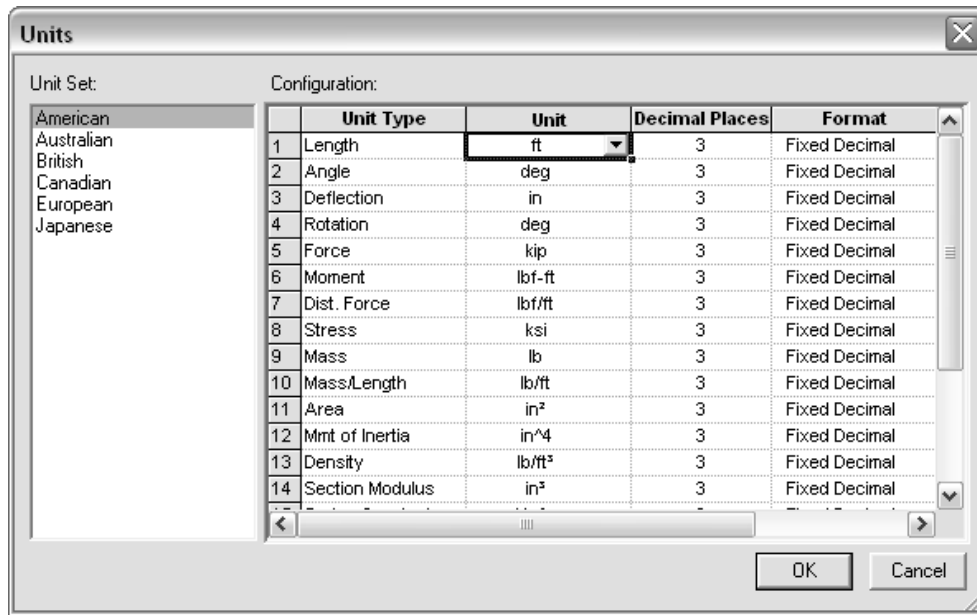
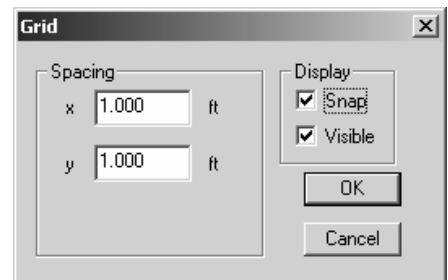


Truss Analysis using Multiframe

1. The software is on the computers in the College of Architecture in Programs under the Windows Start menu (see <https://wikis.arch.tamu.edu/display/HELPDESK/Computer+Accounts> for lab locations). Multiframe is under the Bentley Engineering menu.
2. There are tutorials available on line at <http://www.formsys.com/mflearning> that list the tasks and order in greater detail. The first task is to define the unit system:
 - Choose Units... from the View menu. Unit sets are available, but specific units can also be selected by double clicking on a unit or format and making a selection from the menu.



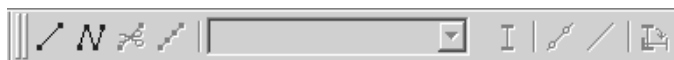
3. To see the scale of the geometry, a grid option is available:
 - Choose Grid... from the View menu



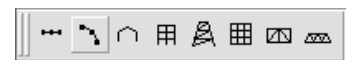
4. To create the geometry, you must be in the Frame window (default). The symbol is the frame in the window toolbar:



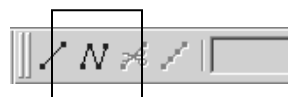
The Member toolbar shows ways to create members:



The Generate toolbar has convenient tools to create typical structural shapes.

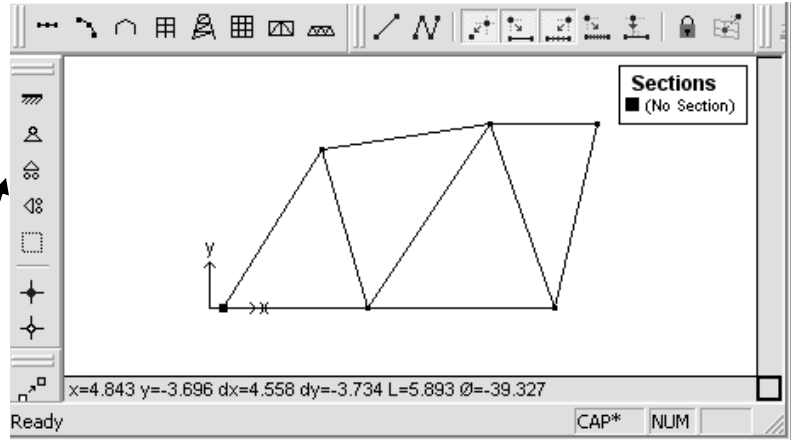


- To create a truss, use the add connected members button:



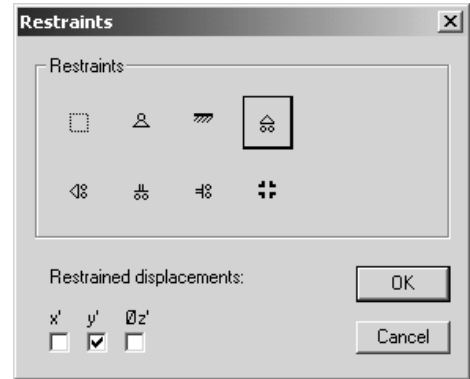
- Select a starting point and ending point with the cursor. The location of the cursor and the segment length is displayed at the bottom of the geometry window. The ESC button will end the segmented drawing. Continue to use the add connected members button. Any time the cursor is over an existing joint, the joint will be highlighted by a red circle.

- The geometry can be set precisely by selecting the joint (drag), and bringing up the joint properties menu (right click) to set the coordinates.



- The support types can be set by selecting the joint (drag) and using the Joint Toolbar (pin shown), or the Frame / Joint Restraint... menu (right click).

NOTE: If the support appears at both ends of the member, you had the member selected rather than the joint. Select the joint to change support for and right click to select the joint restraints menu or select the correct support on the joint toolbar.



The support forces will be determined in the analysis.

5. All members must have sections assigned (see section 6.) in order to calculate reactions and deflections. To use a standard steel section **proceed to step 6**. For custom sections, the section information must be entered. To define a section:

- Choose Edit Sections / Add Section... from the Edit menu
- Type a name for your new section
- Choose group Frame from the group names provided so that the section will remain with the file data
- Choose a shape. The Flat Bar shape is a rectangular section.
- Enter the cross section data.

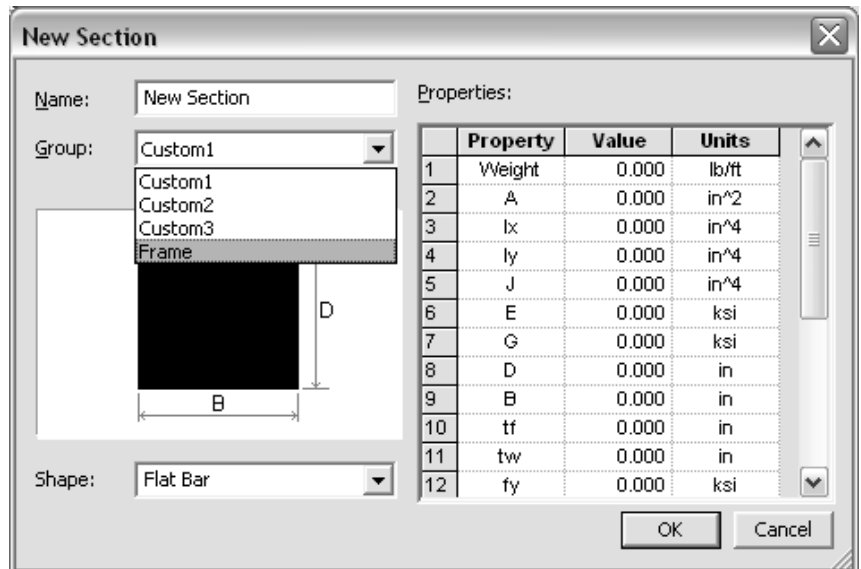


Table values 1-9 must have values for a Flat Bar, but not all are used for every analysis. A recommendation is to put the value of 1 for those properties you don't know or care about. Properties like t_f , t_w , etc. refer to wide flange sections.

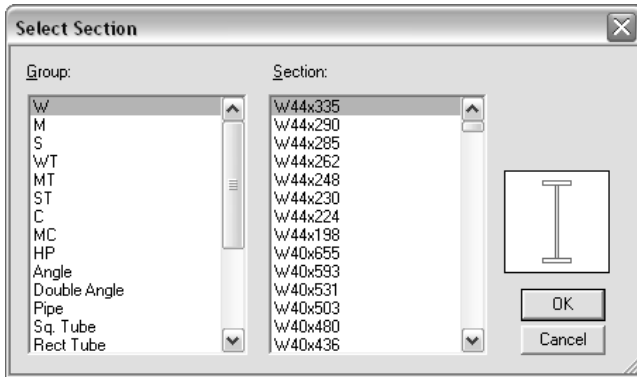
- Answer any query. If the message says there is an error, the section will not be created until the error is corrected.
6. The standard sections library loaded is for the United States. If another section library is needed, use the Open Sections Library... command under the file menu, choose the library folder, and select the SectionsLibrary.slb file.

Select the members (drag to make bold) and assign sections with the Section button on the Member toolbar:

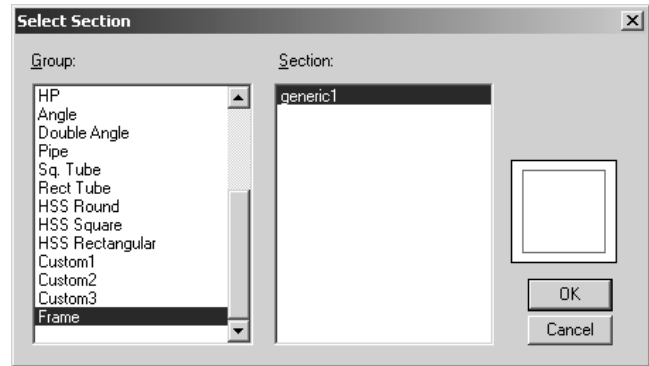


- Choose the group name and section name:

(STANDARD SHAPES)



(CUSTOM)



7. In order for Multiframe to recognize that the truss members are two-force bodies, all joints must be highlighted and assigned as pins with the Pinned Joints button on the Joint toolbar:



8. The truss geometry is complete, and in order to define the load conditions you must be in the Load window represented by the green arrow:



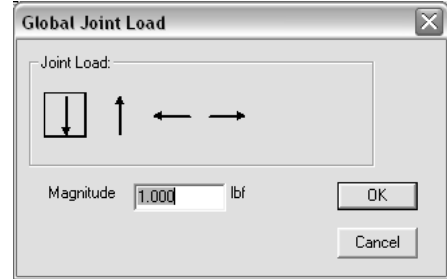
9. The Load toolbar allows a joint to be loaded with a force or a moment in global coordinates, shown by the first two buttons after the display numbers button. It allows a member to be loaded with a distributed load, concentrated load or moment (next three buttons) in global coordinates, as well as loading with distributed or single force or moment in the local coordinate system (next three buttons). It allows a load panel to be loaded with a distributed load in global or local coordinates (last two buttons).



- Choose the joint to be loaded (drag) and select the load type (here shown for point loading):

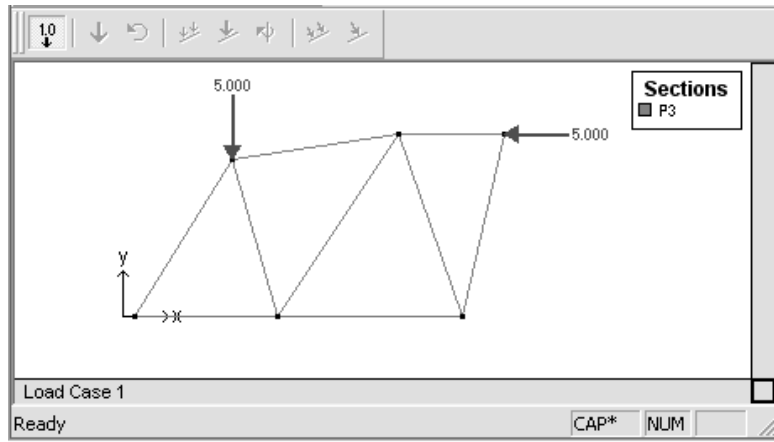


- Choose the direction by the arrow shown. There is no need to put in negative values for downward loading.
- Enter the values of the load



NOTE: Do not put support reactions as applied loads. The analysis will determine the reaction values

Multiframe will automatically generate a grouping called a Load Case named Load Case 1 when a load is created. All additional loads will be added to this load case unless a new load case is defined (Add case under the Case menu).



10. In order to run the analysis after the geometry, member properties and loading has been defined:

- Choose Analyze Linear from the Case menu

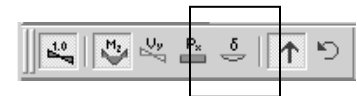
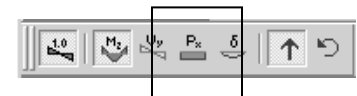
11. If the analysis is successful, you can view the results in the Plot window represented by the red moment diagram:



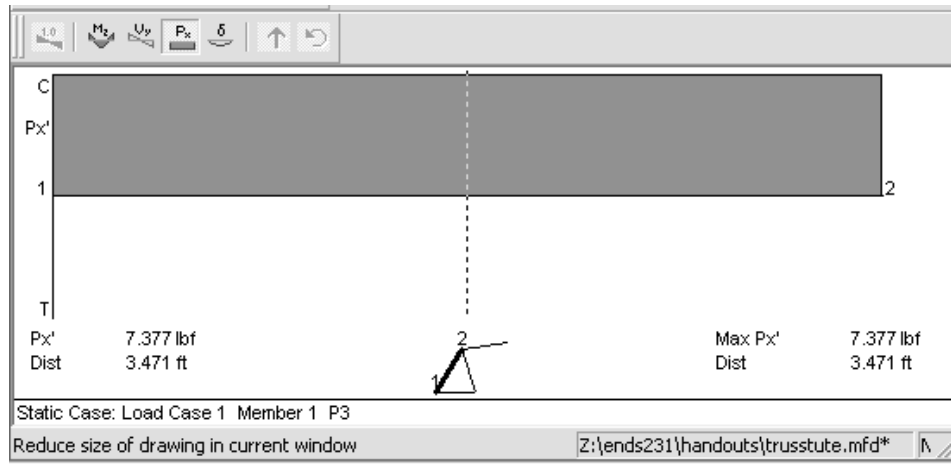
12. The Plot toolbar allows the numerical values to be shown (1.0 button), the reaction arrows to be shown (brown up arrow) and reaction moments to be shown (brown curved arrow):



- To show the axial force diagram, Choose the purple Axial Force button. Tensile members will have “T” by the value (if turned on), while compression members will have “C” by the value
- To show the deflection diagram, Choose the blue Deflection button
- To animate the deflection diagram, Choose Animate... from the Display menu. You can also save the animation to a .avi file by checking the box.



- To see exact values of axial load and deflection, double click on the member and move the vertical cross hair with the mouse. The ESC key will return you to the window.



13. The Data window (D) allows you to view all data “entered” for the geometry, sections and loading. These values can be edited.



14. The Results window (R) allows you to view all results of the analysis including displacements, reactions, member forces (actions) and stresses. These values can be cut and pasted into other Windows programs such as Word or Excel.



Static Case: Load Case 1					
	Joint	Label	Rx' lbf	Ry' lbf	Mz' lbf-ft
1	1		5.000	6.246	0.000
2	2		0.000	-0.000	0.000
3	3		-0.000	-0.000	0.000
4	4		0.000	0.000	0.000
5	5		0.000	-1.246	0.000
6	6		0.000	-0.000	0.000
7	Total (Global)		Rx=5.000	Ry=5.000	

Static Case: Load Case 1						
	Memb	Label	Joint	Px' lbf	Vy' lbf	Mz' lbf-ft
1	1		1	7.377	0.000	0.000
2	1		2	-7.377	0.000	0.000
3	2		2	-0.681	0.000	0.000
4	2		3	0.681	0.000	0.000
5	3		1	1.075	0.000	0.000
6	3		3	-1.075	0.000	0.000
7	4		2	4.157	0.000	0.000
8	4		4	-4.157	0.000	0.000

NOTE: Px' refers to the axial load (P) in the local axis x direction (x').

- To save the file Choose Save from the File menu.
- To load an existing file Choose Open... from the File menu.
- 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).