# Beam Analysis Using Visual Analysis

1. The software is available at the Open Access Labs (OAL) and the Virtual OAL at http://voal.tamu.edu in Programs under the Windows Start menu. The software can also be downloaded for use on student computers. To download, go to https://www.iesweb.com/edu/index.htm and follow the instructions that requires your TAMU email address to obtain the download link. The license is valid for one academic year, and may require registration renewal.

2. There is detailed documentation and tutorials available in the Help menu of the software by choosing Table of Contents. The first task is to define the unit system:

   • Choose Manage Units from the Home tab or use the style listed in the drop down menu. Unit Styles are available for the most common units, although a Custom Style can be defined.
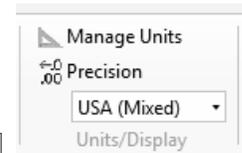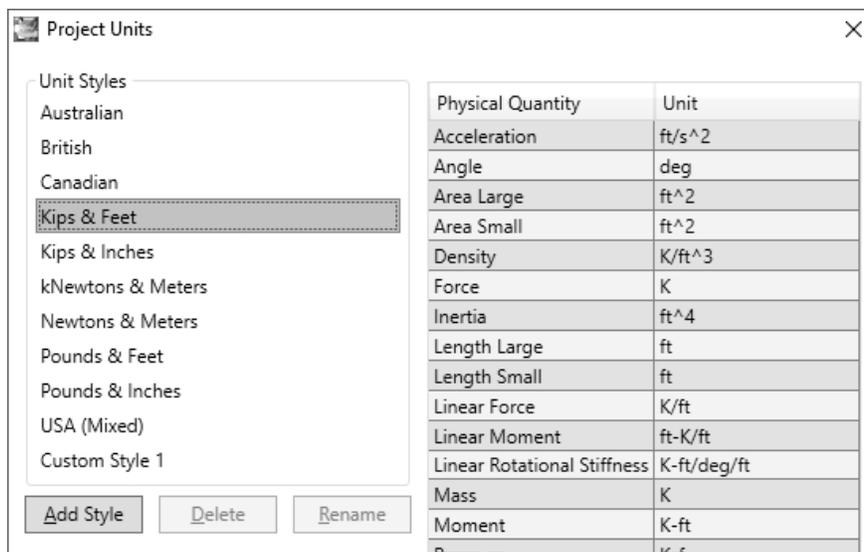
3. To set the scale of the geometry grid, a Grid tab is available in the Project Manager window:

   • Choose Grid, and specify the grid spacing for both axes and how many divisions to be displayed for each axes.

4. To create the geometry, you must be in the Structure tab and Model View. Draw Members allows you to draw line members with the mouse that snap to the grid. You can also define end coordinates with Create Members:
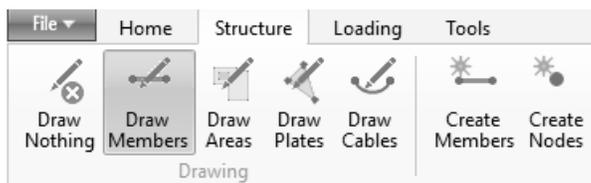

   The Create Tab of the Project Manager shows other objects and systems that can be created. (Hovering over the menu gives instructions.)
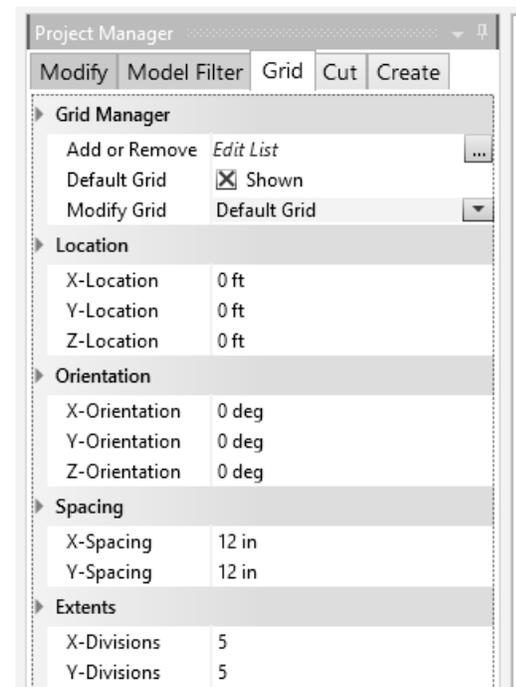
- Select a starting point and ending point with the cursor. The data for the member is displayed at the bottom of the Model View and the Project Manager will show the member properties. You can change the material and select a Shape Type from the Database (or add a Custom Blob). All members drawn after making a selection will have the same material and shape but can be changed.

- Continue to add members if the beam continues across supports or does not have the supports the ends. If you need to make a node for a support on a member, select it and with the right-click menu choose Split Members.

- The geometry can be set precisely by selecting the joint and defining the Location in the Project Manager window with the Modify tab.

- Member ends can snap to grid points or to other points (with care).

- The geometry can be set precisely by selecting the beam member, bringing up the specific menu (right click), choosing Member Properties to set the length. In addition, the Data menu can be used to enter coordinates.

- To support a node, chose the node, and in the Modify tab of the Project Manager choose from the Support sub-item. There are four support types (no roller). A pinned support will fix all the displacements (DX, DY, DZ). A rigid support will fix the displacements and all rotations (RX, RY, RZ). The individual restraints can be chose as well.

- If a hinge connection needs to be included between members, you will need to change the member Connections Type to Simple-Rigid, or Rigid-Simple depending on where the nodes with the hinge is. Be careful to make just *one* member end at the hinge Simple.

- Because a pin support will allow rotation out of plane, there must be rotation restriction out of plane. **At least one support needs to have RX fixed.**

NOTE: The support forces will be determined in the analysis. *Do not add loads at these nodes*.
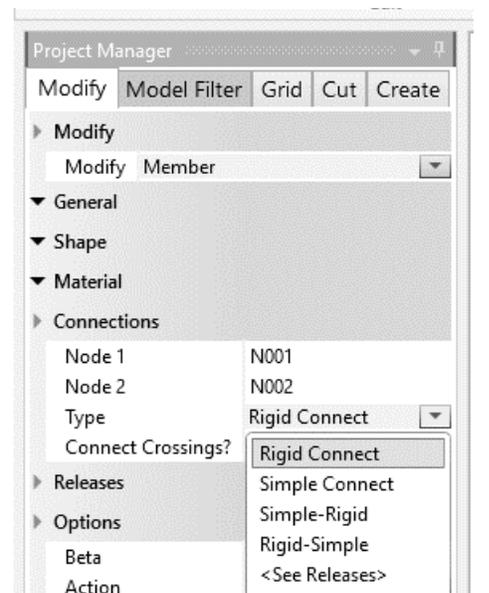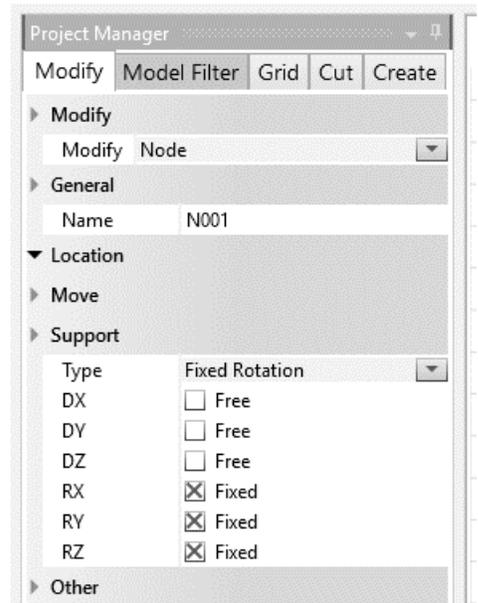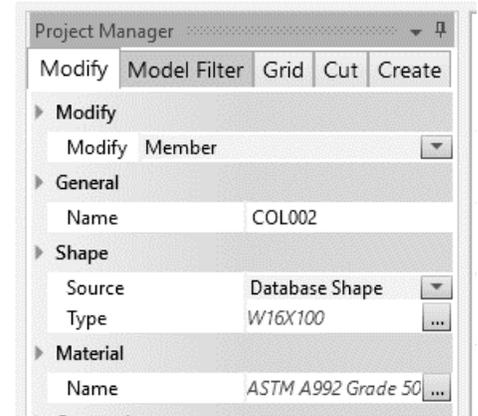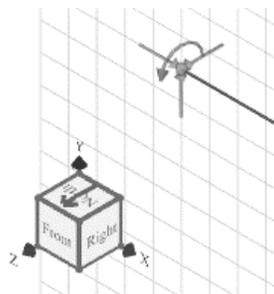
5. The beam geometry is complete, and we need to put on loads. We also might need to tell Visual Analysis that there will be "weightless" members by choosing the Loading tab that shows the service case is "D".
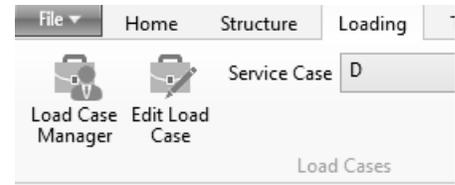
   - Choose Edit Load Case and under the Self Weight Type and choose No Self Weight.
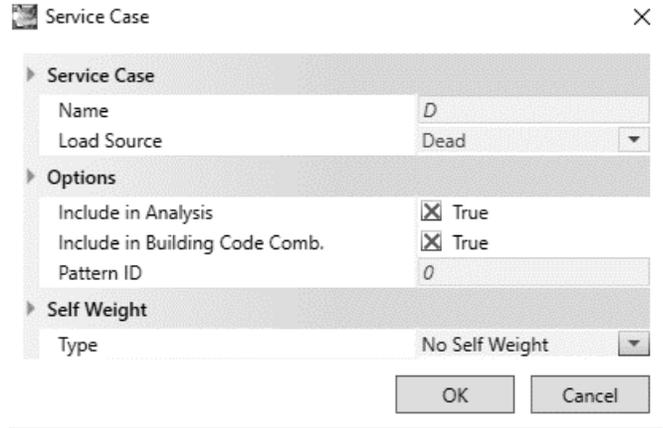
6. The loads that can be applied in the Loading menu are grayed out until the node or member or other element are chosen. If the loads need to be assigned to types, there are defined Service Case types including wind and which direction of the load (ex. W+X). The Load Case manager tool has Load Combinations by building codes.

   - Choose the member or node to be loaded and select the load type (here shown for member loading)

   - Chose the Load Type. Choose the direction. Enter any other load information required

   - Enter the magnitude of the force. A negative value is opposite of the positive global direction.

   - To display the load value, the load graphic must be selected first. Under the Model Filter tab of the Project Manager, select the options under the Show or Hide sub-item.
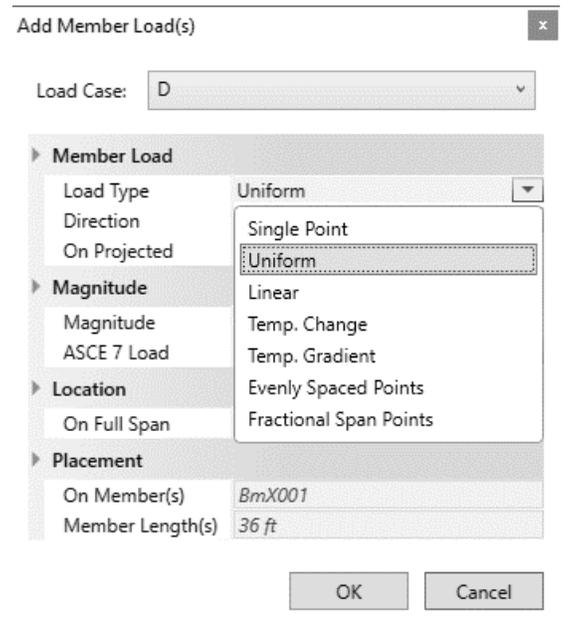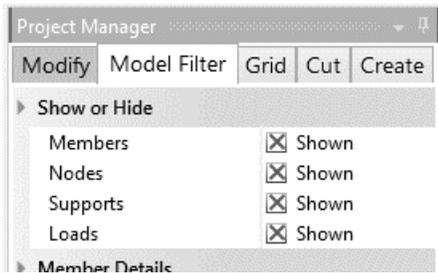
   - If the force units are not what you want, you must exit the loading dialog and manage the units first.

   *NOTE: **Do not** put support reactions as applied loads. The analysis will determine the reaction values.*
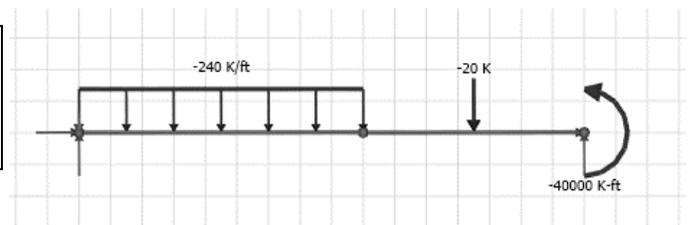
7. To run the linear analysis after the geometry, member
    properties, and loading have been defined, make certain
    that the Modify tab of the Project Manager shows under
    the Analysis sub-item that the Static Method is First
    Order.

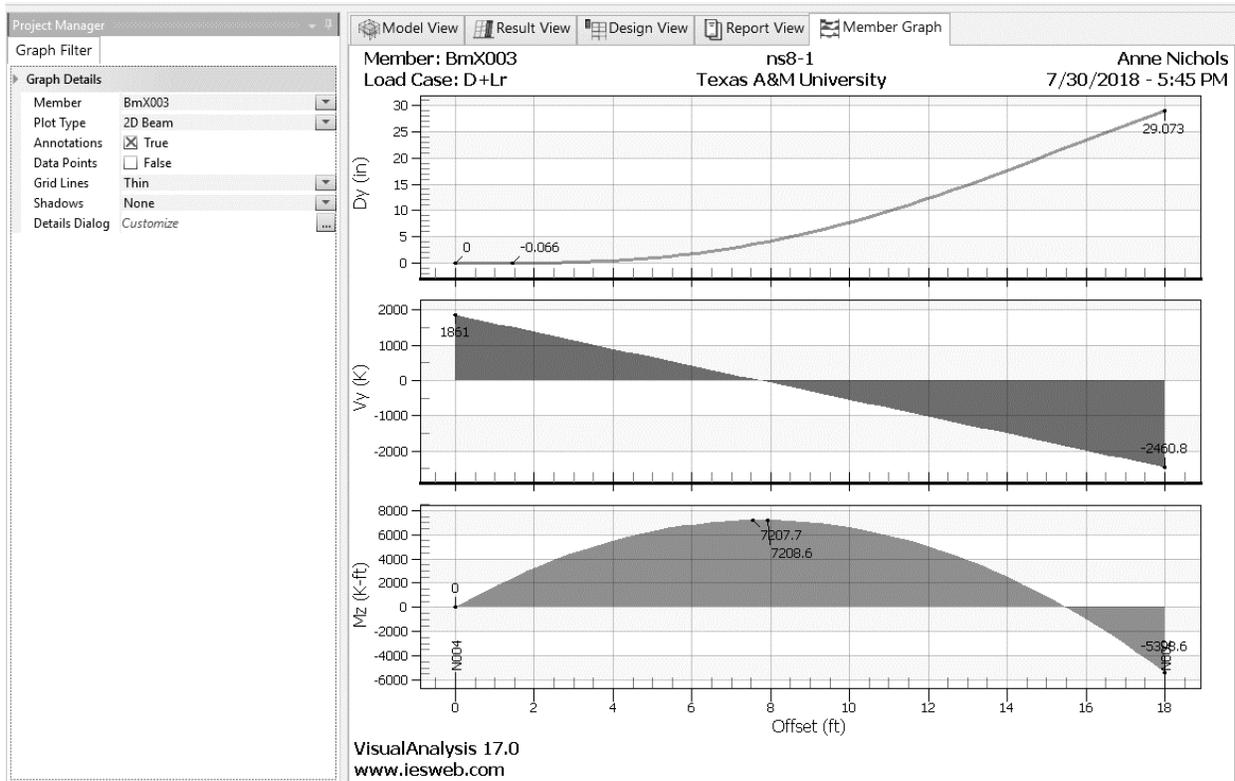   • Choose the Results View tab.

8. If the analysis is successful, there will not be any error
    message displayed. There is a Status menu in the upper
    left corner of the Results View that is useful when the
    analysis is not successful. To see the specific results,
    choose the Results Filter tab in the Project Manager.

    • To show the shear results, choose the Result Type of
      Vy, shear force. To show the bending moment
      results choose the Result Type of Mz, bending
      moment.

    • With the Graphics option of Colors, tensile members will have positive values, while
      compression members will have negative values.
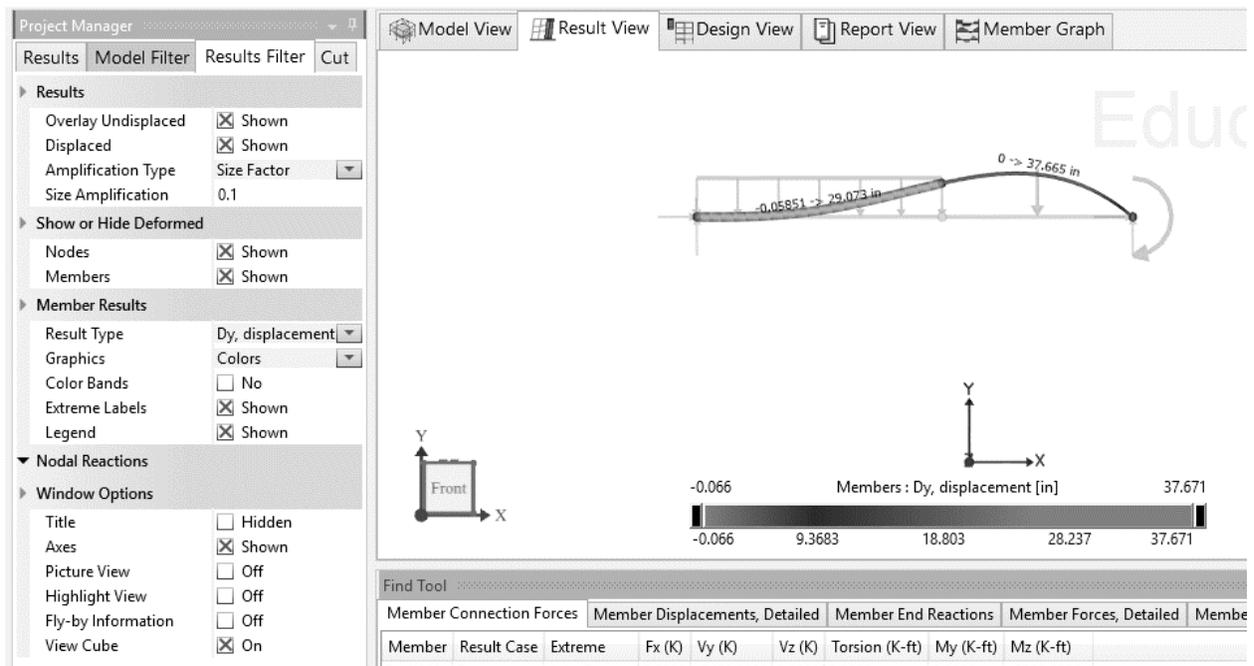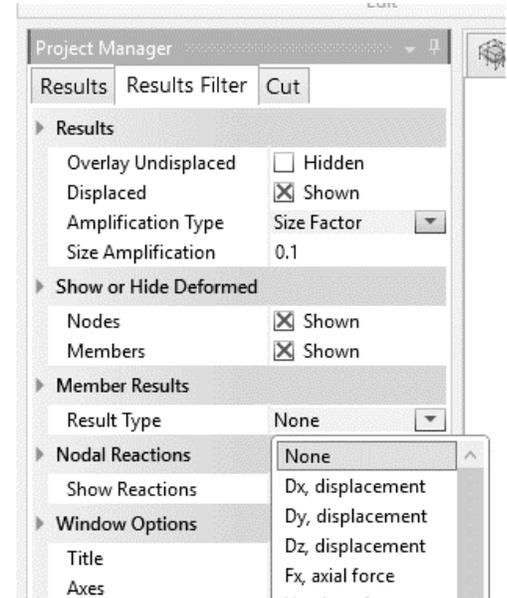
    • With the Graphics option of Diagram, the relative sizes for the Result Type chosen will
      appear as colored bars over the length of the members.

    • To show the undeflected diagram, choose the Overlay Undisplaced box under the Results
      sub-item.

    • To show the forces at the supported nodes, choose Show Reactions under the Nodal
      Reactions sub-item.

    • To animate the deflection diagram, in the Results View tab, choose Choose Animate
      Results from the right-click menu.

- To see exact values of member forces, connection forces, and displacements of a member, chose the member and choose the Member Graph tab. Hover over the point on the plot and the value will be displayed next to the hand pointer icon.

9. The Report View tab allows for a variety of tables to be included by dragging the item onto the place needed in the report view. To include the images from the Model or Results View, use Ctrl+C for copying and Ctrl+V to paste into the report.

10. To save the file choose Save or Save Project from the File menu.

11. To load an existing file choose Open Project from the File menu.

12. To print a plot choose Print or Print Preview Window from the File menu. There is a Page Setup command under the File menu.

13. To print a plot Choose Print Window... from the File menu. As an alternative, you may copy the plot (Ctrl+c) and paste it in a word processing document (Ctrl+v).