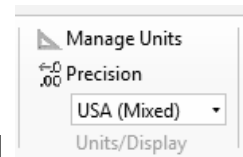
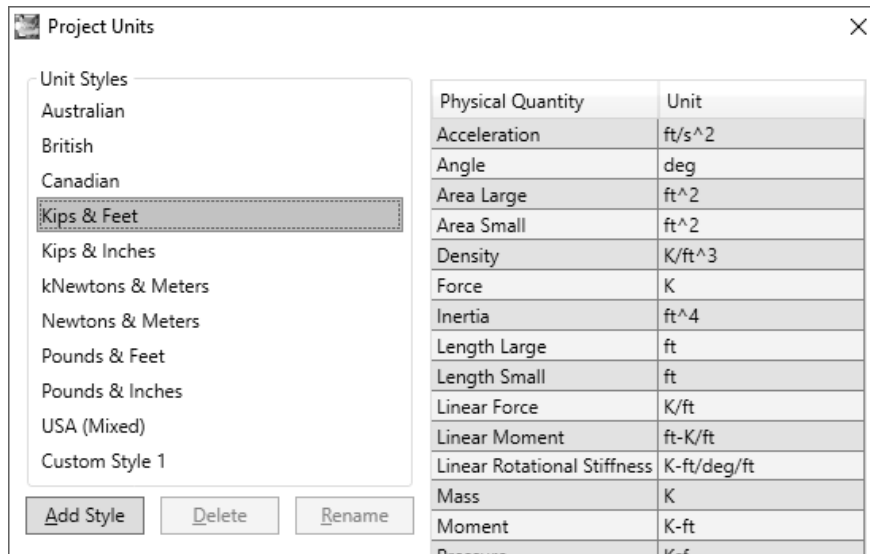
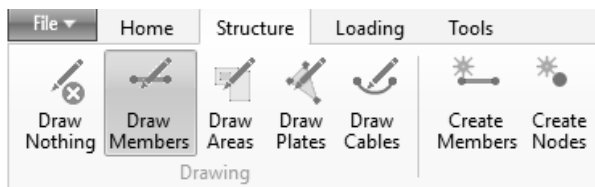


## Truss Analysis using Visual Analysis

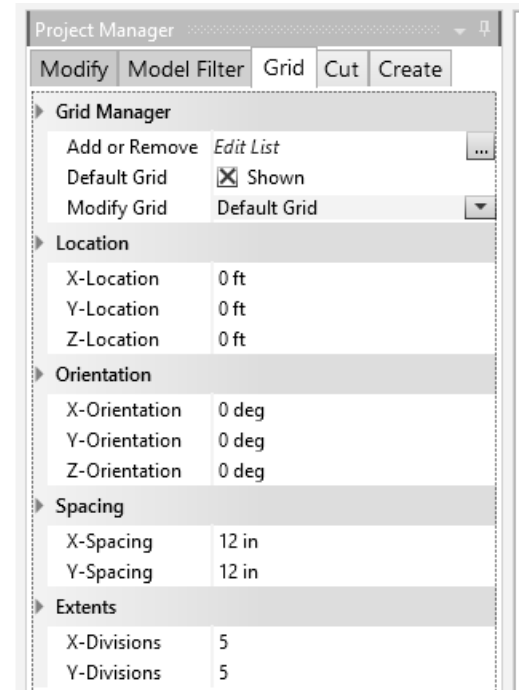
- The software is available at the Open Access Labs (OAL) and the Virtual OAL at <http://voal.tamu.edu> in Programs under the Windows Start menu. The software can also be downloaded for use on student computers. To download, go to <https://www.iesweb.com/edu/index.htm> and follow the instructions that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



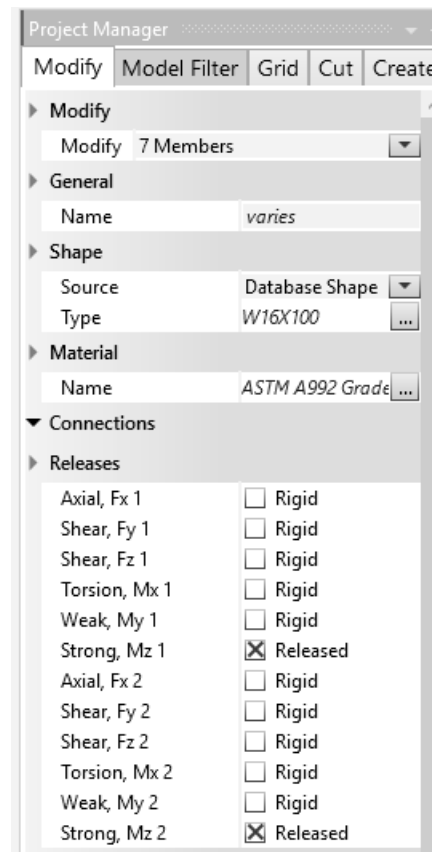
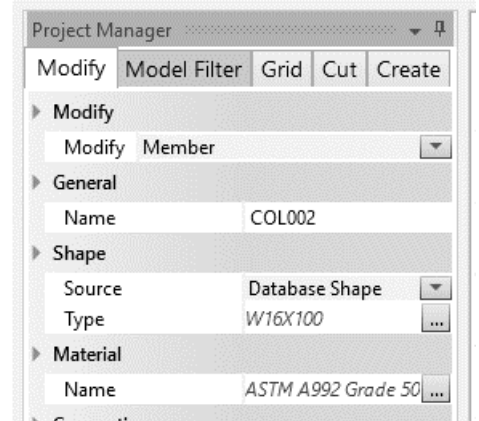
- To set the scale of the geometry grid, a Grid tab is available in the Project Manager window:
  - Choose Grid, and specify the grid spacing for both axes and how many divisions to be displayed for each axes.
- To create the geometry, you must be in the Structure tab and Model View. Draw Members allows you to draw line members with the mouse that snap to the grid. You can also define end coordinates with Create Members:



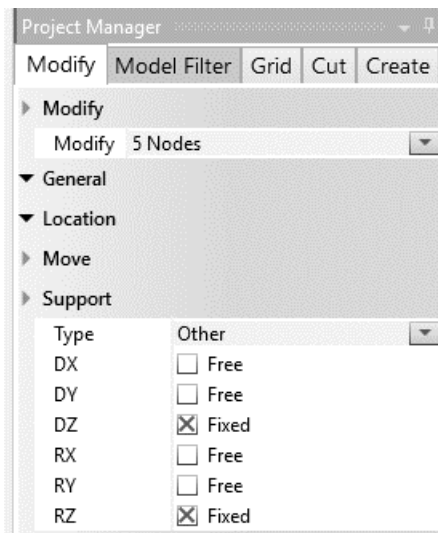
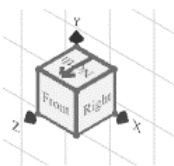
The Create Tab of the Project Manager shows other objects and systems that can be created. (Hovering over the menu gives instructions.)



- 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 a node is created on an existing member, the member will have to be subdivided for the truss by selecting it and with the right-click menu choosing Split Members. Any time the cursor is over an existing node, the node will change color.
- If the grid doesn't allow for a node to fall on a member, you can draw the new member such that it crosses. The crossing point will show up, and you can delete the node that is overhanging to delete the member piece you don't want.
- The geometry can be set precisely by selecting the node 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).
- Select all members by clicking on a member while holding the SHIFT key down or by drawing a boundary by holding the SHIFT key down and DRAGGING LEFT. The Project Manager will display properties to modify. You may need to change the Modify menu to "members".  
In order to make the members 2-force bodies, you will need to check the boxes for Strong, Mz 1 and Strong, Mz 2 to be Released. Circles should appear at the end of each member.



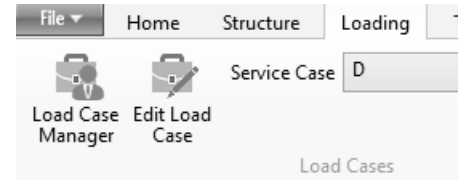
- Select all nodes by clicking a node while holding the SHIFT key down. The Project Manager will display a Support sub-item. In order for Visual Analysis to keep the truss in the x-y plane, check the boxes for DZ and RZ to be Fixed. Purple arrows should appear, and if you chose the axis graphic and rotate it out of plane, you will see purple arrows in the Z direction for each node.



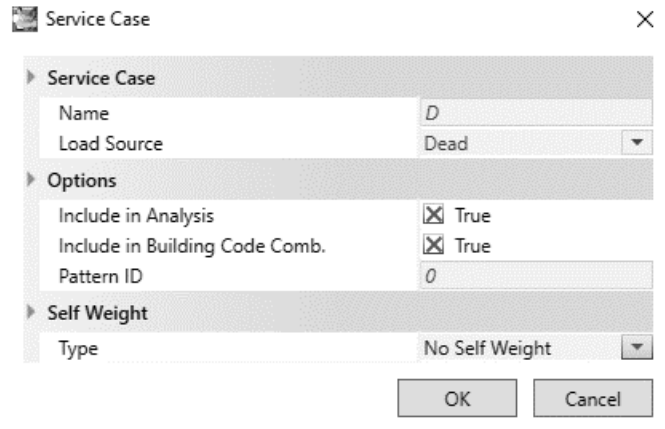
- The node with the triangle model symbol is called a pin support. Choose that node and in the Project Manager under Support, also check DX and DY to be fixed.
- The node with the triangle with wheels model symbol is called a roller support. Choose that node and in the Project Manager under Support, also check DY to be fixed.

NOTE: The support forces will be determined in the analysis. *Do not add loads at these nodes.*

- The truss geometry is complete, and we need to put on loads. We also need to tell Visual Analysis that there will be “weightless” members by choosing the Loading tab that shows the service case is “D”.

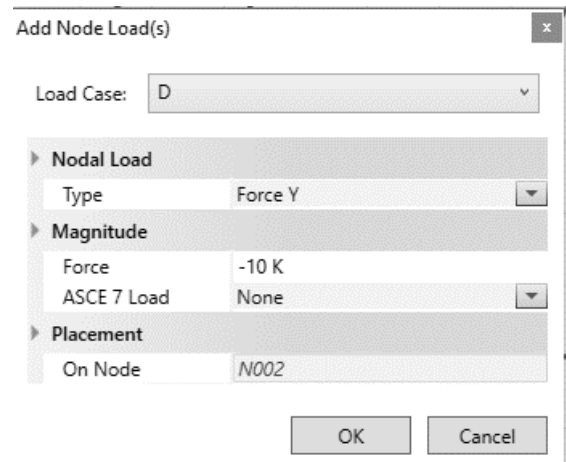


- Choose Edit Load Case and under the Self Weight Type, and choose No Self Weight.

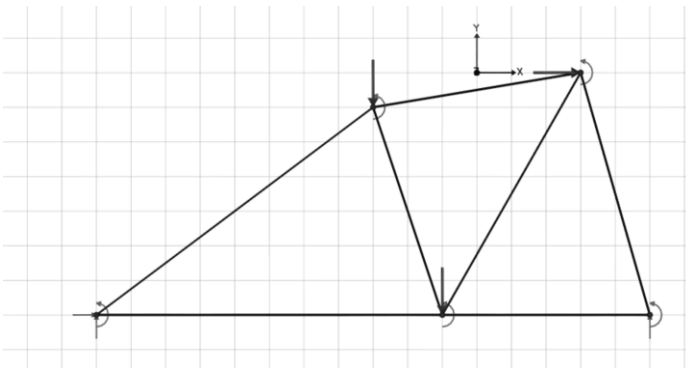


- The loads that can be applied in the Loading menu are grayed out until the node or member or other element are chosen.

- Choose the node to be loaded and select the load type (here shown for nodal loading)
- Choose the Type of Load (Force X or Force Y). Enter the magnitude of the force. A negative value is opposite of the positive global direction.
- 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.**

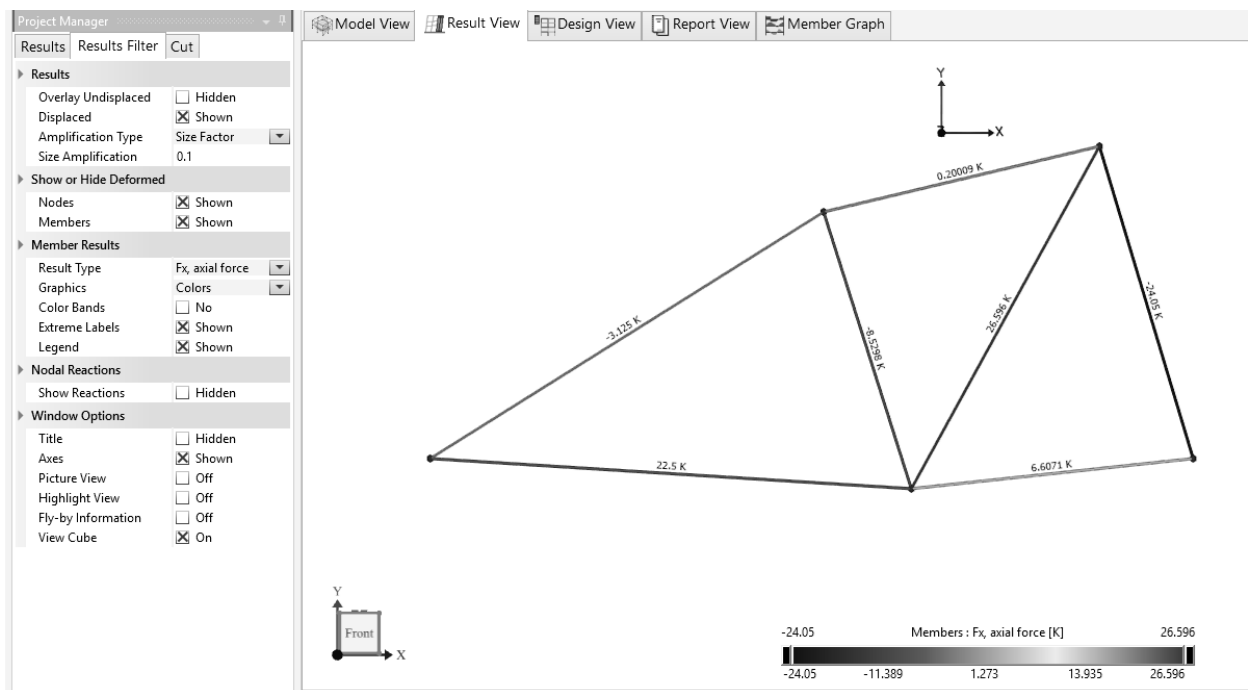
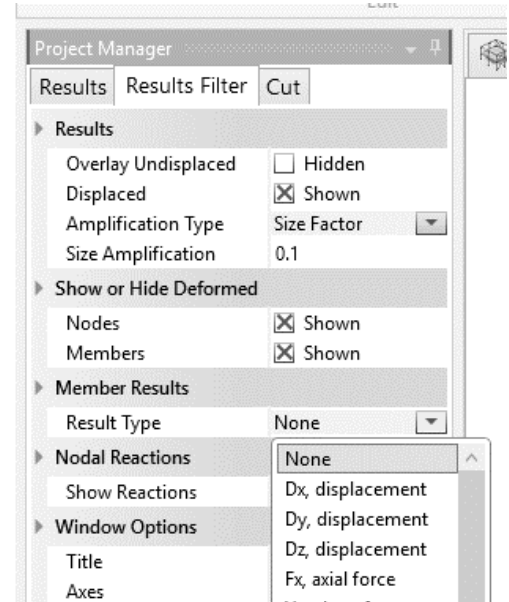


7. To run the linear analysis after the geometry, member properties, and loading have been defined, make certain that the Modify tab of the Project Manager shows under the Analysis sub-item that the Static Method is First Order.

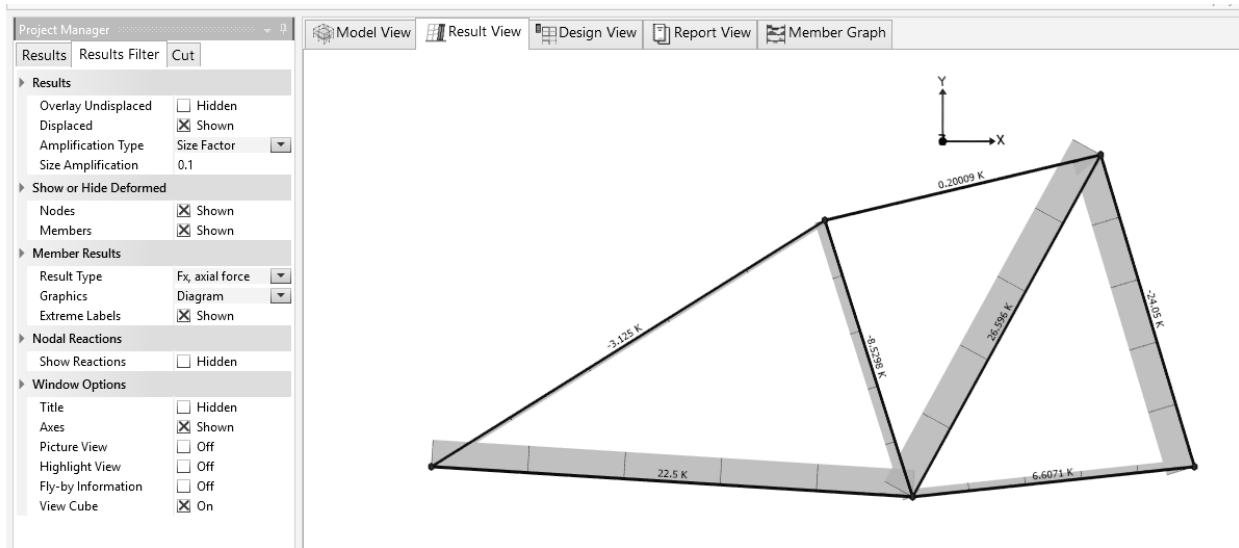
- Choose the Results View tab.

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 axial force results, choose the Result Type of Fx, axial force.
- 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 will appear as colored bars over the length of the members. (graphic next page)
- 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 and displacements, click on the member and the values will be displayed below the view in the Find Tool.



9. The Member Graph will show the three major analysis values for a member.  
NOTE: P refers to the axial load.
10. 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.
11. To save the file choose Save or Save Project from the File menu.
12. To load an existing file choose Open Project from the File menu.
13. To print a plot choose Print or Print Preview Window from the File menu. There is a Page Setup command under the File menu.