
There are no member loads assigned in this example. Member concentrated loads, member distributed loads and pedestrian loads could be assigned here.

Design Input | Control Options

The **Control Options** input section provides options for analysis and design. For this example, leave the options as is.

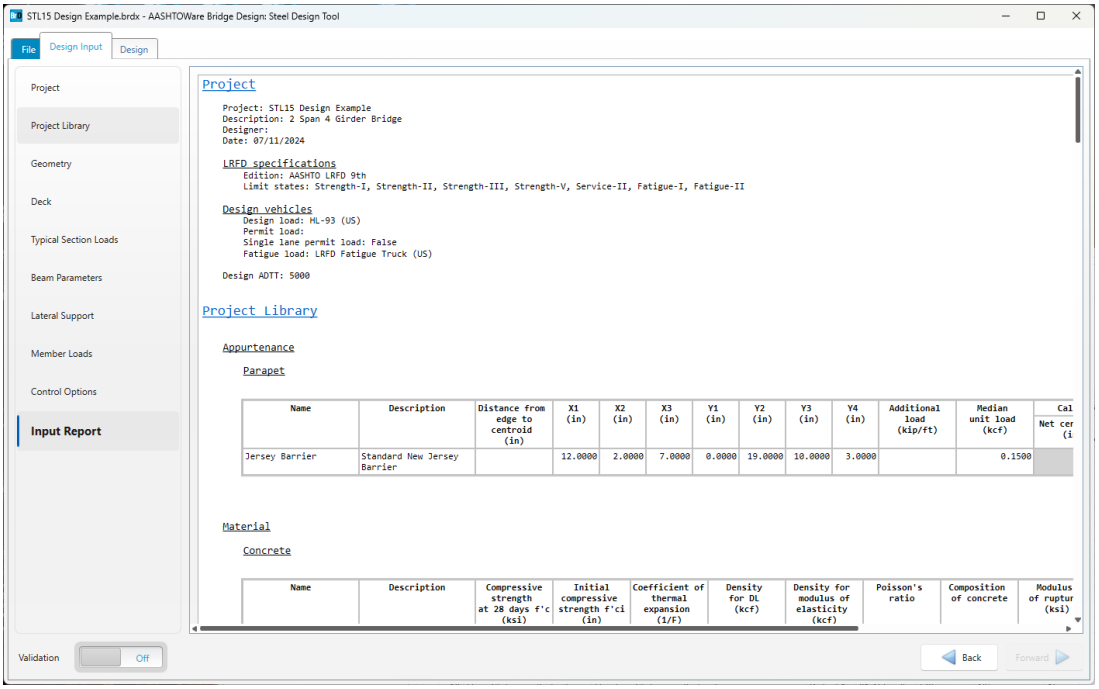


At this point, all design data has been defined. Turn **Validation** on and ensure that there are no validation error marks displayed next to the input section. Otherwise, go back to these sections and resolve the errors.



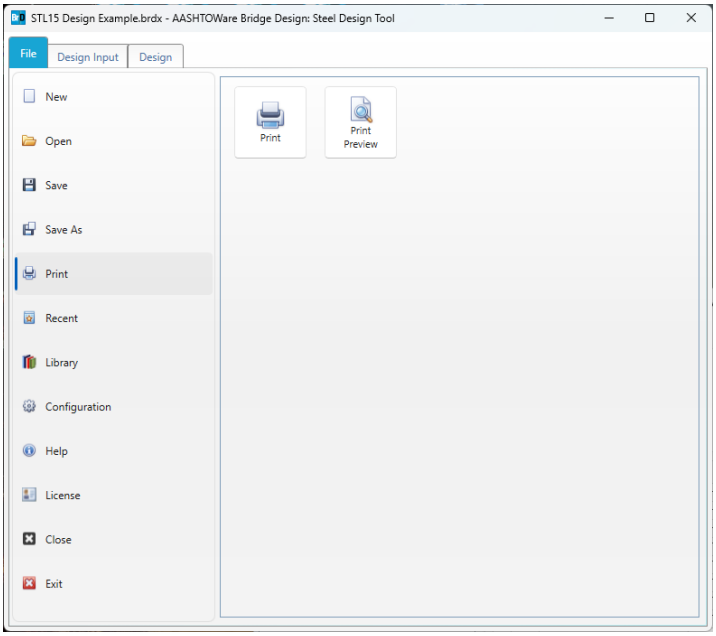
Design Input | Input Report

The **Input Report** section provides a detailed report of the input data.



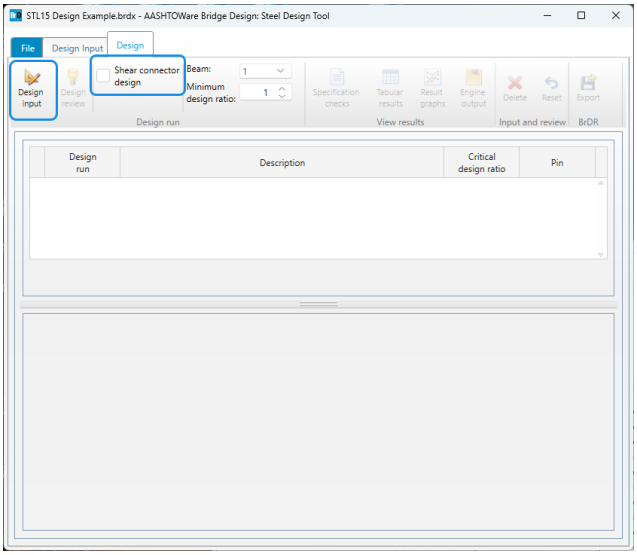
File | Print

The **Print** and **Print Preview** buttons in the **File | Print** section apply to the **Input Report**.



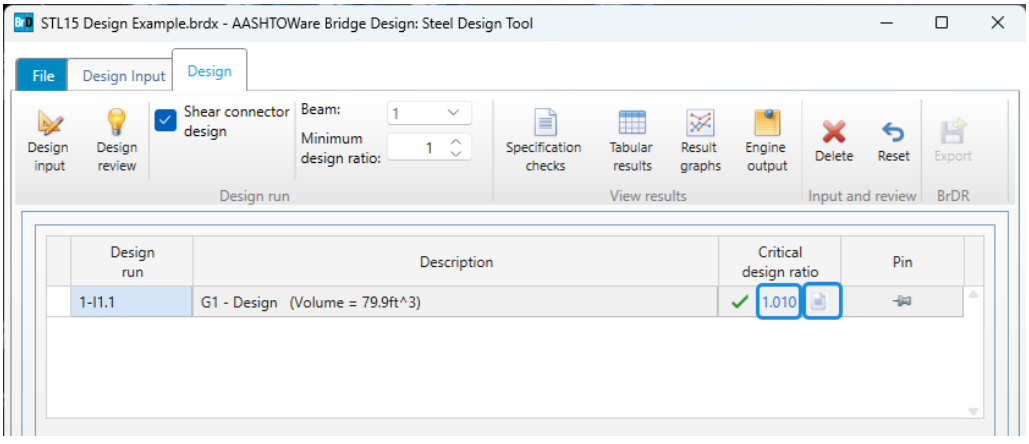
Design | Design Input

After the input data is entered and reviewed, **Design Input** run can be performed by clicking on the **Design Input** button located on the **Design ribbon**. **Design Input** run is based on the input data and produces a design that is displayed in the **Design Run** grid with a brief description and values of the critical design ratios. Select the checkbox for **Shear connector design** to design the shear connectors along with the girder in the composite regions. The beam dropdown is the selection for which girder the program will design and the input for minimum design ratio defines the target design ratio for which the girder will be designed.



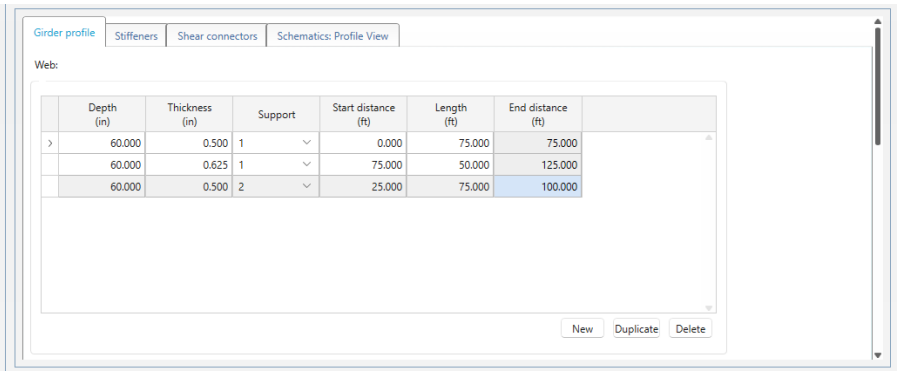
Design | Results Table

The results of the **Design Input** run are displayed in the table. The user can click on the **Critical design ratio** to open the **Analysis engine feedback report**, and on the page icon next to the **Critical design ratio** to open the **Specification Check Summary**.



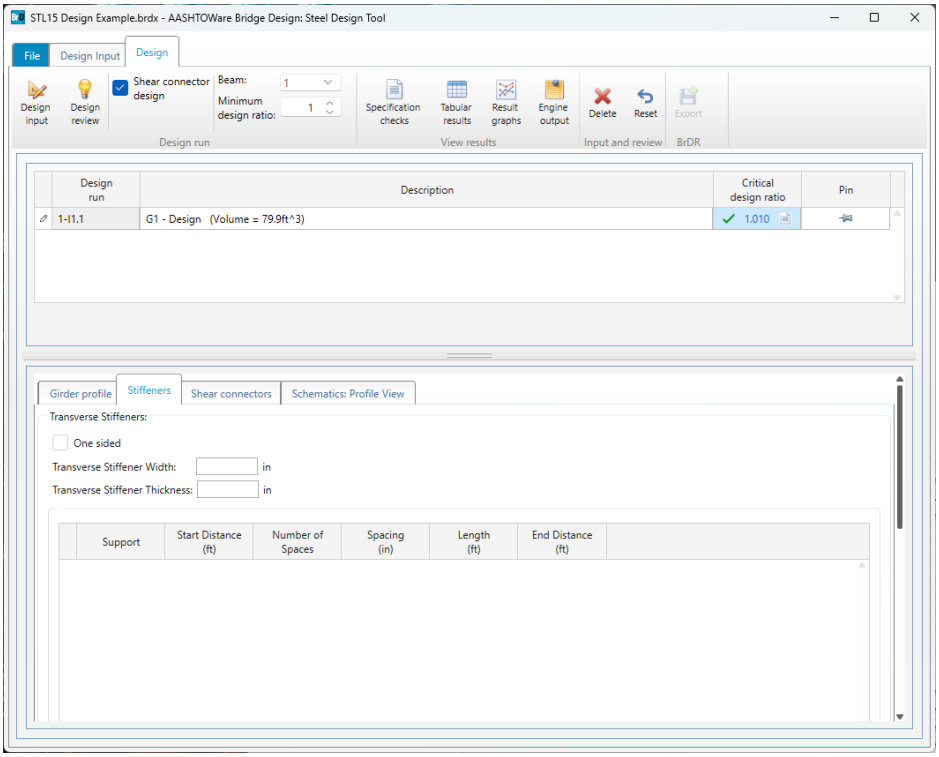
Design | Girder Profile

The **Girder Profile** tab displays the ranges for steel plates along the web and flanges. After a design input run is completed, these tables will display the program computed ranges. The user may modify these ranges and reanalyze the member using the **Design Review** option.



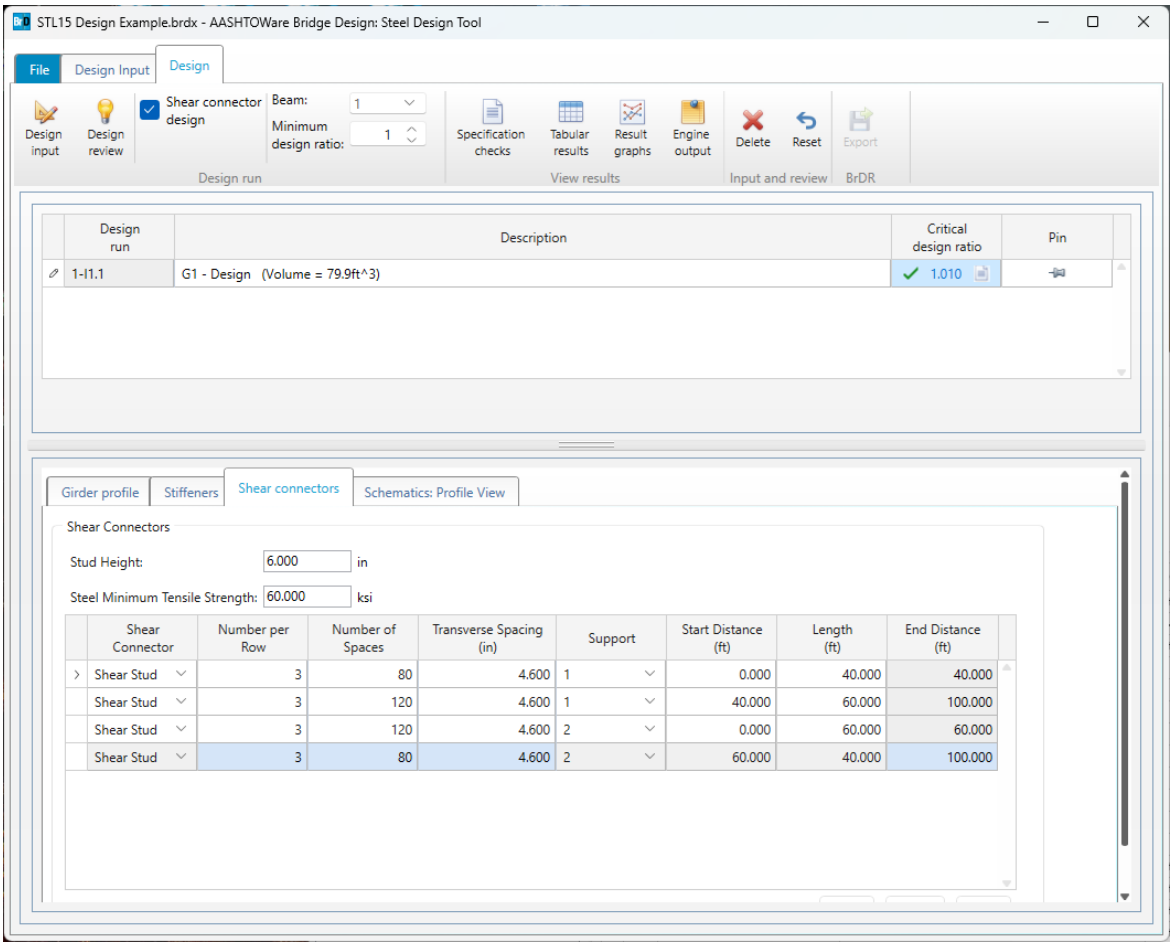
Design | Stiffeners

The **Stiffeners** tab displays the results of the stiffener design. This includes transverse stiffeners and bearing stiffeners. The transverse stiffeners will only be designed when the design input option to **Use Transverse Stiffeners** in the **Design Input | Beam Parameters** window is selected.



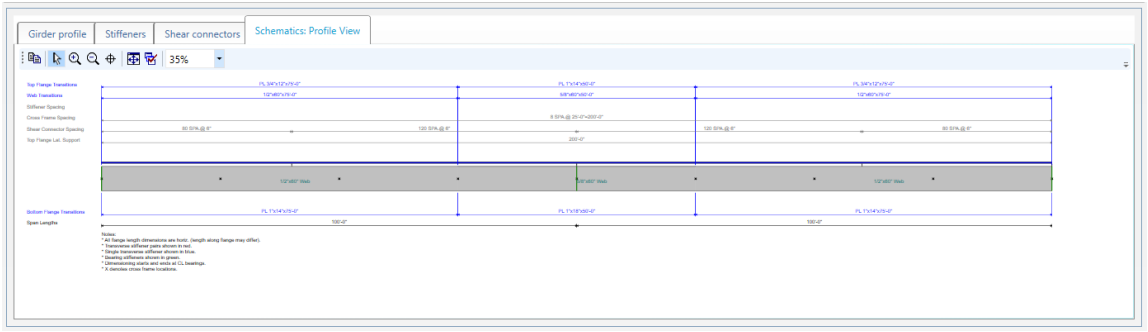
Design | Shear connectors

The **Shear connectors** tab shows the results of the shear connector design if enabled and if the member has composite regions.



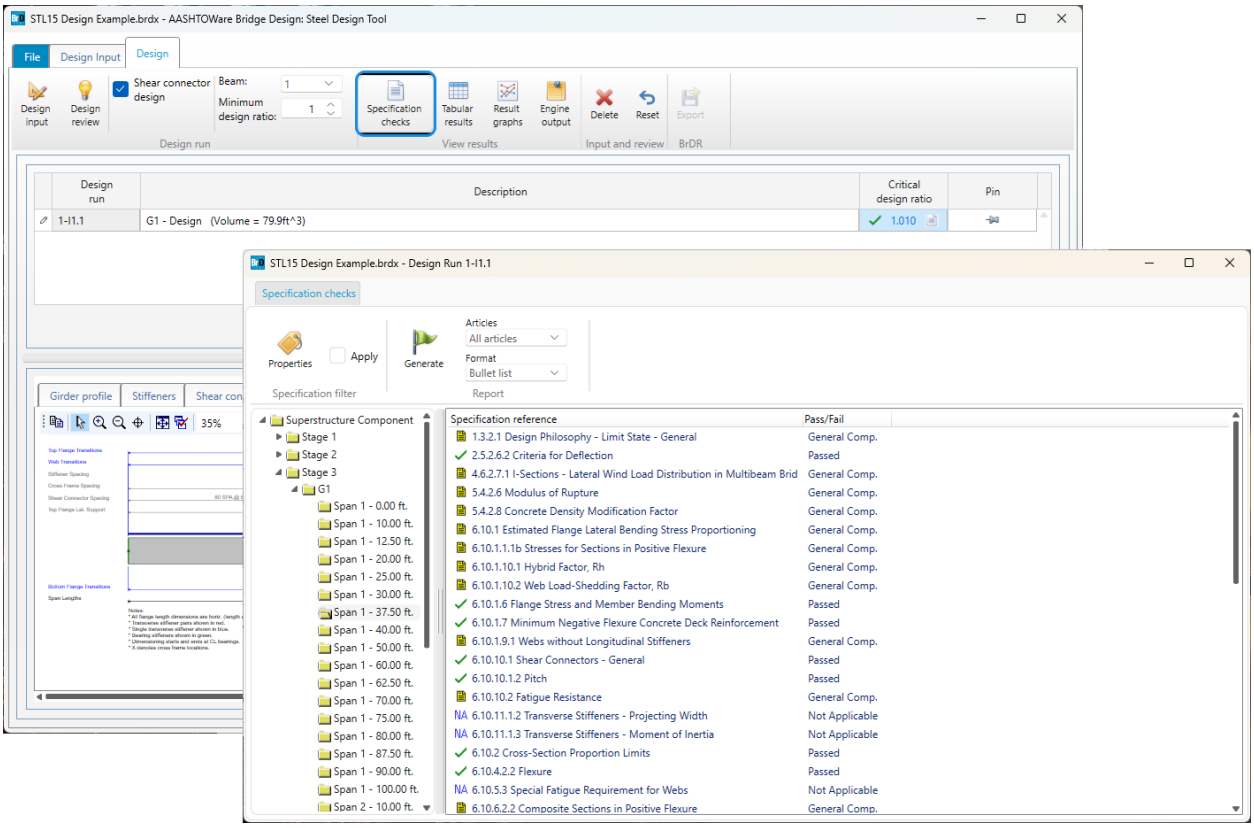
Design | Schematics: Profile View

The **Schematics: Profile View** tab shows a schematic of the girder design.



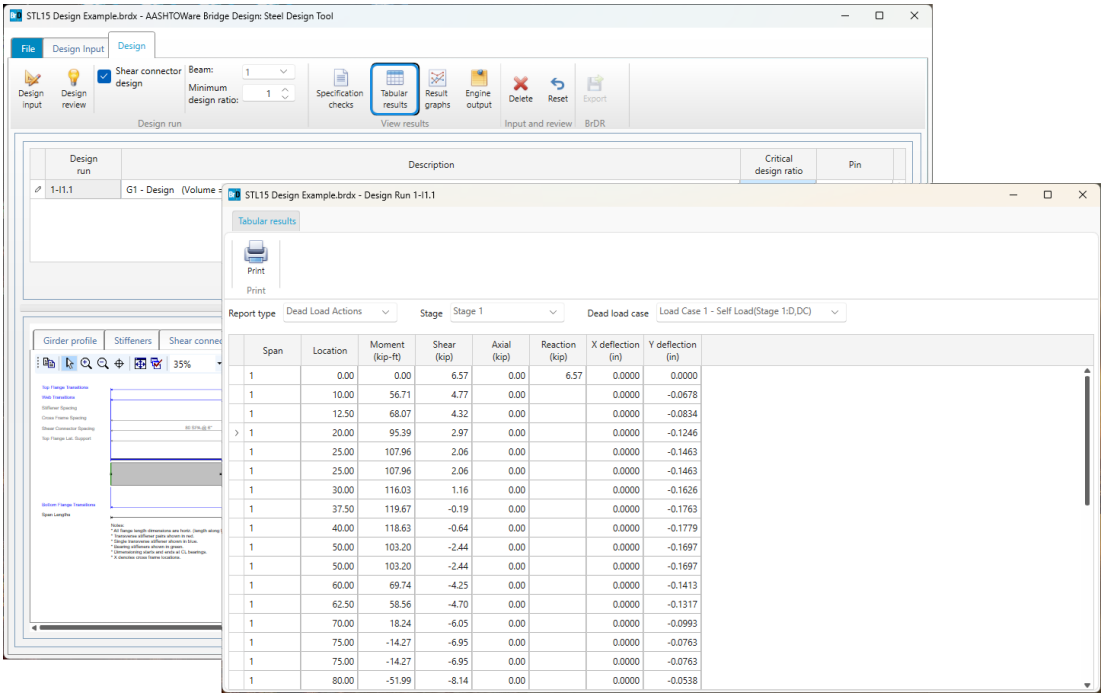
Design | Specification Check

To view the specification check results, click on the **Specification checks** button from the **View results** group of the **Design** ribbon.



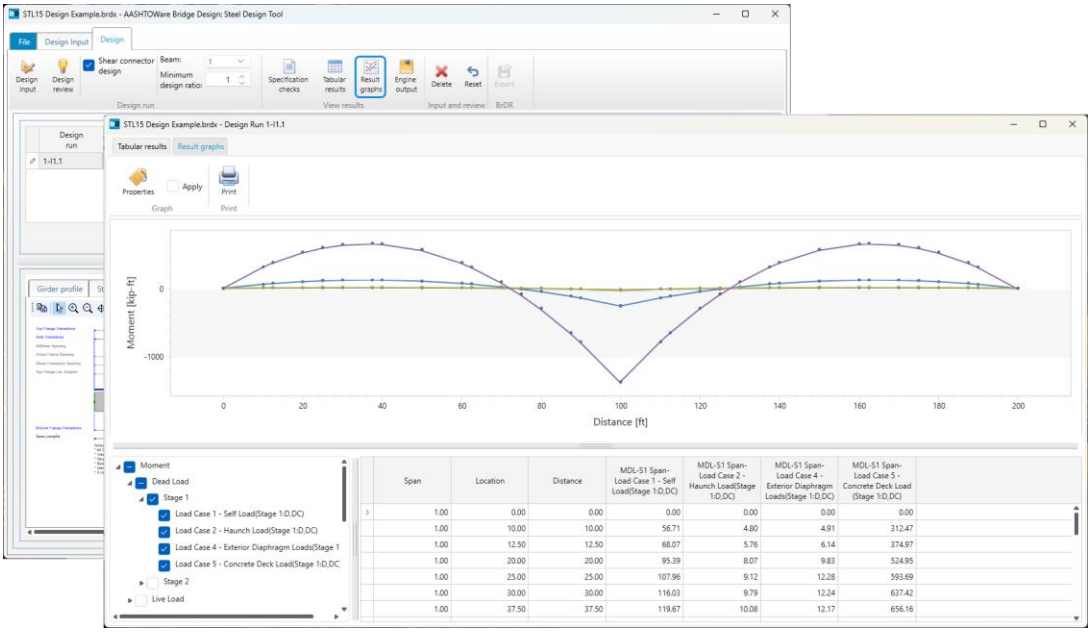
Design | Tabular Results

To view the tabular results, click on the **Tabular results** button from the **View results** group of the **Design** ribbon.



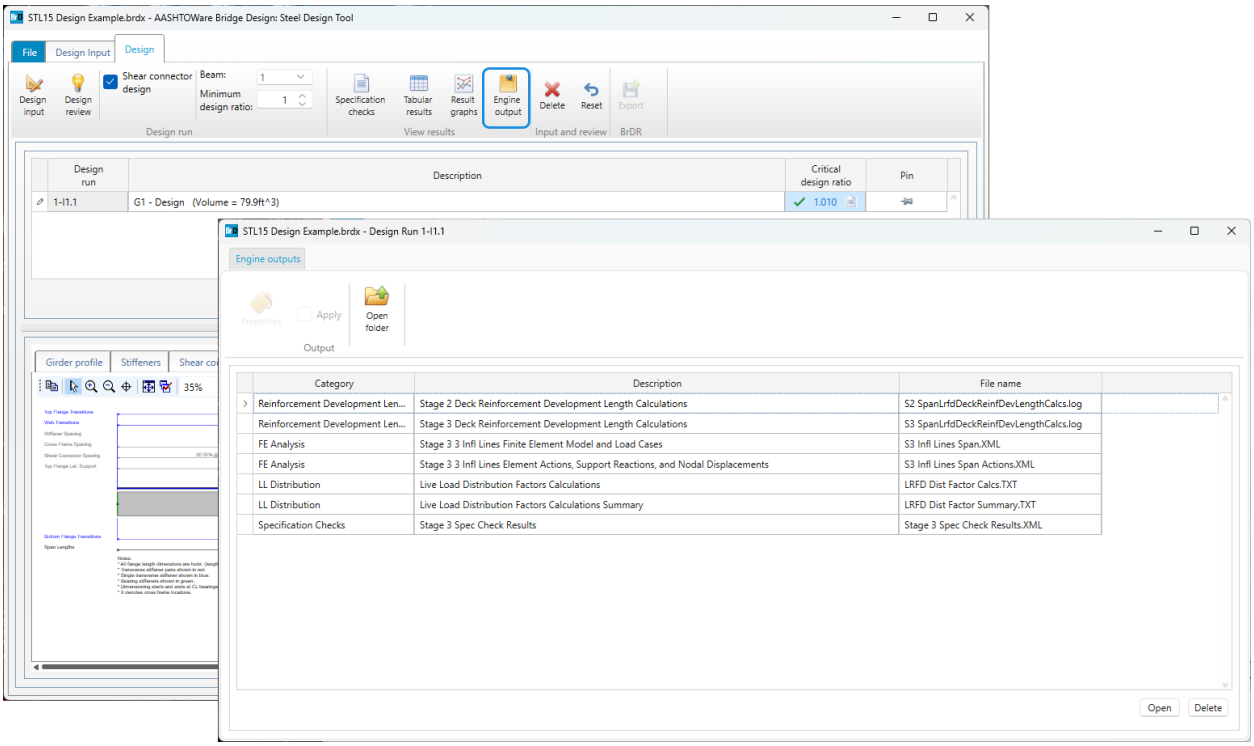
Design | Result Graphs

To view the result graphs, click on the **Result graphs** button from the **View results** group of the **Design** ribbon.



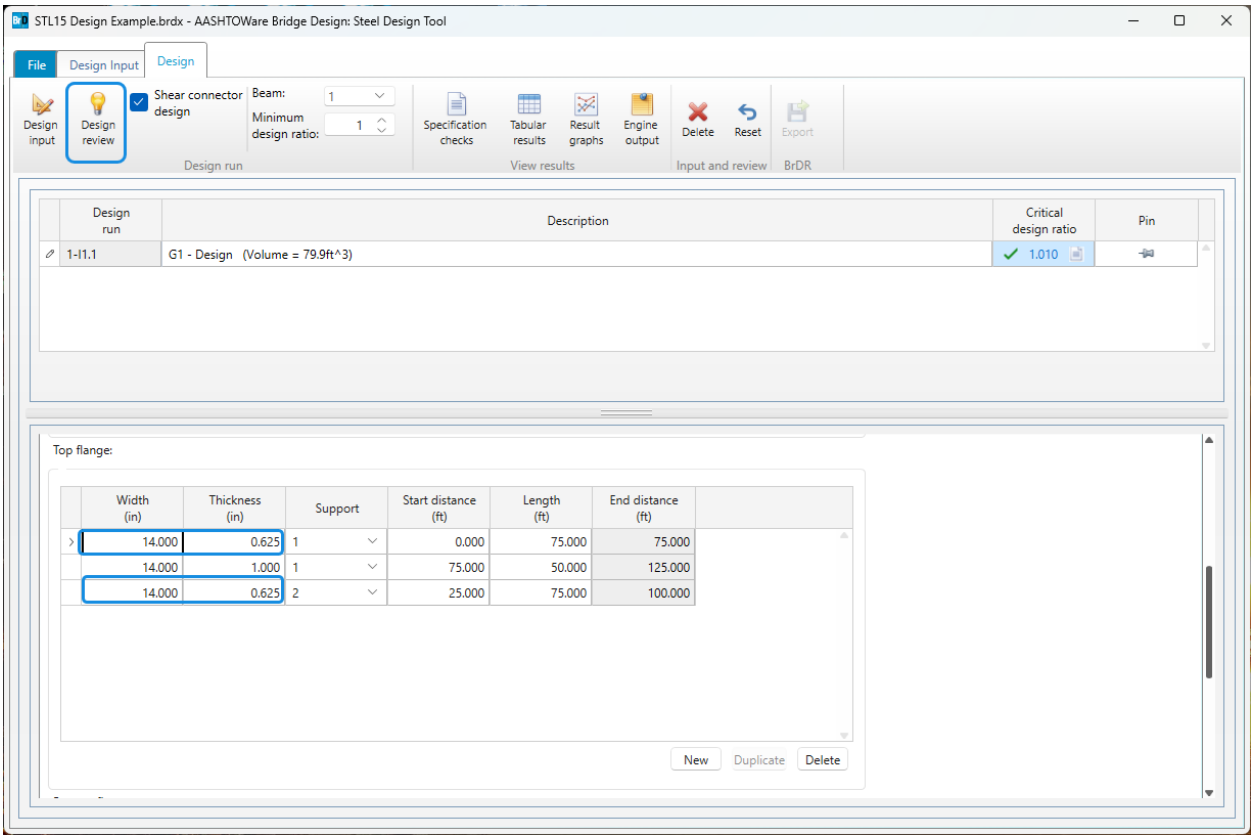
Design | Engine Outputs

To view the contents of the engine output files, click on the **Engine outputs** button from the **View results** group of the **Design** ribbon, and then double-click on the row corresponding to the required file.



Design | Design Review

To illustrate the ability of the program to adjust results of the **Design Input** run, modify the **Top flange** table to define a top flange width of 14 inches for all ranges and a thickness of 0.625 for the first and third range as shown below. Select **Design Review** to analyze this modified design.



After the program finishes performing the design review, it will add another row to the design run grid. The design review runs are indicated with an **R** displayed in the **Design run** column in contrast to an **I** shown in that column for design input runs. The results for the **Design review** runs are displayed and can be reviewed or further modified the same way as design input runs. Additional design input runs can be performed by modifying the input on the **Design Input** tab. Each of the design runs, either input or review, stores a copy of its design input data that is reloaded every time the design input run is selected in the design run grid.