# Beam Analysis using Multiframe

1. The software is on the teaching computers in the College of Architecture in Programs under the Windows Start menu. Multiframe is under the Bentley Engineering menu. It is also available at the Open Access Labs (OAL) and the Virtual OAL at http://voal.tamu.edu

2. There are tutorials available on line at http://www.daystarsoftware.com/support/mftutorials 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.
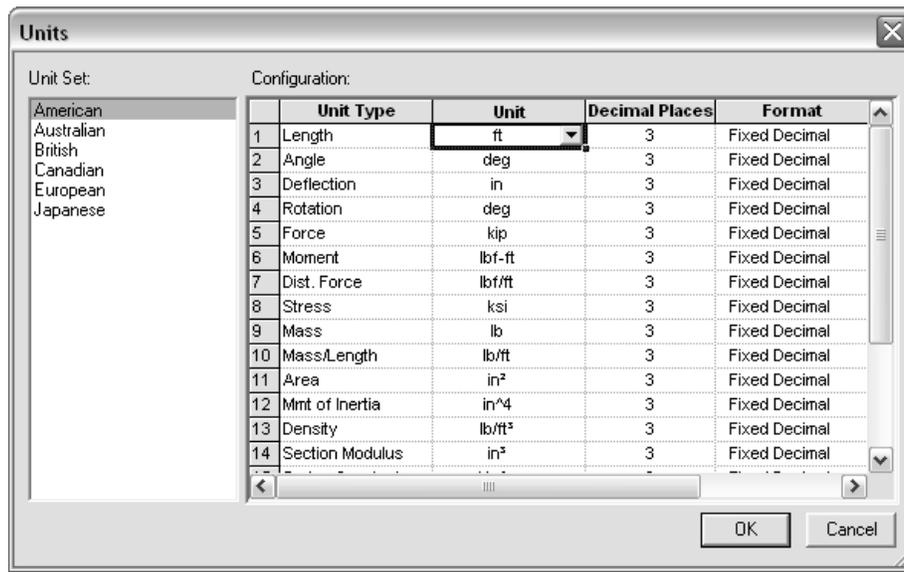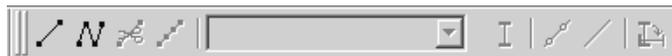

3. To see the scale of the geometry, a grid option is available:

   - Choose Grid… from the View menu
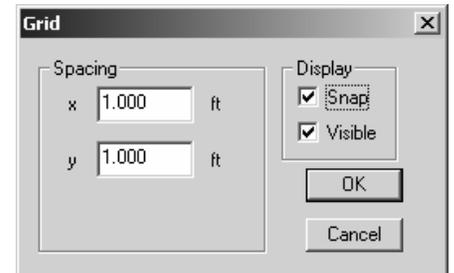
4. To create the geometry, you must be in the Frame window (default). The symbol is the frame in the window toolbar:

   The Member toolbar shows ways to create members:

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


   - To create a beam with supports at one or both ends, use the add member 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.

- To create a beam with supports NOT at the ends, use the add connected members button to create segments between supports and ends

- 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.

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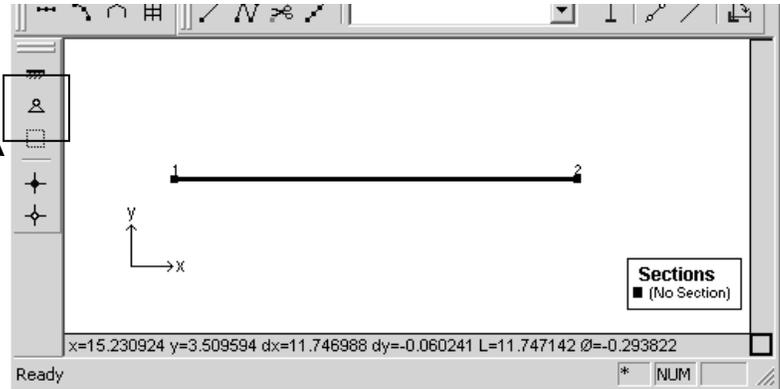
- The support types can be set by selecting the joint (drag) and using the Joint Toolbar (pin shown), or the Frame / Joint Restraint ... menu (right click).

  NOTE: If the support appears at both ends of the beam, you had the beam selected rather than the joint. Select the joint to change the support for and right click to select the joint restraints menu or select the correct support on the joint toolbar.
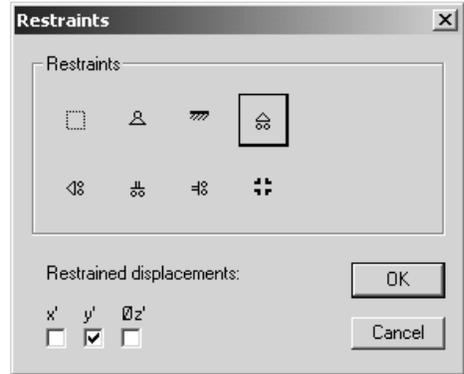
The support forces will be determined in the analysis.

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 <u>Frame</u> 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.
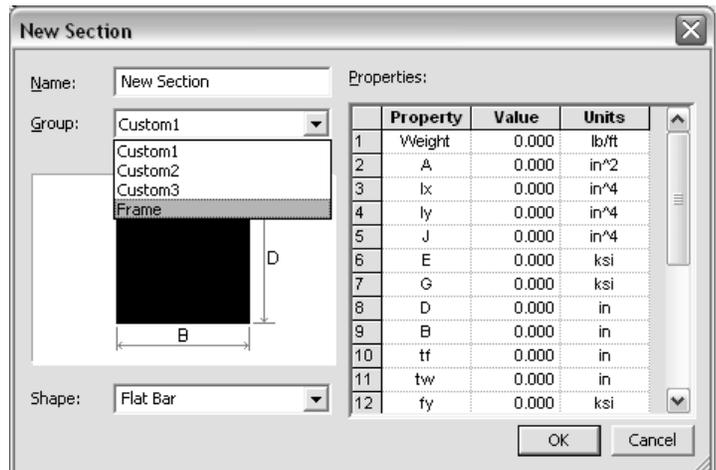
- Enter the cross section data.

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:
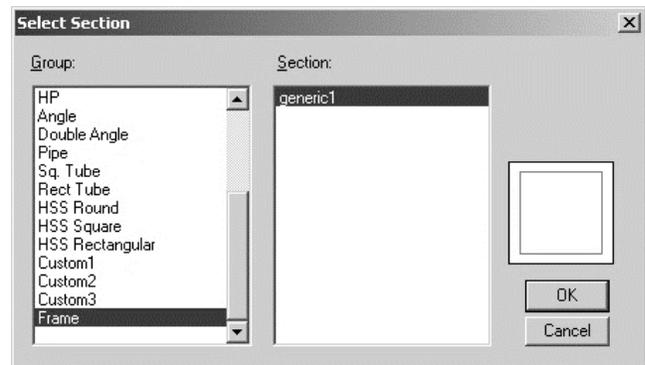

- Choose the group name and section name:
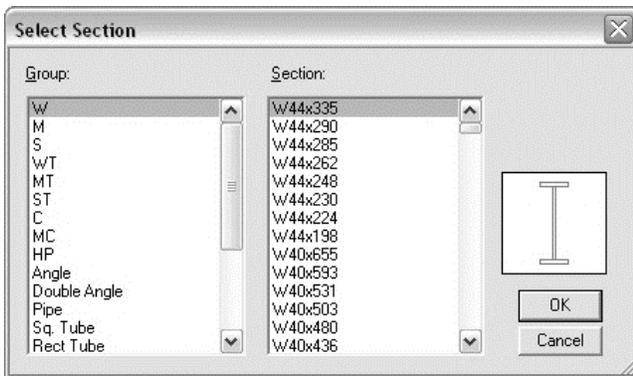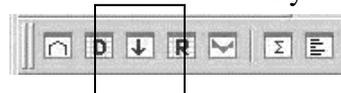
<center>(STANDARD SHAPES)   (CUSTOM)</center>

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

8. 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 member to be loaded (drag) and select the load type (here shown for global distributed loading):

<center>121</center>

- Choose the distribution type and direction. Note that the arrow shown is the direction of the loading. There is no need to put in negative values for downward loading.

- Enter the values of the load and distances (if any). Distances can be entered as a function of the length , i.e. L/2, L/4...

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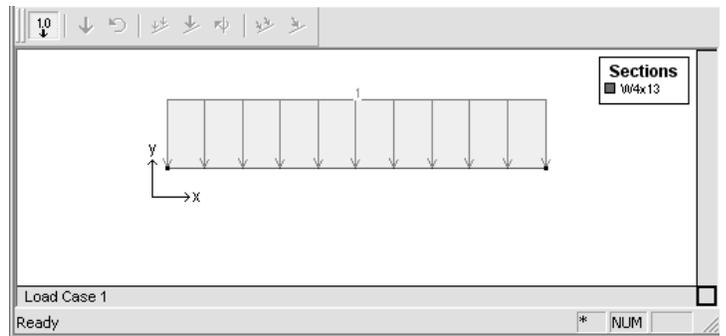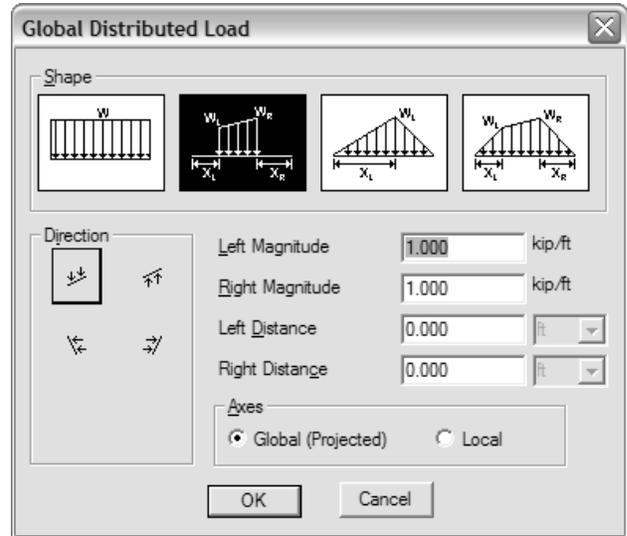
9. In order to run the analysis after the geometry, member properties and loading has been defined:

- Choose Linear from the Analyze menu

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

11. 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):
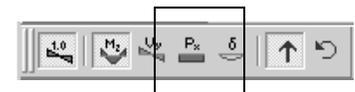

- To show the moment diagram, Choose the red Moment button
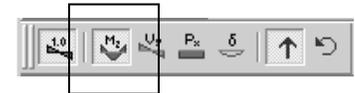
- To show the shear diagram, Choose the green Shear button

- To show the axial force diagram, Choose the purple Axial Force button
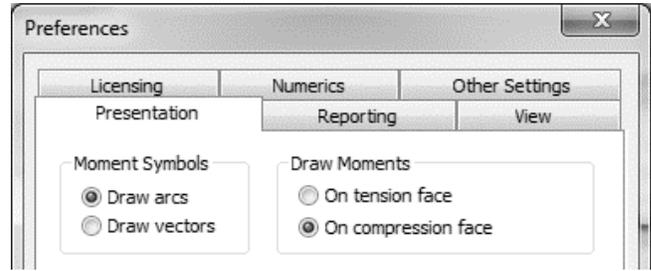

- To show the deflection diagram, Choose the blue Deflection button

- 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 plot the bending moment on the "top" choose Preferences from the Edit menu and under the Presentation tab Draw moments on the compression face

- To see exact values of shear, moment 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.
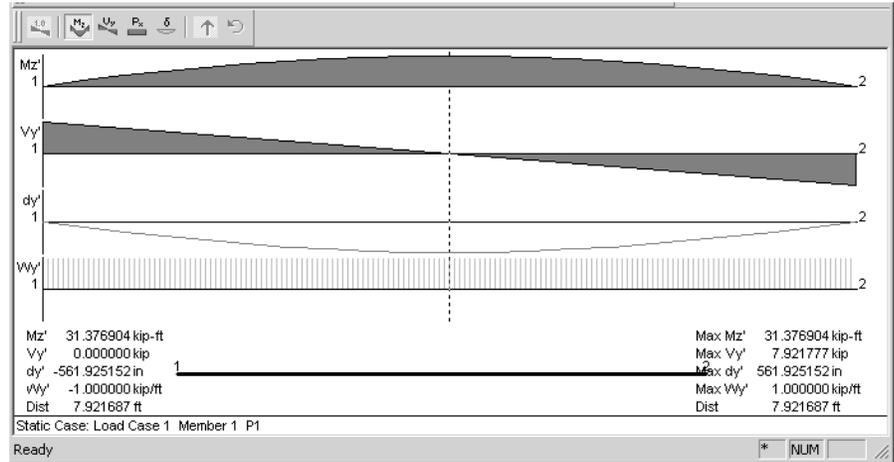
12. The Data window (D) allows you to view all data "entered" for the geometry, sections and loading. These values can be edited.

13. 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' kip	Ry' kip	Mz' lbf-ft
1	1		0.000	7.922	0.000
2	2		0.000	7.922	0.000
3	Total	(Global)	Rx=0.000	Ry=15.844	

Static Case: Load Case 1

	Memb	Label	Joint	Px' kip	Vy' kip	Mz' lbf-ft
1	1		1	0.000	7.922	0.000
2	1		2	0.000	7.922	0.000

NOTE: Px' refers to the axial load (P) in the local axis x direction (x'). Vy' refers to the shear perpendicular to the local x axis, and Mz' refers to the bending moment.

14. To save the file Choose Save from the File menu.

15. To load an existing file Choose Open... from the File menu.

16. 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).