3D Analysis with AASHTOWare Bridge Design and Rating
Here’s what you’ll learn in this presentation:

1. Review of finite element modeling basics
2. Review of generated model
3. Review of the user-interface for steel multi-girder superstructure
4. Review of how the analysis is performed
5. Review of available output
6. Comparison of results for four models with different mesh sizes
3D Analysis with AASHTOWare Bridge Design and Rating

Here’s what you’ll learn in this presentation:

1. Review of finite element modeling basics
2. Review of generated model
3. Review of the user-interface for steel multi-girder superstructure
4. Review of how the analysis is performed
5. Review of available output
6. Comparison of results for four models with different mesh sizes
Beam elements:
- Are used for concrete beams, steel girder flanges, and diaphragms
- Have six degrees of freedom (DOFs) at each node
- Generally recognize only single curvature bending
Review of Finite Element Modeling Basics

**Shell elements:**

- Are used for the steel girder web and the deck
- Have four nodes with six DOFs at each node

Girder Web  
(Shell Element)  
(Typ.)
Deck-to-beam connection:

- Master-slave constraint – used for 3D curved girder systems
- Rigid link connection – used for 3D straight girder systems
- Connects center of gravity of deck to girder top flange
Review of Finite Element Modeling Basics

**Modeling of reinforced concrete sections in 3D:**

- Beam elements used for reinforced concrete beam
- Shell elements used for deck/top flange
- Rigid links used for connection (straight girder)
Modeling of prestressed concrete sections in 3D:

- Beam elements used for prestressed concrete beam
- Shell elements used for deck
- Rigid links used for connection (straight girder)
Review of Finite Element Modeling Basics

Modeling of steel beam with concrete deck in 3D:
• Beam elements used for steel girder flanges
• Shell elements used for deck and steel girder web
• Rigid links used for connection (if straight girder)
**Dead loads:**

- Stage 1 – non-composite dead loads
- Stage 2 – composite dead loads
- Distributed loads are converted to nodal forces
- Discretization of model must be sufficient to ensure series of nodal loads accurately represents distributed load
Review of Finite Element Modeling Basics

*Live loads:*

- Stage 3 – live loads
- Applied to influence surface
- Location of vehicle selected to produce maximum of desired effect
Support conditions:

- **Free bearings** – permit translation in all directions
- **Guided bearings** – permit translation in only one direction, usually either longitudinal or transverse
- **Fixed bearings** – do not permit translation in any direction

For each of these three support conditions, rotation can be provided or limited in many different combinations.
Here’s what you’ll learn in this presentation:

1. Review of finite element modeling basics
2. Review of generated model
3. Review of the user-interface for steel multi-girder superstructure
4. Review of how the analysis is performed
5. Review of available output
6. Comparison of results for four models with different mesh sizes
Definition of elements for curved structures:

• Curvature is represented by straight elements with small kinks at node points
• Elements are not curved
Review of the Generated Model

**Non-skewed model:**

- Deck and beam are divided into elements
- The software allows user to adjust number of shell elements and target aspect ratio for shell elements
**Skewed model:**

- Nodes are defined along the skew
Review of the Generated Model

**Nodes:**

- Numbers each node of generated model
- Defines X, Y, and Z coordinates for each node

### Nodes

<table>
<thead>
<tr>
<th>Node</th>
<th>X (ft)</th>
<th>Y (ft)</th>
<th>Z (ft)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.000</td>
<td>0.000</td>
<td>-16.500</td>
</tr>
<tr>
<td>2</td>
<td>0.000</td>
<td>-0.667</td>
<td>-16.500</td>
</tr>
<tr>
<td>3</td>
<td>0.000</td>
<td>-4.208</td>
<td>-16.500</td>
</tr>
<tr>
<td>4</td>
<td>0.000</td>
<td>-7.750</td>
<td>-16.500</td>
</tr>
<tr>
<td>5</td>
<td>5.459</td>
<td>0.000</td>
<td>-16.479</td>
</tr>
<tr>
<td>6</td>
<td>5.459</td>
<td>-0.667</td>
<td>-16.479</td>
</tr>
<tr>
<td>7</td>
<td>5.459</td>
<td>-4.208</td>
<td>-16.479</td>
</tr>
<tr>
<td>8</td>
<td>5.459</td>
<td>-7.750</td>
<td>-16.479</td>
</tr>
<tr>
<td>9</td>
<td>10.918</td>
<td>0.000</td>
<td>-16.417</td>
</tr>
</tbody>
</table>

The tables on this and the following slides define the model generated based on data entered by the user.
Review of the Generated Model

**Master Slave Node Pairs:**

- Used to define connection between girder and deck for steel curved girders
- Master node is in deck
- Slave node is along girder top flange
- One-to-one correlation between master node and slave node

### Master Slave Node Pairs

<table>
<thead>
<tr>
<th>Master Node</th>
<th>Slave Node</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>6</td>
<td>5</td>
</tr>
<tr>
<td>10</td>
<td>9</td>
</tr>
<tr>
<td>14</td>
<td>13</td>
</tr>
<tr>
<td>18</td>
<td>17</td>
</tr>
<tr>
<td>22</td>
<td>21</td>
</tr>
<tr>
<td>26</td>
<td>25</td>
</tr>
<tr>
<td>30</td>
<td>29</td>
</tr>
<tr>
<td>34</td>
<td>33</td>
</tr>
</tbody>
</table>
Review of the Generated Model

**Beam Elements:**
- Numbers each beam element in the generated model
- Defines start node and end node
- Also defines reference node

**Beam Elements**

<table>
<thead>
<tr>
<th>Beam Element</th>
<th>Start Node</th>
<th>End Node</th>
<th>Reference Node</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>6</td>
<td>2069</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>8</td>
<td>2069</td>
</tr>
<tr>
<td>5</td>
<td>6</td>
<td>10</td>
<td>2069</td>
</tr>
<tr>
<td>8</td>
<td>8</td>
<td>12</td>
<td>2069</td>
</tr>
<tr>
<td>9</td>
<td>10</td>
<td>14</td>
<td>2069</td>
</tr>
<tr>
<td>12</td>
<td>12</td>
<td>16</td>
<td>2069</td>
</tr>
<tr>
<td>13</td>
<td>14</td>
<td>18</td>
<td>2069</td>
</tr>
<tr>
<td>16</td>
<td>16</td>
<td>20</td>
<td>2069</td>
</tr>
<tr>
<td>17</td>
<td>18</td>
<td>22</td>
<td>2069</td>
</tr>
<tr>
<td>20</td>
<td>20</td>
<td>24</td>
<td>2069</td>
</tr>
<tr>
<td>21</td>
<td>22</td>
<td>26</td>
<td>2069</td>
</tr>
<tr>
<td>24</td>
<td>24</td>
<td>28</td>
<td>2069</td>
</tr>
<tr>
<td>25</td>
<td>26</td>
<td>30</td>
<td>2069</td>
</tr>
<tr>
<td>28</td>
<td>28</td>
<td>32</td>
<td>2069</td>
</tr>
<tr>
<td>29</td>
<td>30</td>
<td>34</td>
<td>2069</td>
</tr>
</tbody>
</table>
Review of the Generated Model

Shell Elements:
- Numbers each shell element in generated model
- Defines Node1 through Node4 for each shell element

Shell Elements

<table>
<thead>
<tr>
<th>Shell Element</th>
<th>Node1</th>
<th>Node2</th>
<th>Node3</th>
<th>Node4</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>2</td>
<td>6</td>
<td>7</td>
<td>3</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>7</td>
<td>8</td>
<td>4</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>10</td>
<td>11</td>
<td>7</td>
</tr>
<tr>
<td>7</td>
<td>7</td>
<td>11</td>
<td>12</td>
<td>8</td>
</tr>
<tr>
<td>10</td>
<td>10</td>
<td>14</td>
<td>15</td>
<td>11</td>
</tr>
<tr>
<td>11</td>
<td>11</td>
<td>15</td>
<td>16</td>
<td>12</td>
</tr>
<tr>
<td>14</td>
<td>14</td>
<td>18</td>
<td>19</td>
<td>15</td>
</tr>
<tr>
<td>15</td>
<td>15</td>
<td>19</td>
<td>20</td>
<td>16</td>
</tr>
<tr>
<td>18</td>
<td>18</td>
<td>22</td>
<td>23</td>
<td>19</td>
</tr>
<tr>
<td>19</td>
<td>19</td>
<td>23</td>
<td>24</td>
<td>20</td>
</tr>
<tr>
<td>22</td>
<td>22</td>
<td>26</td>
<td>27</td>
<td>23</td>
</tr>
<tr>
<td>23</td>
<td>23</td>
<td>27</td>
<td>28</td>
<td>24</td>
</tr>
</tbody>
</table>
Review of the Generated Model

Supports:

- Identifies all support nodes
- Defines the following in X, Y, Z directions
  - Translation state (fixed or free)
  - Translation spring constant (kip/in)
  - Rotation state (fixed or free)
  - Rotation spring constant (in-kip/Deg)

<table>
<thead>
<tr>
<th>Support Node</th>
<th>X Translation State</th>
<th>Y Translation State</th>
<th>Z Translation State</th>
<th>X Translation Spring Constant (kip/in)</th>
<th>Y Translation Spring Constant (kip/in)</th>
<th>Z Translation Spring Constant (kip/in)</th>
<th>X Rotation State</th>
<th>Y Rotation State</th>
<th>Z Rotation State</th>
<th>X Rotation Spring Constant (in-kip/Deg)</th>
<th>Y Rotation Spring Constant (in-kip/Deg)</th>
<th>Z Rotation Spring Constant (in-kip/Deg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>Free</td>
<td>Fixed</td>
<td>Fixed</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
<td>Free</td>
<td>Free</td>
<td>Free</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
</tr>
<tr>
<td>152</td>
<td>Free</td>
<td>Fixed</td>
<td>Free</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
<td>Free</td>
<td>Free</td>
<td>Free</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
</tr>
<tr>
<td>352</td>
<td>Free</td>
<td>Fixed</td>
<td>Free</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
<td>Free</td>
<td>Free</td>
<td>Free</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
</tr>
<tr>
<td>496</td>
<td>Free</td>
<td>Fixed</td>
<td>Free</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
<td>Free</td>
<td>Free</td>
<td>Free</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
</tr>
</tbody>
</table>
Inclined Supports:

- Defines constraint type – translational or rotational
- Defines X, Y, and Z components of a 10’ line oriented in the direction of constraint (i.e., oriented perpendicular to the direction of allowable movement)

### Inclined Supports

<table>
<thead>
<tr>
<th>Inclined Support Node</th>
<th>Constraint Type</th>
<th>X (ft)</th>
<th>Y (ft)</th>
<th>Z (ft)</th>
</tr>
</thead>
<tbody>
<tr>
<td>152</td>
<td>Translational</td>
<td>-2.266</td>
<td>0.000</td>
<td>9.740</td>
</tr>
<tr>
<td>352</td>
<td>Translational</td>
<td>-5.043</td>
<td>0.000</td>
<td>8.635</td>
</tr>
<tr>
<td>496</td>
<td>Translational</td>
<td>-6.868</td>
<td>0.000</td>
<td>7.268</td>
</tr>
<tr>
<td>648</td>
<td>Translational</td>
<td>-2.266</td>
<td>0.000</td>
<td>9.740</td>
</tr>
<tr>
<td>848</td>
<td>Translational</td>
<td>-5.043</td>
<td>0.000</td>
<td>8.635</td>
</tr>
<tr>
<td>992</td>
<td>Translational</td>
<td>-6.868</td>
<td>0.000</td>
<td>7.268</td>
</tr>
<tr>
<td>1144</td>
<td>Translational</td>
<td>-2.266</td>
<td>0.000</td>
<td>9.740</td>
</tr>
<tr>
<td>1344</td>
<td>Translational</td>
<td>-5.043</td>
<td>0.000</td>
<td>8.635</td>
</tr>
<tr>
<td>1488</td>
<td>Translational</td>
<td>-6.868</td>
<td>0.000</td>
<td>7.268</td>
</tr>
</tbody>
</table>
**Inclined Supports:**

- Constraints specified in local coordinate system at support
- User defines orientation of local coordinate system as either:
  - Parallel to tangent of member reference line at support
  - Parallel to specified chord angle from the tangent
Review of the Generated Model

Member Releases:

- Generated to model hinges and pinned diaphragm connections
- Provides the following in X, Y, Z directions
  - Translation release (false or true)
  - Rotation release (false or true)

### Member Releases

<table>
<thead>
<tr>
<th>Member</th>
<th>Node</th>
<th>X Translation</th>
<th>Y Translation</th>
<th>Z Translation</th>
<th>X Rotation</th>
<th>Y Rotation</th>
<th>Z Rotation</th>
</tr>
</thead>
<tbody>
<tr>
<td>1969</td>
<td>2</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>true</td>
</tr>
<tr>
<td>1970</td>
<td>498</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>true</td>
</tr>
<tr>
<td>1971</td>
<td>4</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>true</td>
</tr>
<tr>
<td>1971</td>
<td>500</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>true</td>
</tr>
<tr>
<td>1972</td>
<td>4</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>true</td>
</tr>
<tr>
<td>1973</td>
<td>500</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>true</td>
</tr>
<tr>
<td>1974</td>
<td>18</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>true</td>
</tr>
<tr>
<td>1974</td>
<td>514</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>false</td>
<td>true</td>
</tr>
</tbody>
</table>
Review of the Generated Model

Load Case:

- Each load is identified by load case and load ID
- Loads are applied at nodes
- Provides the following in X, Y, Z directions
  - Force (kips)
  - Moment (kip-ft)

Support Reactions

<table>
<thead>
<tr>
<th>Node</th>
<th>X Force (kip)</th>
<th>Y Force (kip)</th>
<th>Z Force (kip)</th>
<th>X Moment (kip-ft)</th>
<th>Y Moment (kip-ft)</th>
<th>Z Moment (kip-ft)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.000</td>
<td>20.370</td>
<td>-0.041</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>114</td>
<td>-1.134</td>
<td>85.058</td>
<td>4.873</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>264</td>
<td>-2.830</td>
<td>86.108</td>
<td>4.847</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>372</td>
<td>0.117</td>
<td>20.331</td>
<td>-0.123</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>375</td>
<td>-0.002</td>
<td>15.725</td>
<td>0.020</td>
<td>0.000</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>
Here’s what you’ll learn in this presentation:

1. Review of finite element modeling basics
2. Review of generated model
3. Review of the user-interface for steel multi-girder superstructure
4. Review of how the analysis is performed
5. Review of available output
6. Comparison of results for four models with different mesh sizes
Review of the User-Interface for Steel Multi-Girder Superstructure

Superstructure Definitions:

• Provides tree structure
• Includes each Member and each Member Alternative
• Provides navigational tool to access each window

The following slides highlight data that is specific to 3D finite element models
Review of the User-Interface for Steel Multi-Girder Superstructure

Girder System Superstructure Definition – Definition Tab

Define Horizontal Curvature Along Reference Line
Review of the User-Interface for Steel Multi-Girder Superstructure

Girder System Superstructure Definition – Definition Tab
Review of the User-Interface for Steel Multi-Girder Superstructure

Girder System Superstructure Definition – Analysis Tab

Define refined vs. speed

Define Longitudinal Loading and Transverse Loading
Review of the User-Interface for Steel Multi-Girder Superstructure

Structure Framing Plan Details – Layout Tab

- Enter Distance from Reference Line to Leftmost Girder
- Summary of Girder Radii
- Define Bearing Alignments (Tangent or Chord with Chord Angle)
- Applies Bearing Alignment Properties to All Members
Diaphragm Definition

Provide all required diaphragm information
For a 3D analysis, this load is used only if it is entered, and if it is not entered, the software will determine the dead load based on the Diaphragm Definition.
Diaphragm Loading Selection

Select diaphragms for influence surface loading in the 3D analysis.
Here’s what you’ll learn in this presentation:

1. Review of finite element modeling basics
2. Review of generated model
3. Review of the user-interface for steel multi-girder superstructure
4. Review of how the analysis is performed
5. Review of available output
6. Comparison of results for four models with different mesh sizes
Review of How the Analysis is Performed

Analysis Settings

Select 3D FEM (for Design Review or Rating) or 3D FEM-Vehicle Path (for Rating only)
Review of How the Analysis is Performed

Analysis Settings – Output Tab

Select AASHTO Engine Reports
Here’s what you’ll learn in this presentation:

1. Review of finite element modeling basics
2. Review of generated model
3. Review of the user-interface for steel multi-girder superstructure
4. Review of how the analysis is performed
5. Review of available output
6. Comparison of results for four models with different mesh sizes
**Review of Available Output**

**List of major sections of output**

- Model Actions report provides moments and shears
- Model, FE Model Graphics, Transverse Loader Patterns available
Review of Available Output

**Model Viewer:**

- Model can be viewed graphically
- Model Viewer permits view from many different vantages
- Ability to select what portions of model are viewed
- Ability to view influence surfaces
Review of Available Output

**User-interface tabular reports:**

- Output can be viewed in tabular reports
- This example presents dead load analysis results

Select for tabular results for dead load effects, live load effects, and ratings
Review of Available Output

**User-interface tabular reports:**
- This example presents live load analysis results
Review of Available Output

**User-interface tabular reports:**

- This example presents load rating results
Review of Available Output

**User-interface graphs:**

- Output can also be viewed as graphs
- This example presents dead load and live load moments

Select for graphical results for dead load and live load effects
Review of Available Output

Specification checks:

- Stages 1, 2, and 3 spec checks can be selected at each node
- Available for selected method (LFR/LFD or LRFR/LRFD)
- Detailed calculations available for each spec check

Select for specification checks for plate
Here’s what you’ll learn in this presentation:

1. Review of finite element modeling basics
2. Review of generated model
3. Review of the user-interface for steel multi-girder superstructure
4. Review of how the analysis is performed
5. Review of available output
6. Comparison of results for four models with different mesh sizes
Compare Results for Four Models with Different Mesh Sizes

**Simple beam model example:**

- Consider a 72-ft long simple-span beam
- Beam depth = 6 feet
- Concentrated load = 10 kips at midspan

\[
M = \frac{PL}{4} = \frac{(10 \text{ kips})(72 \text{ ft})}{4} = 180 \text{ kip-ft}
\]
Simple beam model example:

- Model 1 – one shell element in the depth
- Resulting Moment = 110 kip-ft
Simple beam model example:

- Model 2 – two shell elements in the depth
- Resulting Moment = 144 kip-ft

Figure 2 Model two (refined mesh model)

Moment at the middle point = 17.544 kip x 6ft + 12.84 kip x 4.5 ft - 12.59 kip x 1.5 ft = **144.159 kip-ft**
**Simple beam model example:**

- Model 3 – four shell elements in the depth
- Resulting Moment = 170 kip-ft

Figure 3 Model three

Moment at the middle point = 19.522 kip*6 ft + 11.6368*5.25 + 2.939*2.75 - 3.827*2.25 - 10.869*0.75 = 169.5436 kip-ft
Compare Results for Four Models with Different Mesh Sizes

**Simple beam model example:**

- Model 4 – eight shell elements in the depth
- Resulting Moment = 177 kip-ft

Figure 4 Model four

Moment at the middle point = 29.278 kip*6 ft + 7.18*5.625 + 3.925*4.875 + 2.28*4.125 + 0.612*3.375 - 1.078*2.525 - 2.789*1.875 - 4.536*1.125 - 6.342*0.375 = **177.13 kip-ft**
Simple beam model example:

Analytical solution = 180.0 kip-ft at midspan

Conclusion:
More shell elements along the depth of the beam results in more accurate results.
Compare Results for Four Models with Different Mesh Sizes

Simple beam model example:

Analytical solution = 0.0 kip-ft at support

Same conclusion:
More shell elements along the depth of the beam results in more accurate results.
Here’s what you’ve learned in this presentation:

1. Review of finite element modeling basics
2. Review of generated model
3. Review of the user-interface for steel multi-girder superstructure
4. Review of how the analysis is performed
5. Review of available output
6. Comparison of results for four models with different mesh sizes