AASHTOWare BrDR 7.5.0

3D FEM Analysis Tutorial

Mesh Generation and Dead Load Analysis Example

AASHTOWare Bridge Design and Rating Training
3DFEM5 – Mesh Generation and Dead Load Analysis Example

Topics Covered:

- 3D model deck mesh generation
- Dead load only analysis
- 3D model validation

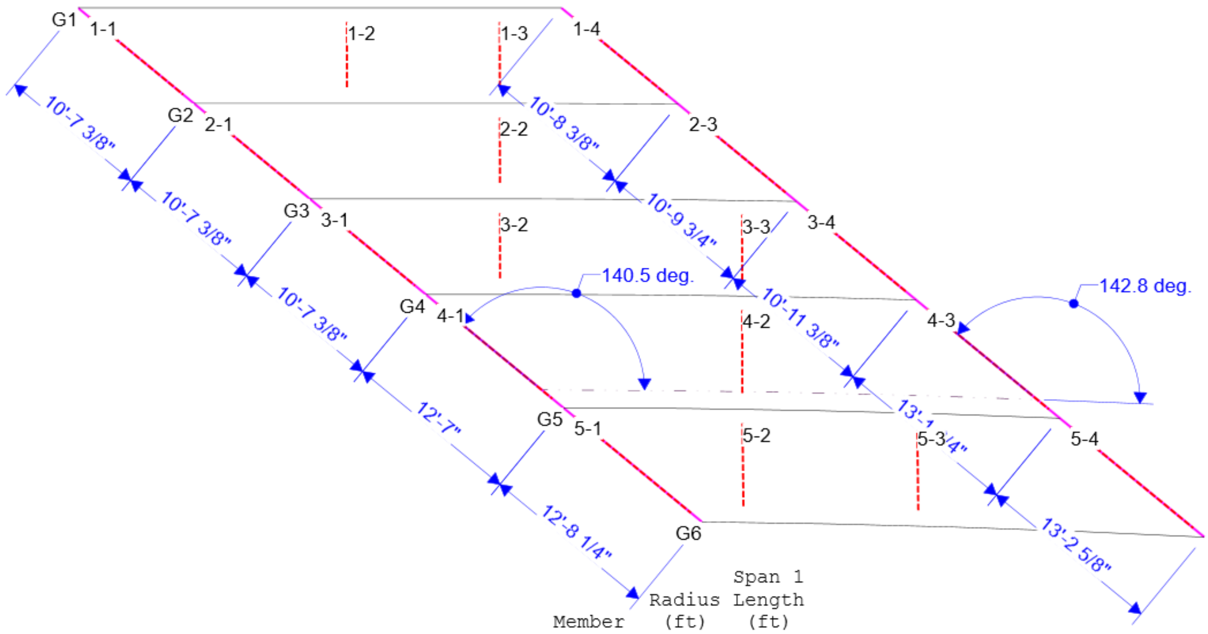
Overview of mesh generation options:

- Number of shell elements
- Target aspect ratio for shell elements
- Node merge tolerance by span

This tutorial describes the data entry for a curved steel girder system bridge in BrDR version 7.5. This tutorial illustrates the process of using the dead load only analysis to determine appropriate mesh generation options for a particular model. The impact of each mesh generation option on the deck shell elements is discussed.

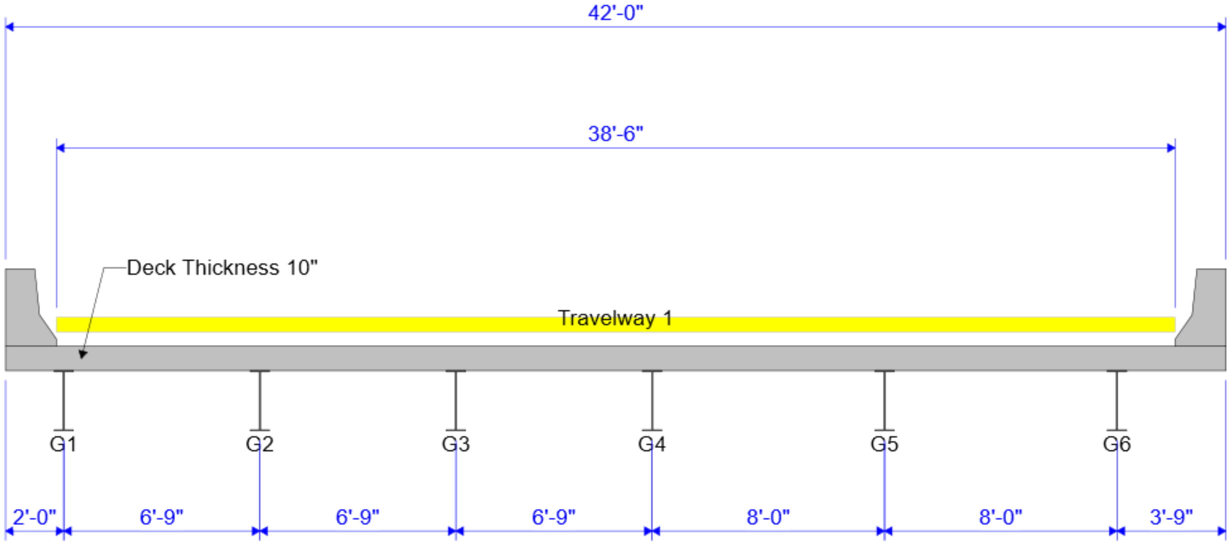
Example Curved Steel Girder Bridge

Framing Plan



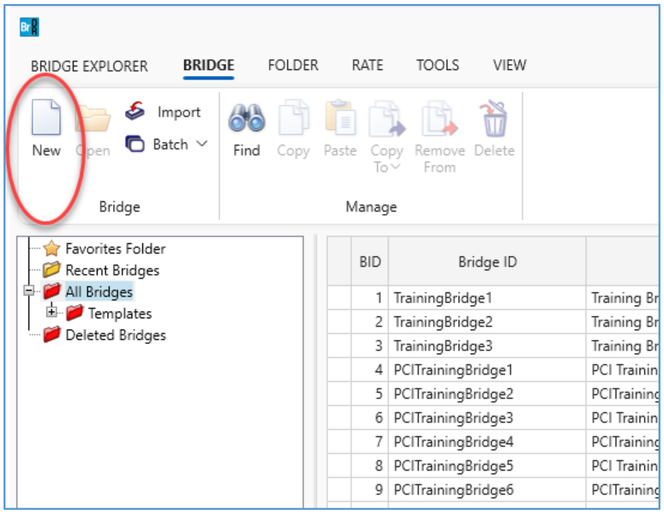
Member	Radius (ft)	Span 1 Length (ft)
G1	927.00	34.14
G2	920.25	34.21
G3	913.50	34.37
G4	906.75	34.63
G5	898.75	35.08
G6	890.75	35.50
Structure Ref.Line	900.00	35.00

Structure Typical Section

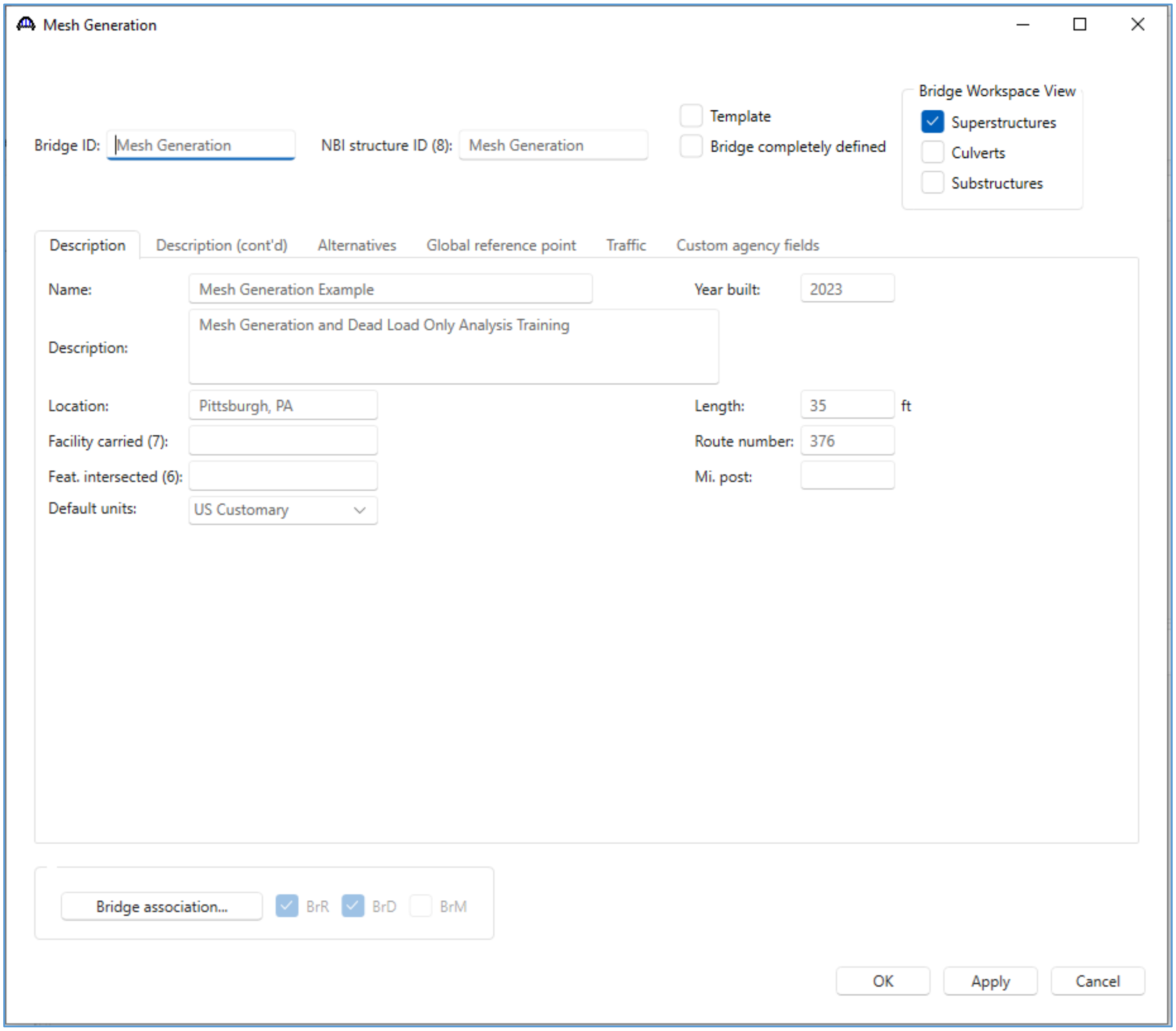


Curved Girder Structure Data Entry

Create a new bridge from the bridge explorer menu.



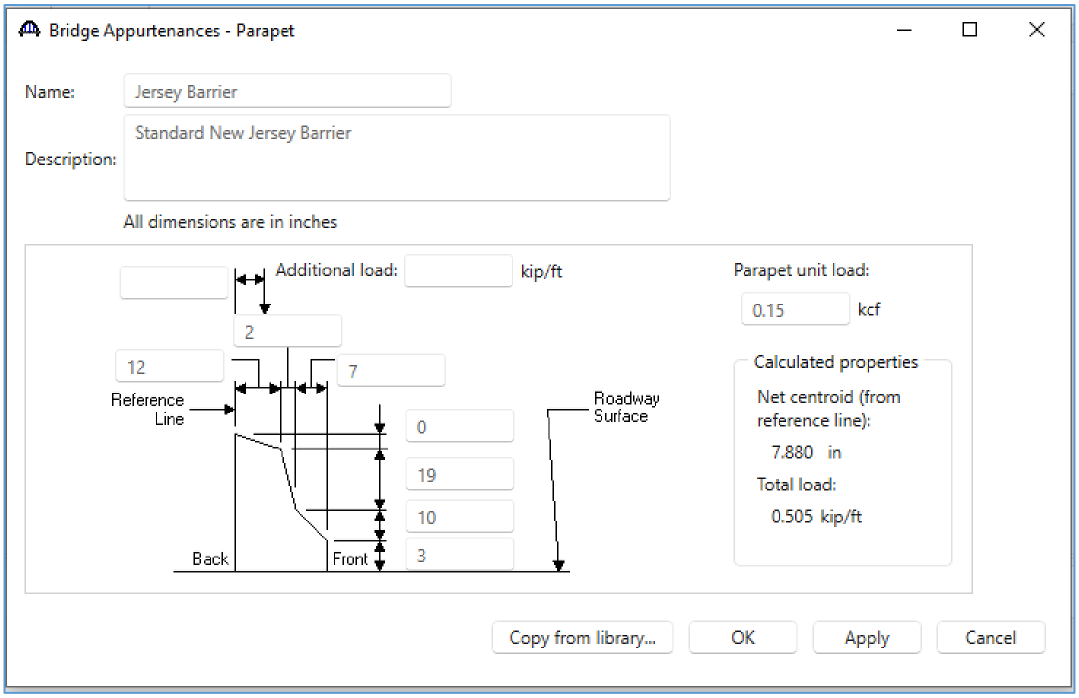
Assign the bridge ID, NBI structure ID, bridge name and other relevant information. Because this bridge will only contain a superstructure, only the 'Superstructures' option needs to be selected in the Bridge Workspace View. After inputting the bridge description, select 'OK' to create the new structure.



First define the components. This structure has 1 appurtenance, 2 steel shapes, a concrete material, and a steel material.

Appurtenances

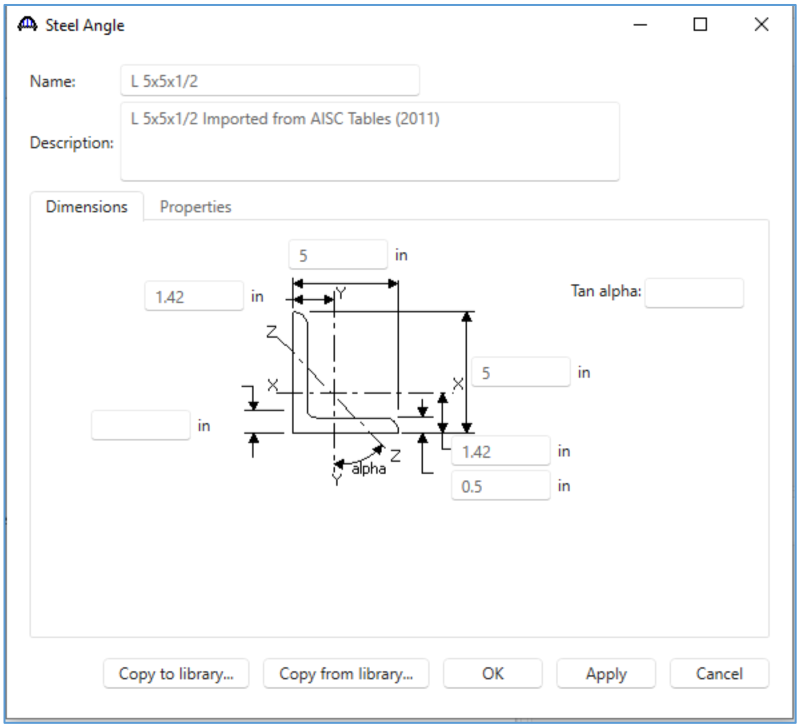
Parapet: (Use 'Copy from library…' to save time.)



Steel Shapes:

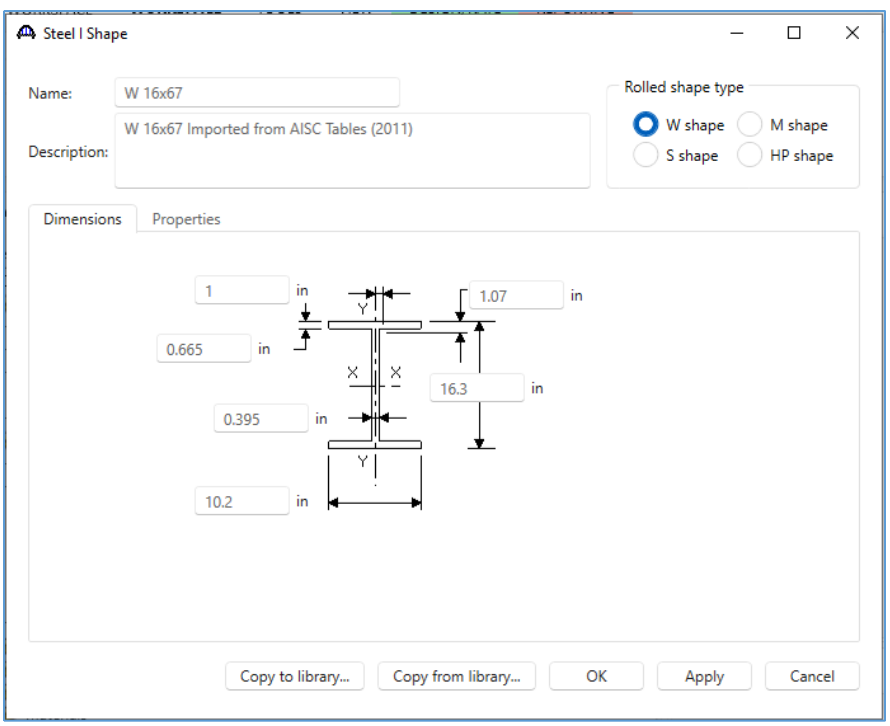
Angles:

L5x5x1/2: (Use 'Copy from library…' to save time.)



I Shapes:

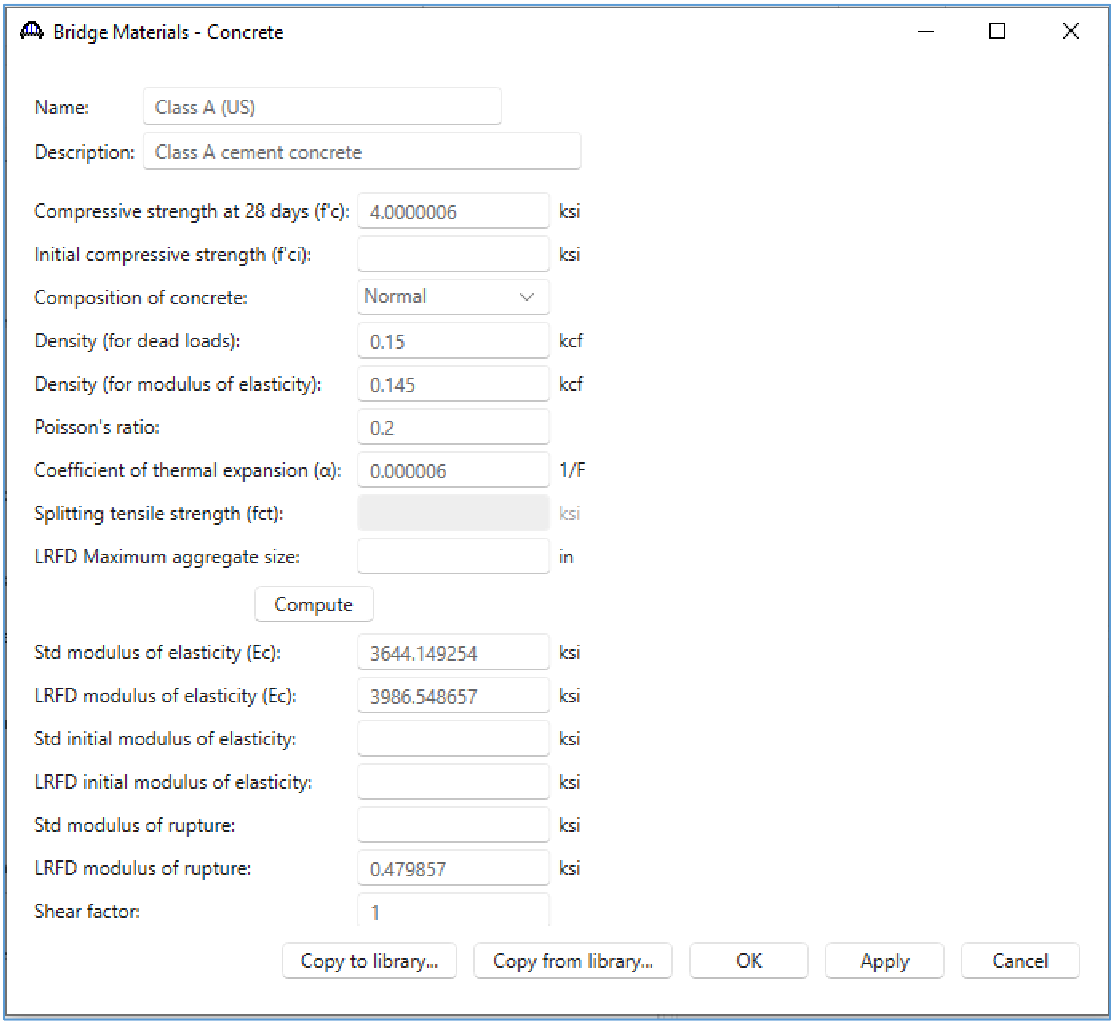
W16x67: (Use 'Copy from library…' to save time.)



Materials:

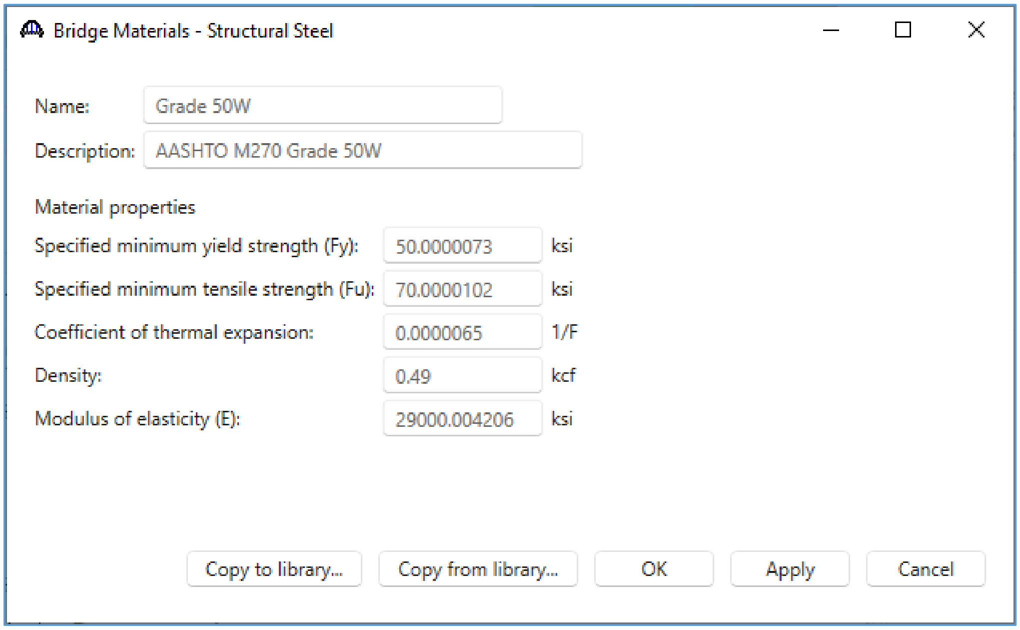
Concrete:

Class A (US): (Use 'Copy from library…' to save time.)

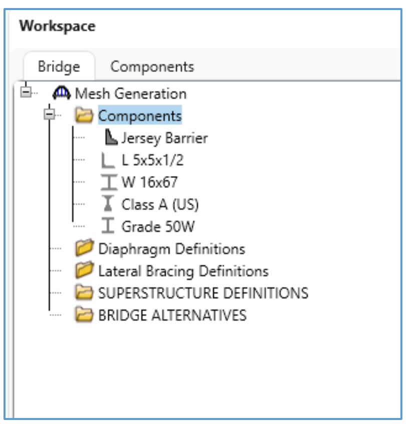


Structural Steel:

Grade 50W: (Use 'Copy from library…' to save time.)

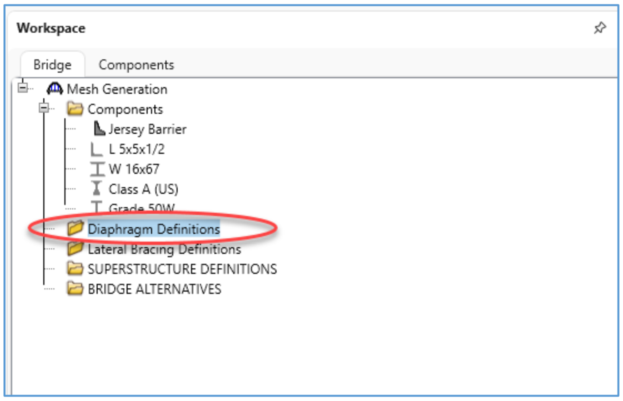


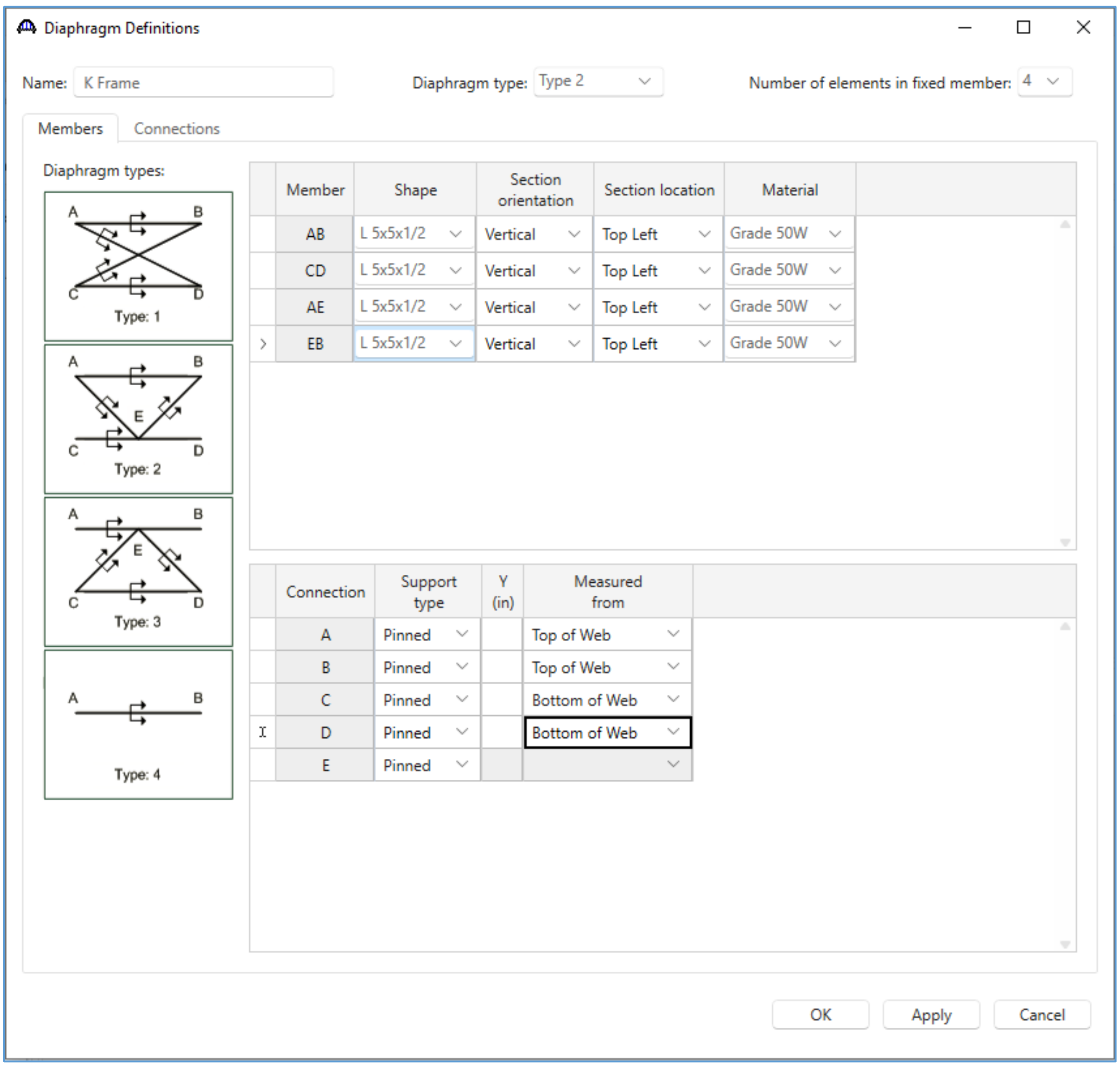
With all the bridge components defined, the bridge workspace tree should now include all the following components.



Diaphragm Definitions

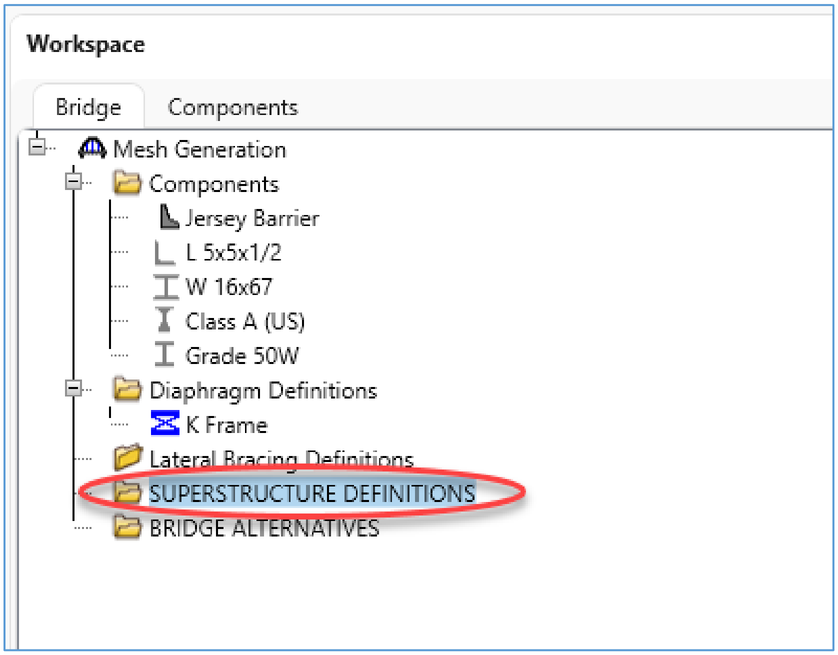
Define the diaphragms. Double click on ‘Diaphragm Definitions’ in the bridge workspace tree to add a new diaphragm definition. This structure has one diaphragm.

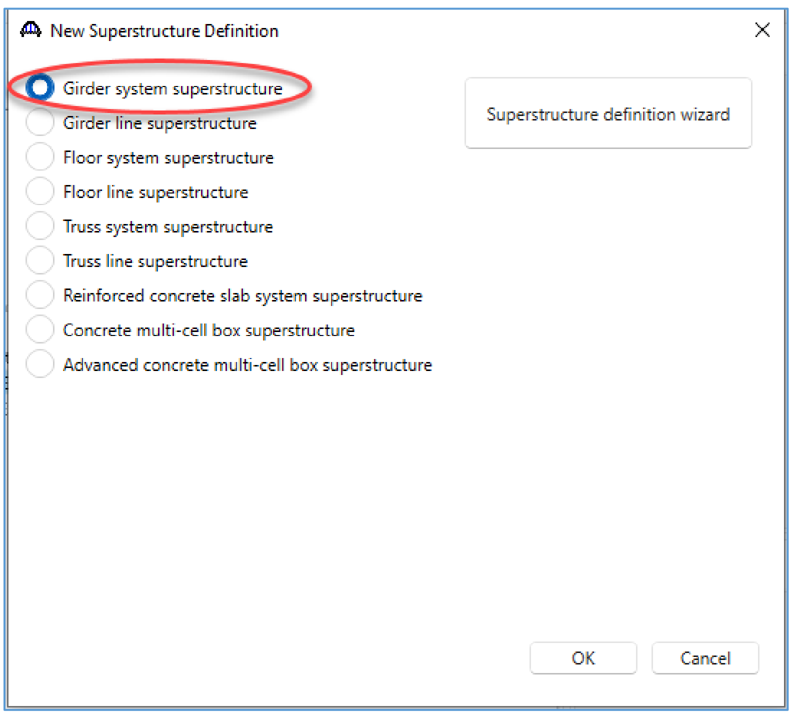




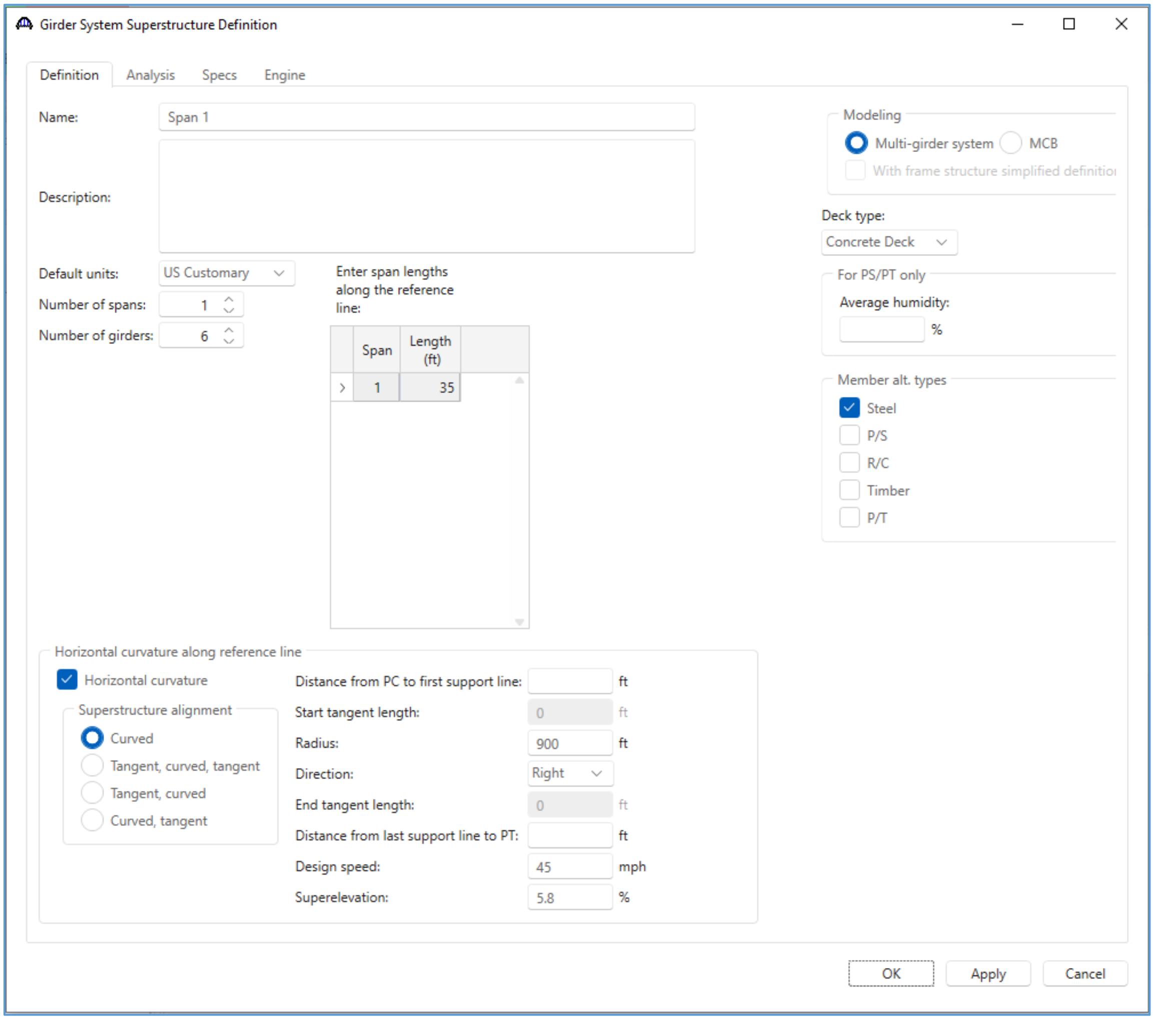
Superstructure Definition

Create a new girder system superstructure definition in the bridge workspace tree.

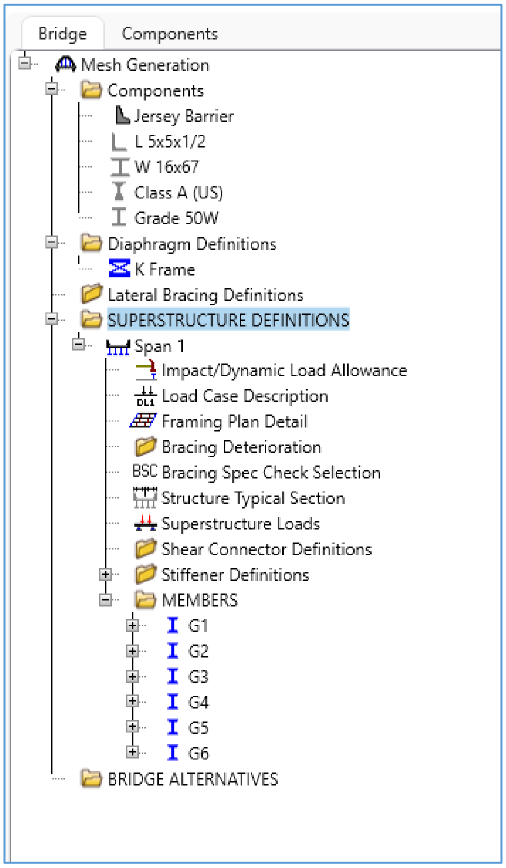




In the **girder system superstructure definition window**, enter the following information. Make sure to input the horizontal curvature. A girder system structure cannot be modified from straight to curved or curved to straight after it has been defined. Leave the default analysis settings under the 'Analysis' tab. These options will be modified later. Select 'OK' to close and save the window.

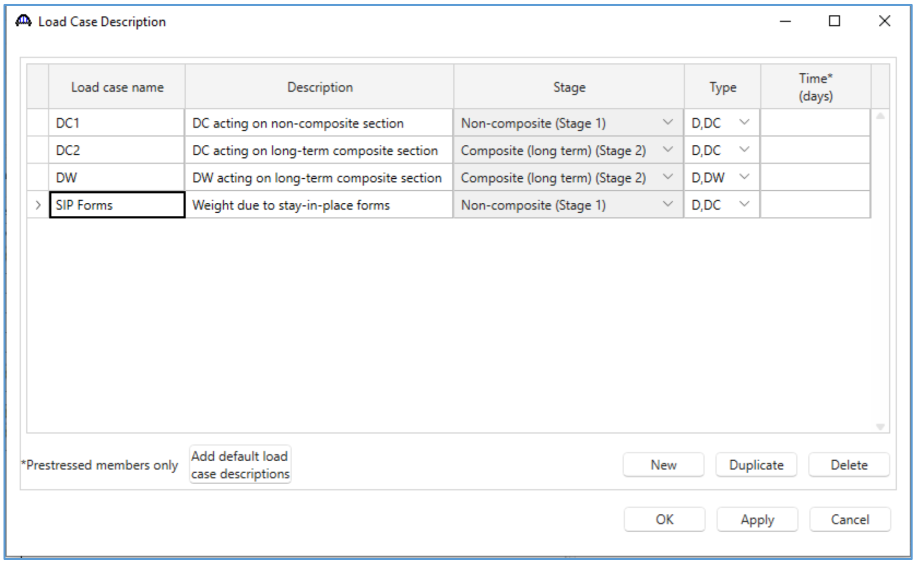


Expand the tree for the new girder system structure definition.



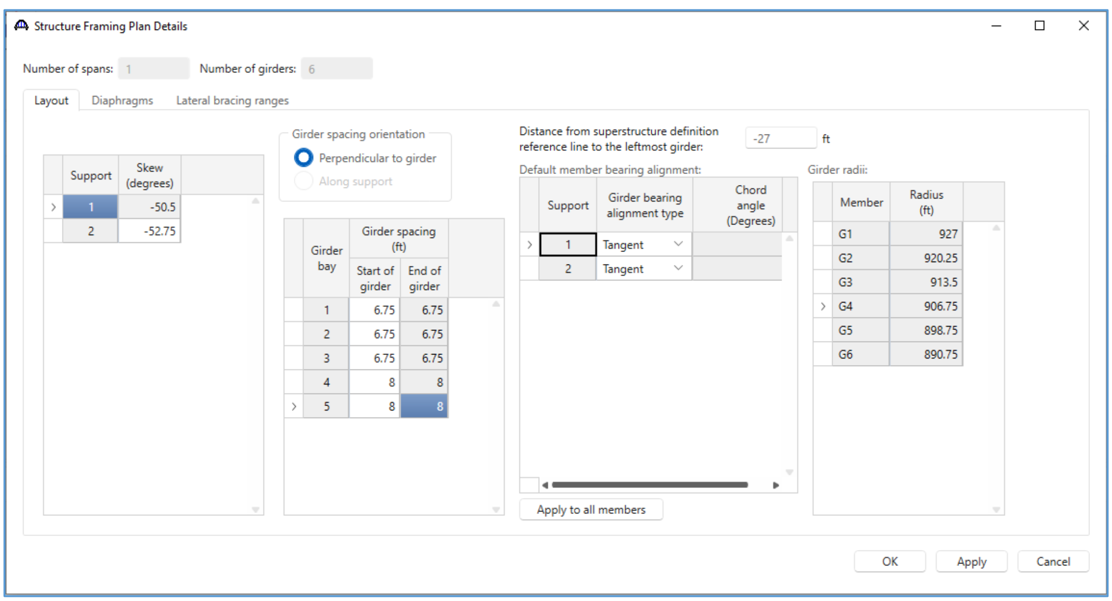
Load Case Description

Add the default load case descriptions for the girder system superstructure.



Framing Plan Detail

Enter the framing plan details in the **Structure Framing Plan Details window**. First, input the following information in the 'Layout' tab. Select 'Apply' to save the layout to memory and keep the window open.

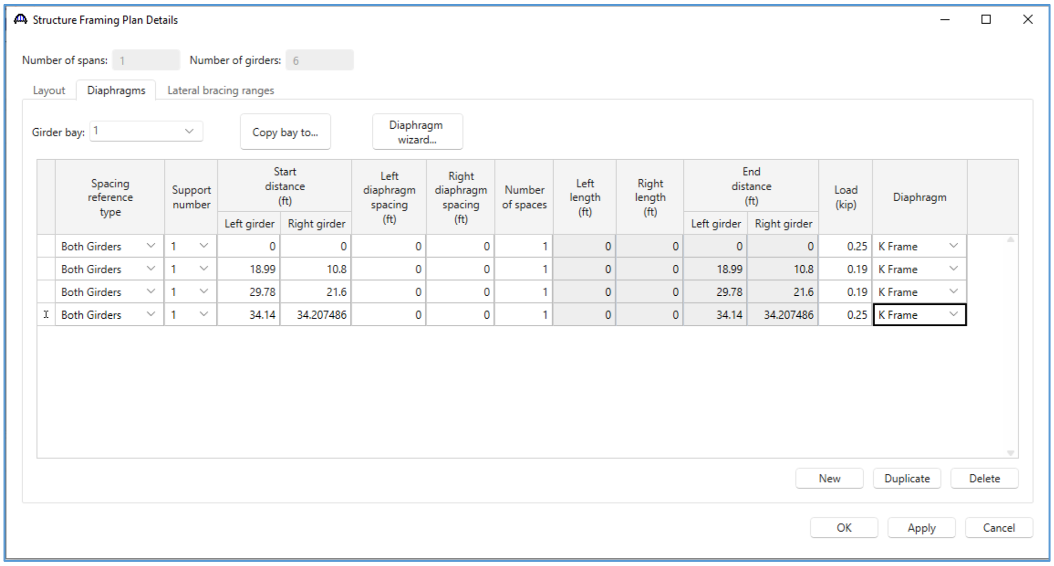


Next input the diaphragms for the girder system structure. The diaphragms must be input within each girder bay. With 6 girders, this structure will have 5 girder bays. When inputting distances along girders, especially for curved structures, try to be precise. The system length tolerance and the structure FEM node tolerance will be used to determine when two points are equal. These tolerances provide some flexibility when inputting distances. Generally, though, it is best to use consistent inputs when entering data in different windows.

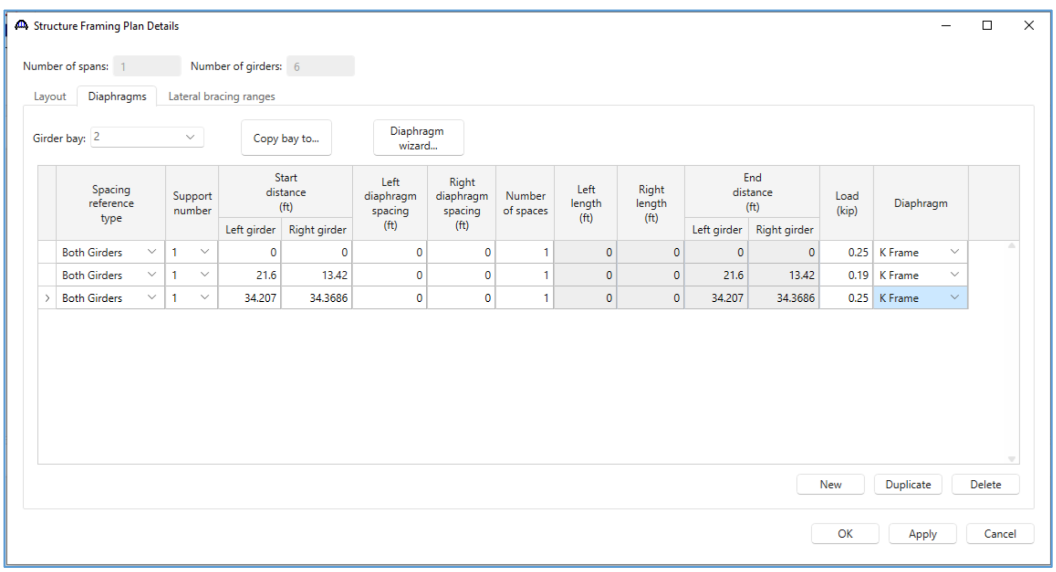
For example, if a girder is 30.0 feet long and the system length tolerance is 0.01 feet, then a diaphragm at 29.97 feet would not be considered at the end of the girder. The finite element model would include nodes at 29.97 ft and 30.0 ft. It is recommended to place the diaphragm at 30.0 feet to match the input for the girder. Using a smaller tolerance, for example, 0.001, would mean separate nodes could be generated at 29.995 ft and 30.000 ft.

Note: The following screenshots were taken after updating the BrDR preferences for the bridge workspace to display the entered number of decimal positions.

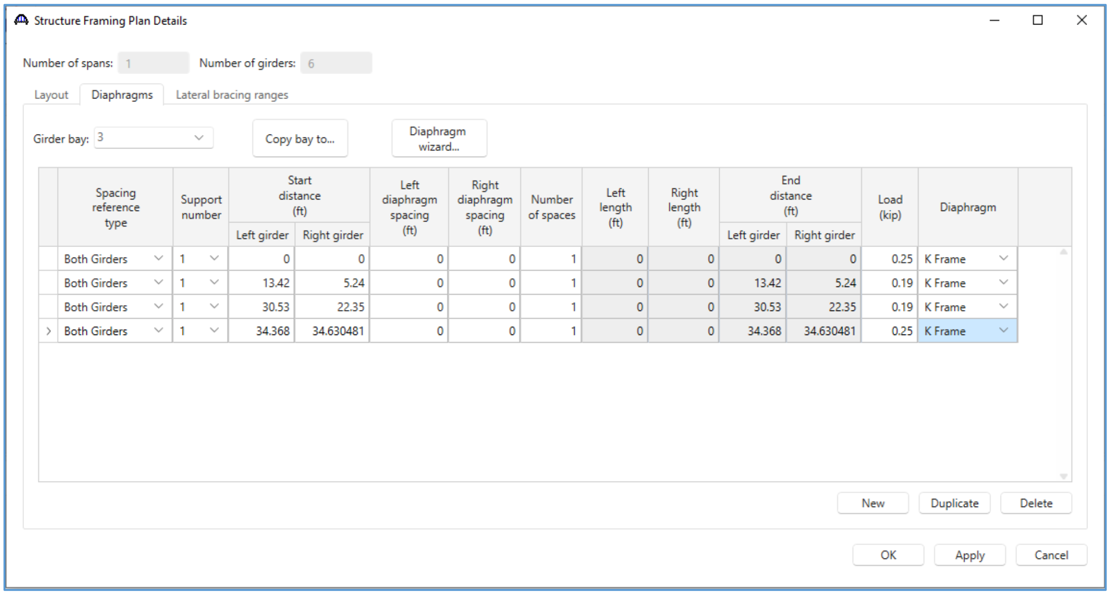
Girder Bay 1:



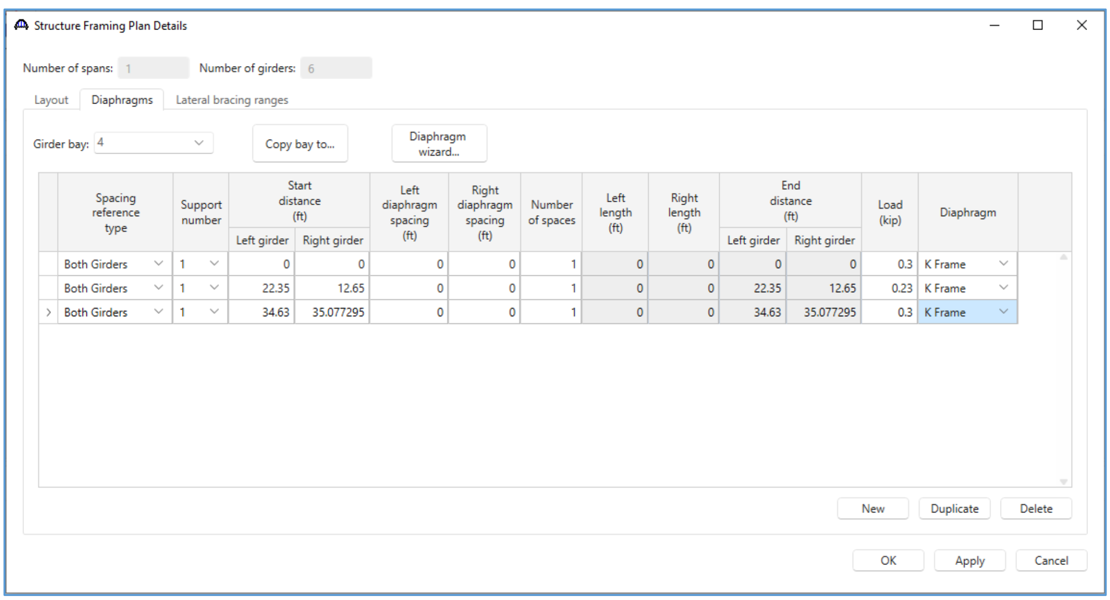
Girder Bay 2:



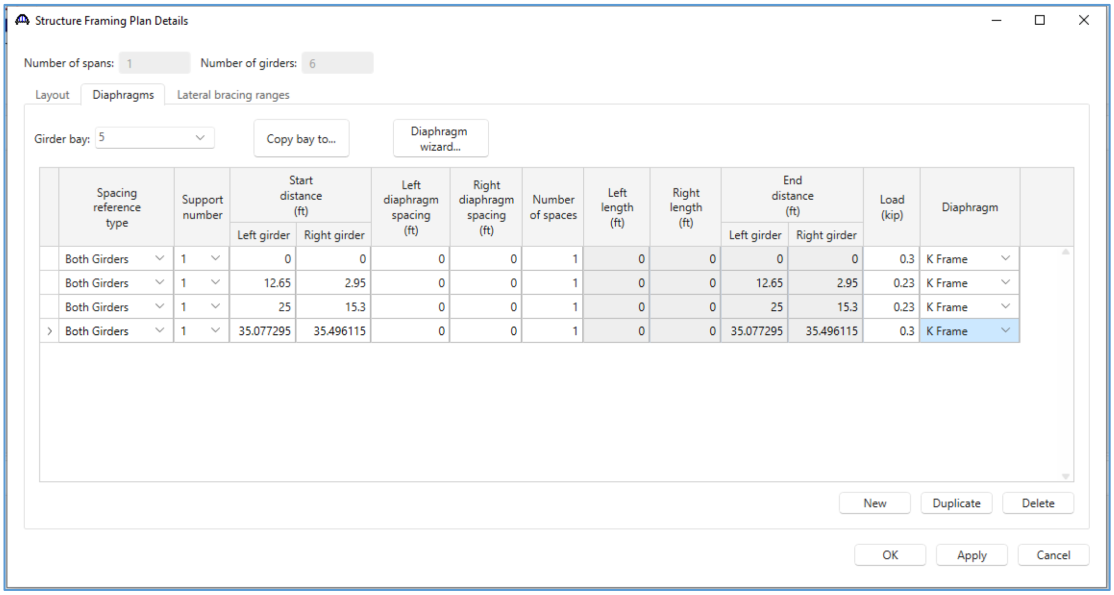
Girder Bay 3:



Girder Bay 4:



Girder Bay 5:



Structure Framing Plan Details

Number of spans: 1 Number of girders: 6

Layout | Diaphragms | Lateral bracing ranges

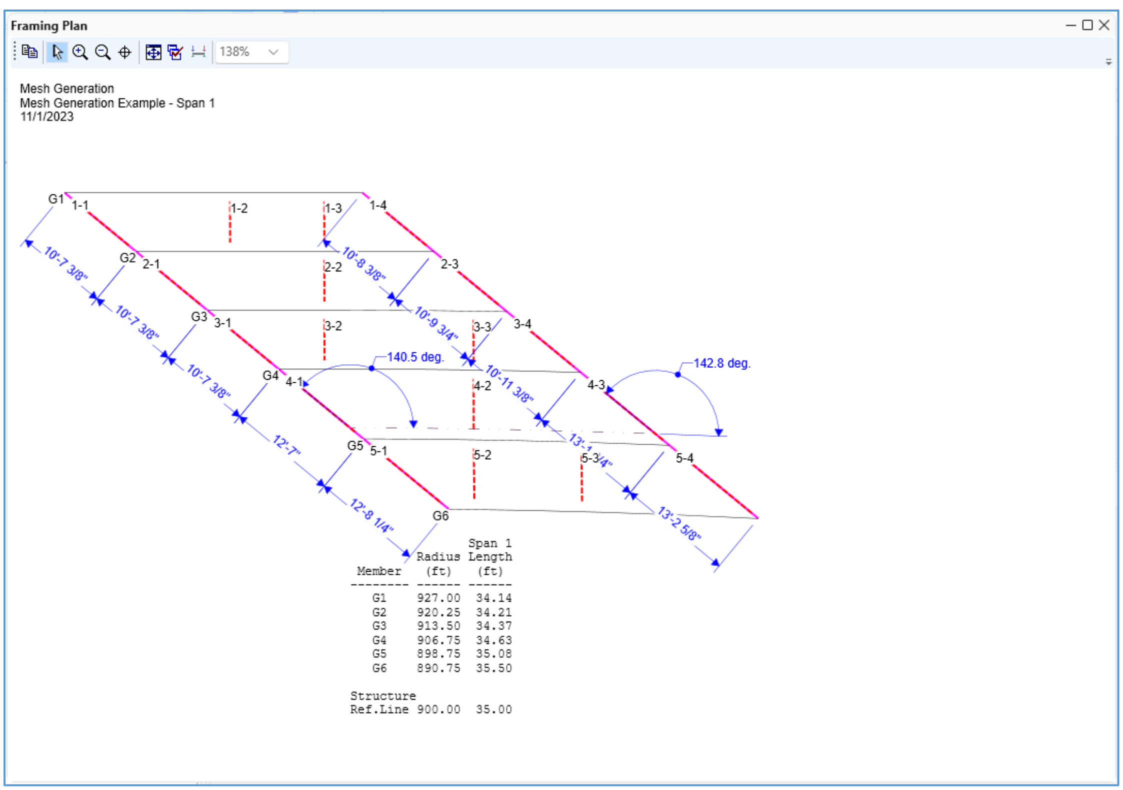
Girder bay: 5 | Copy bay to... | Diaphragm wizard...

Spacing reference type	Support number	Start distance (ft) Left girder	Start distance (ft) Right girder	Left diaphragm spacing (ft)	Right diaphragm spacing (ft)	Number of spaces	Left length (ft)	Right length (ft)	End distance (ft) Left girder	End distance (ft) Right girder	Load (kip)	Diaphragm
Both Girders	1	0	0	0	0	1	0	0	0	0	0.3	K Frame
Both Girders	1	12.65	2.95	0	0	1	0	0	12.65	2.95	0.23	K Frame
Both Girders	1	25	15.3	0	0	1	0	0	25	15.3	0.23	K Frame
Both Girders	1	35.077295	35.496115	0	0	1	0	0	35.077295	35.496115	0.3	K Frame

New | Duplicate | Delete

OK | Apply | Cancel

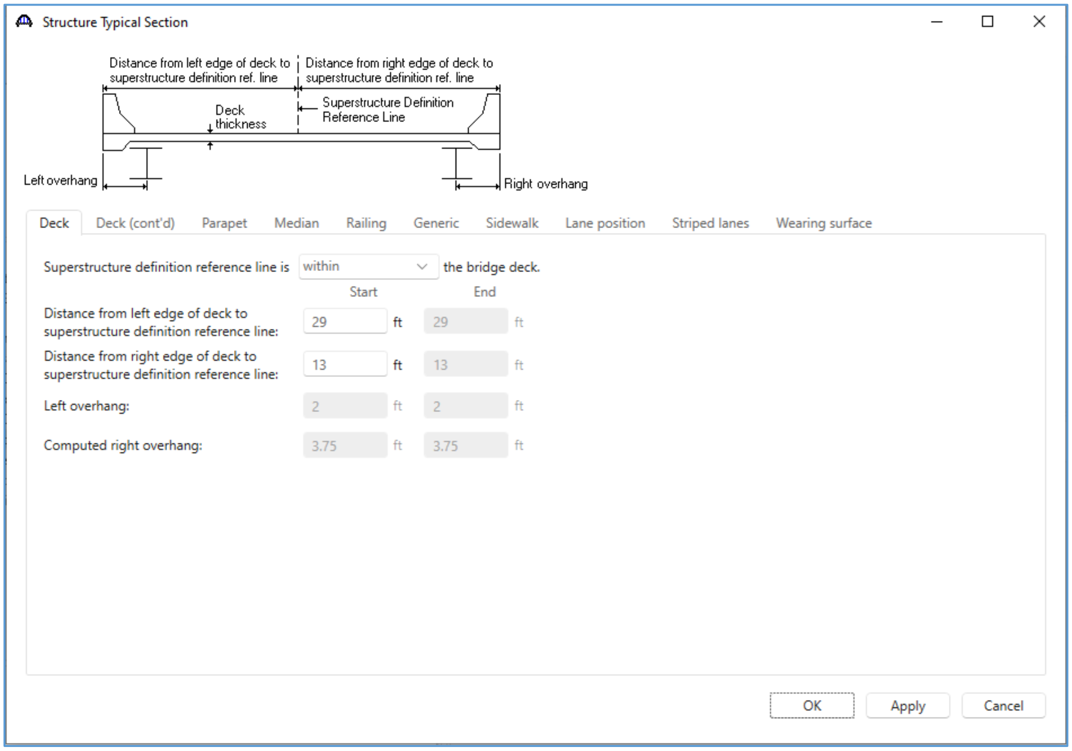
Review the framing plan schematic to verify the framing plan details are correct.



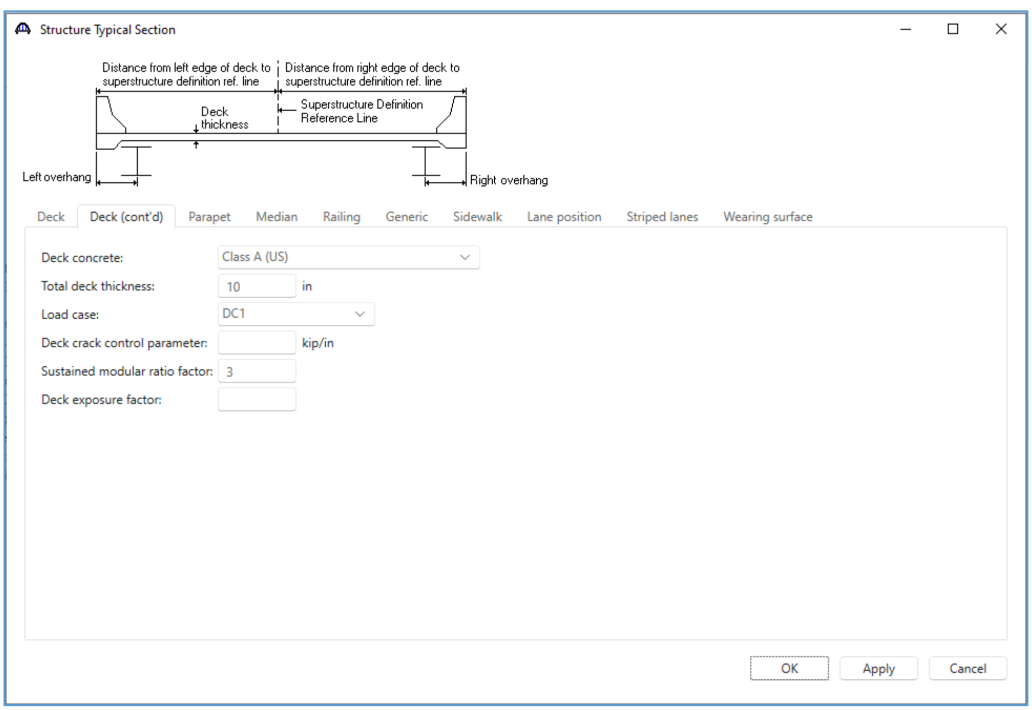
Structure Typical Section

Next, define the structure typical section.

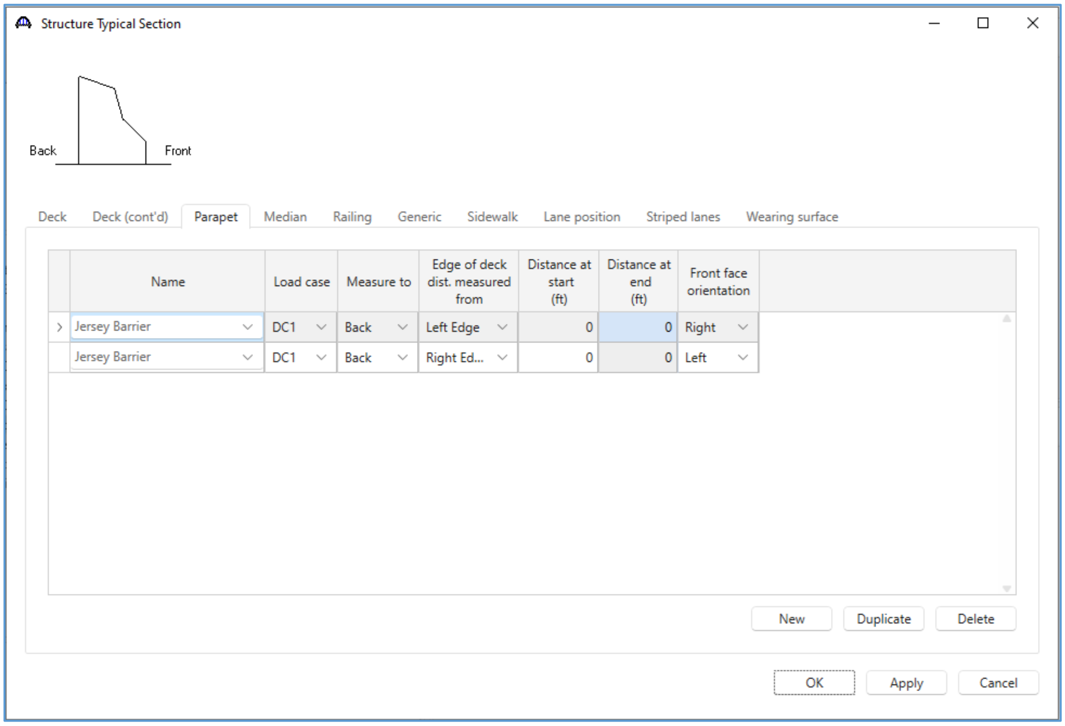
Deck:



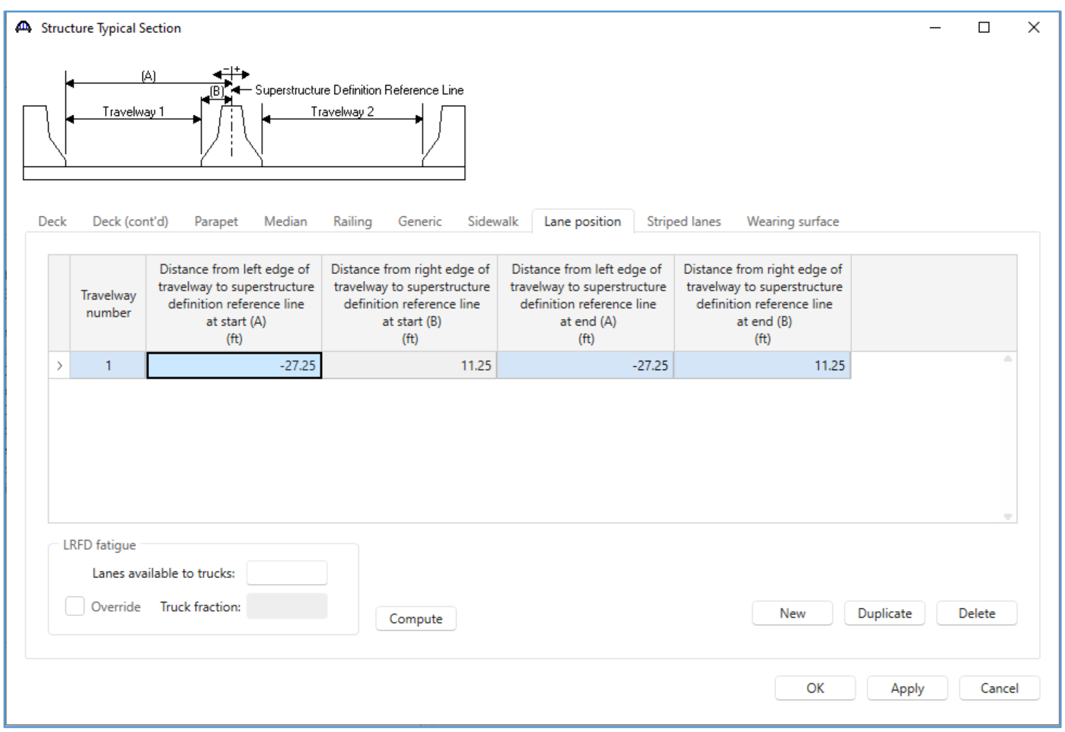
Deck (cont'd):



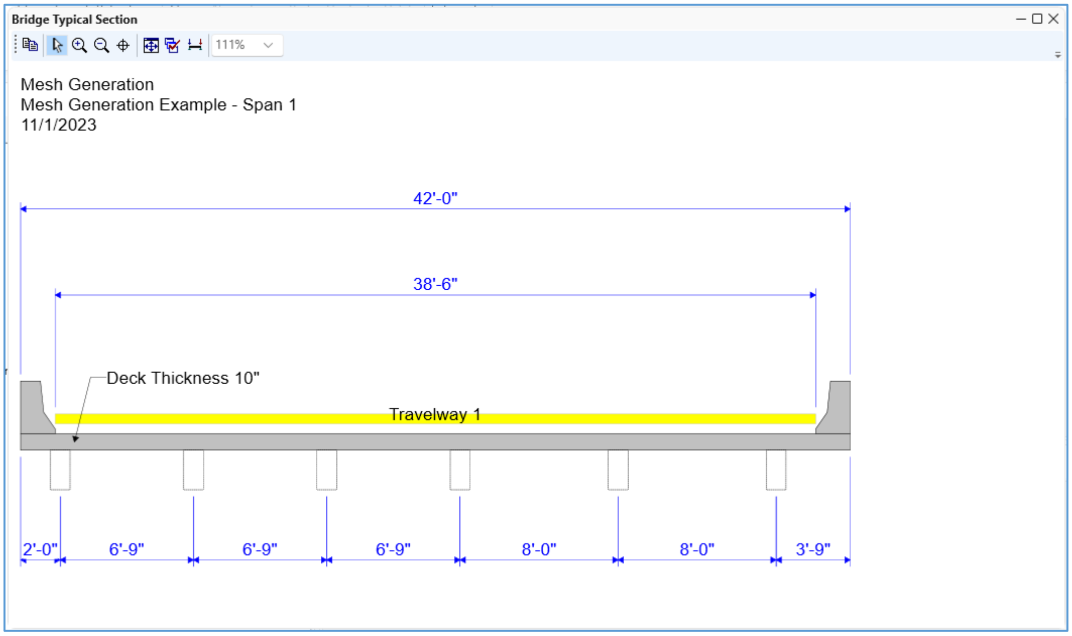
Parapet:



Lane Position:



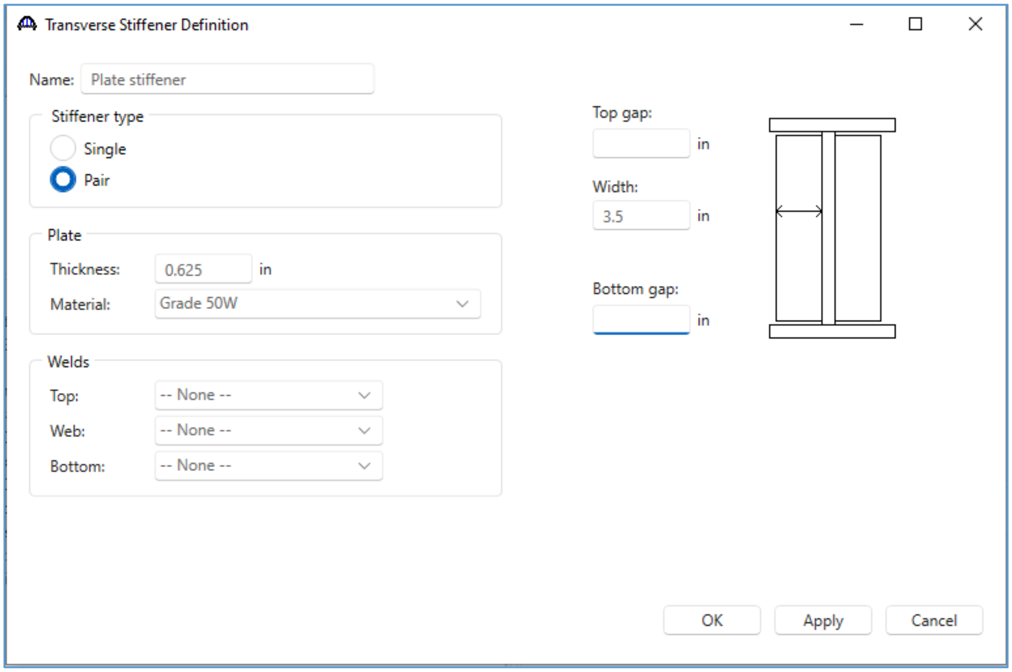
Review the structure typical section schematic to verify the typical section inputs.



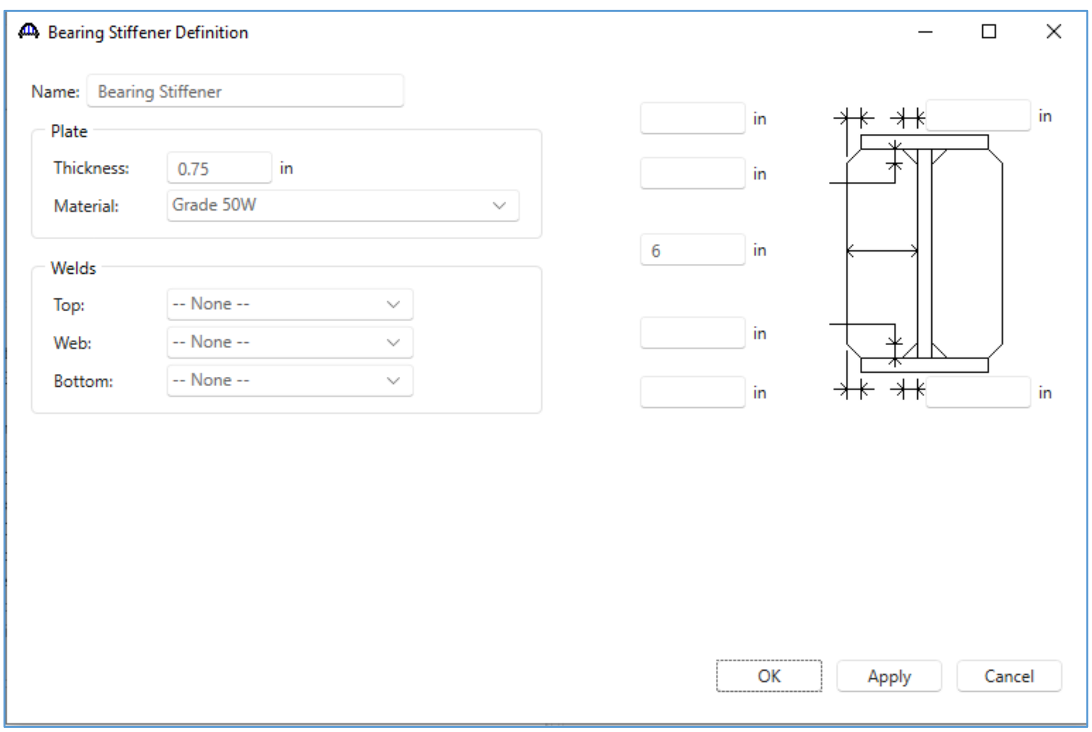
Stiffener Definitions

Add stiffener defintions to the structure. This structure has one transverse stiffener definition and one bearing stiffener definition.

Transverse Stiffener:

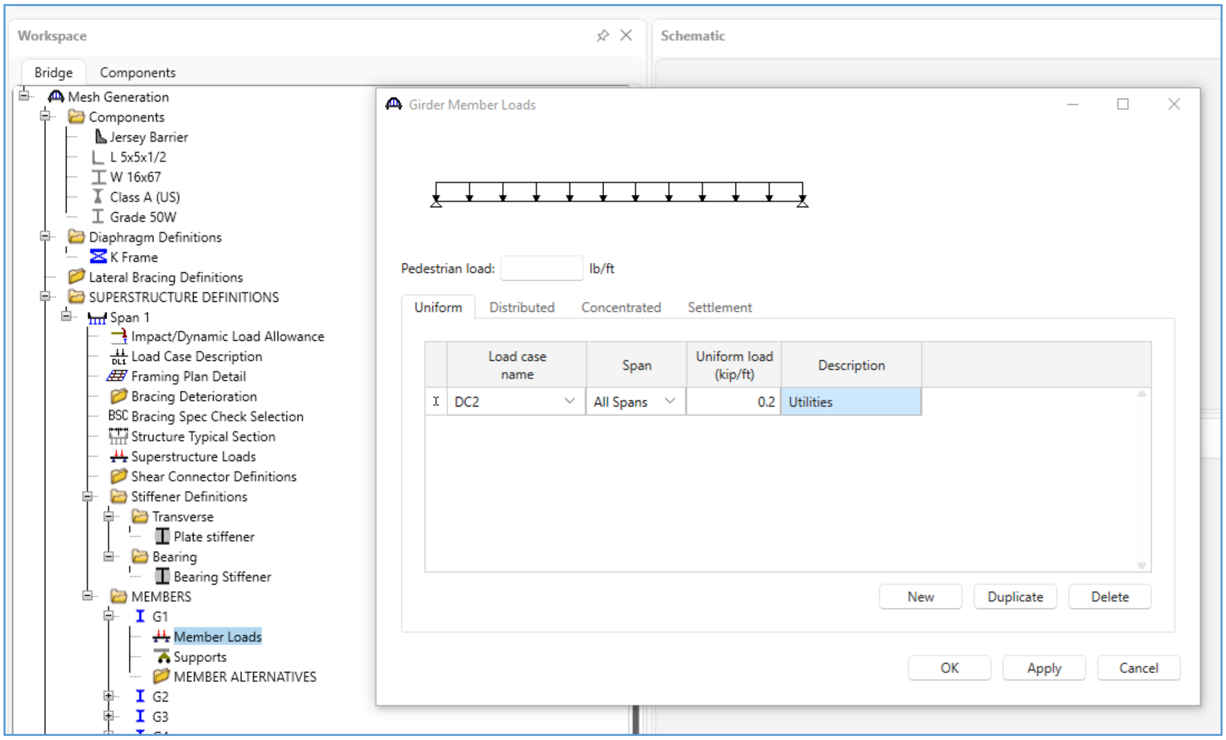


Bearing Stiffener:



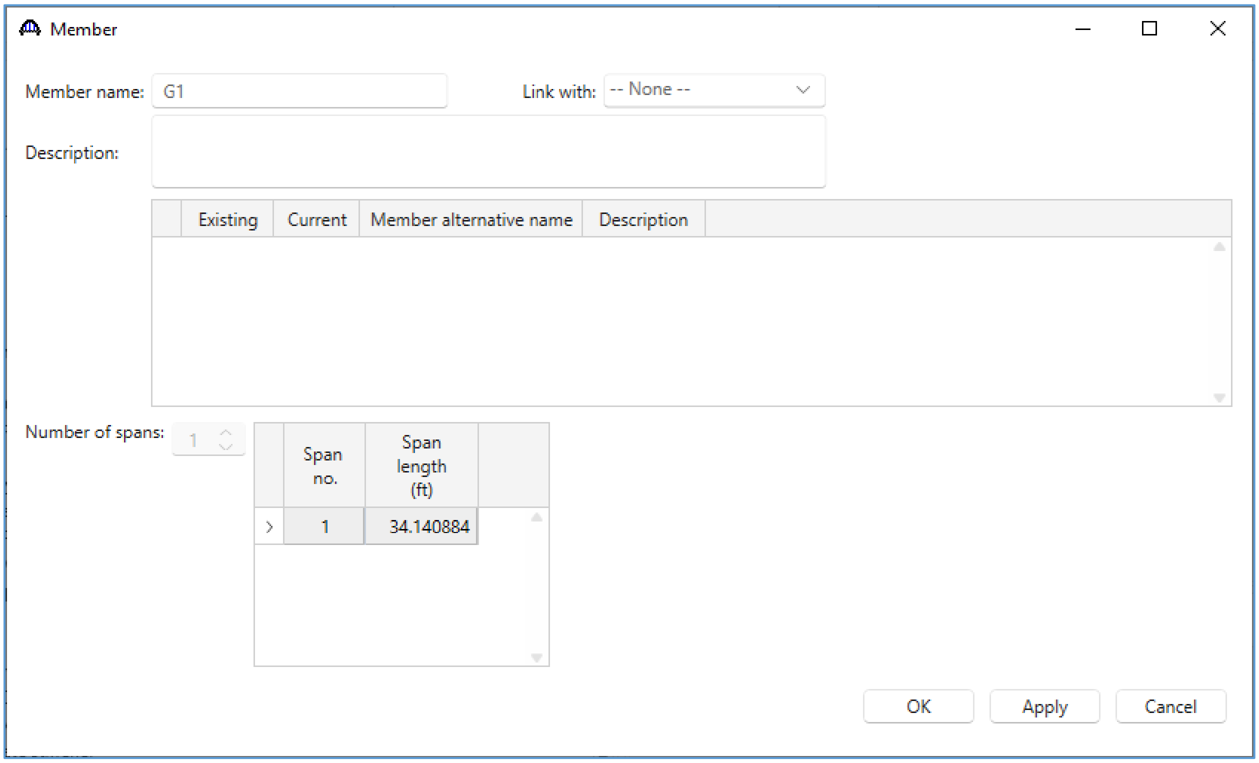
Member Loads

For G1 only, enter an additional member load of 0.2kips/ft.



Member Alternatives

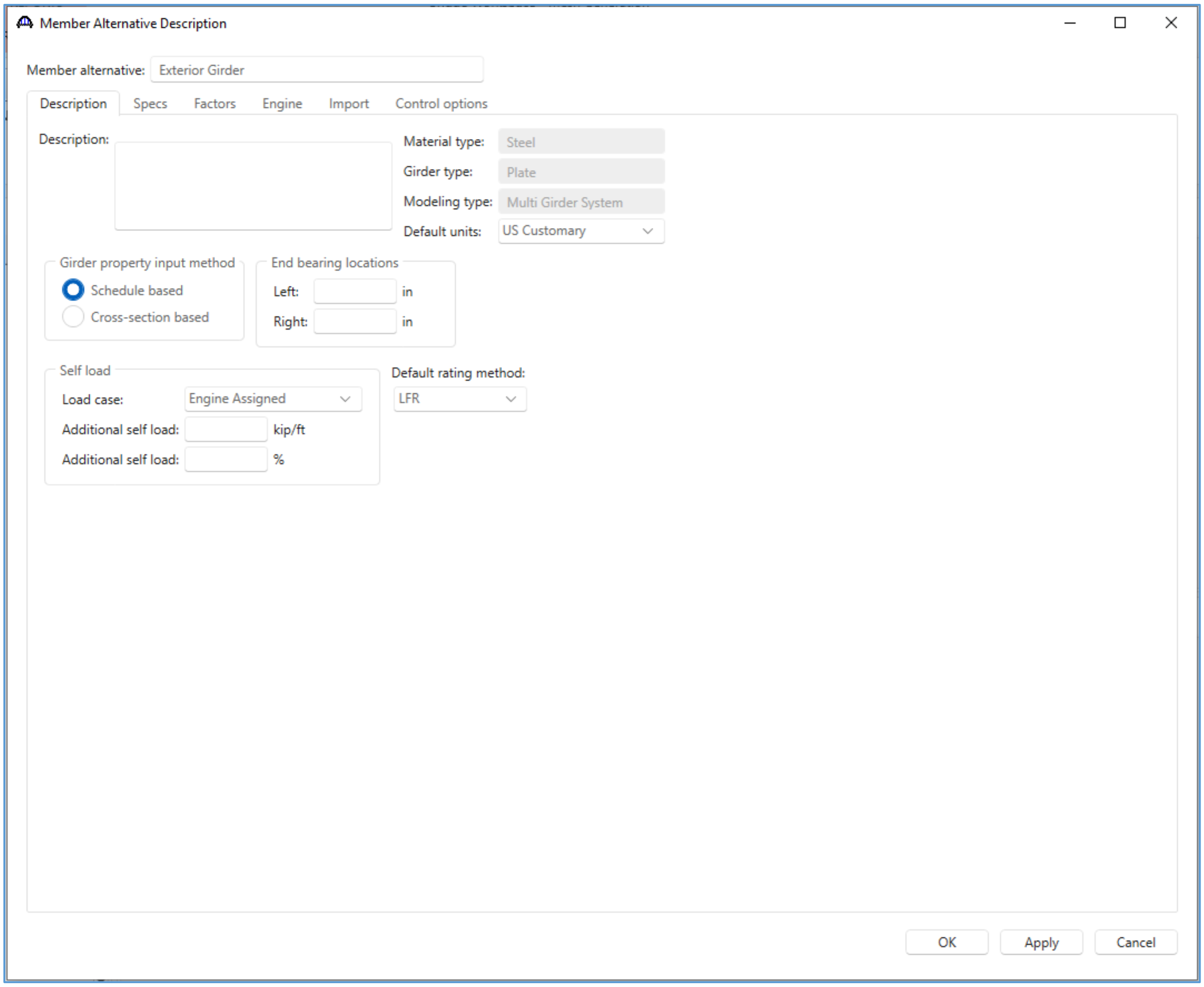
Now add member alternatives for each of the six girders. Since this structure is a curved structure, each girder has a slightly different total length. Each girder has to be input separately. View the calculated length of each girder by opening the girder member definition windows.



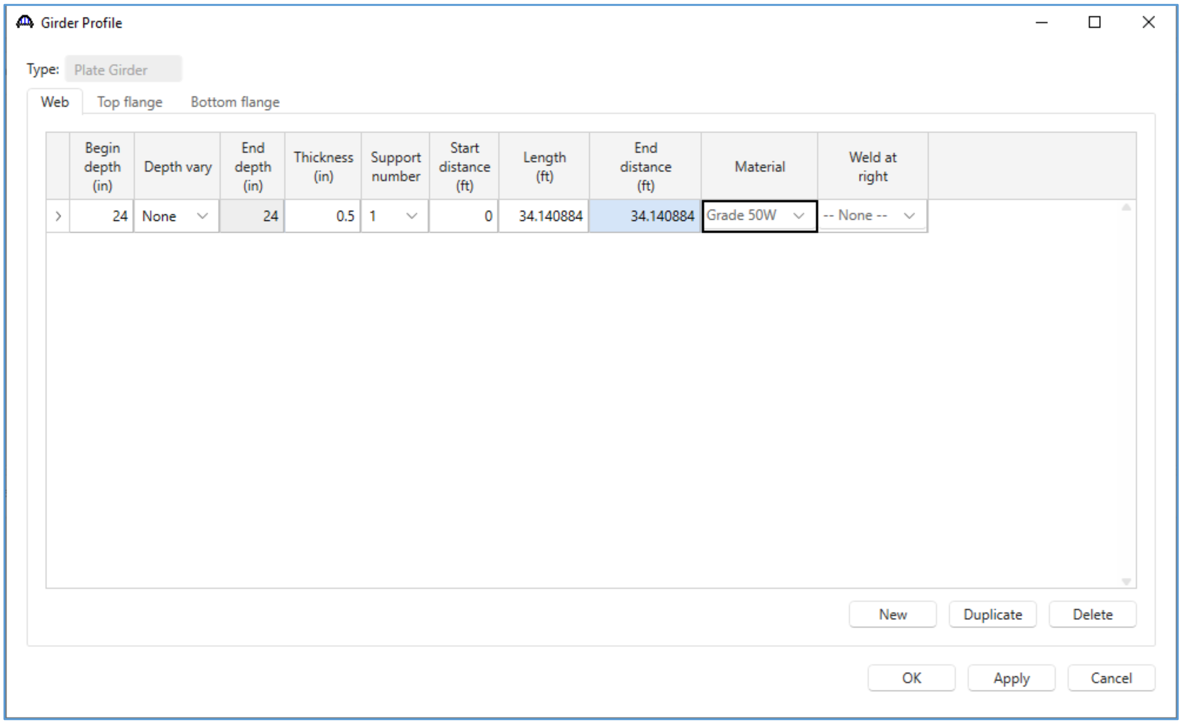
When defining a girder member alternative, it can be helpful to record the total length of the girder member. For a girder member alternative, the girder profile, deck profile, haunch profile and lateral supports will all typically have a total length equal to the girder length.

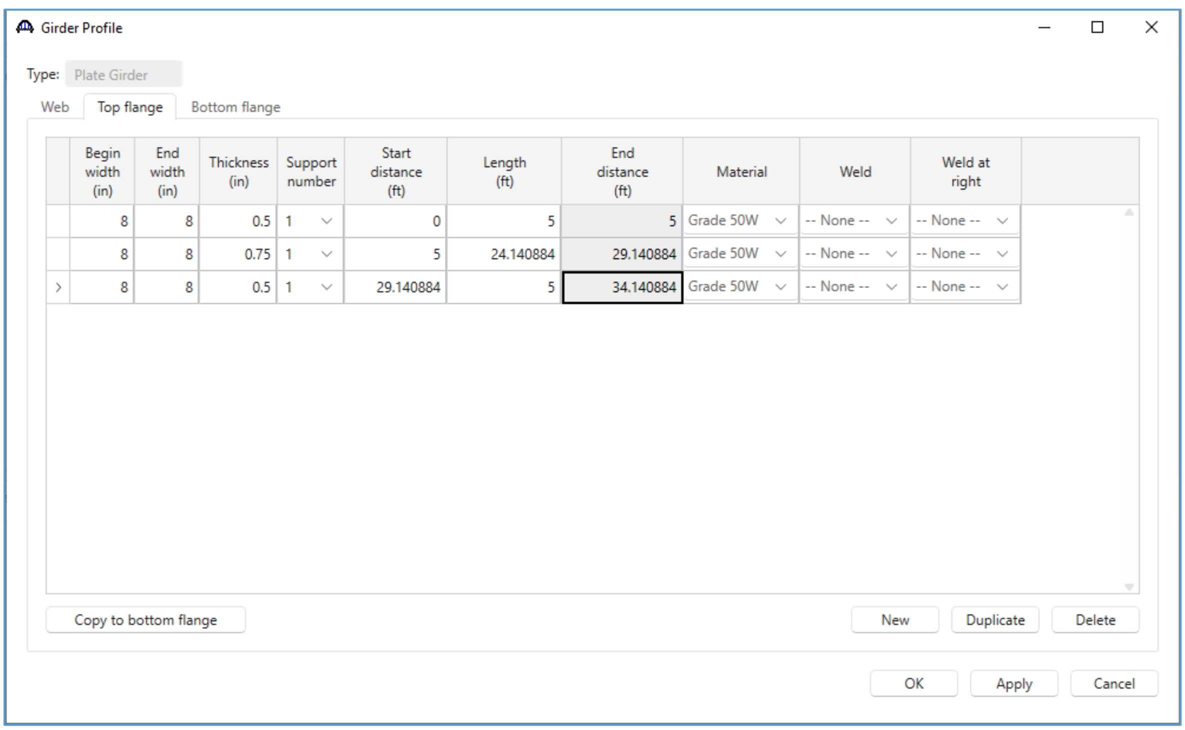
In this structure, all girders are schedule based steel plate girders. The web for each girder has a constant depth of 24” and a width of 0.5”. The top and bottom flanges have a transition at 5 feet from each end of the span. Both flanges have a width of 8” everywhere. The flange thickness for the first and last ranges is 0.5” and the thickness for the middle range is 0.75”.

Girder 1: (Length = 34.140884 ft)
Define the member alternative:

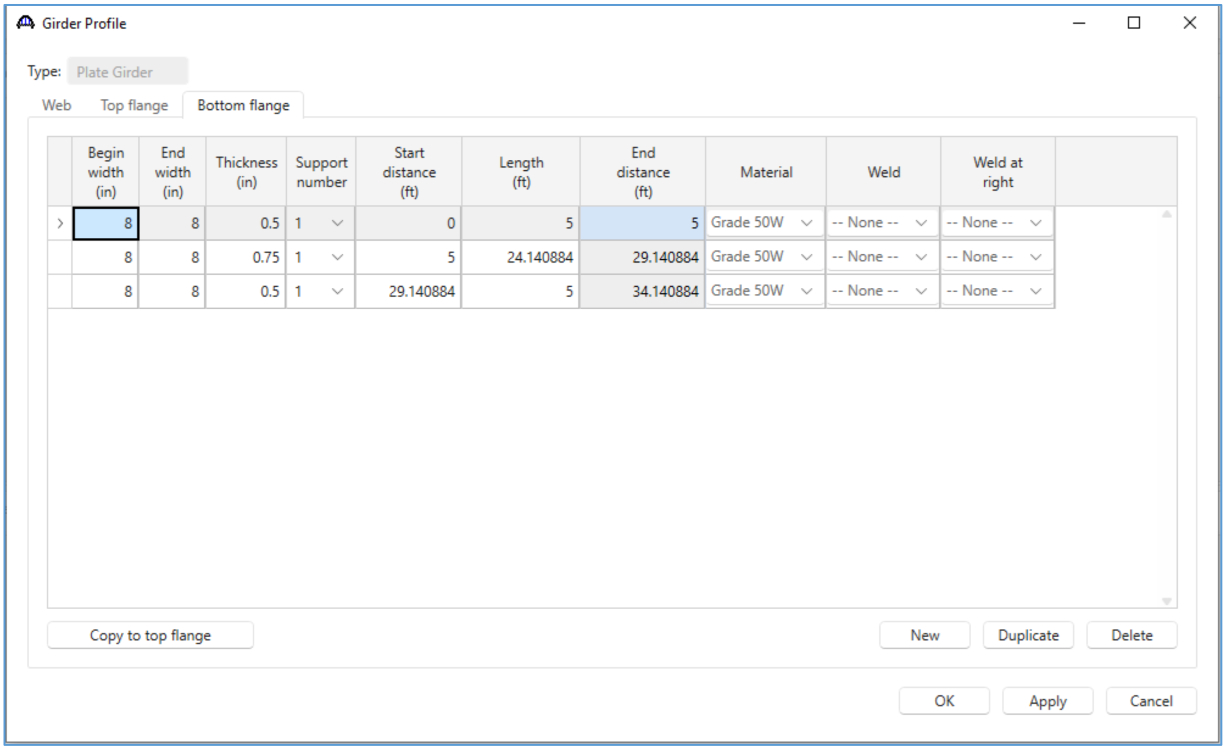


Define the girder profile ranges.



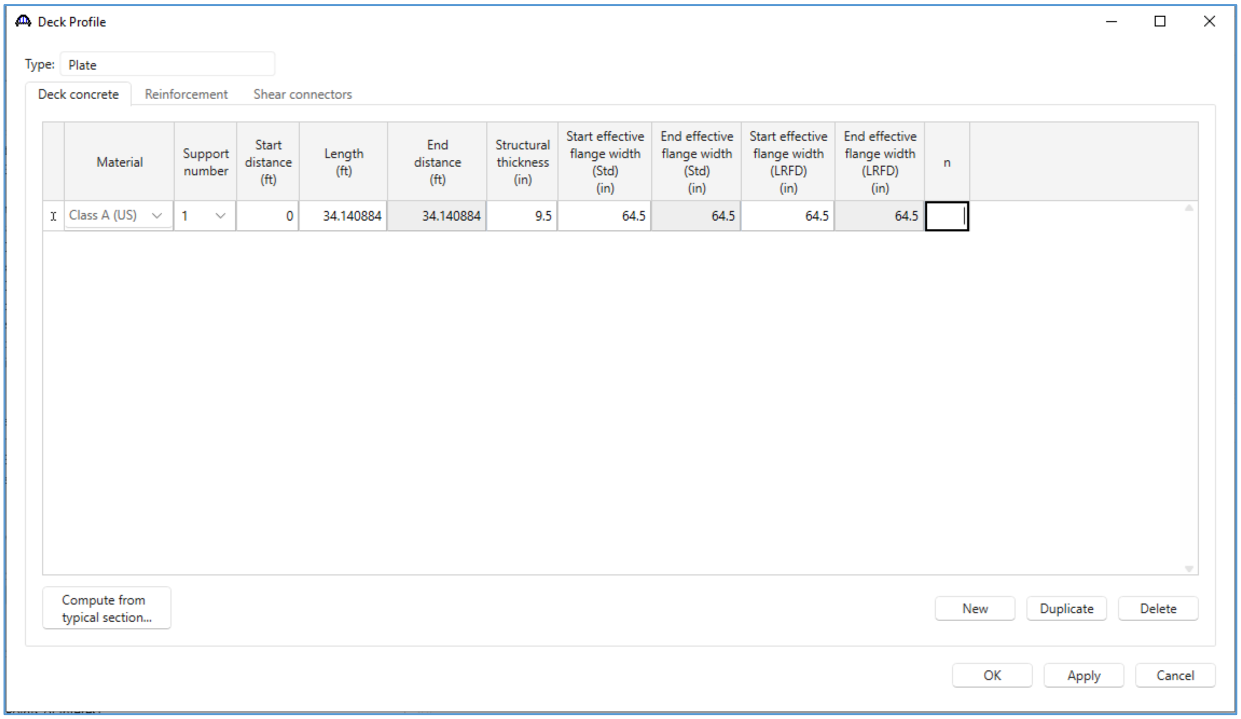


Tip: The top and bottom flanges are the same. Use the ‘Copy to bottom flange’ button to copy the ranges from the top flange to the bottom flange.

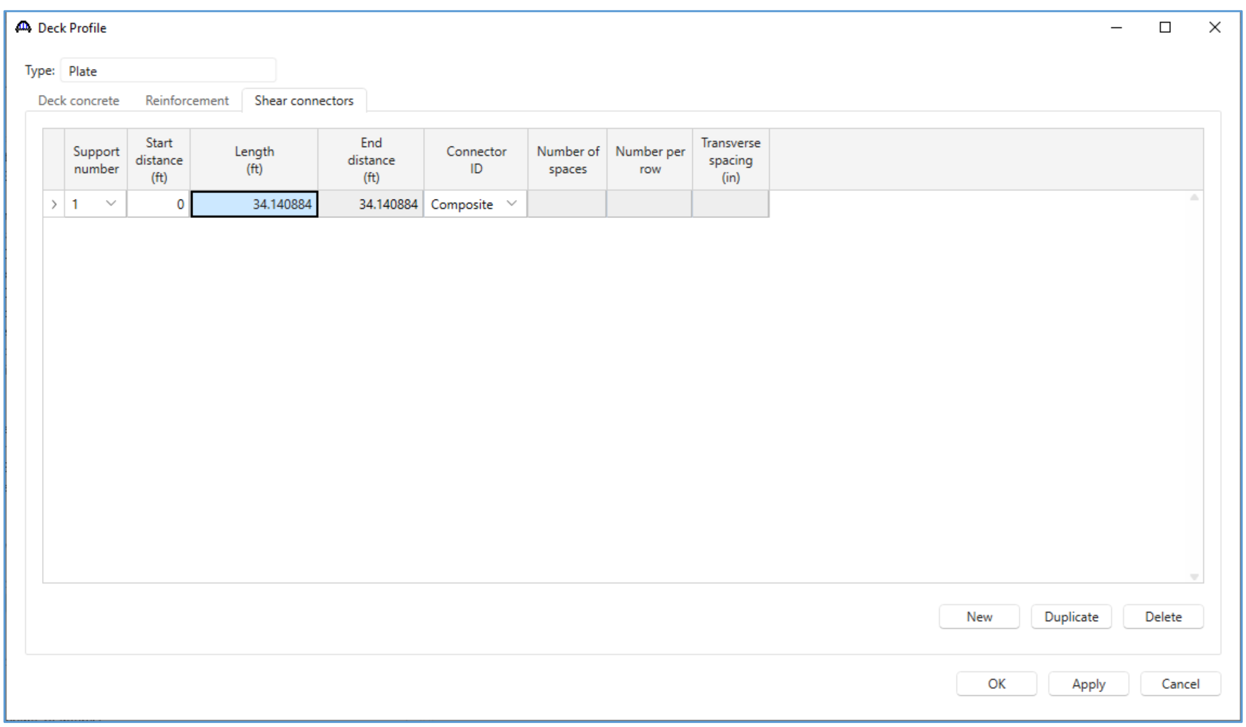

Girder Profile
Type: Plate Girder
Web
Top flange
Bottom flange
Begin width (in)
End width (in)
Thickness (in)
Support number
Start distance (ft)
Length (ft)
End distance (ft)
Material
Weld
Weld at right
8 8 0.5 1 0 5 5 Grade 50W -- None -- -- None --
8 8 0.75 1 5 24.140884 29.140884 Grade 50W -- None -- -- None --
8 8 0.5 1 29.140884 5 34.140884 Grade 50W -- None -- -- None --
Copy to top flange
New
Duplicate
Delete
OK
Apply
Cancel

Deck Profile

Enter the deck profile.

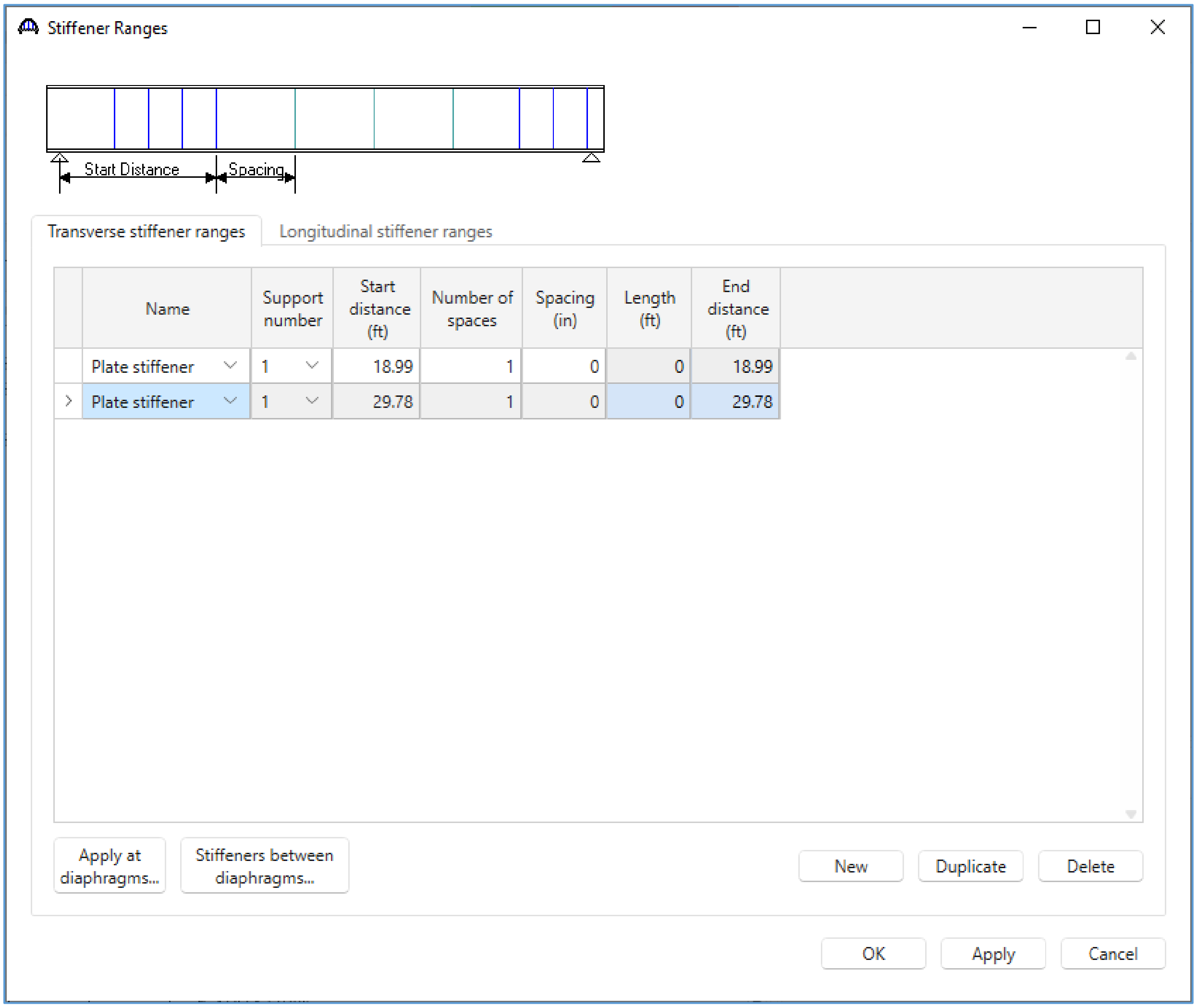


Define the composite region.



Stiffener Ranges

Add the stiffeners. Use the ‘Apply at diaphragms…’ button in the **Stiffener Ranges window** to add the transverse stiffeners at diaphragm locations and the bearing stiffeners at support locations.



Girders 2 – 6:

Follow the same steps as shown for girder 1 to define member alternatives for girders 2 through 6. Refer to the tables below for the girder and deck dimensions. All members should have a composite deck along their full length and have stiffeners at diaphragm locations.

Verify the length of each girder matches the length shown in the table below. If the length is different, review the inputs for the Girder System Superstructure Definition window and the Structure Framing Plan Details window.

Table: Computed Girder Lengths

Girder	Length (ft)
G1	34.140884
G2	34.207486
G3	34.3686
G4	34.630481
G5	35.077295
G6	35.496115

Table: Web Profile Definitions

Girder	Range Start (ft)	Range Length (ft)	Depth (in)	Thickness (in)
G1	0.0	34.140884	24.0	0.5
G2	0.0	34.207486	24.0	0.5
G3	0.0	34.3686	24.0	0.5
G4	0.0	34.630481	24.0	0.5
G5	0.0	35.077295	24.0	0.5
G6	0.0	35.496115	24.0	0.5

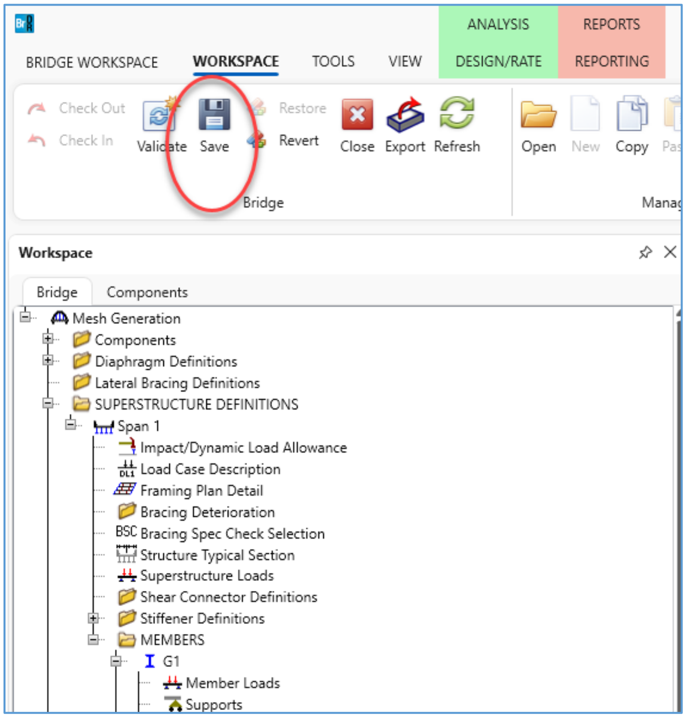
Table: Top and Bottom Flange Profile Definitions

Girder	Range Start (ft)	Range Length (ft)	Width (in)	Thickness (in)
	0.0	5.0	8.0	0.5
G1	5.0	24.140884	8.0	0.75
	29.140884	5.0	8.0	0.5
	0.0	5.0	8.0	0.5
G2	5.0	24.207486	8.0	0.75
	29.207486	5.0	8.0	0.5
	0.0	5.0	8.0	0.5
G3	5.0	24.3686	8.0	0.75
	29.3686	5.0	8.0	0.5
	0.0	5.0	8.0	0.5
G4	5.0	24.630481	8.0	0.75
	29.630481	5.0	8.0	0.5
	0.0	5.0	8.0	0.5
G5	5.0	25.077295	8.0	0.75
	30.077295	5.0	8.0	0.5
	0.0	5.0	8.0	0.5
G6	5.0	25.496115	8.0	0.75
	30.496115	5.0	8.0	0.5

Table: Deck Profile Effective Width

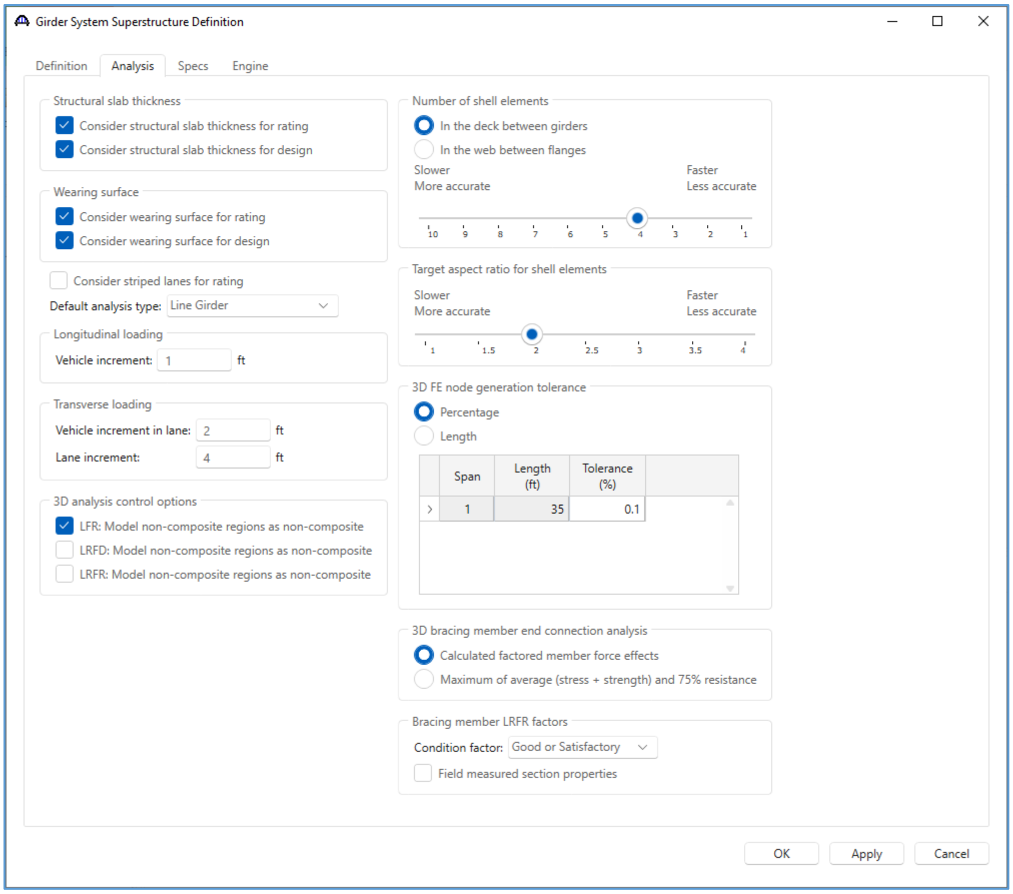
			Standard		LRFD	
Girder	Range Start (ft)	Range Length (ft)	Start Effective Flange Width (in)	End Effective Flange Width (in)	Start Effective Flange Width (in)	End Effective Flange Width (in)
G1	0.0	34.140884	64.5	64.5	64.5	64.5
G2	0.0	34.207486	81.0	81.0	81.0	81.0
G3	0.0	34.3686	81.0	81.0	81.0	81.0
G4	0.0	34.630481	88.5	88.5	88.5	88.5
G5	0.0	35.077295	96.0	96.0	96.0	96.0
G6	0.0	35.496115	93.0	93.0	93.0	93.0

This completes the data entry for the curved girder system structure. Now would be a good time to save the bridge to the database.



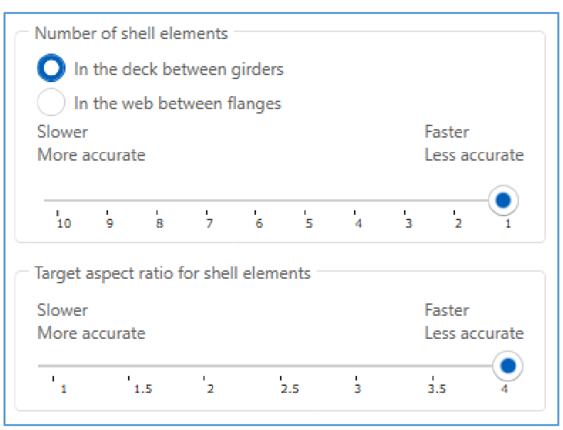
Mesh Generation and Dead Load Only Analysis

Return to the 'Analysis' tab of the Girder System Superstructure Definition window.



Use the 'F1' key to open the window help. The help window includes diagrams and descriptions to describe how these options are used in the analysis. Review the help for the different analysis settings shown in this window.

Modify the deck shell options to generate the coarsest mesh. This may not yield the most accurate results. The results from the coarse mesh can become a benchmark by which to compare finer meshes. Since the analysis will run fastest with the coarse mesh, this will also serve as a quick test to verify the model inputs before attempting a larger analysis.



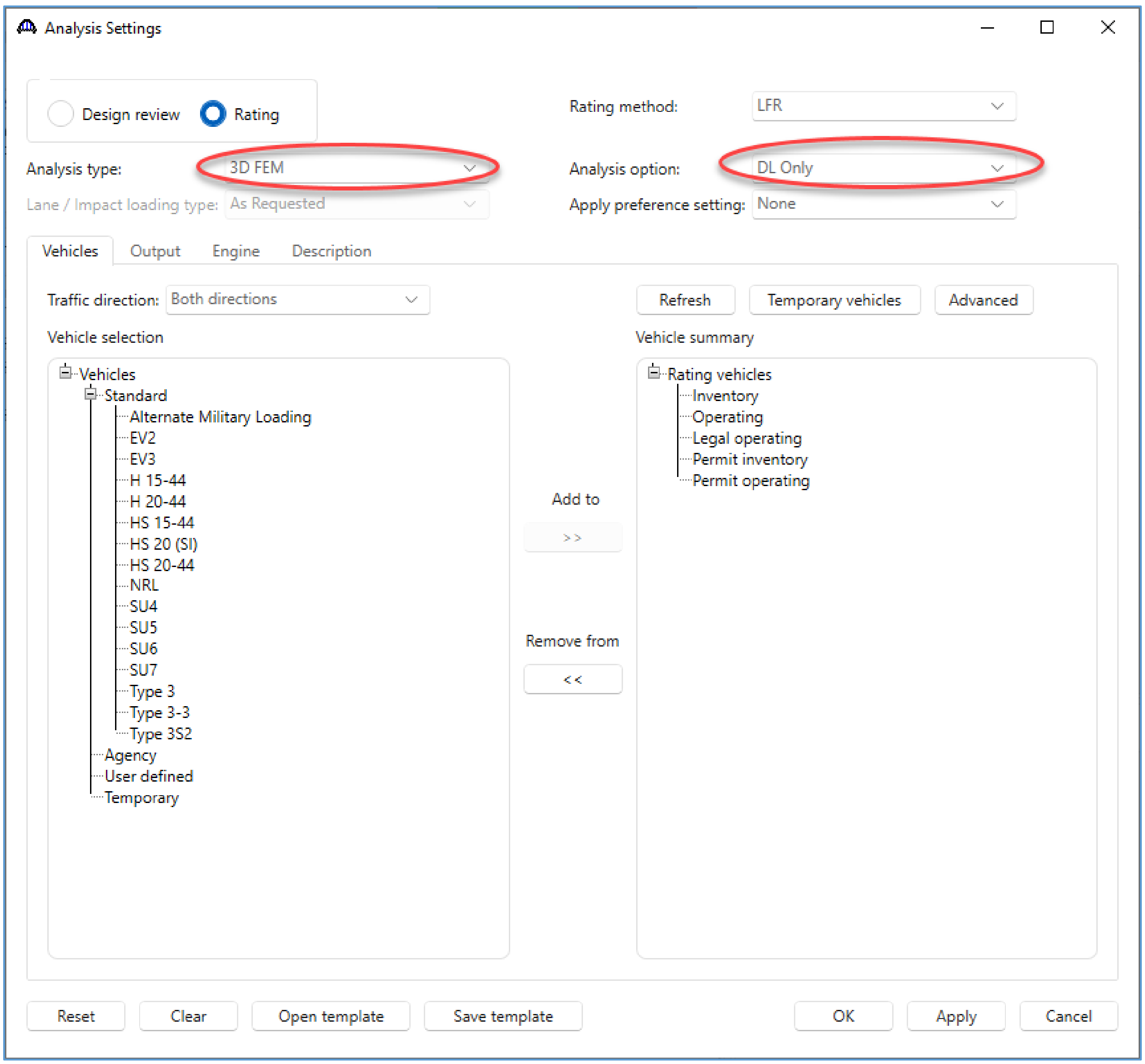
Save the coarse shell mesh generation options to memory by clicking 'OK' on the window.

The dead load only analysis can be used to quickly compare the accuracy of finite element models with increasing levels of mesh refinement. Once an appropriate mesh is selected for the dead load analysis, the live load analysis can be run with greater confidence in the results.

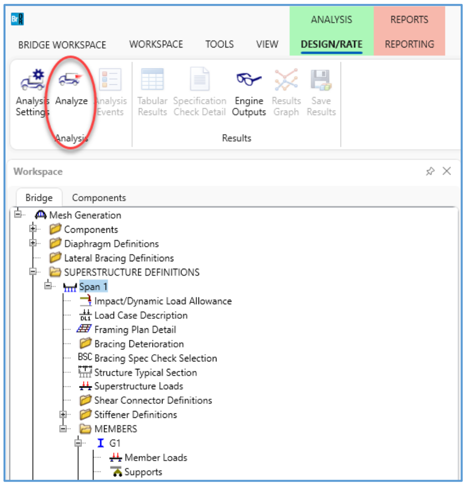
Several techniques can be used to find acceptable options for the mesh generation.

- Inspect the finite element model, checking for appropriate aspect ratios and minimal element distortion.
- Verify the model geometry accurately represents the structure geometry.
- Compare dead load results. Select several dead load cases. Review the results plots for different members to verify the computed actions are reasonable.
- Copy the results from the tabular results and compare the dead load at different levels of mesh refinement. If the differences in dead load between trials is sufficiently small, further mesh refinement may not be necessary.

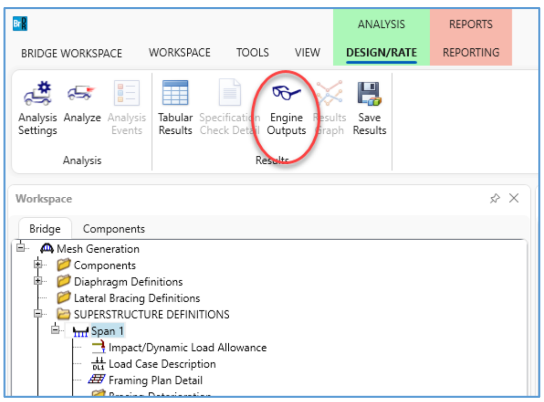
Open the analysis settings window and select a 3D dead load only analysis. For a dead load only analysis, vehicles do not need to be added to the Vehicle summary.



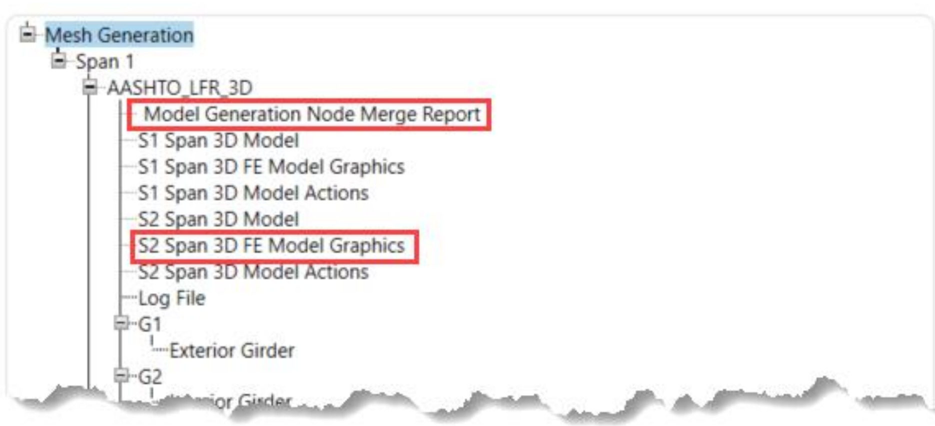
Analyze the girder system superstructure.



After the analysis is complete, open the **Engine Outputs window**.



The S2 Span 3D FE Model Graphics opens a viewer which displays the stage 2 composite finite element model, and the Model Generation Node Merge Report opens a text report on the mesh generation for the girders. Open both and review the output.



Node Merge Report

The node merge report has several sections. The summary section indicates if all nodes in the final mesh are within the desired merge tolerance. When using a larger tolerance to merge nodes, the program may have to exceed the tolerance to preserve section change or support locations. As a result, some points such as diaphragms or tenth points could be shifted a slightly larger distance to maintain equal nodes on each girder. If this occurs the summary section will highlight these points.

The member nodes section reports the process of adding points for each girder member. Points that are within the system length tolerance are merged and considered at the same location.

ModelGenNodeMergeReport - Notepad

File Edit Format View Help

```
Model Generation Node Merge Report
==================================

Summary
-------

SUCCESS: All girder nodes are defined within the desired tolerance!

Member Nodes
------------

System Length Tolerance = 0.001000 (ft)

G1
--

    Distance     Source Type    Node Type      Added At    Span    Member Node
      (ft)                                       (ft)                 In Span
-----------------------------------------------------------------------------
    0.000000     SUPPORT        NEW            0.000000     1          1
    0.000000     DIAPHRAGM      MERGE          0.000000     1
    0.000000     SECTION CHANGE MERGE          0.000000     1
    0.000000     TENTH          MERGE          0.000000     1
    3.414088     TENTH          NEW            3.414088     1          2
    5.000000     SECTION CHANGE NEW            5.000000     1          3
    6.828177     TENTH          NEW            6.828177     1          4
   10.242265     TENTH          NEW           10.242265     1          5
   13.656354     TENTH          NEW           13.656354     1          6
   17.070442     TENTH          NEW           17.070442     1          7
   18.990000     DIAPHRAGM      NEW           18.990000     1          8
   20.484530     TENTH          NEW           20.484530     1          9
   23.898619     TENTH          NEW           23.898619     1          10
   27.312707     TENTH          NEW           27.312707     1          11
   29.140884     SECTION CHANGE NEW           29.140884     1          12
   29.780000     DIAPHRAGM      NEW           29.780000     1          13
   30.726796     TENTH          NEW           30.726796     1          14
   34.140884     SUPPORT        NEW           34.140884     1          15
   34.140884     TENTH          MERGE         34.140884     1
   34.140884     SECTION CHANGE MERGE         34.140884     1
   34.140000     DIAPHRAGM      MERGE         34.140884     1
```

The final section of the report lists the nodes along each girder after merging nodes to be at equal percentages along each span of each girder in the structure. When the source type for a node is “Added for equal percent”, a node was added to the girder to match the percent for a required node along a different girder.

ModelGenNodeMergeReport - Notepad

File Edit Format View Help

Structure Nodes

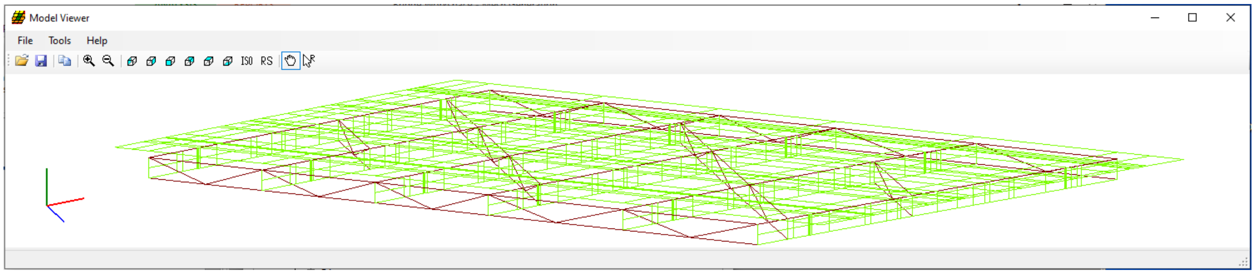
Span	Length (ft)	Percent Tolerance (%)	Equivalent Length Tolerance (ft)
1	35.000000	0.100000	0.035000

G1

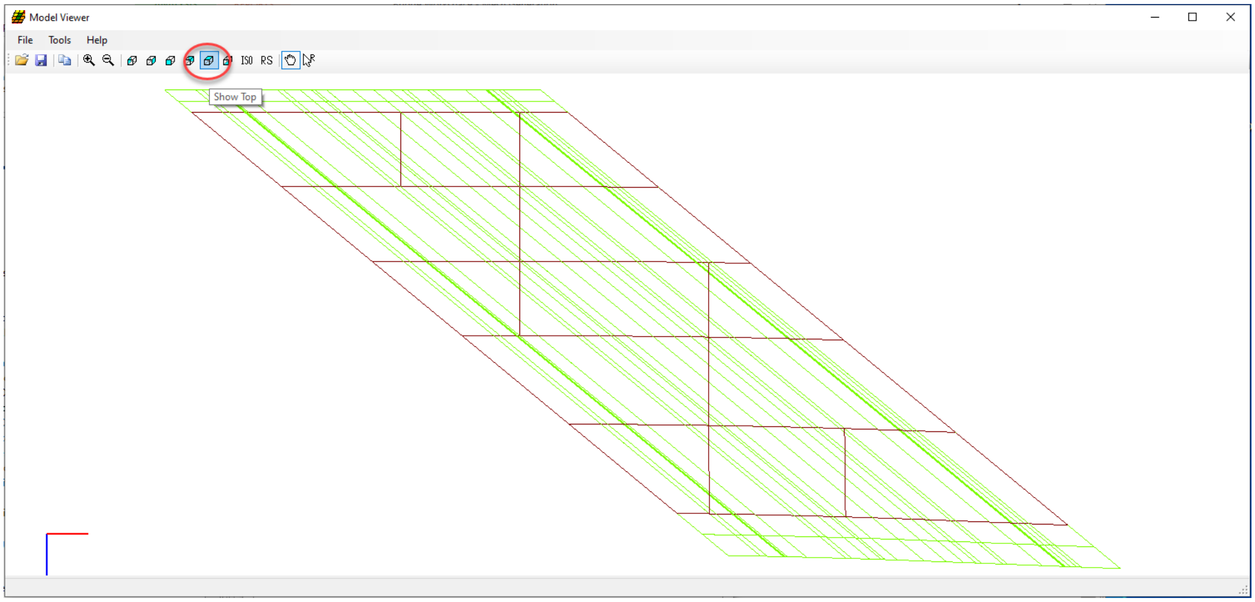
Node	Member Nodes	Distance (ft)	X (ft)	Y (ft)	Z (ft)	Span	Percent (%)	Max Shift (ft)	Source Type
1	1	0.000000	-32.753619	0.000000	-27.000000	1	0.000000	0.000000	SECTION CHANGE, SUPPORT, TENTH, DIAPI
2	n/a	2.837370	-29.916249	0.000000	-27.000000	1	8.310769	n/a	ADDED FOR EQUAL PERCENT
3	2	3.414088	-29.339531	0.000000	-27.000000	1	10.000000	0.000000	TENTH
4	n/a	4.809102	-27.944518	0.000000	-27.000000	1	14.086049	n/a	ADDED FOR EQUAL PERCENT
5	n/a	4.866522	-27.887097	0.000000	-27.000000	1	14.254235	n/a	ADDED FOR EQUAL PERCENT
6	n/a	4.929311	-27.824308	0.000000	-27.000000	1	14.438148	n/a	ADDED FOR EQUAL PERCENT
7	3	5.000000	-27.753619	0.000000	-27.000000	1	14.645198	0.000000	SECTION CHANGE
8	n/a	5.165918	-27.587701	0.000000	-27.000000	1	15.131179	n/a	ADDED FOR EQUAL PERCENT
9	4	6.828177	-25.925442	0.000000	-27.000000	1	20.000000	0.000000	TENTH
10	5	10.242265	-22.511354	0.000000	-27.000000	1	30.000000	0.000000	TENTH
11	n/a	10.778972	-21.974647	0.000000	-27.000000	1	31.572036	n/a	ADDED FOR EQUAL PERCENT
12	n/a	12.312300	-20.441319	0.000000	-27.000000	1	36.063214	n/a	ADDED FOR EQUAL PERCENT
13	n/a	13.331083	-19.422536	0.000000	-27.000000	1	39.047270	n/a	ADDED FOR EQUAL PERCENT
14	6	13.656354	-19.097266	0.000000	-27.000000	1	40.000000	0.000000	TENTH
15	n/a	14.715851	-18.037768	0.000000	-27.000000	1	43.103309	n/a	ADDED FOR EQUAL PERCENT
16	7	17.070442	-15.683177	0.000000	-27.000000	1	50.000000	0.000000	TENTH
17	8	18.990000	-13.763619	0.000000	-27.000000	1	55.622461	0.000000	DIAPHRAGM
18	9	20.484530	-12.269089	0.000000	-27.000000	1	60.000000	0.000000	TENTH
19	n/a	21.557945	-11.195675	0.000000	-27.000000	1	63.144073	n/a	ADDED FOR EQUAL PERCENT
20	n/a	22.034021	-10.719598	0.000000	-27.000000	1	64.538519	n/a	ADDED FOR EQUAL PERCENT
21	10	23.898619	-8.855000	0.000000	-27.000000	1	70.000000	0.000000	TENTH

3D Model Graphics

Use the toolbar in the Model Viewer to scale, pan, and rotate the finite element model.



To better view the meshing of the deck shell elements, use the 'Show Top' option in the ribbon.

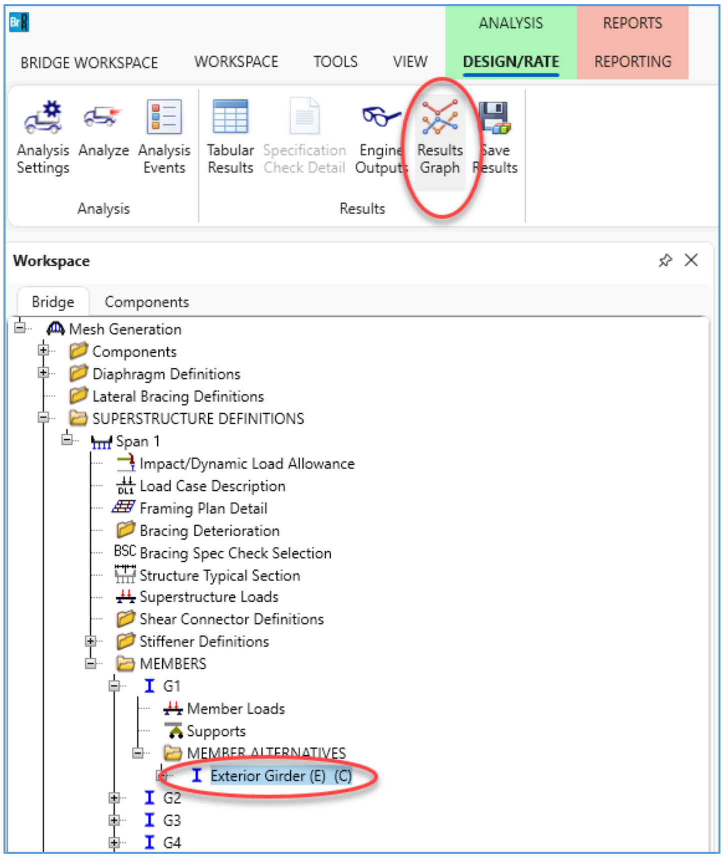


From this view we can see the deck is composed of a series of closely spaced elements. Adding additional elements in the deck between the girders would improve the mesh.

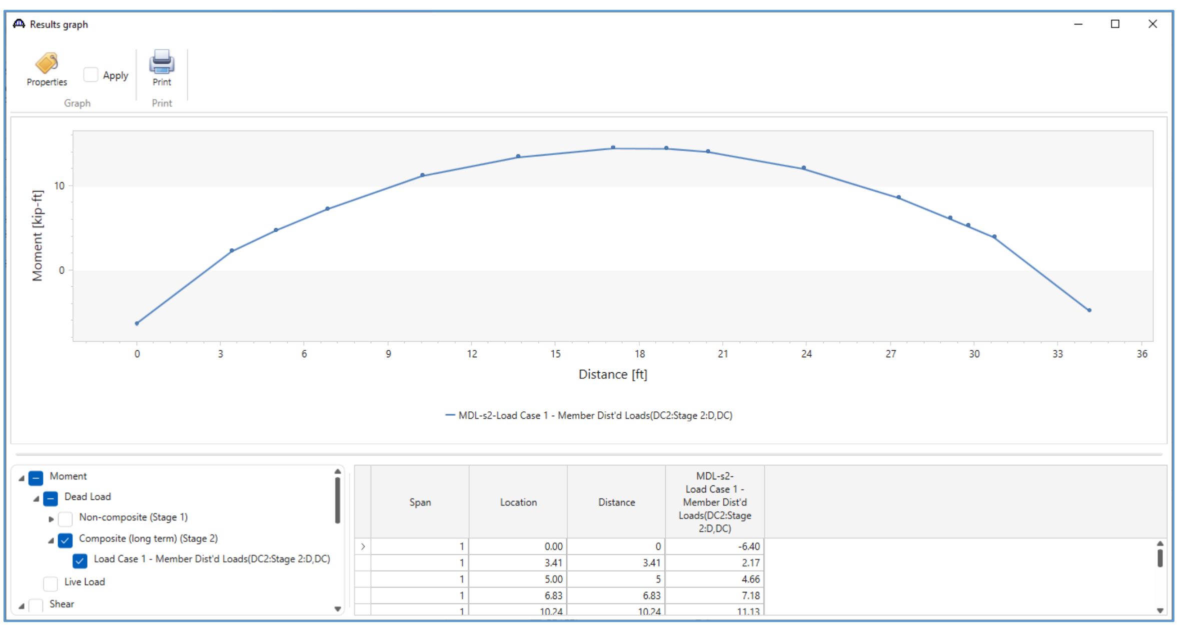
The finite element model for this mesh has 1156 total nodes.

Results Graph

To review the results plot, select a member alternative in the bridge workspace tree and open the **Results Graph** from the Design/Rate ribbon.



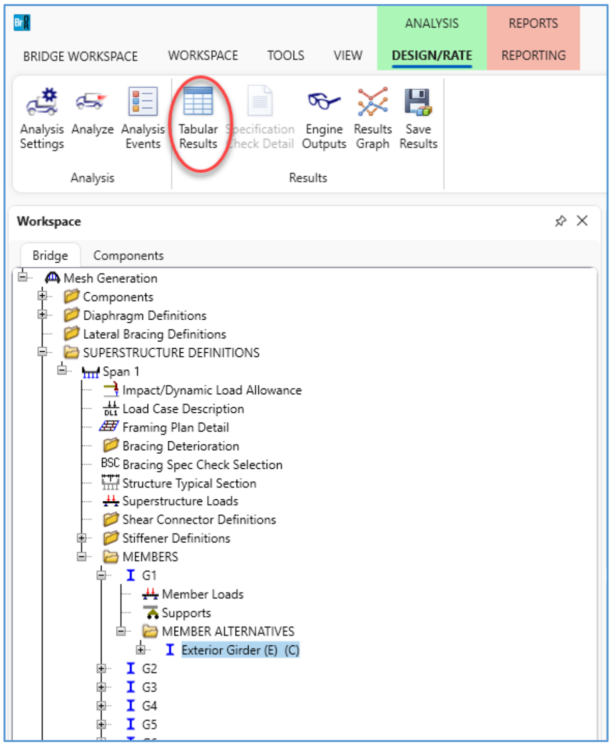
This is the moment from the additional member uniform load assigned to G1 for the utilities.

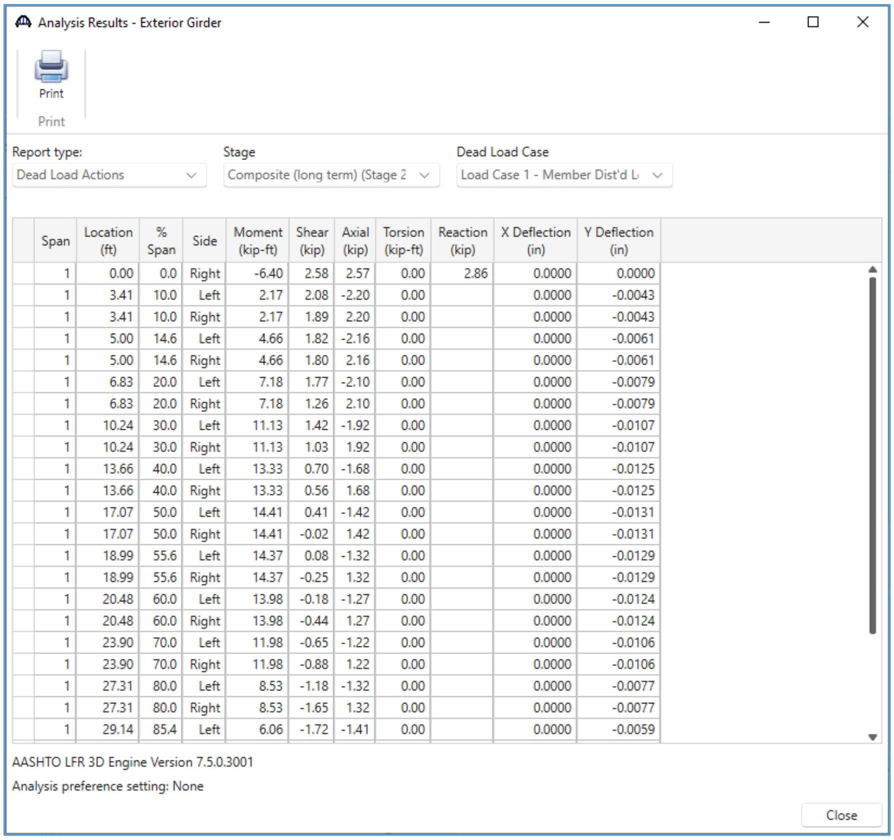


Span	Location	Distance	MDL-s2-Load Case 1 - Member Dist'd Loads(DC2:Stage 2:D,DC)
1	0.00	0	-6.40
1	3.41	3.41	2.17
1	5.00	5	4.66
1	6.83	6.83	7.18
1	10.24	10.24	11.13

Tabular Results

The same load case data is available in a tabular format in the **Tabular Results window**.





Span	Location (ft)	% Span	Side	Moment (kip-ft)	Shear (kip)	Axial (kip)	Torsion (kip-ft)	Reaction (kip)	X Deflection (in)	Y Deflection (in)
1	0.00	0.0	Right	-6.40	2.58	2.57	0.00	2.86	0.0000	0.0000
1	3.41	10.0	Left	2.17	2.08	-2.20	0.00		0.0000	-0.0043
1	3.41	10.0	Right	2.17	1.89	2.20	0.00		0.0000	-0.0043
1	5.00	14.6	Left	4.66	1.82	-2.16	0.00		0.0000	-0.0061
1	5.00	14.6	Right	4.66	1.80	2.16	0.00		0.0000	-0.0061
1	6.83	20.0	Left	7.18	1.77	-2.10	0.00		0.0000	-0.0079
1	6.83	20.0	Right	7.18	1.26	2.10	0.00		0.0000	-0.0079
1	10.24	30.0	Left	11.13	1.42	-1.92	0.00		0.0000	-0.0107
1	10.24	30.0	Right	11.13	1.03	1.92	0.00		0.0000	-0.0107
1	13.66	40.0	Left	13.33	0.70	-1.68	0.00		0.0000	-0.0125
1	13.66	40.0	Right	13.33	0.56	1.68	0.00		0.0000	-0.0125
1	17.07	50.0	Left	14.41	0.41	-1.42	0.00		0.0000	-0.0131
1	17.07	50.0	Right	14.41	-0.02	1.42	0.00		0.0000	-0.0131
1	18.99	55.6	Left	14.37	0.08	-1.32	0.00		0.0000	-0.0129
1	18.99	55.6	Right	14.37	-0.25	1.32	0.00		0.0000	-0.0129
1	20.48	60.0	Left	13.98	-0.18	-1.27	0.00		0.0000	-0.0124
1	20.48	60.0	Right	13.98	-0.44	1.27	0.00		0.0000	-0.0124
1	23.90	70.0	Left	11.98	-0.65	-1.22	0.00		0.0000	-0.0106
1	23.90	70.0	Right	11.98	-0.88	1.22	0.00		0.0000	-0.0106
1	27.31	80.0	Left	8.53	-1.18	-1.32	0.00		0.0000	-0.0077
1	27.31	80.0	Right	8.53	-1.65	1.32	0.00		0.0000	-0.0077
1	29.14	85.4	Left	6.06	-1.72	-1.41	0.00		0.0000	-0.0059

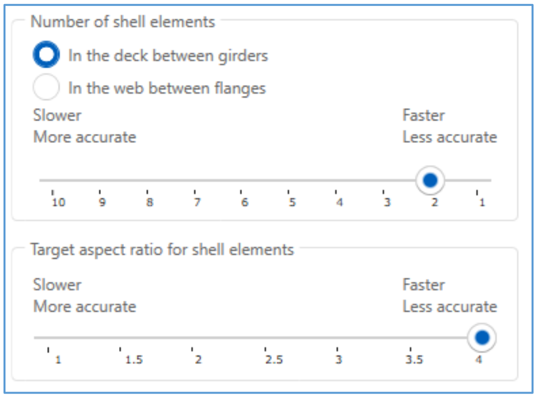
This table can easily be copied into another program to compare the results for different levels of mesh refinement. To select the entire grid, click the top leftmost cell in the table header. To select a portion of the grid, click and drag over the desired cells. Copy the selected cells using the Ctrl-C on your keyboard and paste into another program using Ctrl-V.

NOTE:

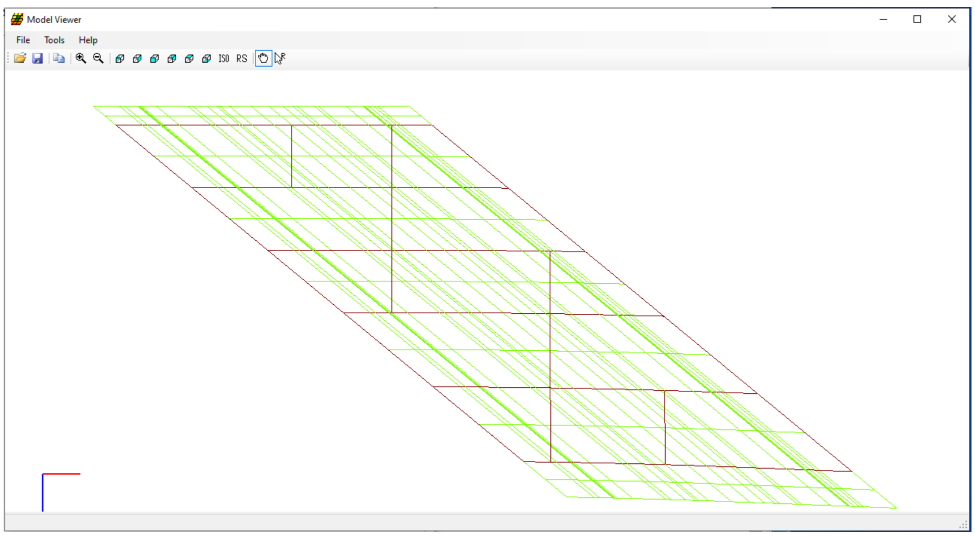
Even though this is a single span structure, the 3D analysis may compute non-zero moments at the supports. These moments at the supports are the result of the structure geometry and the modeled support conditions. A pinned 3D support is assumed to constrain the displacement of the bottom flange of the girder. This support is slightly different from a line girder analysis where a girder is supported at the neutral axis and the bottom flange is free to lengthen under positive flexure.

In the following steps we compare the computed moment at location 0.00 ft as an example of one method of validating the mesh refinement. This location was selected as an example only because it is at the top of the table, any location or action type could be considered.

Refine the finite element mesh by doubling the number of elements in the deck between the girders.

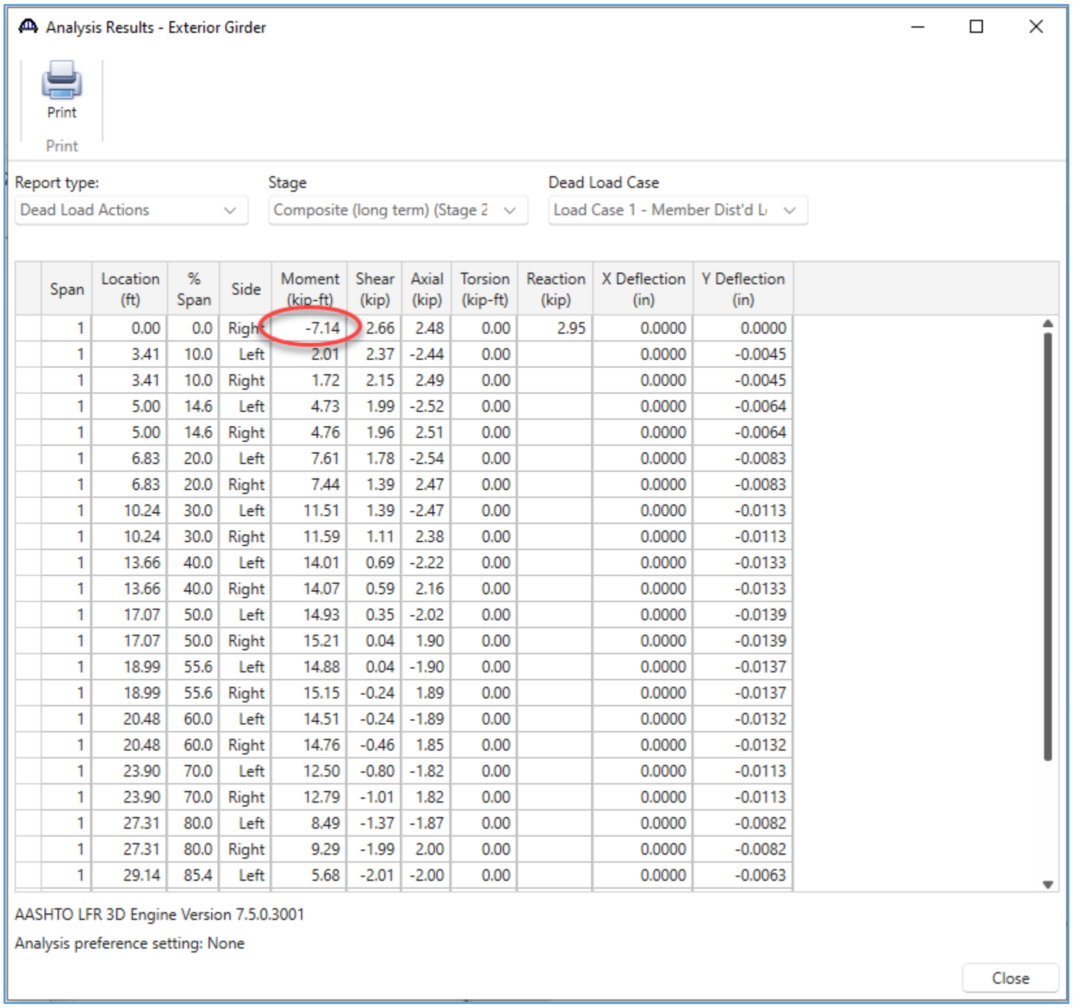


Save the analysis settings and re-analyze the structure with the dead load only analysis.



The model is improved with double the number of elements between girders. Comparing the moment for the applied uniform dead load from utilities at the first support, the computed moment is about 11.5% larger than in the first case. With such a large difference in moment, consider further mesh refinement.

This finite element model has 1311 total nodes, about 13% more than the coarse finite element model.


Analysis Results - Exterior Girder

Print

Report type: Dead Load Actions

Stage: Composite (long term) (Stage 2

Dead Load Case: Load Case 1 - Member Dist'd L

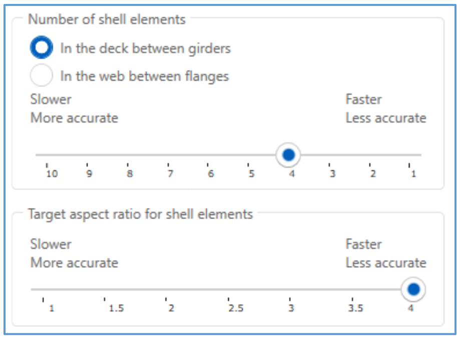
Span	Location (ft)	% Span	Side	Moment (kip-ft)	Shear (kip)	Axial (kip)	Torsion (kip-ft)	Reaction (kip)	X Deflection (in)	Y Deflection (in)
1	0.00	0.0	Right	-7.14	2.66	2.48	0.00	2.95	0.0000	0.0000
1	3.41	10.0	Left	2.01	2.37	-2.44	0.00		0.0000	-0.0045
1	3.41	10.0	Right	1.72	2.15	2.49	0.00		0.0000	-0.0045
1	5.00	14.6	Left	4.73	1.99	-2.52	0.00		0.0000	-0.0064
1	5.00	14.6	Right	4.76	1.96	2.51	0.00		0.0000	-0.0064
1	6.83	20.0	Left	7.61	1.78	-2.54	0.00		0.0000	-0.0083
1	6.83	20.0	Right	7.44	1.39	2.47	0.00		0.0000	-0.0083
1	10.24	30.0	Left	11.51	1.39	-2.47	0.00		0.0000	-0.0113
1	10.24	30.0	Right	11.59	1.11	2.38	0.00		0.0000	-0.0113
1	13.66	40.0	Left	14.01	0.69	-2.22	0.00		0.0000	-0.0133
1	13.66	40.0	Right	14.07	0.59	2.16	0.00		0.0000	-0.0133
1	17.07	50.0	Left	14.93	0.35	-2.02	0.00		0.0000	-0.0139
1	17.07	50.0	Right	15.21	0.04	1.90	0.00		0.0000	-0.0139
1	18.99	55.6	Left	14.88	0.04	-1.90	0.00		0.0000	-0.0137
1	18.99	55.6	Right	15.15	-0.24	1.89	0.00		0.0000	-0.0137
1	20.48	60.0	Left	14.51	-0.24	-1.89	0.00		0.0000	-0.0132
1	20.48	60.0	Right	14.76	-0.46	1.85	0.00		0.0000	-0.0132
1	23.90	70.0	Left	12.50	-0.80	-1.82	0.00		0.0000	-0.0113
1	23.90	70.0	Right	12.79	-1.01	1.82	0.00		0.0000	-0.0113
1	27.31	80.0	Left	8.49	-1.37	-1.87	0.00		0.0000	-0.0082
1	27.31	80.0	Right	9.29	-1.99	2.00	0.00		0.0000	-0.0082
1	29.14	85.4	Left	5.68	-2.01	-2.00	0.00		0.0000	-0.0063

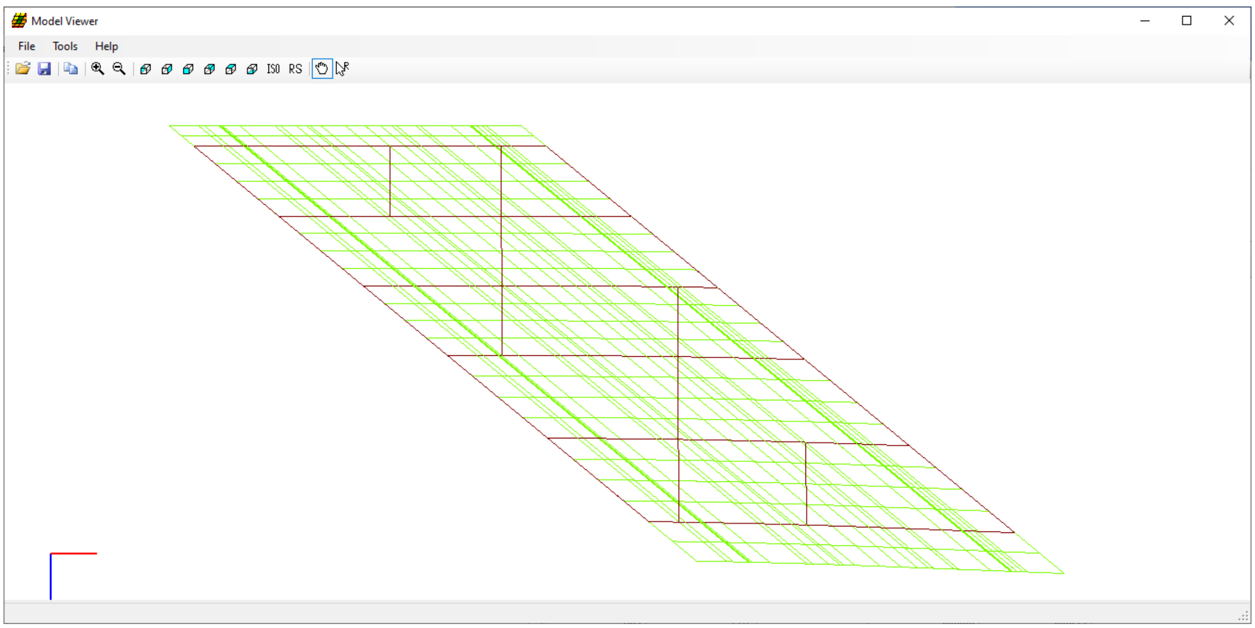
AASHTO LFR 3D Engine Version 7.5.0.3001

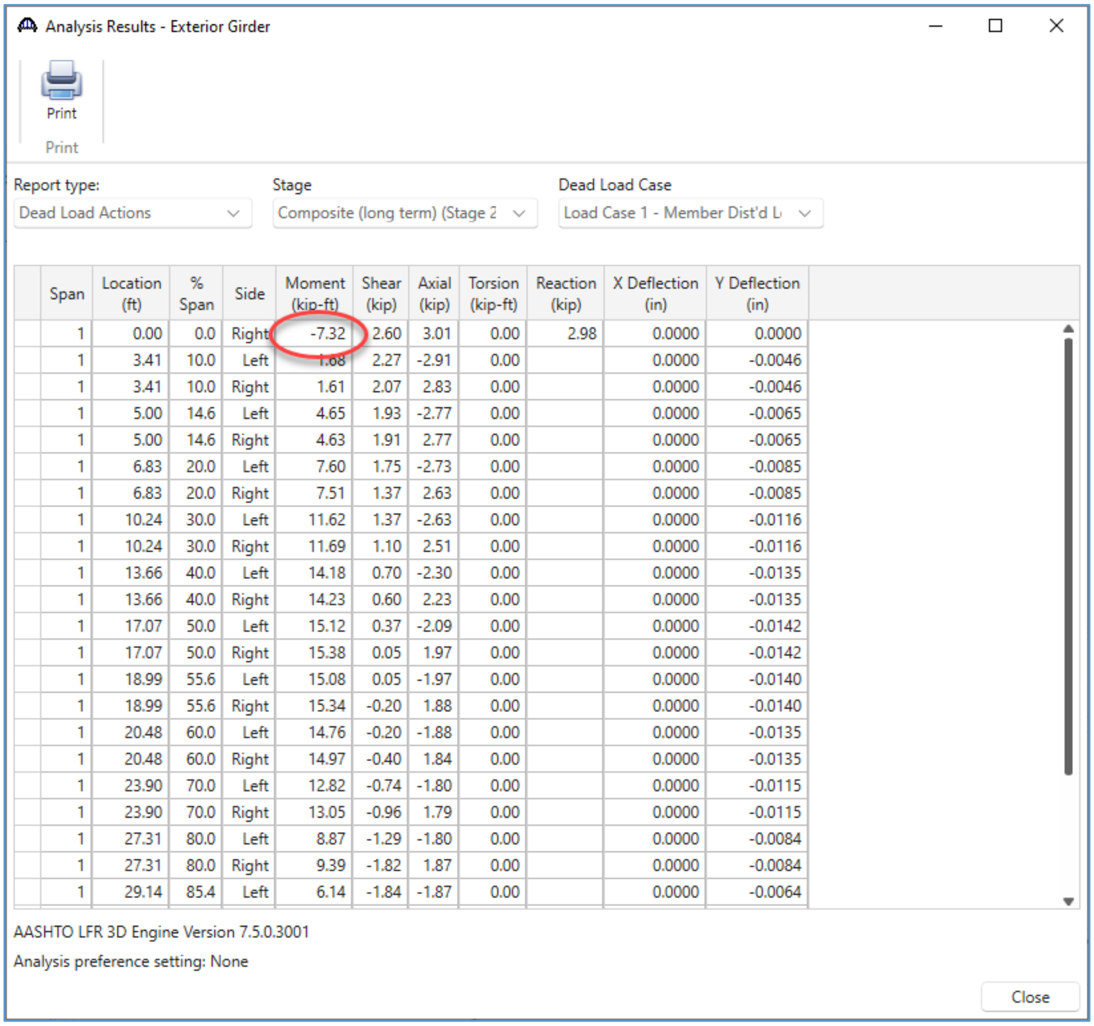
Analysis preference setting: None

Close

Try doubling the number of elements between girders again, from 2 elements to 4 elements.







Analysis Results - Exterior Girder

Print

Print

Report type: Dead Load Actions

Stage: Composite (long term) (Stage 2

Dead Load Case: Load Case 1 - Member Dist'd L

Span	Location (ft)	% Span	Side	Moment (kip-ft)	Shear (kip)	Axial (kip)	Torsion (kip-ft)	Reaction (kip)	X Deflection (in)	Y Deflection (in)
1	0.00	0.0	Right	-7.32	2.60	3.01	0.00	2.98	0.0000	0.0000
1	3.41	10.0	Left	1.68	2.27	-2.91	0.00		0.0000	-0.0046
1	3.41	10.0	Right	1.61	2.07	2.83	0.00		0.0000	-0.0046
1	5.00	14.6	Left	4.65	1.93	-2.77	0.00		0.0000	-0.0065
1	5.00	14.6	Right	4.63	1.91	2.77	0.00		0.0000	-0.0065
1	6.83	20.0	Left	7.60	1.75	-2.73	0.00		0.0000	-0.0085
1	6.83	20.0	Right	7.51	1.37	2.63	0.00		0.0000	-0.0085
1	10.24	30.0	Left	11.62	1.37	-2.63	0.00		0.0000	-0.0116
1	10.24	30.0	Right	11.69	1.10	2.51	0.00		0.0000	-0.0116
1	13.66	40.0	Left	14.18	0.70	-2.30	0.00		0.0000	-0.0135
1	13.66	40.0	Right	14.23	0.60	2.23	0.00		0.0000	-0.0135
1	17.07	50.0	Left	15.12	0.37	-2.09	0.00		0.0000	-0.0142
1	17.07	50.0	Right	15.38	0.05	1.97	0.00		0.0000	-0.0142
1	18.99	55.6	Left	15.08	0.05	-1.97	0.00		0.0000	-0.0140
1	18.99	55.6	Right	15.34	-0.20	1.88	0.00		0.0000	-0.0140
1	20.48	60.0	Left	14.76	-0.20	-1.88	0.00		0.0000	-0.0135
1	20.48	60.0	Right	14.97	-0.40	1.84	0.00		0.0000	-0.0135
1	23.90	70.0	Left	12.82	-0.74	-1.80	0.00		0.0000	-0.0115
1	23.90	70.0	Right	13.05	-0.96	1.79	0.00		0.0000	-0.0115
1	27.31	80.0	Left	8.87	-1.29	-1.80	0.00		0.0000	-0.0084
1	27.31	80.0	Right	9.39	-1.82	1.87	0.00		0.0000	-0.0084
1	29.14	85.4	Left	6.14	-1.84	-1.87	0.00		0.0000	-0.0064

AASHTO LFR 3D Engine Version 7.5.0.3001

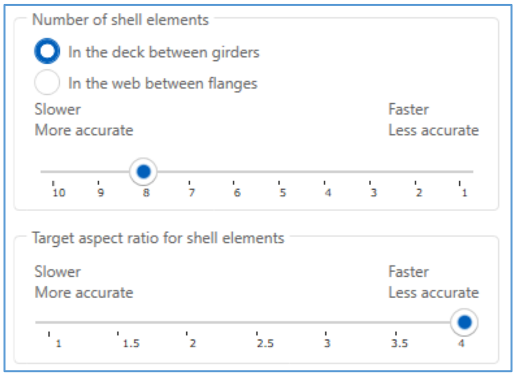
Analysis preference setting: None

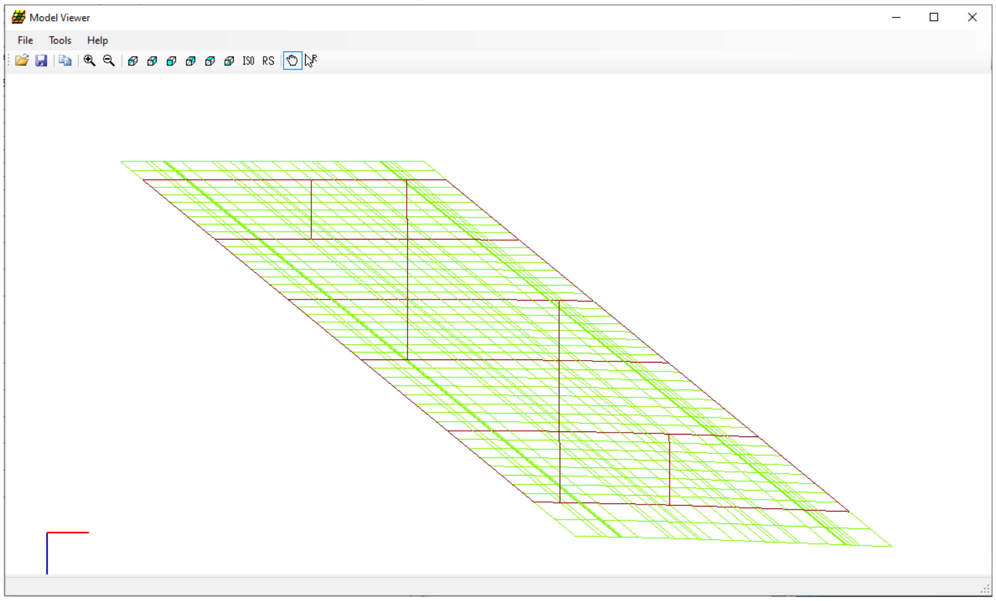
Close

Comparing the same load case, the moment at the first support increased by about 2.5% from the previous trial.

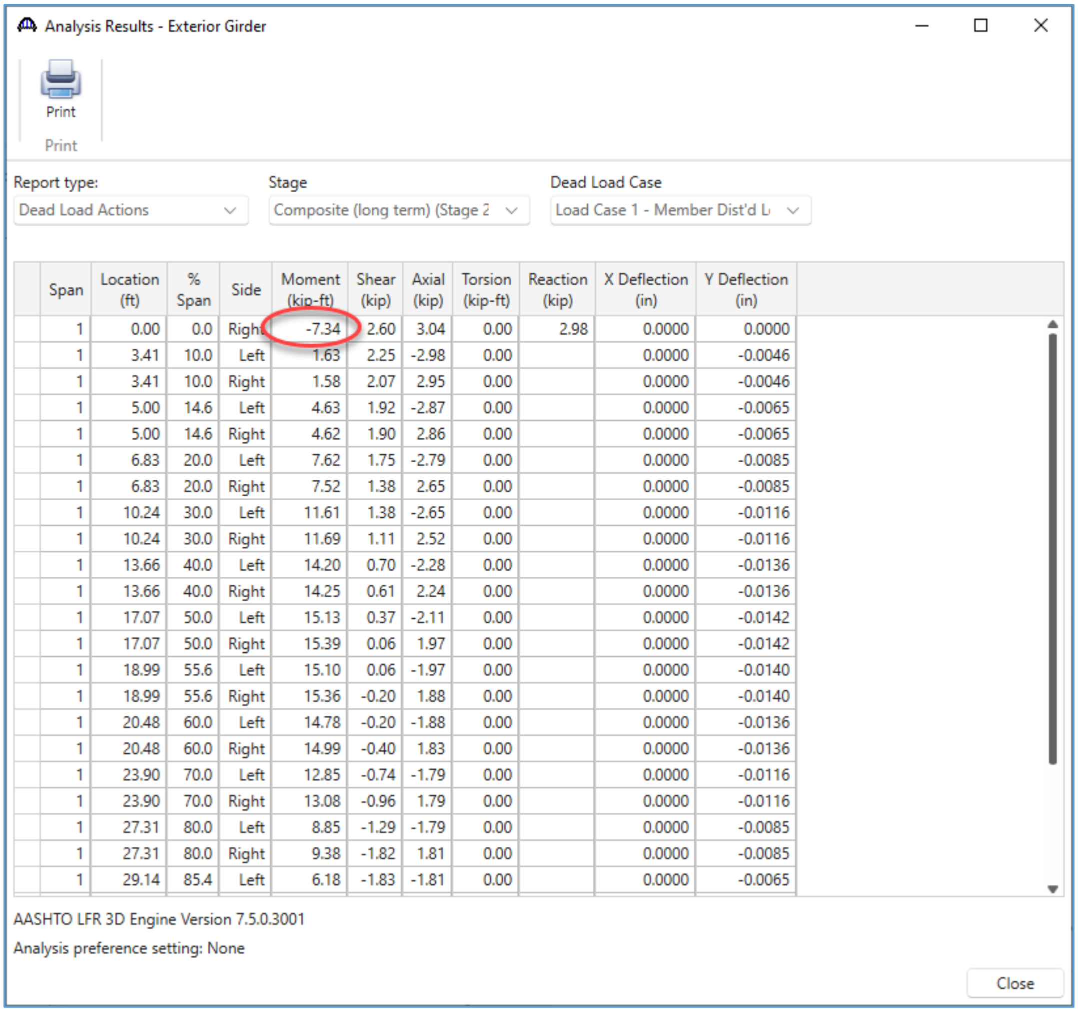
This finite element model has 1621 total nodes, almost 24% more than the previous finite element model.

Repeat this process and double the number of elements between the girders from 4 to 8.





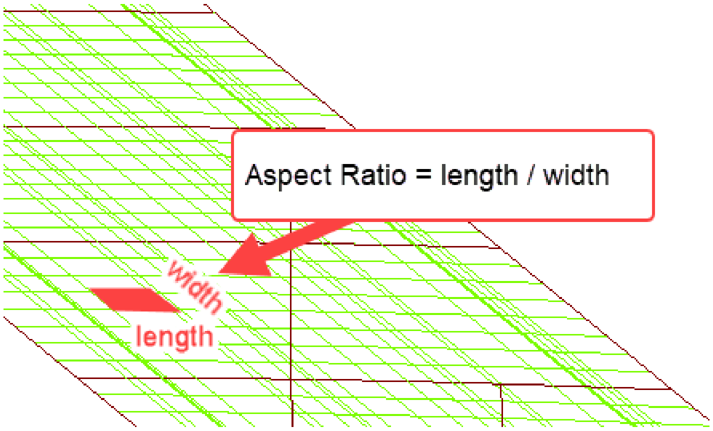
This finite element model has 2241 total nodes, or 38% more than the model with 4 elements between girders.



Span	Location (ft)	% Span	Side	Moment (kip-ft)	Shear (kip)	Axial (kip)	Torsion (kip-ft)	Reaction (kip)	X Deflection (in)	Y Deflection (in)
1	0.00	0.0	Right	-7.34	2.60	3.04	0.00	2.98	0.0000	0.0000
1	3.41	10.0	Left	1.63	2.25	-2.98	0.00		0.0000	-0.0046
1	3.41	10.0	Right	1.58	2.07	2.95	0.00		0.0000	-0.0046
1	5.00	14.6	Left	4.63	1.92	-2.87	0.00		0.0000	-0.0065
1	5.00	14.6	Right	4.62	1.90	2.86	0.00		0.0000	-0.0065
1	6.83	20.0	Left	7.62	1.75	-2.79	0.00		0.0000	-0.0085
1	6.83	20.0	Right	7.52	1.38	2.65	0.00		0.0000	-0.0085
1	10.24	30.0	Left	11.61	1.38	-2.65	0.00		0.0000	-0.0116
1	10.24	30.0	Right	11.69	1.11	2.52	0.00		0.0000	-0.0116
1	13.66	40.0	Left	14.20	0.70	-2.28	0.00		0.0000	-0.0136
1	13.66	40.0	Right	14.25	0.61	2.24	0.00		0.0000	-0.0136
1	17.07	50.0	Left	15.13	0.37	-2.11	0.00		0.0000	-0.0142
1	17.07	50.0	Right	15.39	0.06	1.97	0.00		0.0000	-0.0142
1	18.99	55.6	Left	15.10	0.06	-1.97	0.00		0.0000	-0.0140
1	18.99	55.6	Right	15.36	-0.20	1.88	0.00		0.0000	-0.0140
1	20.48	60.0	Left	14.78	-0.20	-1.88	0.00		0.0000	-0.0136
1	20.48	60.0	Right	14.99	-0.40	1.83	0.00		0.0000	-0.0136
1	23.90	70.0	Left	12.85	-0.74	-1.79	0.00		0.0000	-0.0116
1	23.90	70.0	Right	13.08	-0.96	1.79	0.00		0.0000	-0.0116
1	27.31	80.0	Left	8.85	-1.29	-1.79	0.00		0.0000	-0.0085
1	27.31	80.0	Right	9.38	-1.82	1.81	0.00		0.0000	-0.0085
1	29.14	85.4	Left	6.18	-1.83	-1.81	0.00		0.0000	-0.0065

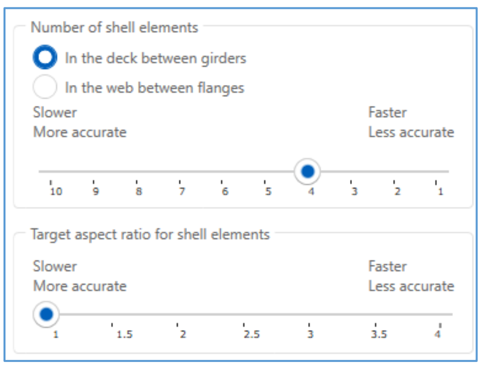
In this case there was only a 0.27% change in moment. At this point, further refinement may not be necessary. Using 4 elements and 8 elements yields approximately the same result, but 4 elements will analyze faster than the 8-element mesh.

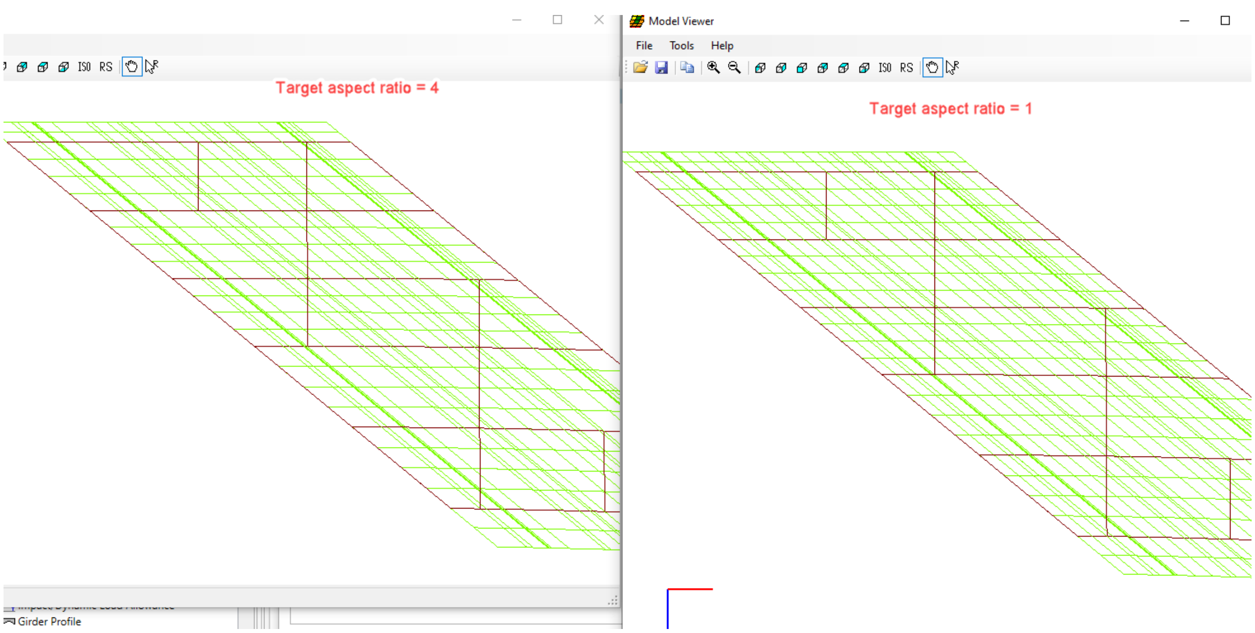
The target aspect ratio is the next shell meshing option. It is always measured in the longitudinal direction.



Because this model has a short span length and a lot of closely spaced POIs, the aspect ratio has not been a problem in the finite element model.

To demonstrate the effect of changing the target aspect ratio, re-analyze the structure with 4 elements between girders and a target aspect ratio of 1.





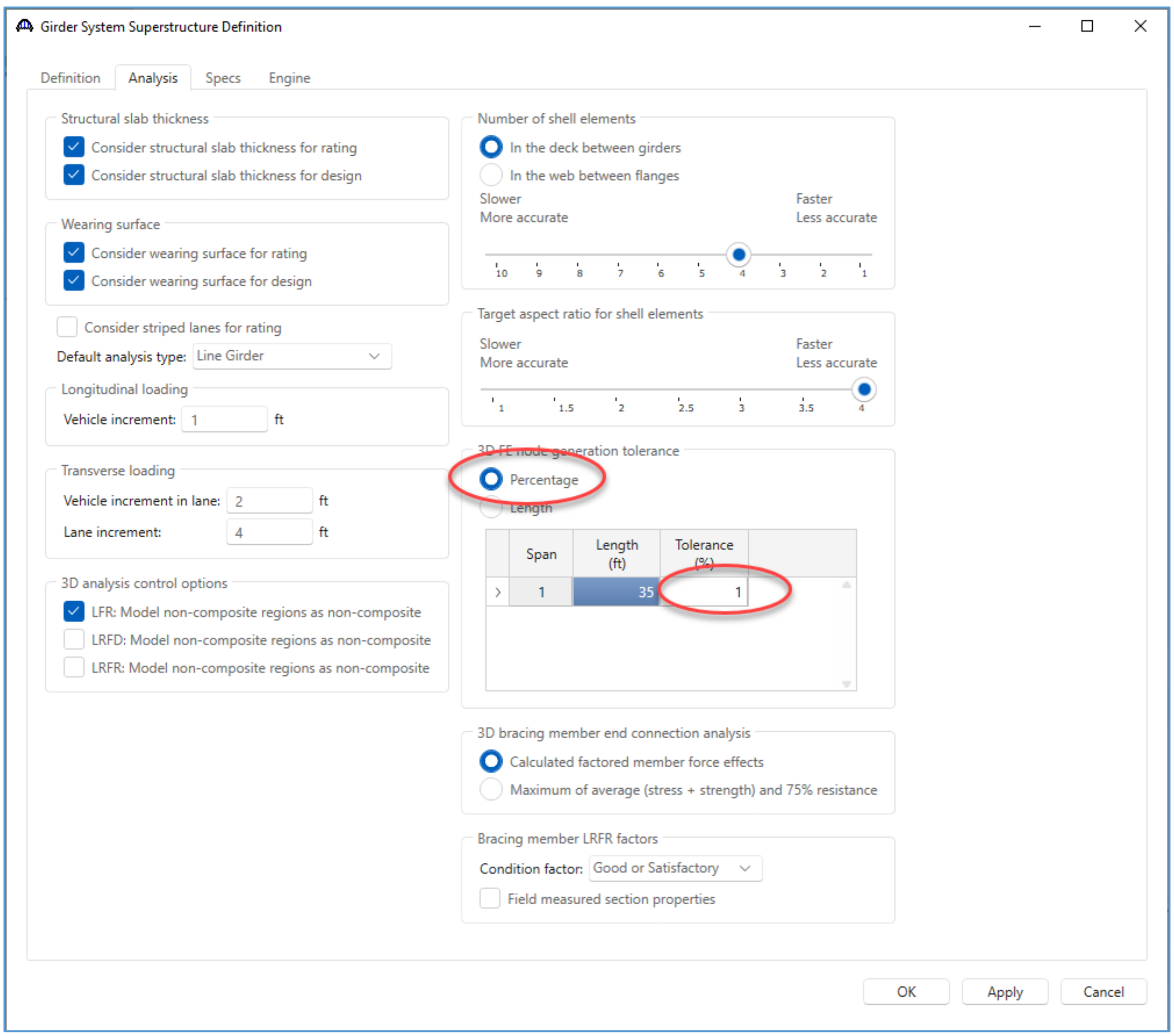
With the target aspect ratio of 1, none of the elements have a length in the longitudinal direction greater than their width. In this case the target aspect ratio does not have a significant impact on the finite element model. For other structures the target aspect ratio could significantly impact the results. This model has 1817 total nodes, which is 12% more than the model generated with a target aspect ratio of 4.

The final option for the deck shell mesh generation is the node merge tolerance. By default, the tolerance is 0.1% of the span length. In previous versions of BrDR, before version 7.5, the tolerance could not be modified. New in version 7.5, the node merge tolerance can be modified for each span and can be input as a length or as a percent.

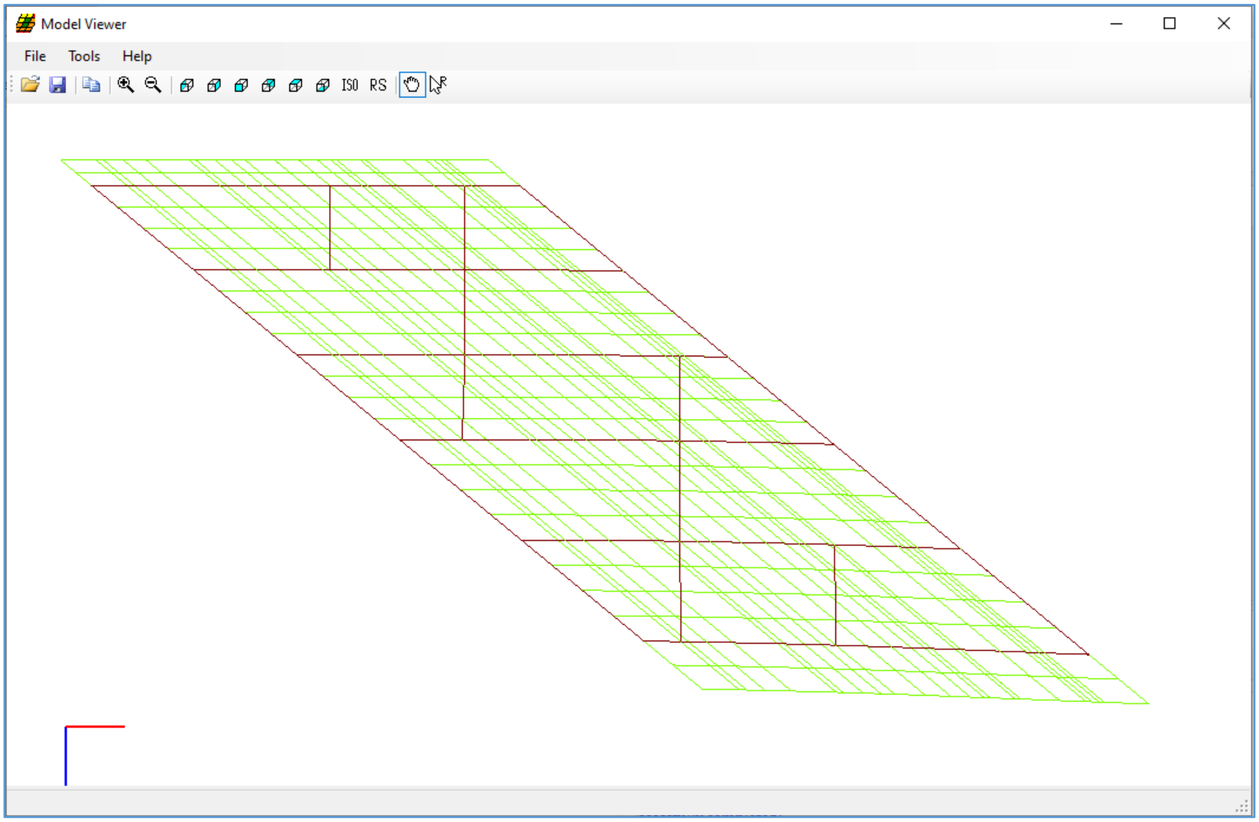
For this example, there are a lot of closely spaced shell elements created around the flange transitions. The default tolerance does not work well in this case because the span is short. For a longer span, such as 250 feet, this 0.1% tolerance would merge nodes within 3 inches. But for this short span, only 35 feet, the default tolerance is equivalent to less than 1/2".

Increase the node merge tolerance to avoid generating a lot of small shell elements. Modify the node merge tolerance to 1% of the span length. Use 4 elements between the girders and a target aspect ratio of 4.

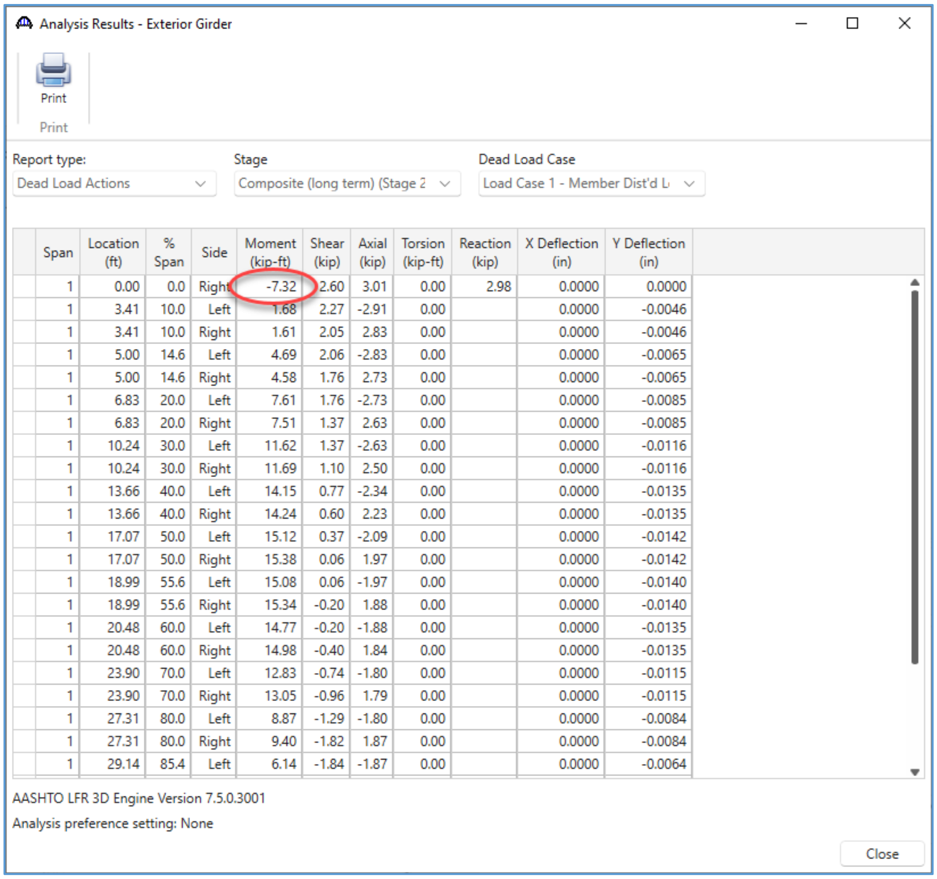
When defining the node merge tolerance, the tolerance can be input as a length or percent of span. If the tolerance is input as a length, the program will convert the length to an equivalent percent of span using the span length.



The deck mesh does not have the small shell elements with this larger tolerance.



The analysis results are similar to the results with the smaller tolerance. This finite element model uses 1229 total nodes, 20% fewer than the corresponding model with a 0.1% merge tolerance.


Analysis Results - Exterior Girder

Print

Report type: Dead Load Actions

Stage: Composite (long term) (Stage 2

Dead Load Case: Load Case 1 - Member Dist'd L

Span	Location (ft)	% Span	Side	Moment (kip-ft)	Shear (kip)	Axial (kip)	Torsion (kip-ft)	Reaction (kip)	X Deflection (in)	Y Deflection (in)
1	0.00	0.0	Right	-7.32	2.60	3.01	0.00	2.98	0.0000	0.0000
1	3.41	10.0	Left	1.68	2.27	-2.91	0.00		0.0000	-0.0046
1	3.41	10.0	Right	1.61	2.05	2.83	0.00		0.0000	-0.0046
1	5.00	14.6	Left	4.69	2.06	-2.83	0.00		0.0000	-0.0065
1	5.00	14.6	Right	4.58	1.76	2.73	0.00		0.0000	-0.0065
1	6.83	20.0	Left	7.61	1.76	-2.73	0.00		0.0000	-0.0085
1	6.83	20.0	Right	7.51	1.37	2.63	0.00		0.0000	-0.0085
1	10.24	30.0	Left	11.62	1.37	-2.63	0.00		0.0000	-0.0116
1	10.24	30.0	Right	11.69	1.10	2.50	0.00		0.0000	-0.0116
1	13.66	40.0	Left	14.15	0.77	-2.34	0.00		0.0000	-0.0135
1	13.66	40.0	Right	14.24	0.60	2.23	0.00		0.0000	-0.0135
1	17.07	50.0	Left	15.12	0.37	-2.09	0.00		0.0000	-0.0142
1	17.07	50.0	Right	15.38	0.06	1.97	0.00		0.0000	-0.0142
1	18.99	55.6	Left	15.08	0.06	-1.97	0.00		0.0000	-0.0140
1	18.99	55.6	Right	15.34	-0.20	1.88	0.00		0.0000	-0.0140
1	20.48	60.0	Left	14.77	-0.20	-1.88	0.00		0.0000	-0.0135
1	20.48	60.0	Right	14.98	-0.40	1.84	0.00		0.0000	-0.0135
1	23.90	70.0	Left	12.83	-0.74	-1.80	0.00		0.0000	-0.0115
1	23.90	70.0	Right	13.05	-0.96	1.79	0.00		0.0000	-0.0115
1	27.31	80.0	Left	8.87	-1.29	-1.80	0.00		0.0000	-0.0084
1	27.31	80.0	Right	9.40	-1.82	1.87	0.00		0.0000	-0.0084
1	29.14	85.4	Left	6.14	-1.84	-1.87	0.00		0.0000	-0.0064

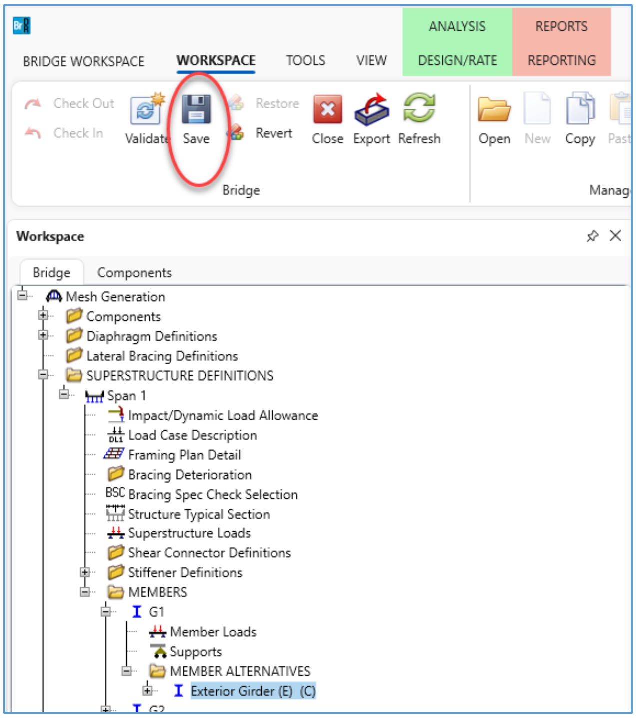
AASHTO LFR 3D Engine Version 7.5.0.3001

Analysis preference setting: None

Close

Increasing the node merge tolerance can negatively impact the accuracy of the finite element model, especially if the tolerance is large. The merge tolerance essentially permits the finite element model to slightly differ from the input structure in order to generate a more regular grid of elements. In the above case, if the span length was 100 ft, the 1% merge tolerance would correspond to 1 ft. A finite element model with that tolerance may model diaphragm locations or flange transitions as far as 1 ft away from the actual distance defined along a member. When modifying the model generation tolerance, the FE model should be inspected to verify that the appropriate beam element properties are used along the lengths of the girders. Verify the input tolerance does not cause section changes to be skipped or produce a finite element model with geometry substantially different from the actual structure.

Once an acceptable finite element model is found, save the bridge to the database with the updated analysis options.



The live load analysis and spec checking can now be performed on this structure. The live load analysis and spec checking require more time to complete than the dead load analysis. Use the dead load only analysis first to find the best mesh generation options.