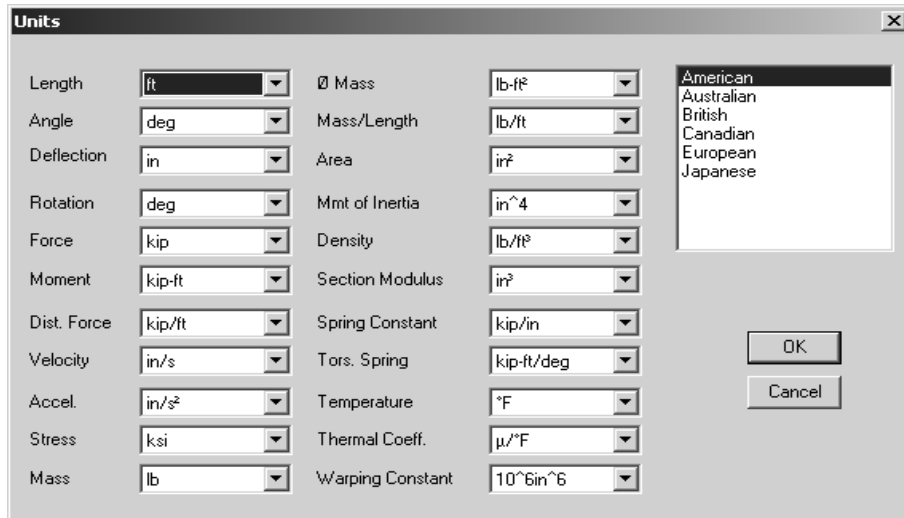


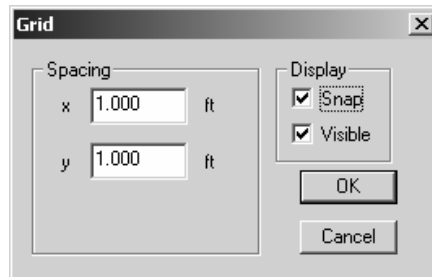
Beam Analysis using Multiframe2D

1. The software is on the computers in the college computing lab (<http://thelab.tamu.edu>) in Programs under the Windows Start menu. Multiframe 2D is under the Multiframe menu.
2. There is a tutorial in the Help menu (Chapter 1 – 2D Tutorial) that lists the tasks and order in greater detail. The first task is to define the unit system:
 - Choose Units... from the View menu



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

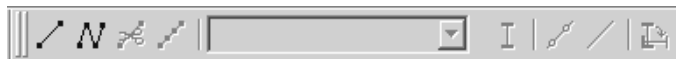
- Choose Grid... from the View menu



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:



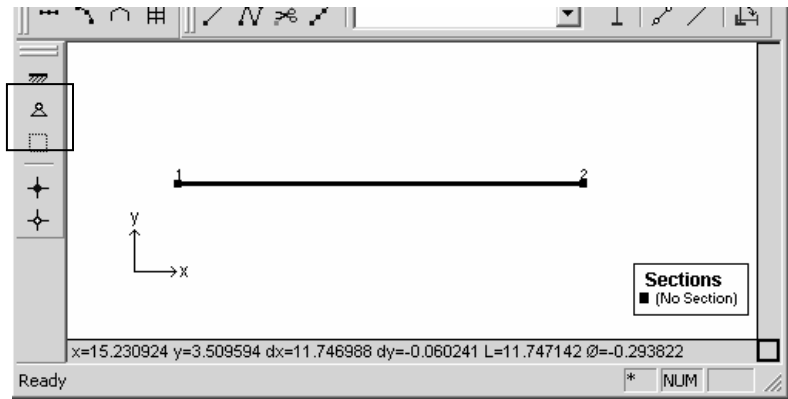
The Generate toolbar has convenient tools to create typical structural shapes.

- To create a beam, use the add member button:

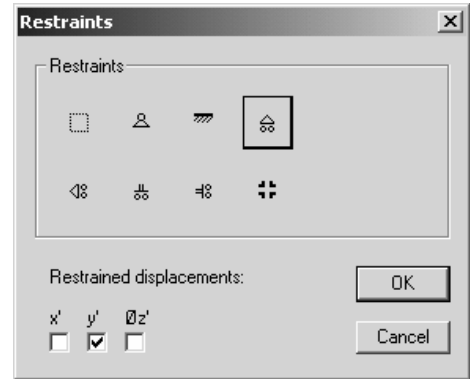


- Select a starting point and ending point with the cursor. The location of the cursor and the segment length is displayed at the bottom of the geometry window.

- 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.



- The support types can be set using the Joint Toolbar (pin shown), and the Frame / Joint Restraint ... menu.



5. All members must have sections assigned in order to calculate deflections. There is a list of standard steel sections, but for any other type the 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 a group from the four group names provided
 - Choose a shape. The Flat Bar shape is a rectangular section.
 - Enter the cross section data.

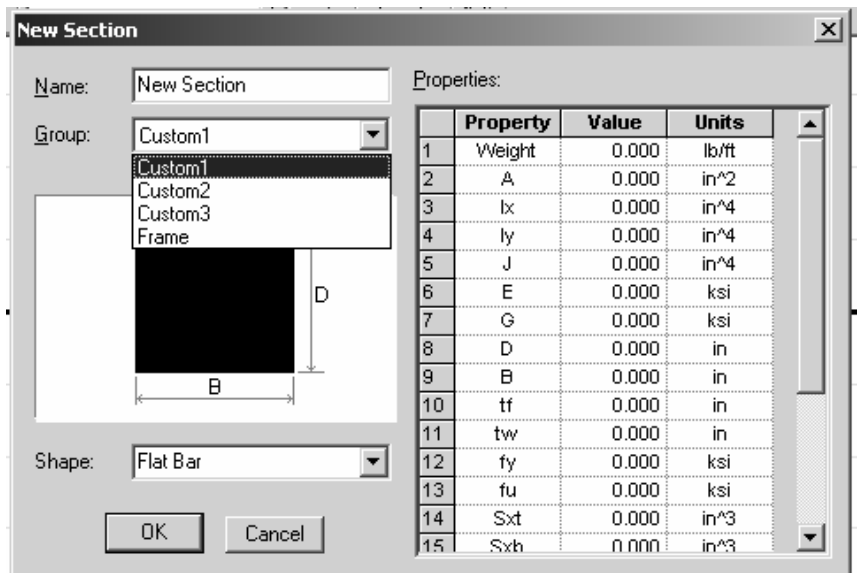
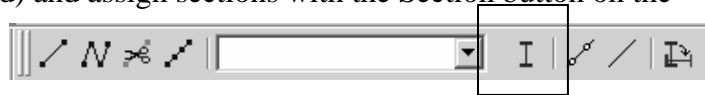


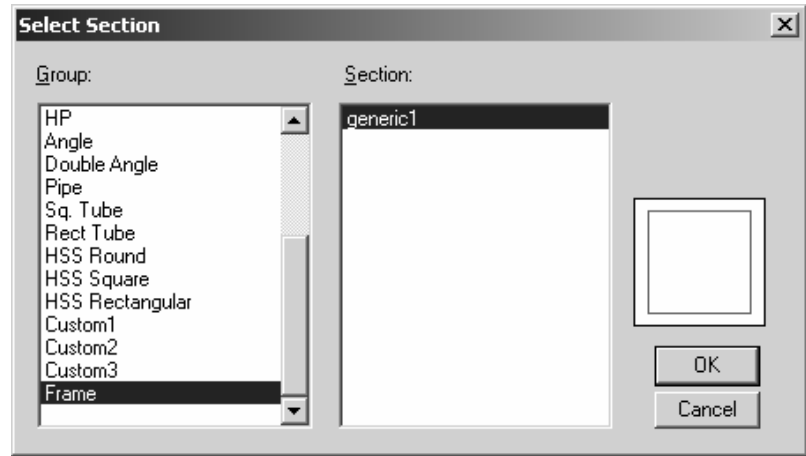
Table values 1-9 must have values for a Flat Bar, but not all are used for every analysis. A recommendation is to put the value of 1 for those properties you don't know or care about. Properties like t_f , t_w , etc. refer to wide flange sections.

- Answer any query. If the message says there is an error, the section will not be created until the error is corrected.

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



- Choose the group name and section name:



- 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:



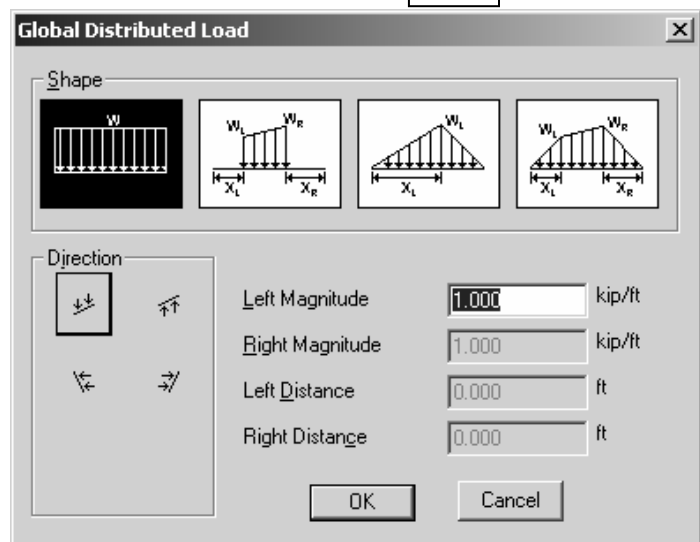
- The Load toolbar allows a joint to be loaded with a force or a moment in global coordinates, shown by the first two buttons. It allows a member to be loaded with a distributed load, concentrated load or moment (next three buttons) in global coordinates, as well as loading with distributed or single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



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



- 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...

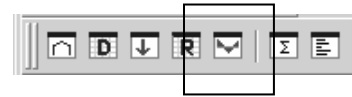


Multiframe will automatically generate a grouping called a Load Case named Load Case 1 when a load is created. All additional loads will be added to this load case unless a new load case is defined (Add case under the Case menu).

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

- Choose Analyze Linear from the Case menu

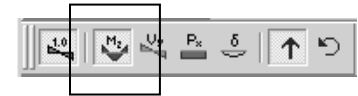
10. If the analysis is successful, you can view the results in the Plot window represented by the red moment diagram:



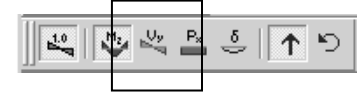
11. The Plot toolbar allows the numerical values to be shown (1.0 button), the reaction arrows to be shown (brown up arrow) and reaction moments to be shown (brown curved arrow):



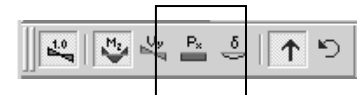
- To show the moment diagram, Choose the red Moment button



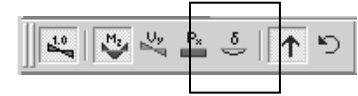
- To show the shear diagram, Choose the green Shear button



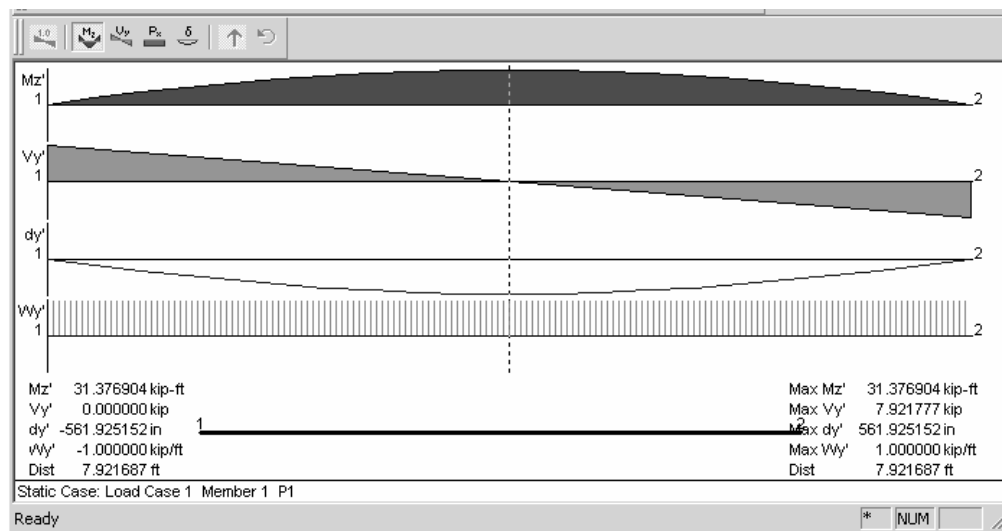
- To show the axial force diagram, Choose the purple Axial Force button



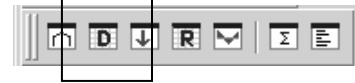
- To show the deflection diagram, Choose the blue Deflection button



- 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 see exact values of shear, moment and deflection, double click on the member and move the vertical cross hair with the mouse.



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.



14. To save the file Choose Save from the File menu.
15. To load an existing file Choose Open... from the File menu. The warning that appears is just reminding you that your model is 2D.

	Joint	Label	Rx' kip	Ry' kip	Mz' kip-ft
1	1		0.000000	7.921777	-0.000000
2	2		0.000000	7.921777	0.000000
3	Total (Global)		Rx=0.000000	Ry=15.843	

Ready * NUM