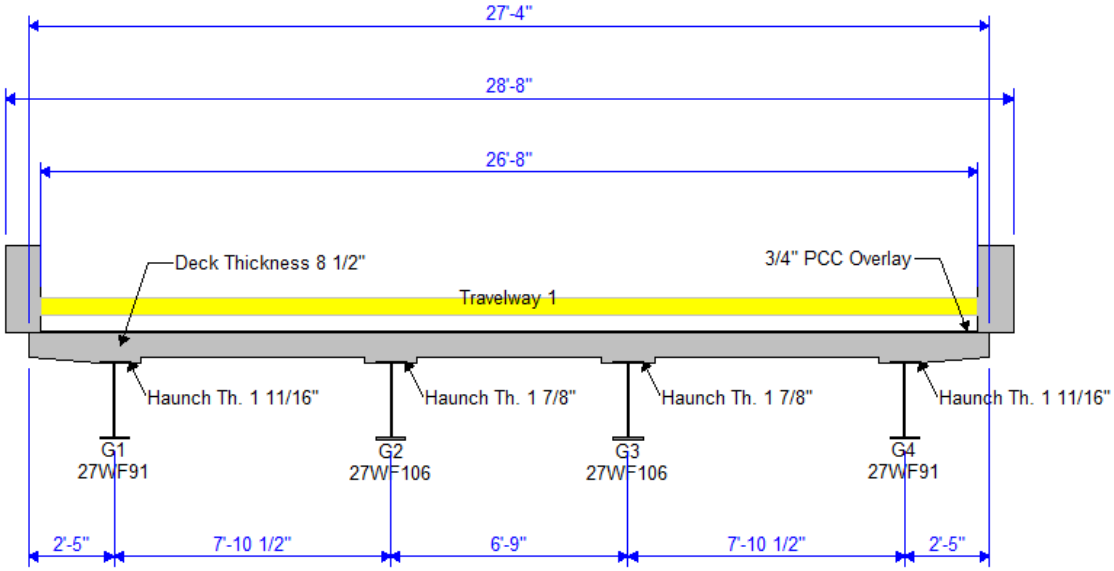
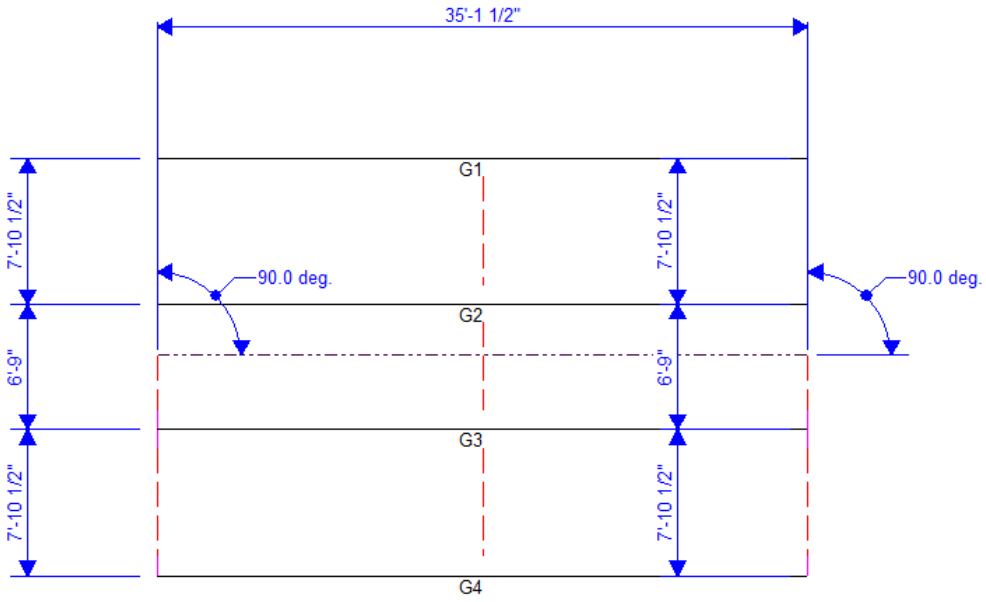
AASHTOWare BrDR 7.5.0

3D FEM Analysis Tutorial

Single Span Steel3D Example


27'-4"
28'-8"
26'-8"
Deck Thickness 8 1/2"
3/4" PCC Overlay
Travelway 1
Haunch Th. 1 11/16"
Haunch Th. 1 7/8"
Haunch Th. 1 7/8"
Haunch Th. 1 11/16"
G1
27WF91
G2
27WF106
G3
27WF106
G4
27WF91
2'-5"
7'-10 1/2"
6'-9"
7'-10 1/2"
2'-5"


35'-1 1/2"
G1
G2
G3
G4
7'-10 1/2"
6'-9"
7'-10 1/2"
90.0 deg.
90.0 deg.
7'-10 1/2"
6'-9"
7'-10 1/2"

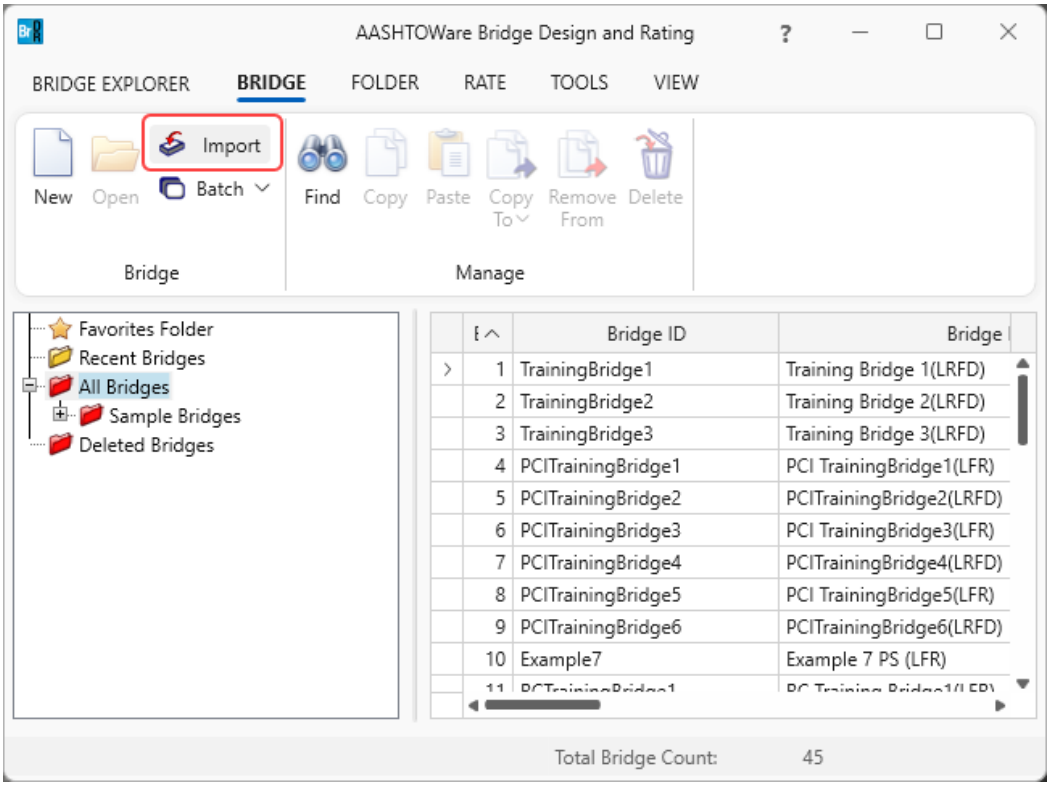
Topics Covered

- 3D analysis of a single span steel rolled beam bridge
- Diaphragm definitions
- 3D analysis settings
- 3D model

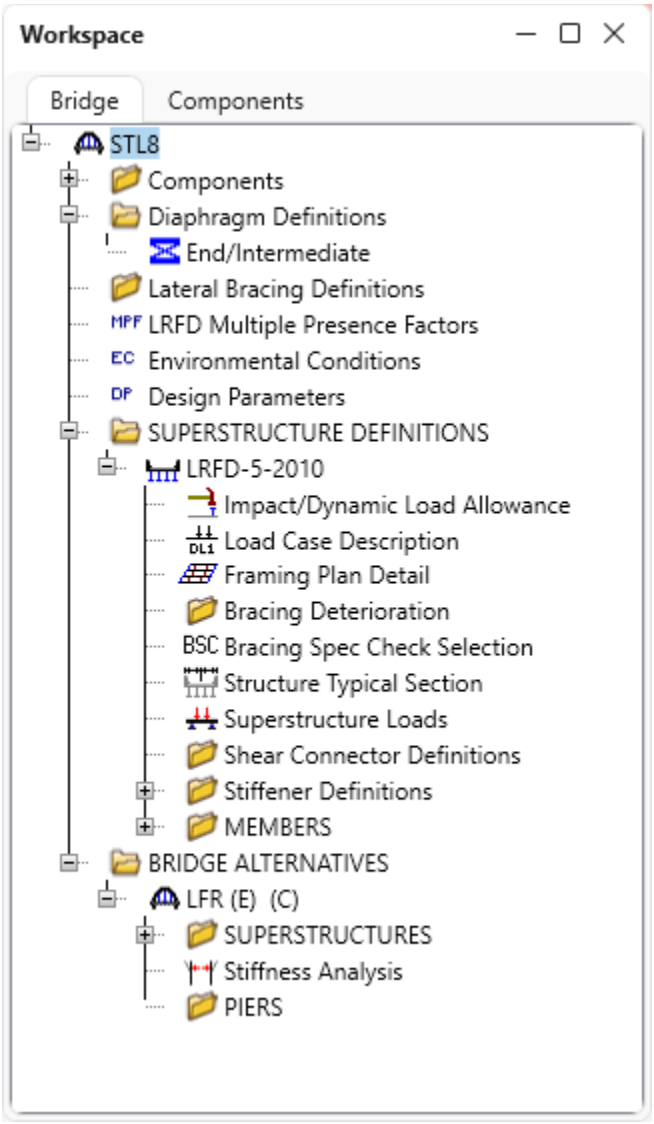
This example describes performing a 3D FEM analysis of a single span steel rolled beam structure.

3D analysis of a single span steel rolled beam bridge

From the **Bridge Explorer** window, import the **3DFEM2-Single-Span-Steel-3D-Example.xml** file provided with this tutorial into **BrDR**.

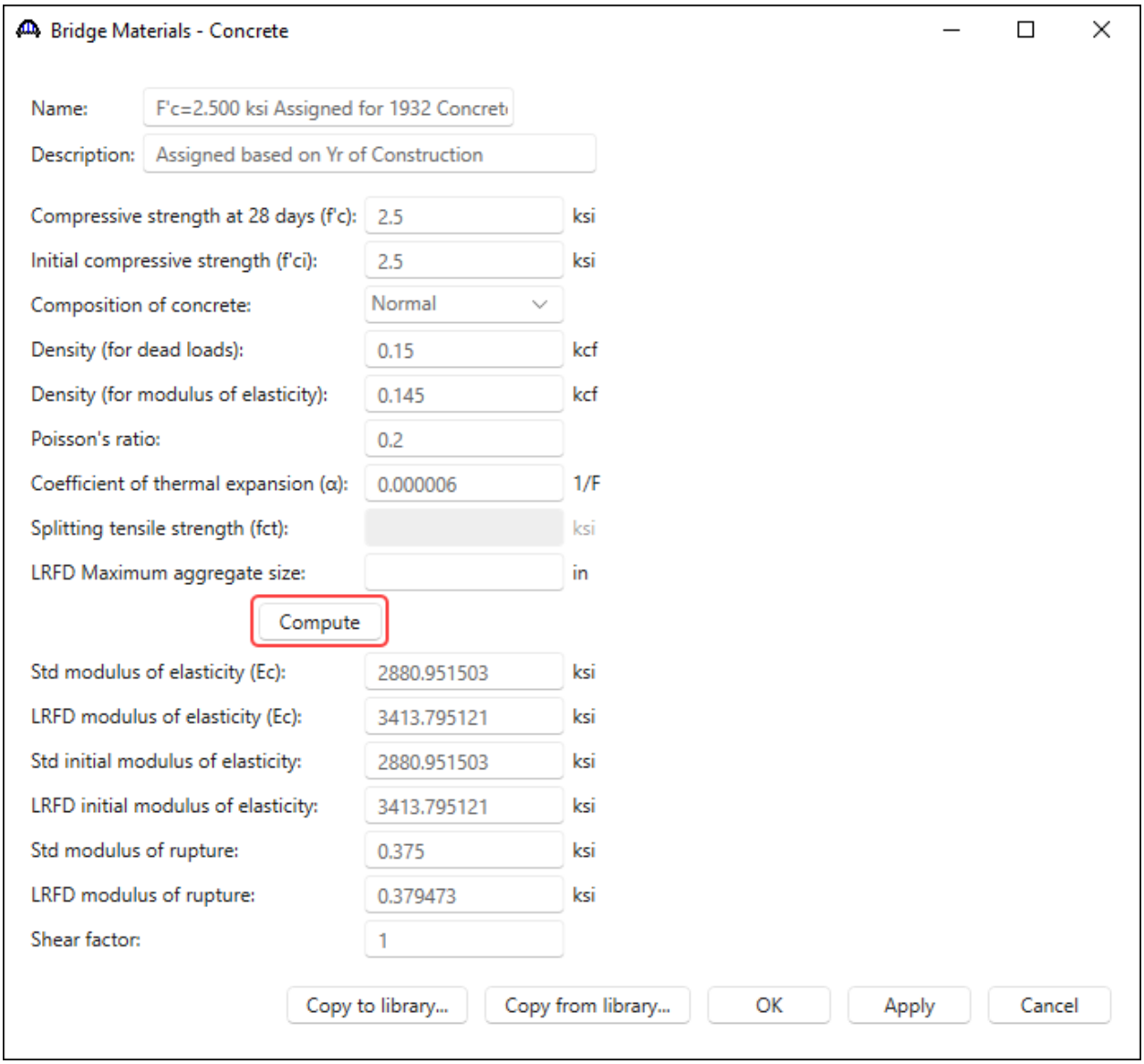


The partially expanded **Bridge Workspace (BWS)** tree is shown below.



Bridge Materials – Concrete

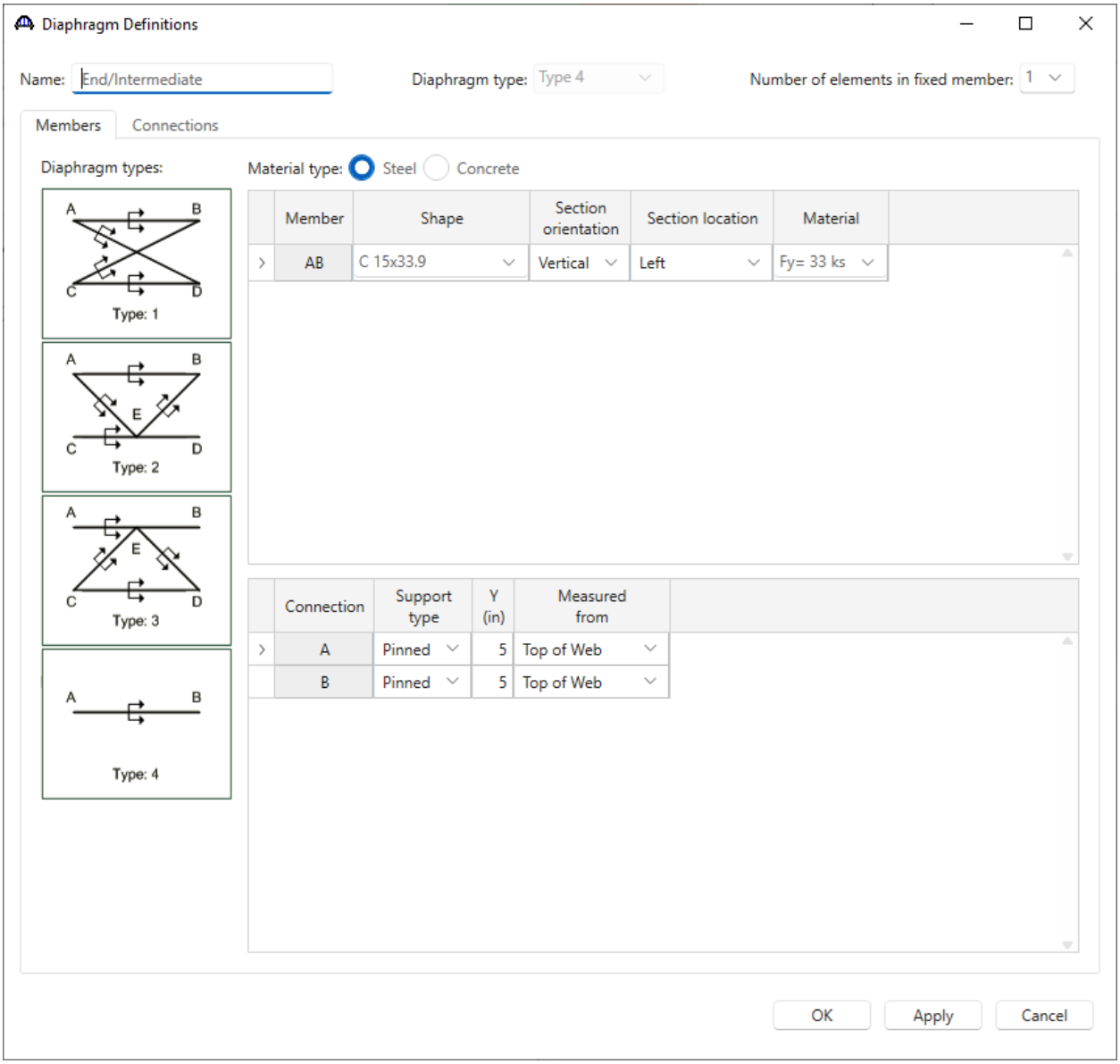
Navigate to the **Components** tab of the **Bridge Workspace** and expand the **Materials** -> **Concrete** folders. Double click on the **Class A (US)** concrete material. Click on the **Compute** button to compute the values based on the input.



Click **OK** to apply the data and close the window.

Diaphragm Definitions

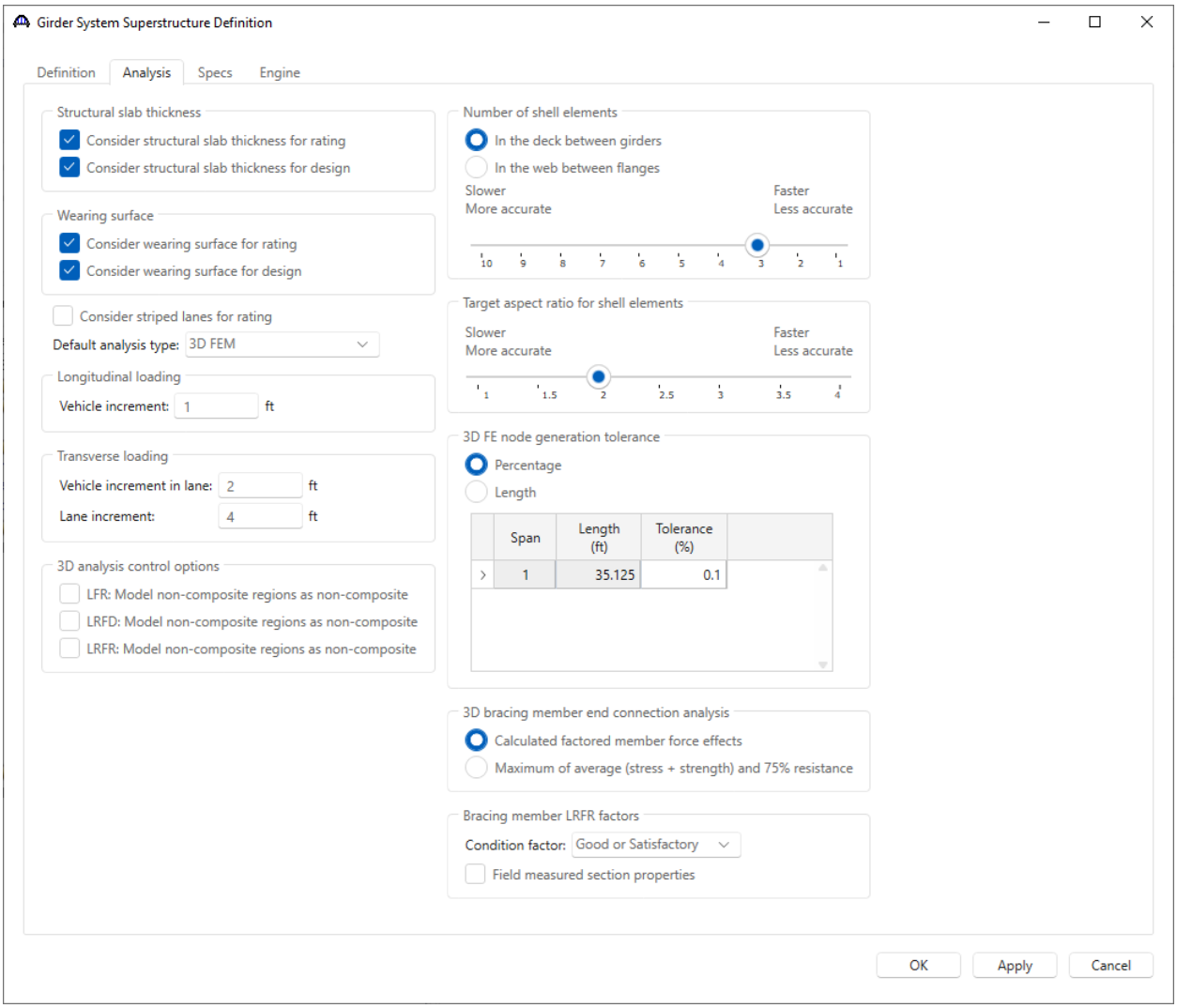
To view the diaphragm definition that will be used for this example, under the **Diaphragm Definitions** node, double click on **End/Intermediate** to open the **Diaphragm Definitions** window. This window describes a diaphragm definition that can be assigned to locations in the **Framing Plan Detail** window. No changes are required in this window.



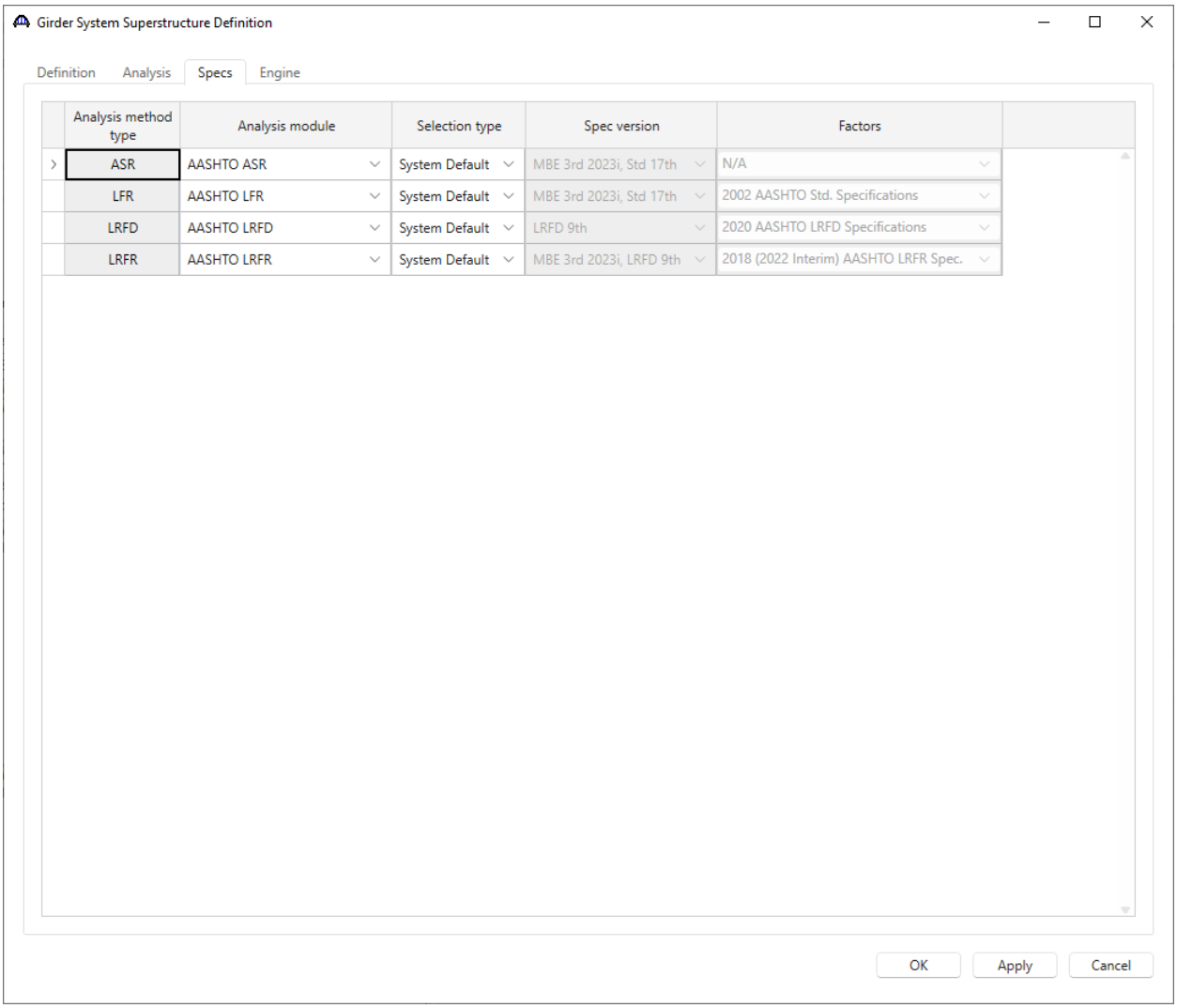
Steel bridges may contain any of the 4 types of diaphragm definitions. Concrete bridges may only contain **Type 4** diaphragm definitions.

Girder System Superstructure Definition

Double click on the **LRFD-5-2010** superstructure definition and navigate to the **Analysis** tab of this window. This window contains the following settings to control the 3D analysis. Users can control the influence surface loading in the longitudinal and transverse loading group boxes as well as the generated mesh through the shell element options.



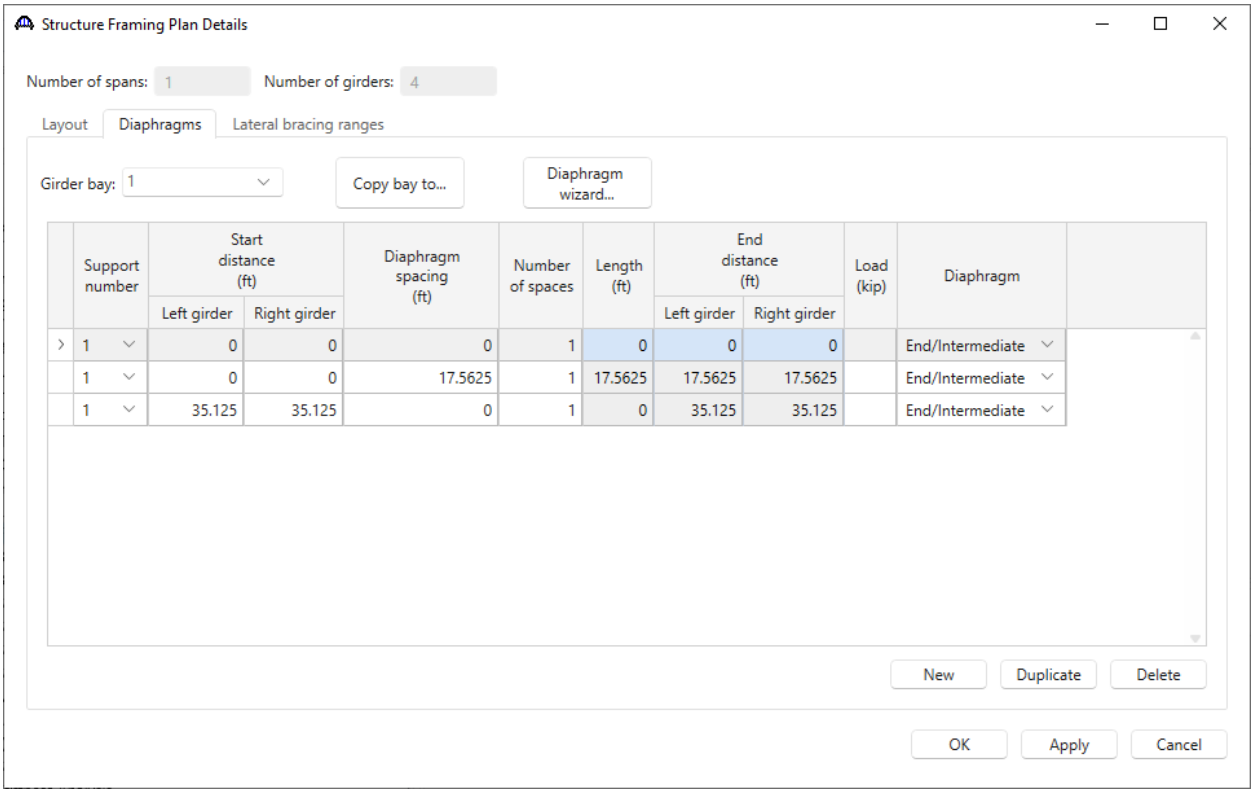
Similarly navigate to the **Specs** tab of this window. The analysis of all member alternatives in the superstructure definition will use the following engine and specification set on this tab.



No changes are required in this window. Click **Cancel** to close this window without applying any changes if made erroneously.

Framing Plan Details

Double click on the **Framing Plan Details** node in the **BWS** tree and navigate to the **Diaphragms** tab to see how diaphragm definitions are assigned to the framing plan. The weight of the diaphragms will be computed by the software and applied to the 3D model. The software uses the dead load listed within the **Load** column (for both line girder and 3D analysis) and will not establish the weight based on the diaphragm definition. If the **Load** column is left blank, the dead load of the diaphragm will be calculated using the details of the diaphragm definition.

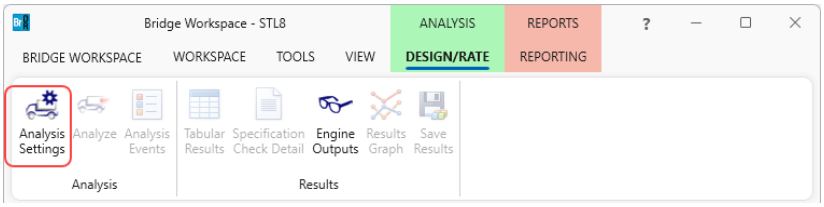


No changes are required in this window. Click **Cancel** to close this window without applying any changes if made erroneously.

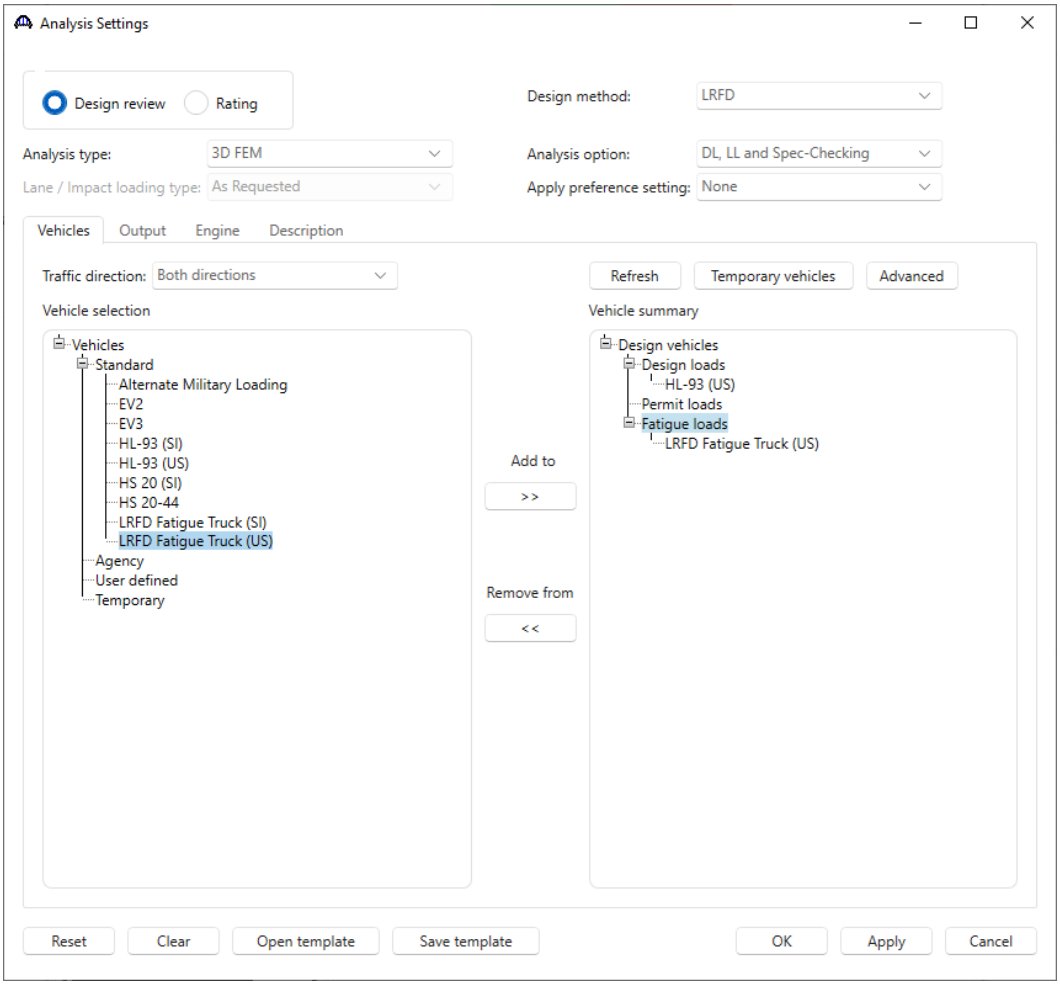
3D Analysis Settings

LRFD Design Review

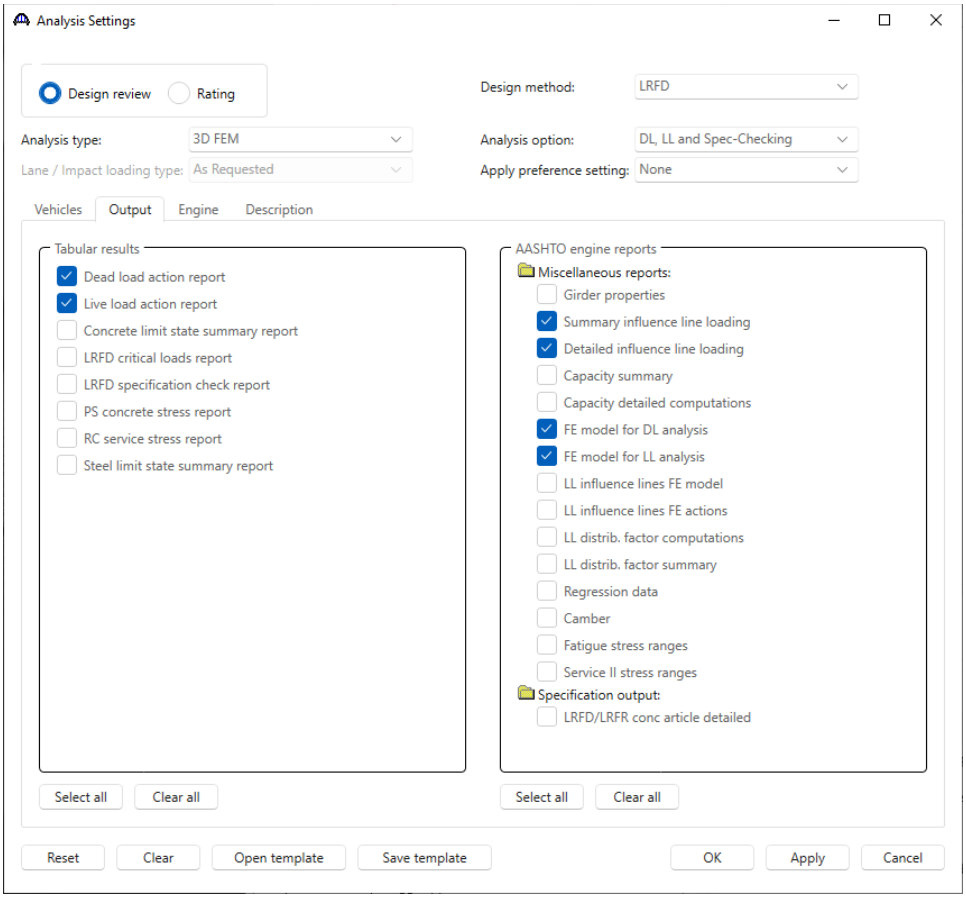
To run a **3D LRFD design review** on the superstructure definition, from the **Analysis** group of the **DESIGN/RATE** ribbon click on **Analysis Settings** button to open the **Analysis Settings** window as shown below.



Select the following setttings for a 3D LRFD design review.



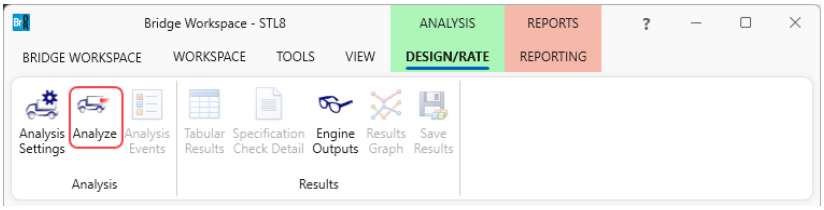
Navigate to the **Output** tab of this window and apply the following settings.



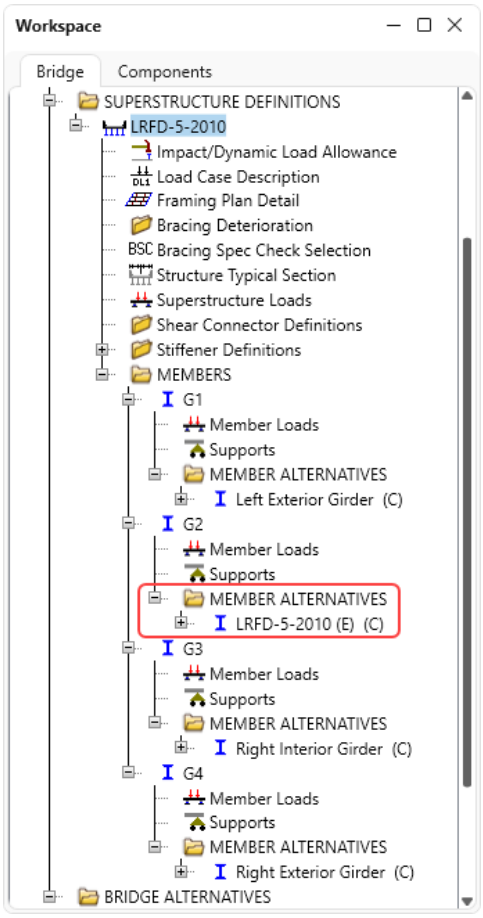
Click **OK** to apply these settings for the design review and close the window.

Design Review

With the **LRFD-5-2010** superstructure selected, click the **Analyze** button from the **Analysis** group of the **DESIGN/RATE** ribbon.

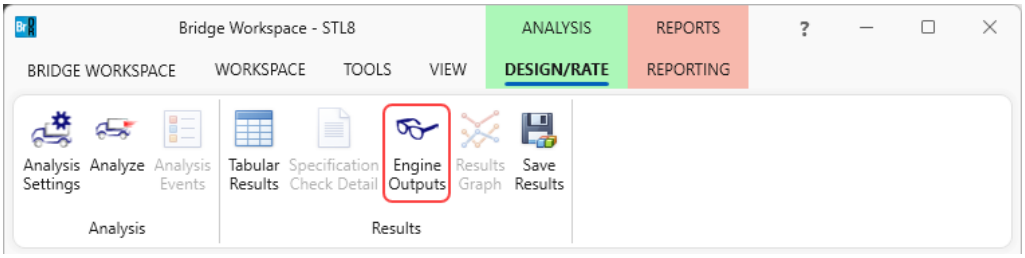


The software develops the 3D model using the member alternative marked as Existing **(E)** for each member. If the member does not have a member alternative marked as Existing and only has 1 member alternative, that member alternative is used for the 3D model. If the member has no member alternative marked as Existing and more than 1 member alternative, the analysis will not be performed. Spec checking and rating is only performed for member alternatives marked as Existing. For this sample bridge, the spec checking will only be performed for the member alternative for member **G2**.

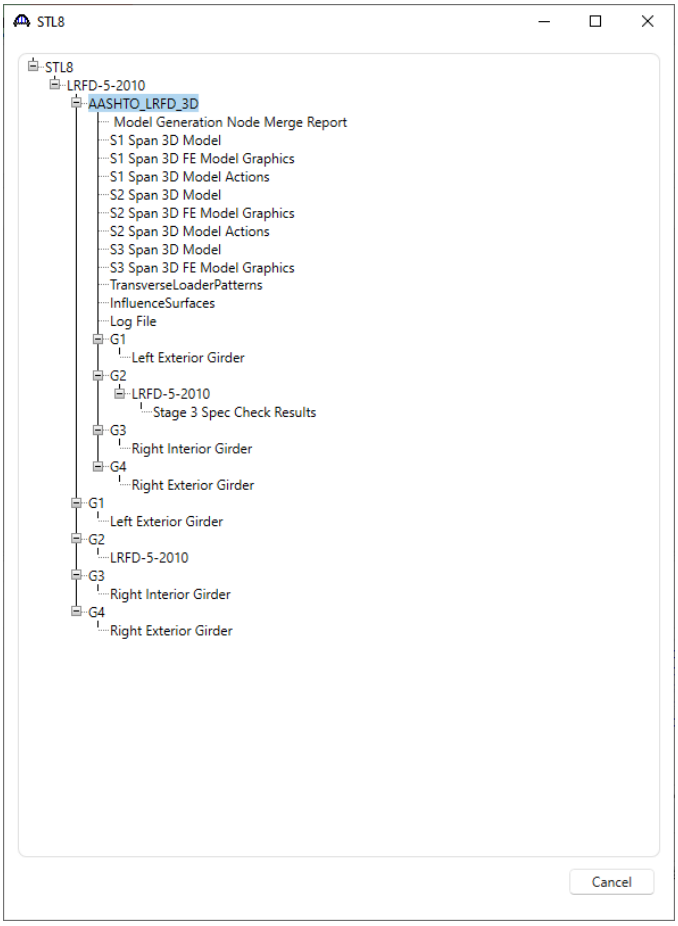


Engine Outputs

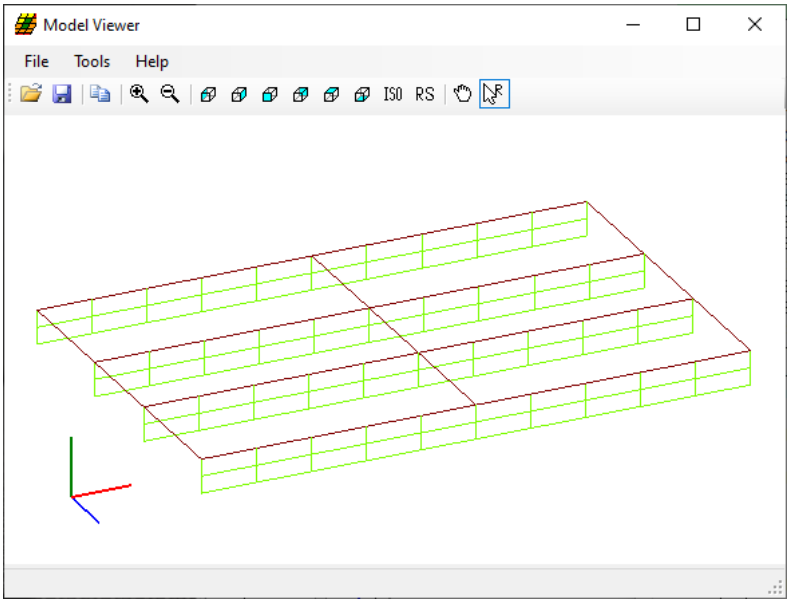
To view **Engine Output** files, click on the **Engine Outputs** button from the **Results** group of the **DESIGN/RATE** ribbon as shown below.



The following shows the output files created by the 3D LRFD design review. Similar files are created for a 3D Std rating.

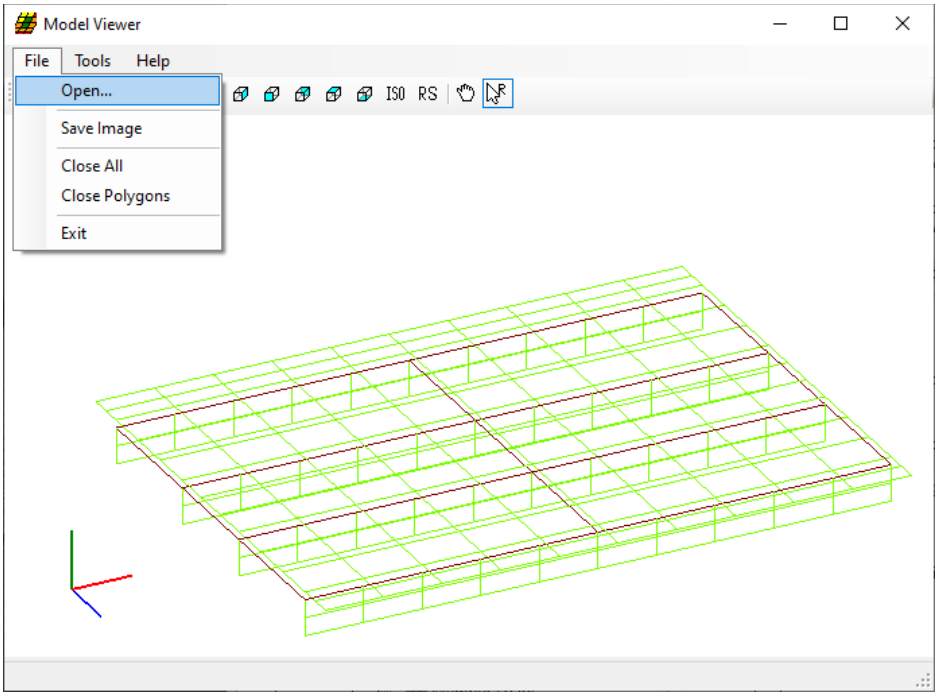


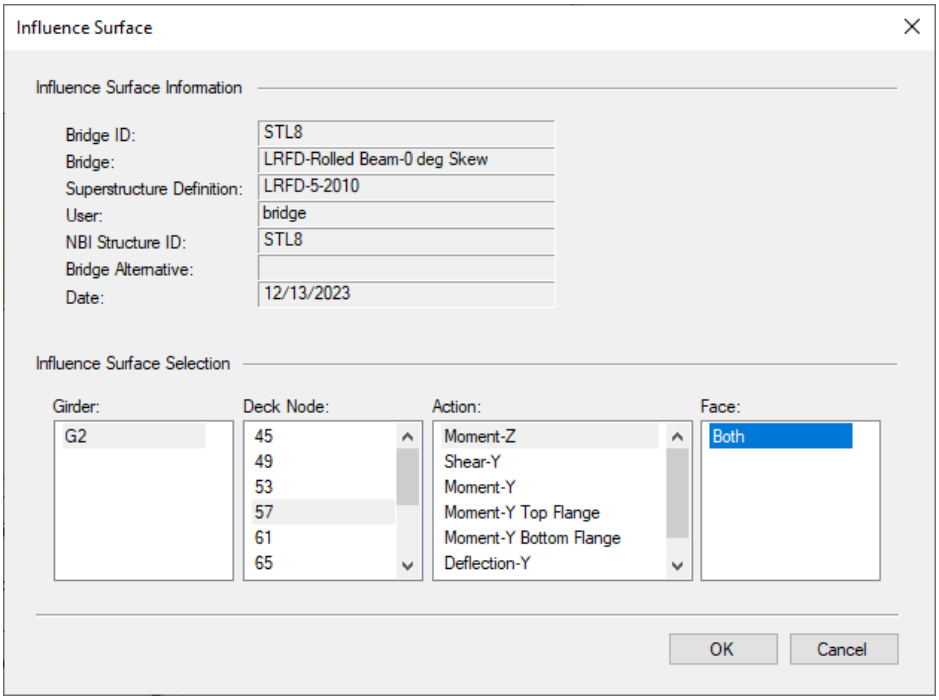
The **3D Model** files list the data for the models including nodes, members, properties, and loads. The **3D Model Actions** files list the FE results (reactions, element actions, displacements) for the models. The **Model Graphics** files can be opened to graphically view the FE models. The following shows the graphics for the Stage 1 model which contains the steel beams and diaphragms.

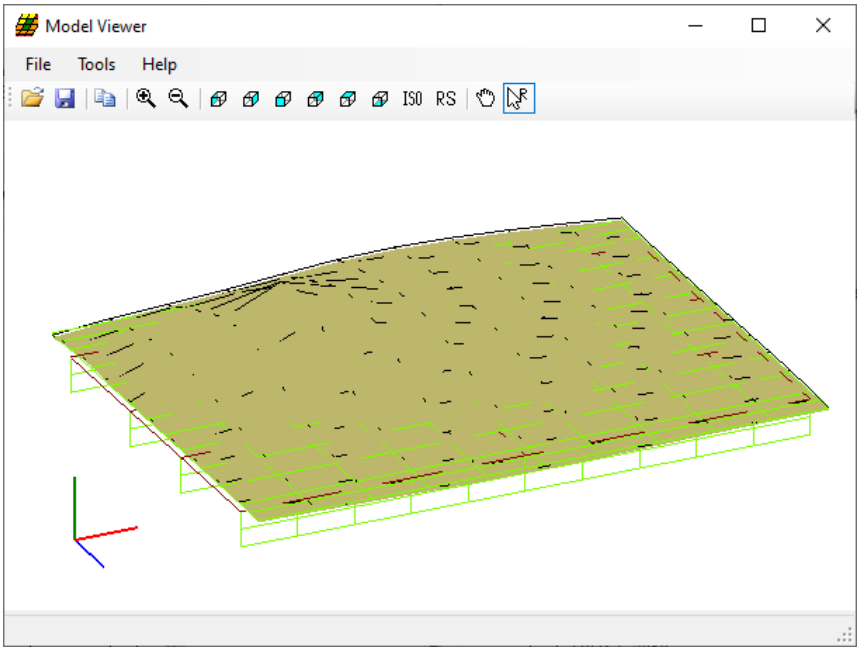


Node and element numbers can be turned on from the **Tools** menu. The mouse controls manipulation of the view. Zoom by rolling the mouse wheel. Translate by pushing down the mouse wheel. Rotate by pushing down the left mouse button.

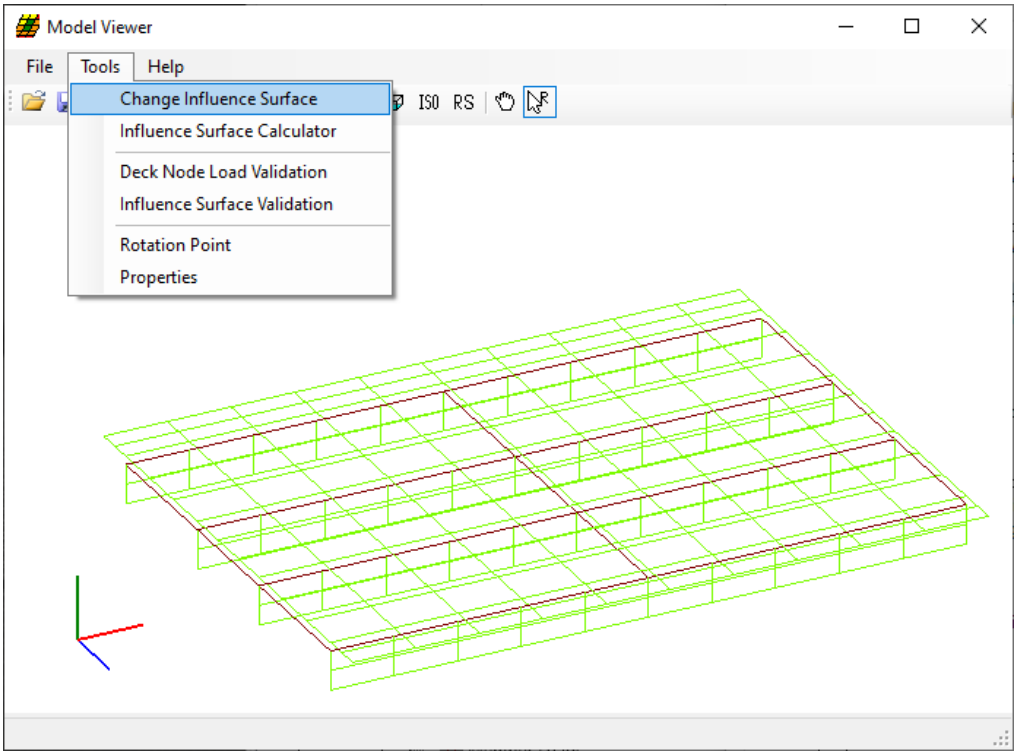
The generated influence surfaces for the unit live loading can be viewed by selecting the **Stage 3** Graphics model and then selecting **File** -> **Open** -> **InfluenceSurfaces.sur**. This opens the Influence Surface window as shown below.

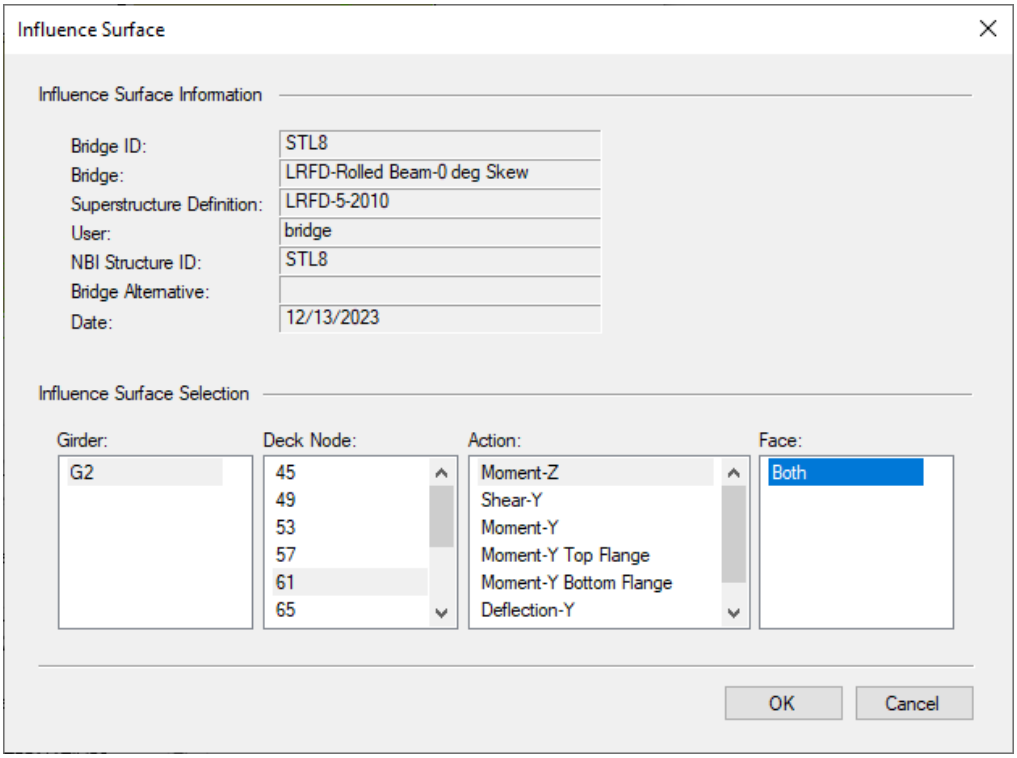


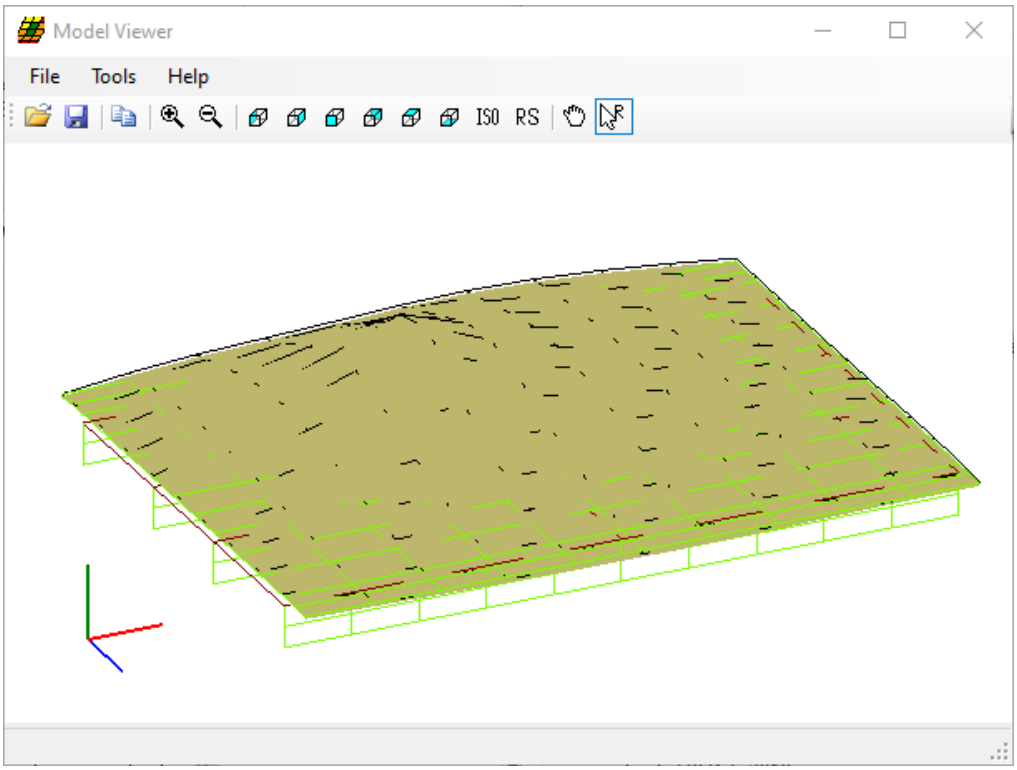




An influence surface for viewing can be chosen by selecting **Tools -> Change Influence Surface** and then selecting desired actions

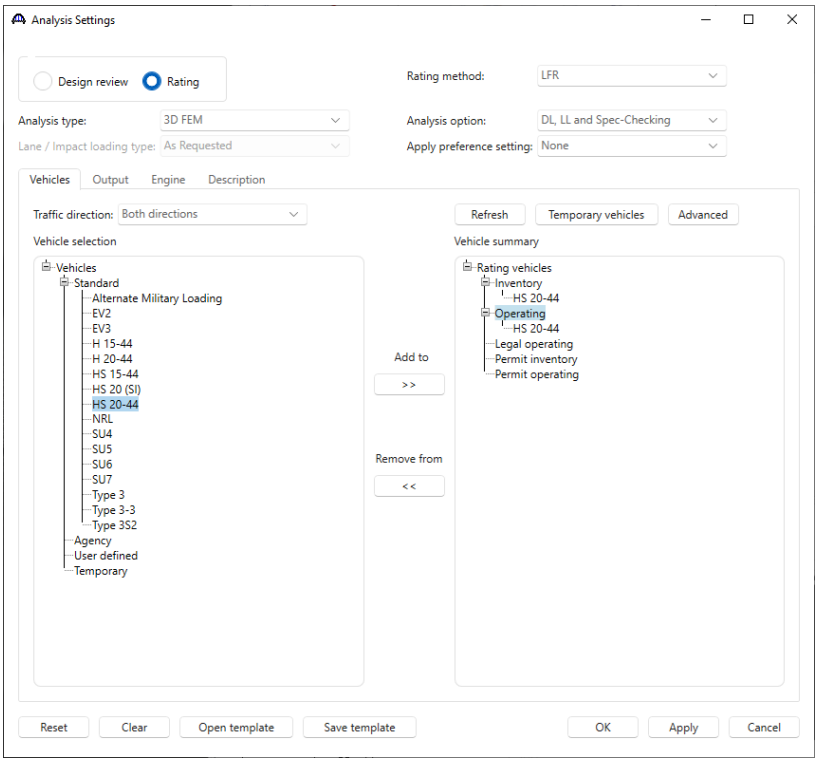


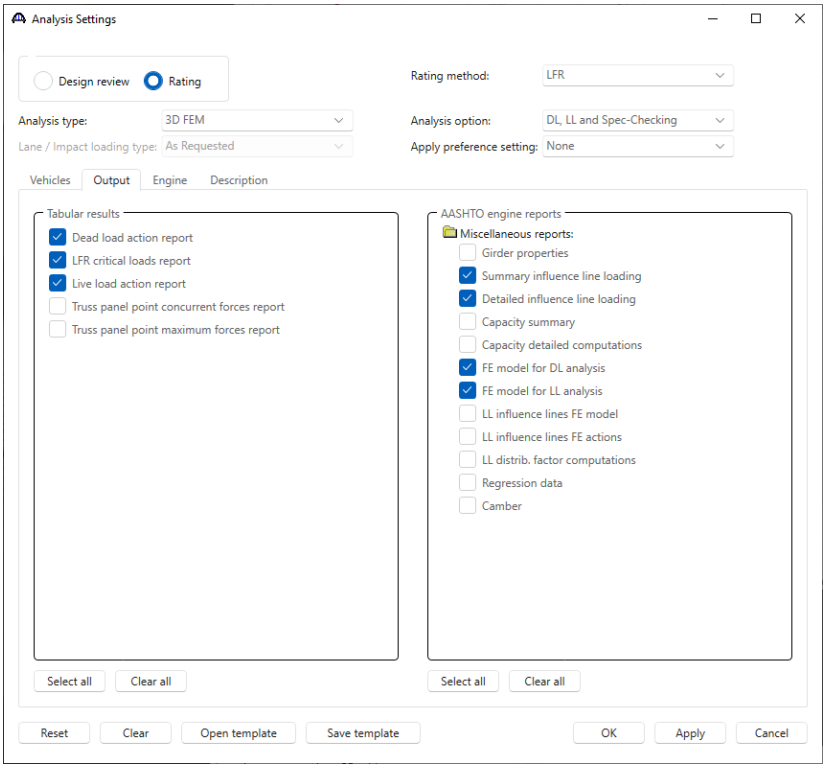

Influence Surface
Influence Surface Information
Bridge ID:
STL8
Bridge:
LRFD-Rolled Beam-0 deg Skew
Superstructure Definition:
LRFD-5-2010
User:
bridge
NBI Structure ID:
STL8
Bridge Alternative:
Date:
12/13/2023
Influence Surface Selection
Girder:
G2
Deck Node:
45
49
53
57
61
65
Action:
Moment-Z
Shear-Y
Moment-Y
Moment-Y Top Flange
Moment-Y Bottom Flange
Deflection-Y
Face:
Both
OK
Cancel


Model Viewer
File
Tools
Help
ISO
RS

LFR Rating

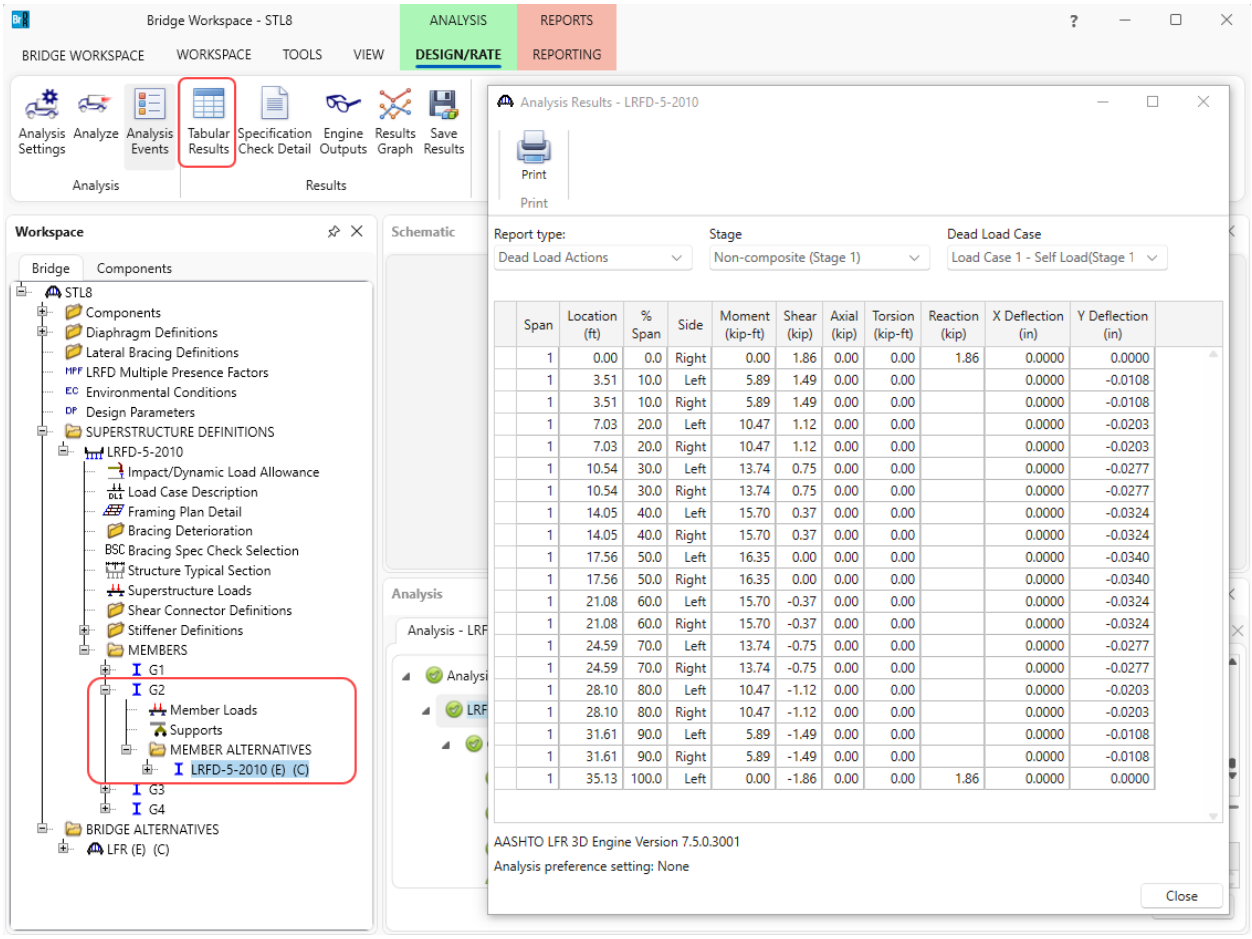
Similarly, apply the following settings in the **Analysis Settings** window to run an **HS20 LFR** analysis and run a rating analysis.

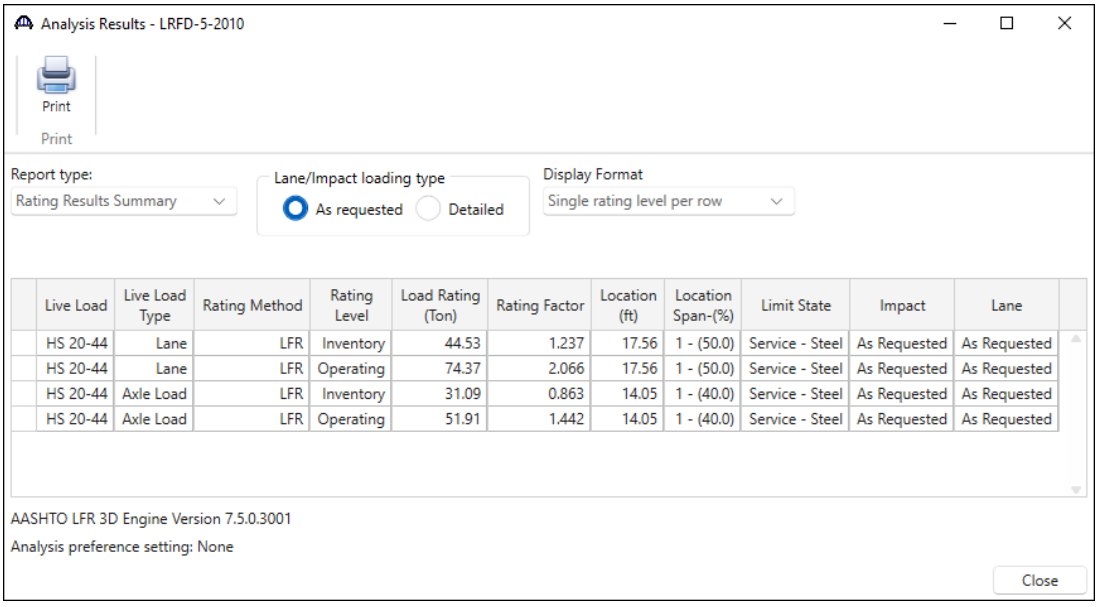




Tabular Results

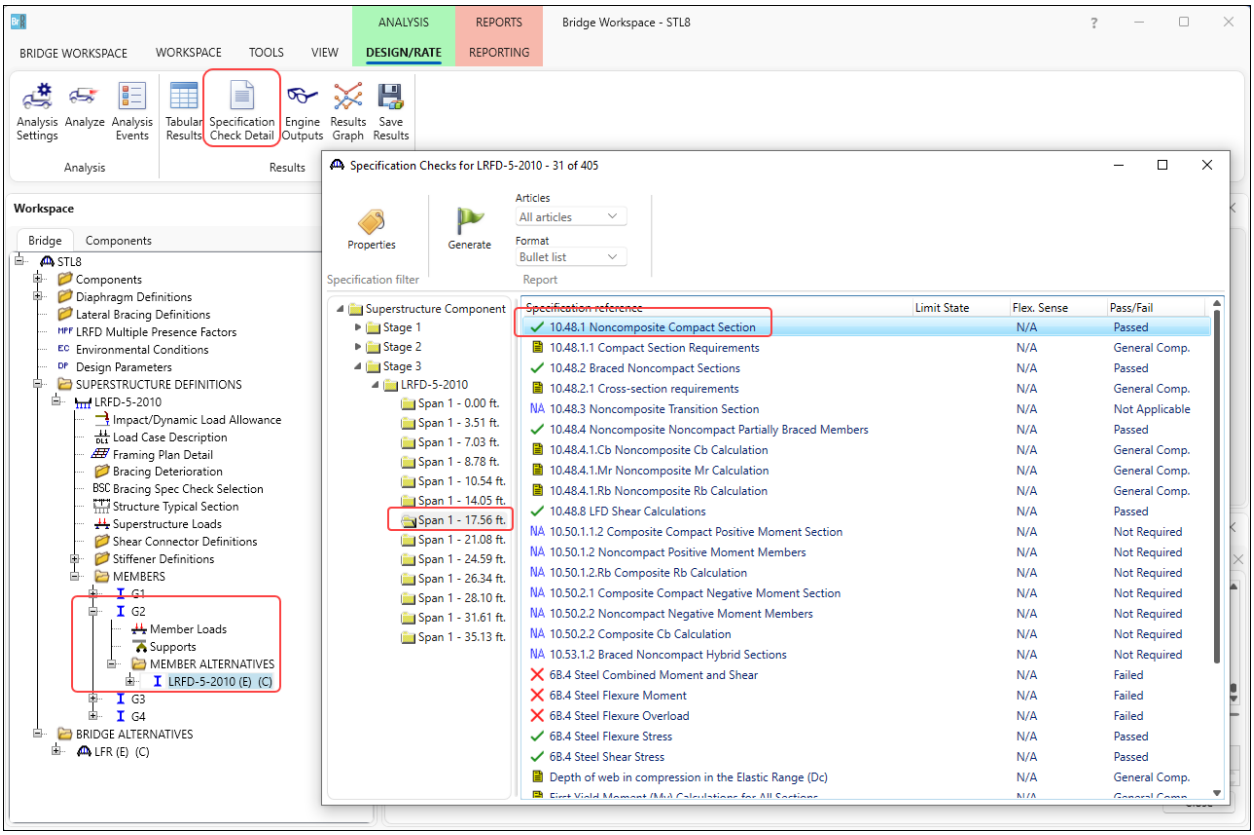
After the analysis completes, open the **Tabular Results** window by clicking on the **Tabular Results** button from the **Results** group of the **DESIGN/RATE** ribbon while the member alternative of member **G2** is selected.





Specificatiton Check Details

Specification Check Details can be viewed for the bracing members by clicking on the **Specification Check Detail** button from the **Results** group of the **DESIGN/RATE** ribbon.



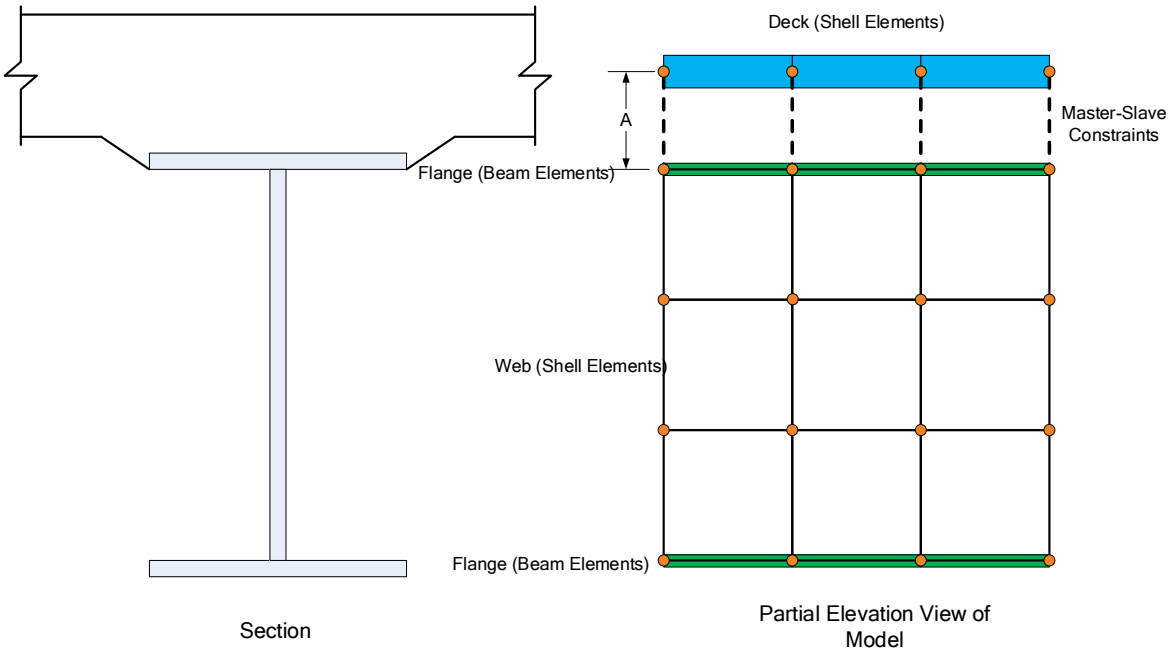
3D Model

The modeling techniques used are the result of a survey of researchers and practitioners and review of several software packages.

Steel Members

The model for a girder system structure with steel girders is comprised of the following:

- Shell elements for the deck
- Beam elements for the top and bottom flanges.
- Shell elements for the web. Web shell elements are divided into equal segments for web shells. The shell nodes may be adjusted to match diaphragm connections.
- For curved structures, curvature is represented by straight elements with small kinks at node points instead of curved elements.
- Master-slave constraints that connect the top flange to middle of deck. The distance 'A' as shown in figure below is measured from the center of the deck to the center of gravity of the top flange (including cover plates) at the start of G1. The same value is used everywhere for all girders to maintain horizontal elements.



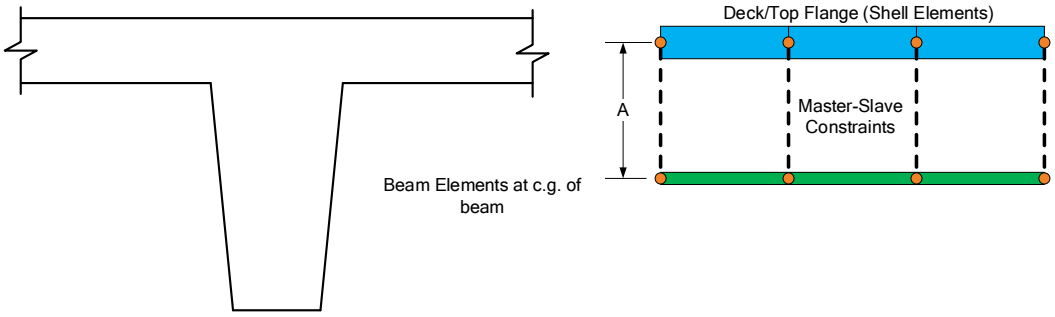
Reinforced Concrete Members

A Stage 1 FE model is generated for the dead loads on the reinforced concrete beam. A Stage 3 FE model is generated for the live load analysis. Reinforced concrete beams only have 1 stage so the Stage 1 and Stage 3 models contain the same cross section properties.

The model generated by the export to the AASHTO LRFD analysis engine will contain node points at the following locations:

- Cross section property change points
- Span tenth points
- Support locations
- User defined points of interest

For reinforced concrete Tee beam cross sections, the cross-section is modeled as a beam element to represent the girder web. For reinforced concrete I beam cross sections, the cross-section is modeled as a beam element to represent the girder web and the bottom flange. Shell elements are used to represent the deck, and master-slave constraints are applied to connect the beam elements to the deck elements. For the RC Tee beam, the distance 'A' shown in figure below is measured from the center of gravity of the deck (top flange) to the center of gravity of the web. For the RC I beam, the distance 'A' is measured from the center of gravity of the deck (top flange) to the center of gravity of the section composed of the web and the bottom flange. Concrete bridges are considered composite over the entire length without consideration of how user specifies composite in the UI.



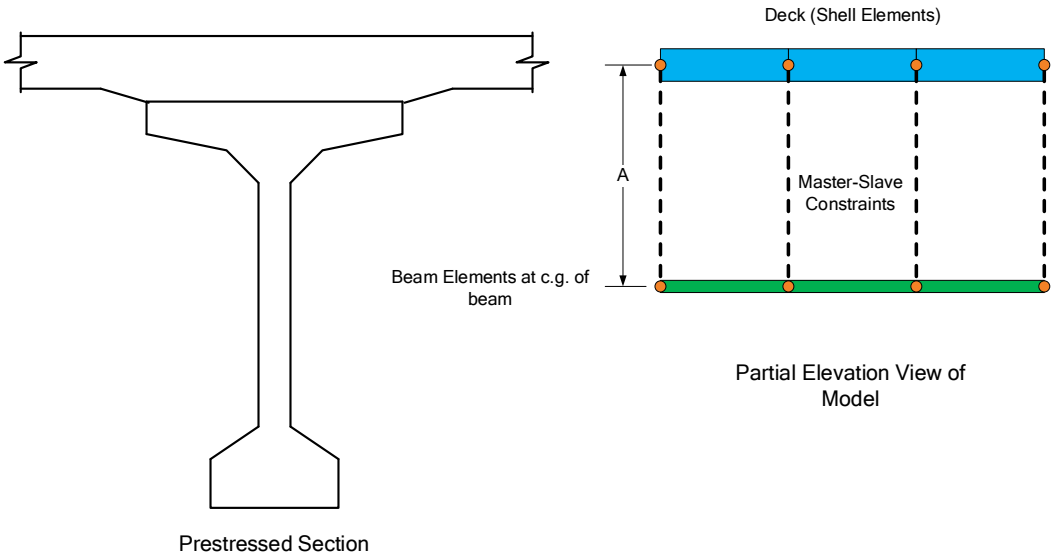
Reinforced Concrete Section

Partial Elevation View of Model

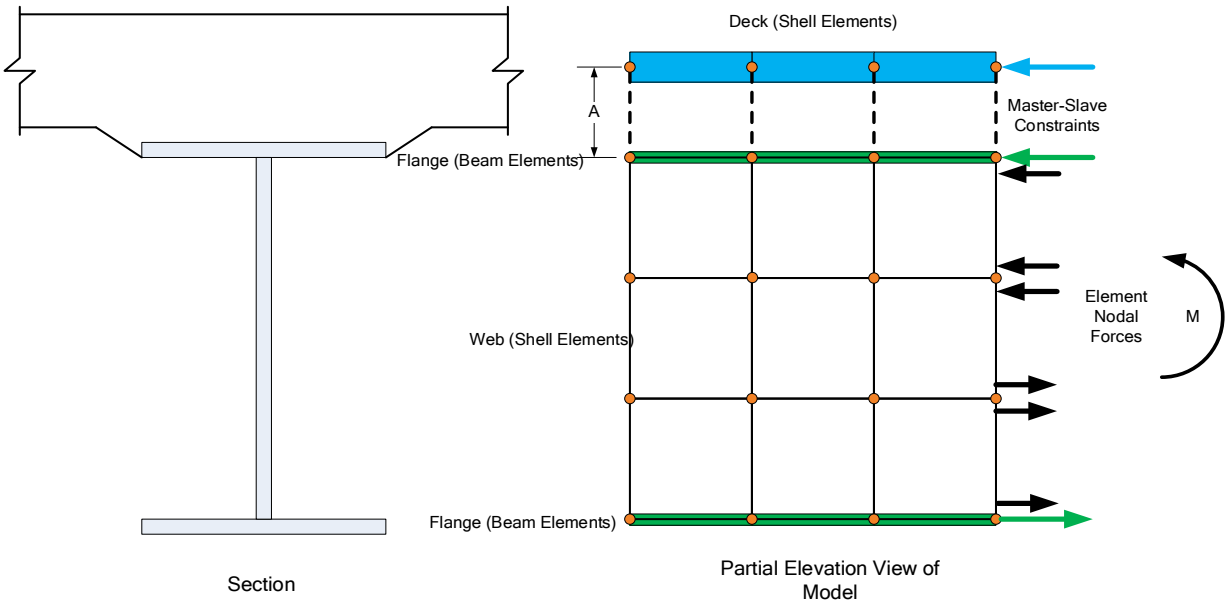
Prestressed Concrete Beams

Three stage models are generated for prestressed concrete beams. A Stage 1 FE model is generated for the beam dead load and non-composite dead loads. A Stage 2 FE model is generated for dead loads applied to the long-term composite section properties. A Stage 3 FE model is generated for the live load analysis.

For prestressed cross sections, for composite sections, the cross-section is modeled as a beam element to represent the girders, shell elements to represent the deck, and master-slave constraints to connect the beam elements to the deck elements. The distance 'A' shown in figure below is measured from the center of gravity of the deck to the center of gravity of the P/S beam.



The moment at a beam cross section is calculated by solving the equilibrium equations at that section. This moment is then used in the specification check articles in the same way that it would for a line girder analysis.



Mesh Generation

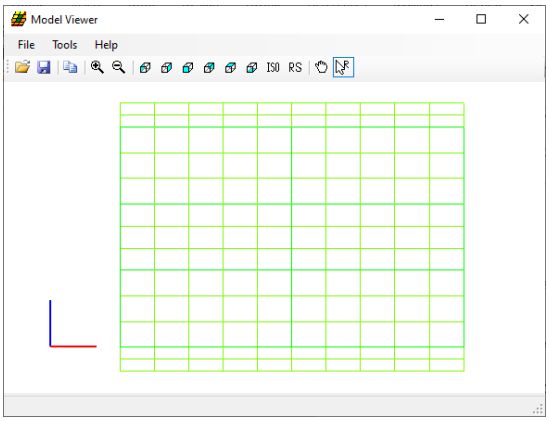
The FE model created by BrDR will contain nodes at the following locations.

- Cross section property change points
- Span tenth points
- Support locations
- Diaphragm locations
- User defined points of interest

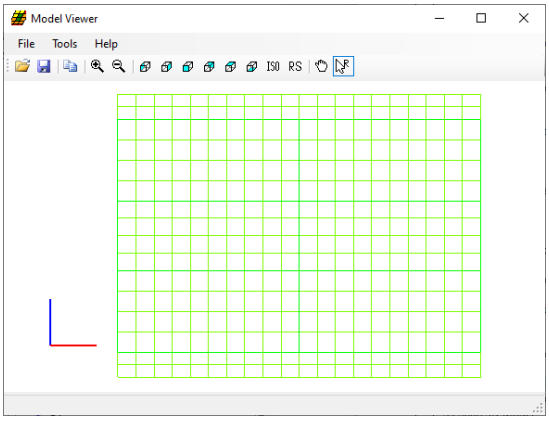
The user controls the mesh generation by the controls previously shown on the **Superstructure Definition: Analysis** tab. The software creates the mesh following the number of elements selected between beams or within the web of a steel beam and the target aspect ratio entered by the user. The presence of nodes at the locations listed above may result in some elements falling outside the target aspect ratio.

The following plan views show how the mesh for this example can be controlled by the user.

3 shells between beams, target aspect ratio = 2



4 shells between beams, target aspect ratio = 1



Loading

The program computes all the dead loads acting on the beam including the self-weight of the beam, user defined appurtenances on the structure typical section, wearing surfaces, diaphragms, and user defined member loads. Composite dead loads are applied directly to the deck shells in the 3D model in their actual location. They are not distributed to the girders based on the choices available in the Superstructure Loads window.

The Stage 3 FE model is loaded with unit loads at each deck node within the travelway to generate influence surfaces for the beam. Lane positions and combinations are determined based on the travelway and the transverse loading parameters set by the user on the Superstructure Definition: Analysis Settings tab. The influence surfaces are then loaded with the selected vehicles to find the maximum live load effects.