

Tutorial for RISA-2D Educational

For use with *Fundamentals of Structural Analysis*, Fourth Edition
by Kenneth M. Leet, Chia-Ming Uang, and Anne Gilbert

1. OVERVIEW OF RISA-2D EDUCATIONAL COMPUTER PROGRAM

The educational version of RISA-2D, a computer program developed by RISA Technologies for use with *Fundamentals of Structural Analysis*, Fourth Edition, by Leet, Uang, and Gilbert, is a first-order, interactive computer program for the analysis of two-dimensional (planar) structures, such as beams, trusses, and frames.

RISA-2D Educational was developed to make the definition, solution, and modification of two-dimensional determinate or indeterminate structures simple and fast. The program allows users to draw and edit a model of the structure on the computer screen. The final results—reactions, axial force, shear, and moment diagrams, as well as the deflected shape—can be displayed graphically. This educational version, limited to structures with maximum of 50 joints and 50 members, will permit the student to analyze the majority of problems in a basic course in structural analysis.

RISA-2D Educational performs a *first-order* analysis, which assumes that joint displacements are small enough so that a valid analysis can be based on the *initial geometry* of the structure. Since the program does not account for the effect of additional forces and deflections produced by large joint displacements, *forces* and *deflections* of slender members carrying compression or compression and moment (such as long slender arches or columns in rigid frames) may be underestimated significantly if the structure is flexible and the applied loads are large.

The computer solution is based on the direct stiffness method, covered in Chapters 17 and 18 of the textbook. In this procedure, the stiffness of each element of the structure is calculated. These stiffnesses are then combined to produce the global stiffness matrix of the structure. The global stiffness matrix is solved for the applied loads to calculate joint deflections, which in turn are used to calculate internal forces and deformations of the individual element.

To introduce RISA-2D Educational, we will use the program to analyze first a truss and then a frame. You need to download a self-extracting file containing both the RISA Educational software and this tutorial from <http://www.mhhe.com/leet>. A "Help" file is also provided for a more detailed description of the program's features.

2. INTRODUCTION

This tutorial describes the use of RISA-2D Educational software for the analysis of planar structures. When the computer program is opened, three items will appear on the computer screen (see Figure 1):

- Three rows of symbols and a row of commands along the top of the screen
- A rectangular grid of lines on which the structure will be depicted. The origin of the x-y coordinate system is positioned at the lower left corner of the grid.
- A **Data Entry** box appears on the top right side of the screen. The data required to define the structure, its loads, and its supports are entered here.

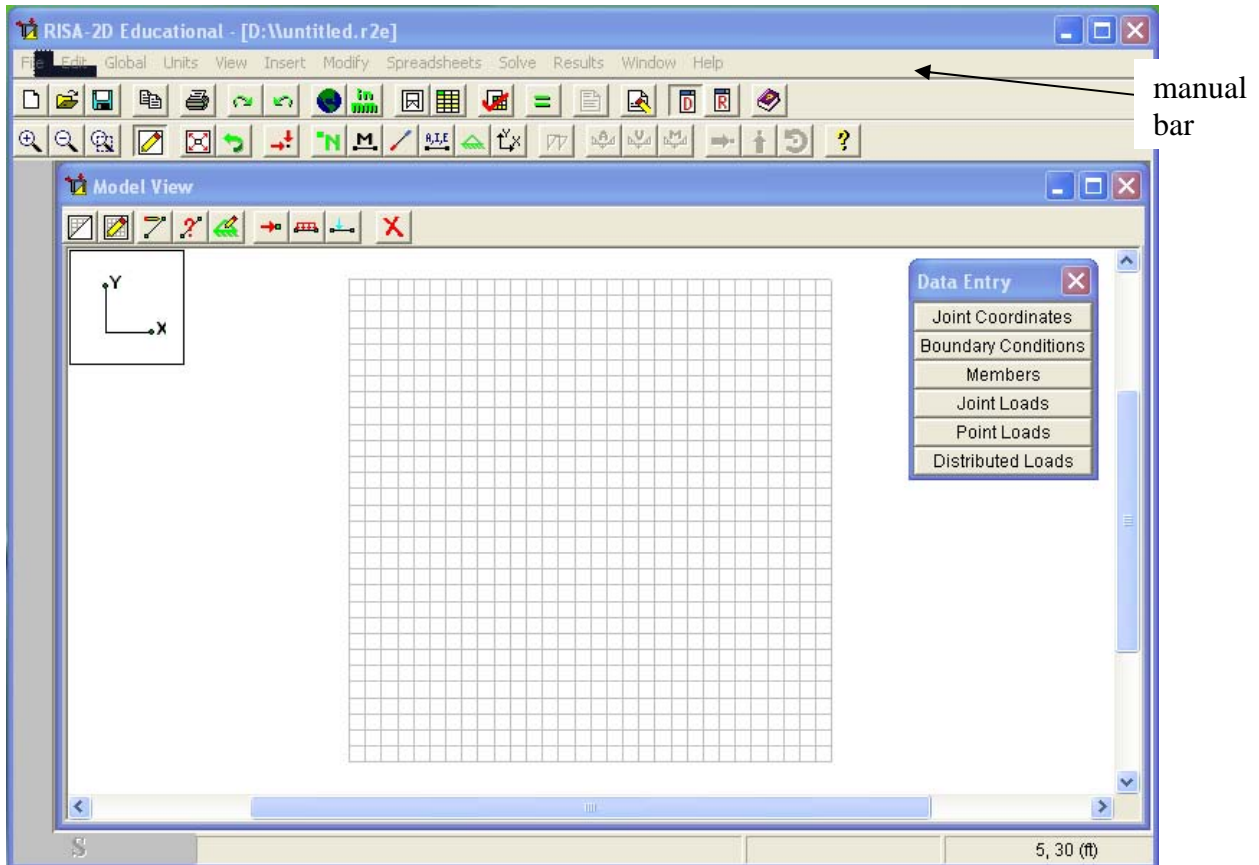


Figure 1

As data is inserted into the six tables in the **Data Entry** box, a figure of the structure, its supports, and the design loads will automatically appear on the grid. This figure permits the designer to verify visually the accuracy of the input data.

To perform the computer analysis, click on **Solve** at the top of the screen. (If for any reason a structure is unstable, the computer program will flash a warning on a red screen and will not carry out the analysis.) The program then produces tables that list values of support reactions, joint deflections, and internal forces in members. There are additional commands to label members and joints, to sketch the deflected shape to an exaggerated scale, and to draw shear, moment, and axial load diagrams.

3. TRUSS ANALYSIS

The planar truss in Figure 2 is indeterminate to the first degree, but it is not necessary to establish the degree of indeterminacy when using the computer program. The area of all members is 1.2 in^2 , and Young's Modulus, E , is equal to 29,000 ksi.

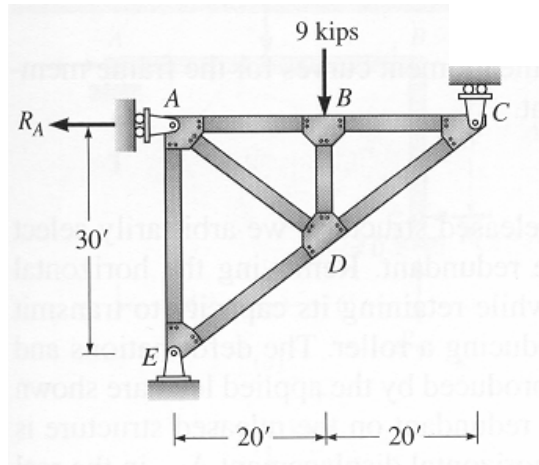


Figure 2

Figure 3

To begin the analysis, click on **Global** (top left of screen). The table **Global Parameters** (Figure 3) provides space for the **Title** of the analysis, the name of the **Designer**, and the **Number of Sections** along the member's length. Since we wish to base the analysis on the assumption that the truss bars are pin-ended and only carry axial load, we set the number of sections equal to 2 to establish the forces at each end of the truss bars.

If the designer does not specify the number of sections, the program will automatically specify 5 equally spaced sections along the member's axis as the default value. For beams and frames, the program will compute the magnitude of the shear, moment, and axial load at each section.

The box **Notes** provides space for additional descriptive comments, such as the grade of steel, controlling design code, and so on. Click **OK** when entries are completed.

- (1) To establish the units, click **Units** at the top of the screen to activate the **Units Selection** toolbar (Figure 4), which provides several options:
 - You can specify **Use CONSISTENT Units**. For this option, you must express all physical quantities—length, area, modulus of elasticity E , and so on—in terms of consistent units, for example, inches and kips.
 - For practical applications, the program provides a more convenient way of handling unit conversions by allowing the designer to click on either **Standard Metric** or **Standard Imperial** for U.S. customary units (USCU). The designer specifies the units for each variable, and the program will automatically handle the conversion internally to place all units on a consistent basis. We choose **Standard Imperial** in our example since all member properties are expressed in USCU.
 - Or if you wish, set units in convenient dimensions (as in Figure 4), and then click **Convert Existing Data for Any Units Changes**. Click **OK** once you have completed the table.

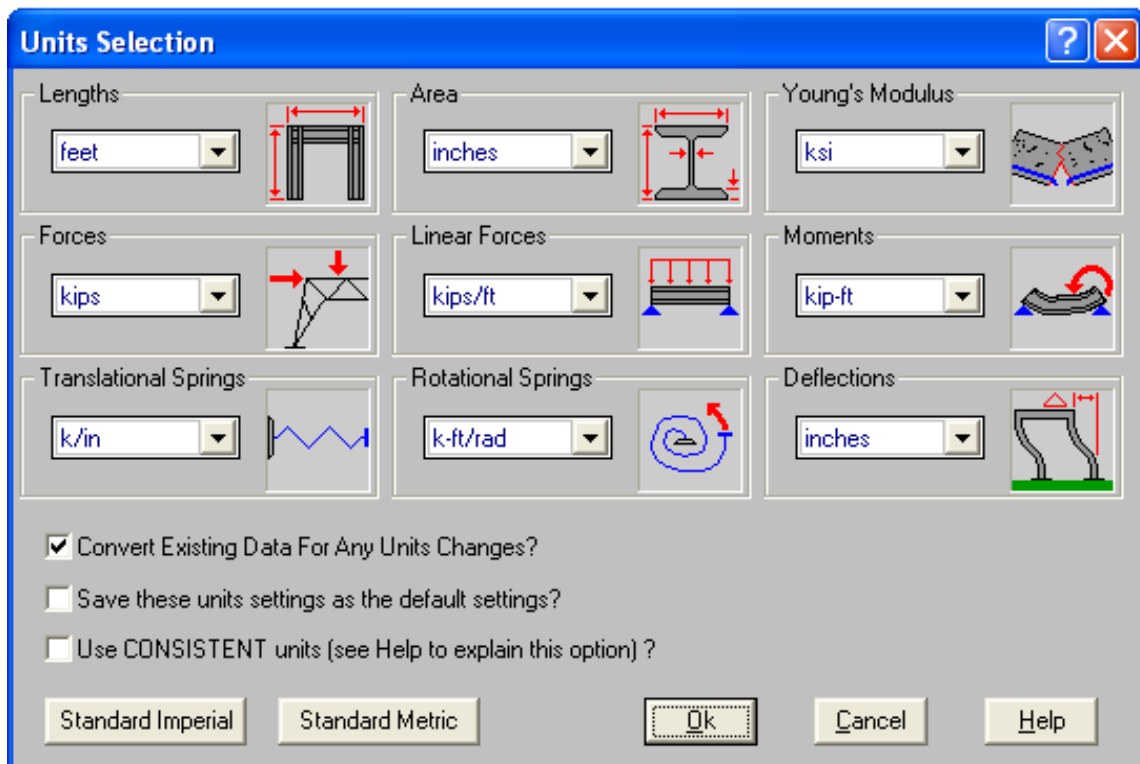


Figure 4

- (2) To establish the grid dimensions, click **Modify** (top screen) and select **Grid**. A **Define Drawing Grid** table appears (Figure 5). This table shows that the program assigns default values of 30 units, 1 ft in length, to the y-direction and 30 units, 1 ft in length, to the x-direction. The default value also positions the origin of the

coordinate system at $x = 0$ and $y = 0$, which is the lower left corner. The right-hand rule is used such that counterclockwise rotation is positive.

If the dimensions of the structure fit in the grid, click **OK**. If the dimensions of the structure exceed those of the grid, the grid dimensions can be modified by filling in the lines under **X-Axis (ft)** and **Y-Axis (ft)** with a revised scale. Considering the overall dimensions of the structure in Figure 2, we can change the default setting to 4@10 ft and 6@5 ft in the X and Y directions, respectively, such that all joints will fall on the grid to facilitate visual inspection.

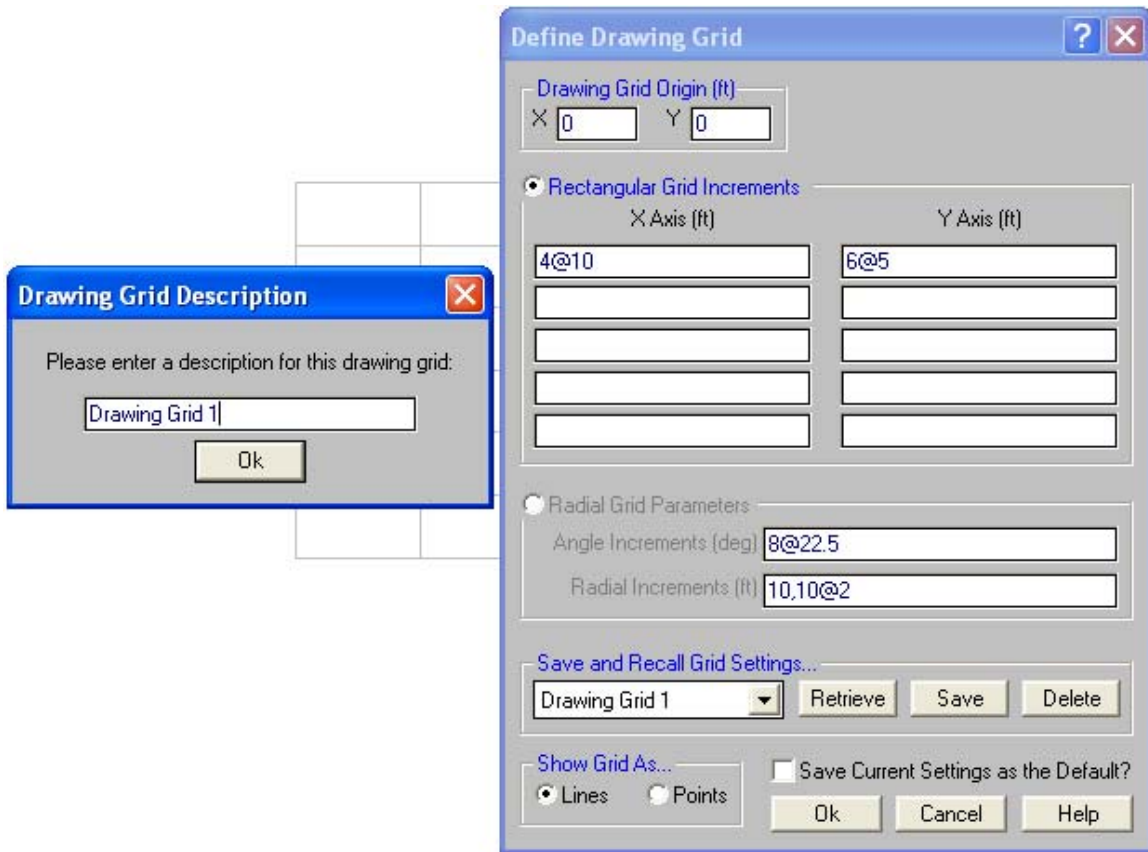


Figure 5

- (3) Next fill in the appropriate tables contained in the **Data Entry** box (Figure 6). Starting at the top, click on **Joint Coordinates** to open the **Joint Coordinates** Table (Figure 7) to define the position of each joint.

Follow the instruction in the **Joint Coordinates** window to define each joint. The program by default labels each joint sequentially as N1, N2, . . . , etc. But you can rename each joint, as long as the joint name does not start with a number (e.g., 1N). We arbitrarily label joint A as N1 at $x = 0$ and $y = 0$, and a black dot appears at the lower left corner (i.e., the origin) of the grid. Next click the **Enter** key on the keyboard to produce the second line for joint B (labeled as N2). Insert $x = 0$ and $y =$

30; click **Enter** again. Alternatively, click **Edit** from the manual bar or right click the mouse and select **Insert Line** to add additional joints. Once you have completed the table, all the joints will appear as black dots on the screen. Close the table by pressing the **X** at the top right corner of the table.

To label the joints on the screen, click **View** at the top of the screen and select **Joint Labels**.

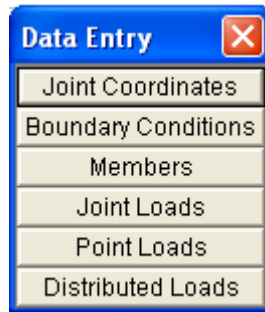


Figure 6

	Label	X [m]	Y [m]
1	N1	0	0
2	N2	0	30
3	N3	20	30
4	N4	40	30
5	N5	20	15

Figure 7

- (4) To produce the table that defines the joint restraints (Figure 8), return to the **Data Entry** box and click **Boundary Conditions**.
- Press the **Entry** key to produce the first line labeled Joint N1.
 - Click on the space under column X, which turns green and produces a red arrow on the right side of the cell.
 - Click the red arrow to produce a box labeled **Set Boundary Condition** (Figure 8).
 - Since the hinge support at Joint 1 provides restraint in the horizontal or x-direction, click **fixed**.
 - Click **OK** and **Reaction** will appear in Column X.
 - Click in the space under Column Y; again click on the red arrow; again click on **fixed**, indicating no displacement in the y-direction.
 - Click on **OK** and on the space labeled **Rotation**.
 - Since the pin does not provide rotational restraint, click **Free** and then **OK**. (You can skip this step if no restraint is provided in that particular direction.) When you

complete this line of the chart, **Reaction** will appear in the X and Y columns, and a pin support will appear on the screen at Joint 1.

- Next click **Enter** to produce the next line for the boundary conditions at the next supported joint (N2).

After this line is complete, a short horizontal line will appear on the grid at Joint N2, representing the horizontal reaction provided by the roller (Figure 11). Complete the table for the vertical restraint at Joint N4 and observe that a short vertical line, representing the vertical reaction, appears on the graph. When you are finished specifying reactions, close the box by checking the **X** at the top right corner. For final results, see Figure 8.

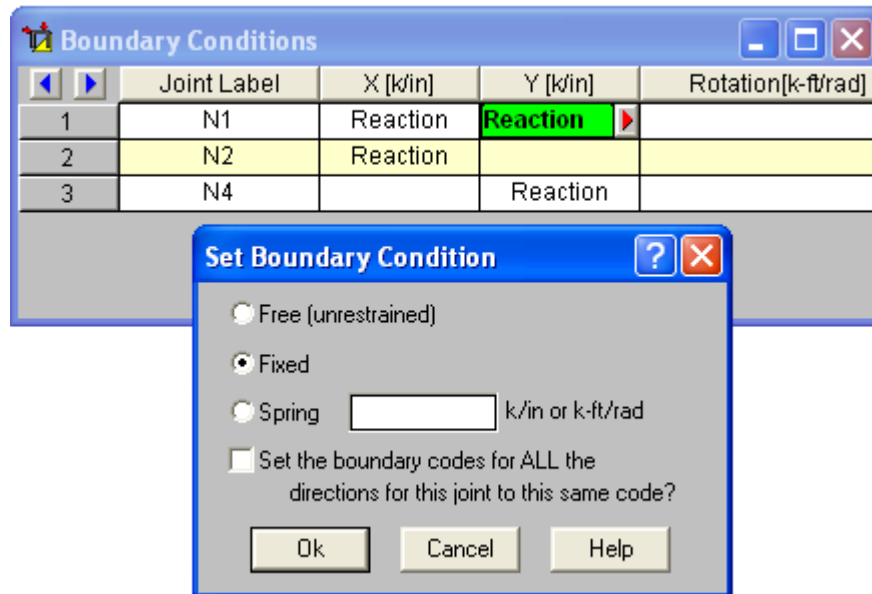


Figure 8

- (5) To insert the properties of members, return to **Data Entry** and click on **Members**.
- Open the table by clicking on any key or the **Entry** key, and the line for Member 1 (M1) opens (Figure 9). The I joint can be the lower numbered joint, N1, and the J joint is the joint N2 at the other end.
 - If we wish to analyze the truss, assuming that the joints are rigid, we must provide each member's area, A , as well as the moment of inertia, I .
 - If we want to simplify the analysis and assume that the joints are pinned, we only have to provide the areas of the members and we can leave the default value $I = 100$ (the computer program does not consider the value of I in the analysis if pins are specified at each end of the truss bars).
 - A default value (= 29,000 ksi) of Young's modulus, E , is given. If needed, change the value of E .
 - The next two columns, called **I Release** and **J Release**, are provided to insert pins at the ends of the member where they connect to the joint. To insert a pin, click on

the space below the release. A red arrow appears. Click on the red arrow, and **Set Boundary Condition** appears.

- To produce a pin joint, click **Pinned**. To produce a rigid joint, click **Fully Fixed**.
- Click **OK** to complete the modification. The word **PIN** will appear in each column.

In most trusses, members are slender, and the magnitude of the bar forces will be about the same, regardless of the degree of fixity at the member ends at joints. However, if the joints of a shallow massive truss are fabricated by welding or with high strength bolts and thick gusset plates, bending stresses may be significant, and the assumption of fixed ends at joints is appropriate. If you select pinned joints, the program will insert a small open circle at the end of each member.

In the last column labeled **Length**, the member length will be computed automatically by the computer program. If the length does not appear, click the space under **Length**. Continue entering member properties by clicking **Enter** to produce the data for the remaining bars. To label members, click **View** and run down to **Member labels**. If the label doesn't appear, try again.

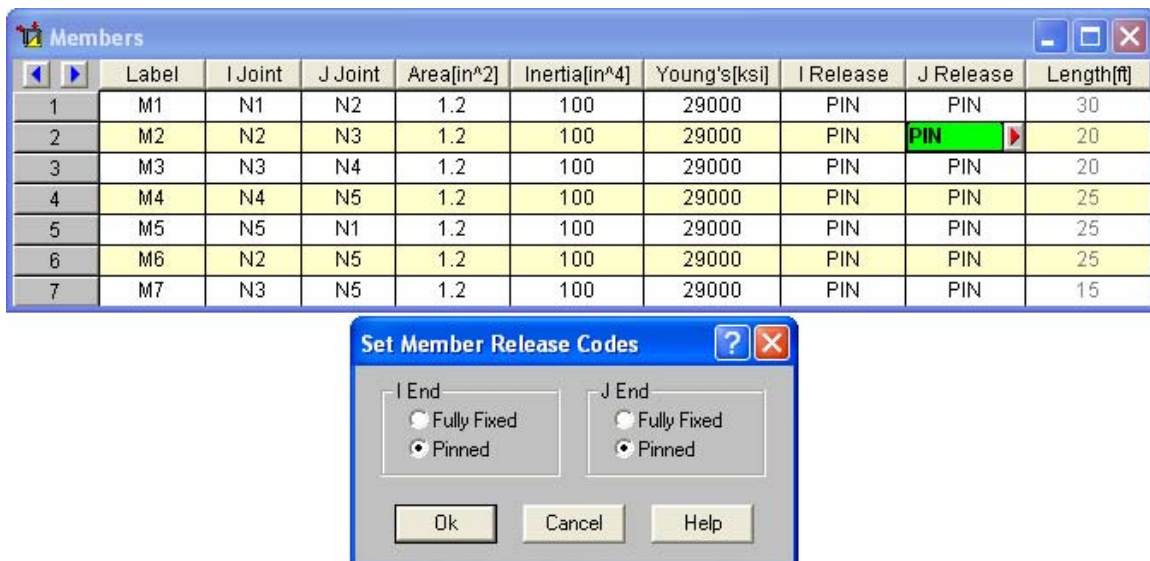


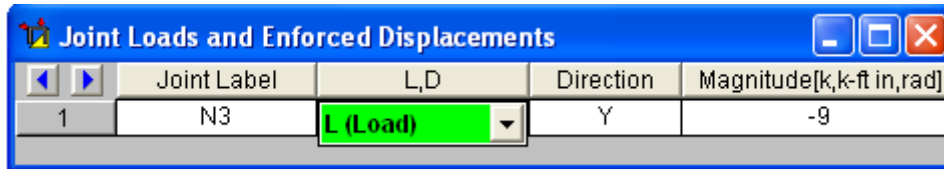
Figure 9

(6) To specify the loads that act on the structure, use the last three entries in the **Data Entry** toolbar, **Joint Loads**, **Point Loads**, and **Distributed Loads**. In the case of a truss, we often add an allowance to the joint load to account for the weight of the members framing into a joint. Since the truss is loaded at the joints only, click on **Joint Loads** to obtain the **Joint Loads and Enforced Displacement** Table (Figure 10).

- In the first column, insert the joint number (i.e., N3) where the load is applied.
- In the second column, specify **L** for load. [You can specify **D (Displacement)** in the second column for problems that involve support settlements.]

- In the **Direction** column, specify Y or X.
- In the last column, labeled **Magnitude**, specify the magnitude.

If the load acts down or to the left—the negative directions for the coordinate system—insert a *minus sign* before the load. If a joint is loaded by a force with both the x and y components, you will need two lines to specify the components. After the **Joint Load and Enforced Displacement** Table is finished, close it by clicking **X**. Click on **View** and select **Loads**, the loads will appear on the structure shown on the screen (Figure 11).



	Joint Label	L,D	Direction	Magnitude[k,k-ft in,rad]
1	N3	L (Load)	Y	-9

Figure 10

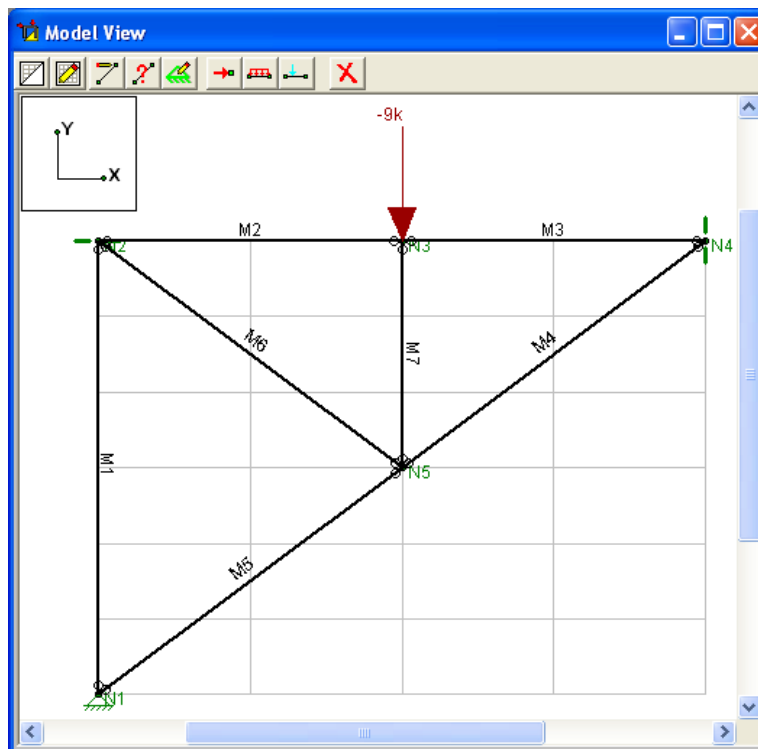





Figure 11

- (7) To analyze the truss, click on **Solve** (or the  icon) at the top of the screen in Figure 1. Results, including member forces, support reactions, and displacement of joints, are tabulated in tables and viewed by clicking on the labels of the **Results** toolbar (Figure 12). As was mentioned earlier, if the data supplied is incomplete or the structure is unstable, the program will issue a warning.

To plot the deflected shape, click on the  icon from the manual bar. Figure 14 shows an exaggerated sketch of the deflected shape in color. Plot the axial force along each member by clicking  on the manual bar (see Figure 15). Values of the member forces can be obtained by clicking **Member Forces** from the **Results** toolbar. As for the sign convention, the signs of these results correspond to the member's local axes, using the right hand rule. The left side forces at each section location are displayed. There are three force values for each section location. These are axial force, shear and moment. As can be seen in Figure 13, the section forces listed at any given section are the *left side* forces. For axial forces, compressive is positive. For moments, counter-clockwise around the member axis is positive. (Moment and shear are equal to zero for pin-connected truss members.)

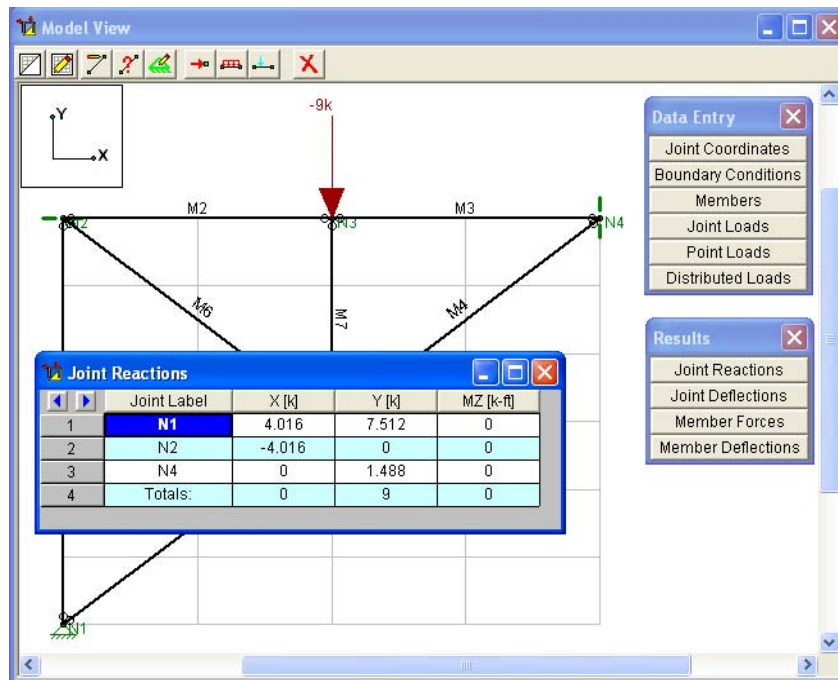


Figure 12

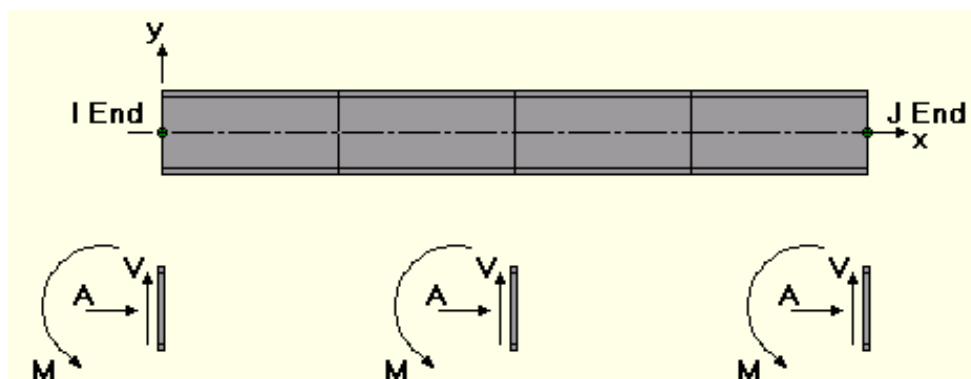


Figure 13

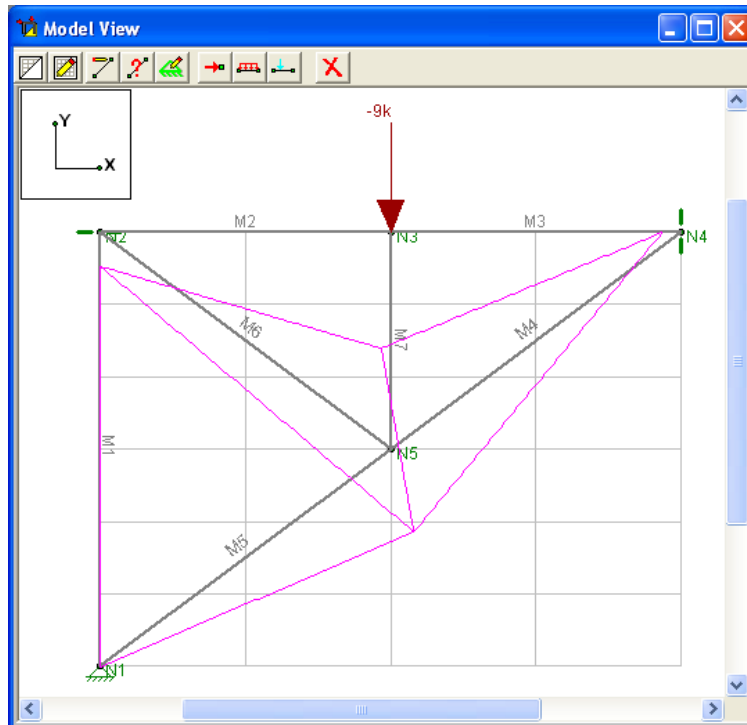


Figure 14

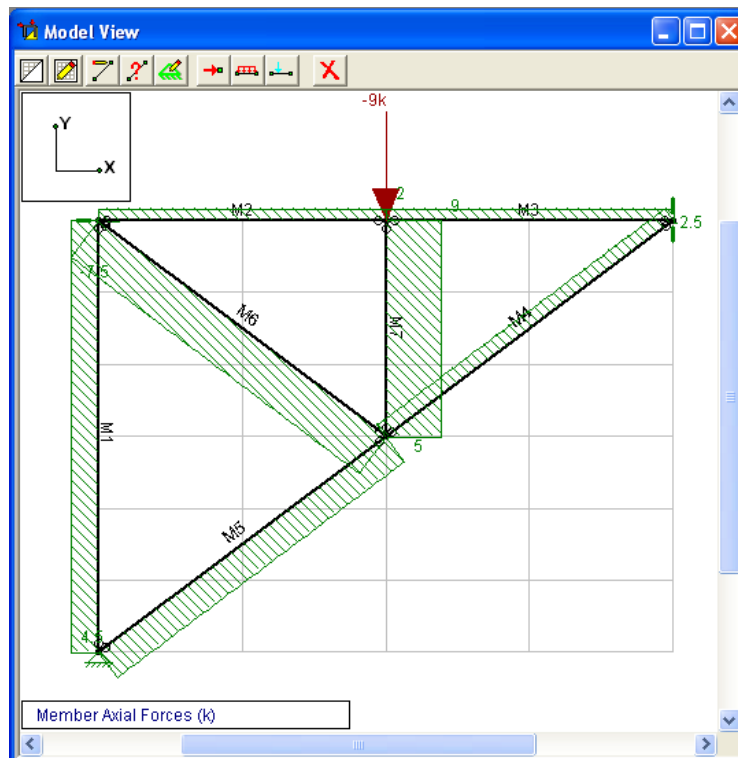







Figure 15

- (8) Note that creating the model and specifying loading as described in step (3) through step (6) can also be performed graphically. Figure 16 shows the icons that can be used for this purpose. For example, clicking on the third icon () allows you to specify both the joints and members. The support conditions can be specified by clicking on , and the loadings can be specified by clicking on either one of   .

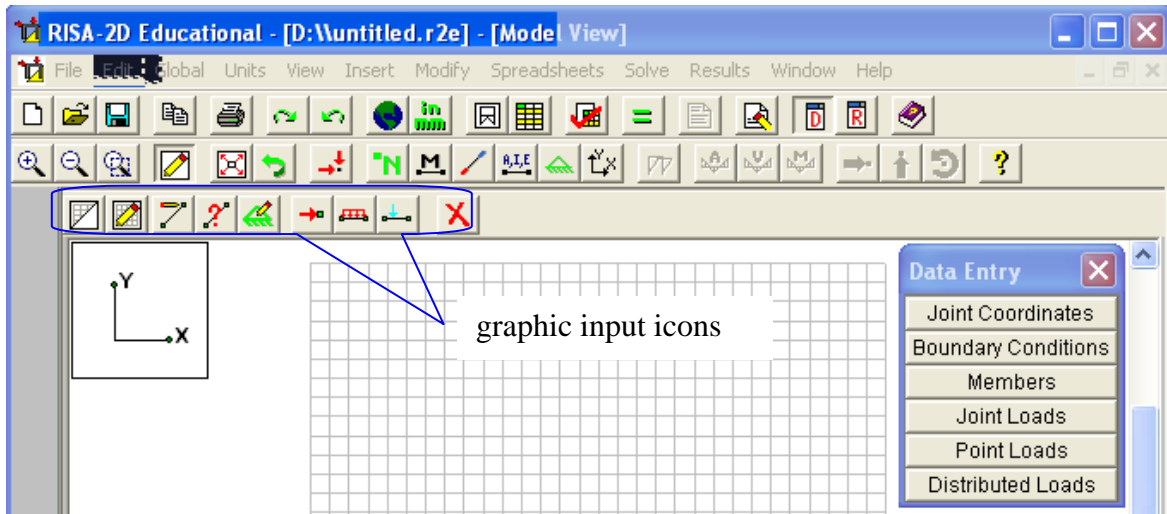


Figure 16

FRAME ANALYSIS

Consider the 2-dimensional frame in Figure 17. It is statically indeterminate to the sixth degree. Assuming that the value of I is 500 in^4 , the area of member AB is 15 in^2 , the area of the remaining members is 10 in^2 , and a Young's modulus of $29,000 \text{ ksi}$, the analysis is summarized below.

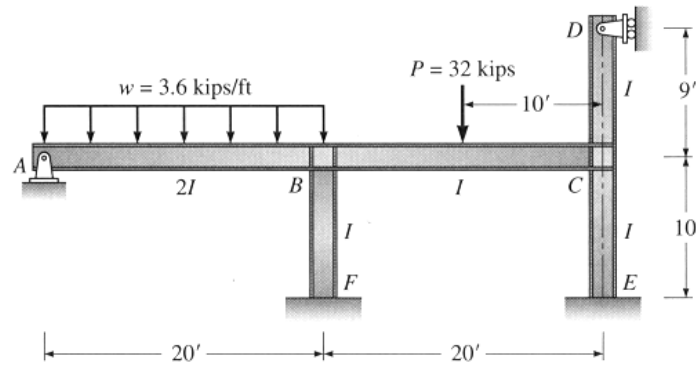


Figure 17

- (1) Follow steps (1) and (2) in the previous section to provide general information. Use the default value (= 5) for the **Number of Sections** so that internal forces at 5 equally spaced locations along each member will be reported. The frame is composed of 6 joints and 5 members. In step 5, change the default grid settings to $40@1 \text{ ft}$ and $19@1 \text{ ft}$ in the X and Y directions, respectively, such that all joints of the frame fall on the grid.
- (2) Follow step (3) to enter the joint coordinates (see Figure 18). Alternatively, you can follow step (8) to specify both joints and members graphically.

	Label	X [ft]	Y [ft]
1	A	0	10
2	B	20	10
3	C	40	10
4	D	40	19
5	E	40	0
6	F	20	0

Figure 18

- (3) Follow step (4) to provide information for the **Boundary Conditions**. Since joints E and F are fix-ended, set the boundary codes for all the directions (X, Y, and rotation) as **Fixed** (see Figure 19).

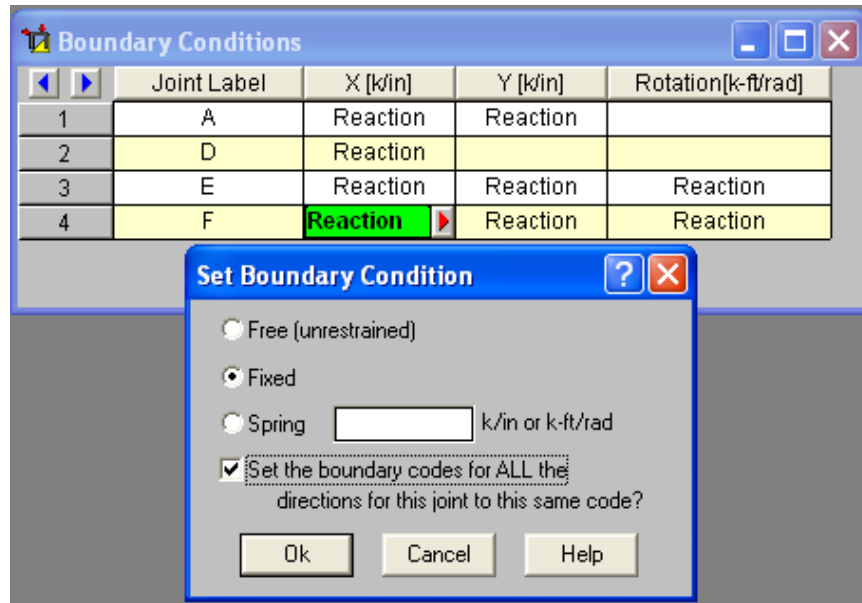


Figure 19

- (4) Click **Members** in the **Data Entry** toolbar to specify member data, which include the member labels, joint labels at both ends, area, moment of inertia, and Young's modulus (see Figure 20). Note that shearing deformation of the member is ignored in this educational version. If it is desired to ignore the axial deformation of the flexural member, you can specify a large value for the member area.

	Label	I Joint	J Joint	Area[in ²]	Inertia[in ⁴]	Young's[...]	I Release	J Release	Length[ft]
1	M1	A	B	15	1000	29000			20
2	M2	B	C	10	500	29000			20
3	M3	F	B	10	500	29000			10
4	M4	E	C	10	500	29000			10
5	M5	C	D	10	500	29000			9

Figure 20

- (5) Skip **Joint Loads** from the **Data Entry** toolbar because this example does not have joint loads. Instead, click **Point Loads** from the **Data Entry** toolbar to specify the 32-kip point load that acts on member BC (see Figure 21). Click **Distributed Loads** from the **Data Entry** toolbar to specify the uniformly distributed load that acts on member AB (see Figure 22). The data entry is now complete. Click **View** from the manual bar and select **Loads** to show graphically the applied loads (Figure 23).

Note that you can select the loading direction as **X**, **Y**, **x**, or **y** in the **Direction** field when specifying either the point load or the distributed load. Directions X and Y

refer to the global coordinate system (see Figure 1), while directions x and y refer to the local coordinate system of a member. As can be seen from Figure 24, the local x-axis corresponds to the member centerline. The positive direction of this local x-axis is from I joint towards J joint. The local z-axis is always normal to the plane of the model with positive z being towards you. The local y-axis is then defined by the right-hand rule. When a member is inclined, it is sometimes more convenient to specify the point load or transverse load in the local coordinate system.

Member Label	Direction	Magnitude[k,k-ft]	Location[ft]
1	Y	-32	10

Figure 21

Member Label	Direction	Start Magnitude[k/ft]	End Magnitude[k/ft]	Start Location[ft]	End Location[ft]
1	Y	-3.6	-3.6	0	20

Figure 22

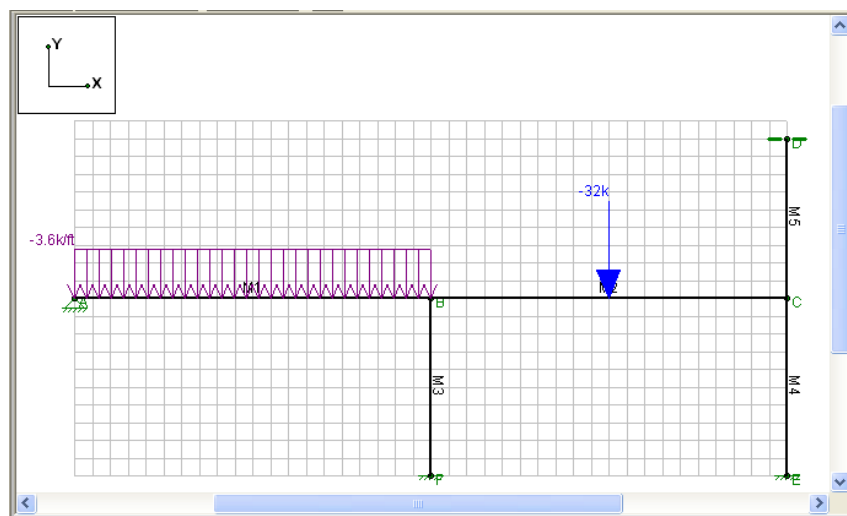


Figure 23

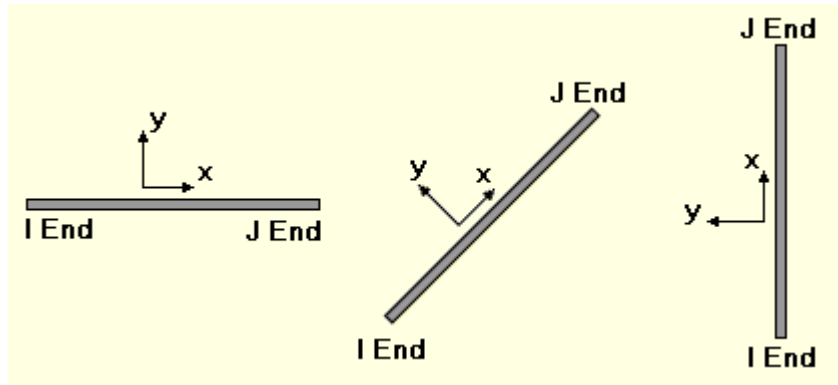


Figure 24

- (6) Now click **Solve** from the manual bar to perform the structural analysis. Click **Joint Reactions** from the **Results** toolbar to view the reaction forces (see Figure 25). Click **Joint Deflections** for the deflections and rotation at each joint (see Figure 26). Click **Member Deflections** if you are interested in the deflections of the members (see Figure 27). The member internal forces at equally spaced sections along each member can be viewed by clicking **Member Forces** (see Figure 28). The sign convention of the internal forces is defined in Figure 13.

Joint Reactions				
	Joint Label	X [k]	Y [k]	MZ [k-ft]
1	A	6.987	28.718	0
2	D	3.065	0	0
3	E	-4.213	13.52	13.237
4	F	-5.839	61.762	18.817
5	Totals:	0	104	32.053

Figure 25

Joint Deflections				
	Joint Label	X [in]	Y [in]	Rotation [rad]
1	A	0	0	-3.654e-3
2	B	-.004	-.026	1.031e-3
3	C	-.005	-.006	7.774e-4
4	D	0	-.006	-4.554e-4
5	E	0	0	0
6	F	0	0	0

Figure 26

Member Section Deflections				
	Member Label	Sec	x [in]	y [in]
1	M1	1	0	0
2		2	0	-.189
3		3	-.002	-.243
4		4	-.003	-.148
5		5	-.004	-.026
6	M2	1	-.004	-.026
7		2	-.004	-.076
8		3	-.004	-.167
9		4	-.005	-.103
10		5	-.005	-.006
11	M3	1	0	0
12		2	-.006	-.005
13		3	-.013	-.014
14		4	-.019	-.014
15		5	-.026	.004
16	M4	1	0	0
17		2	-.001	-.004
18		3	-.003	-.009
19		4	-.004	-.009
20		5	-.006	.005
21	M5	1	-.006	.005
22		2	-.006	.018
23		3	-.006	.019
24		4	-.006	.012
25		5	-.006	0

Figure 27

Member Section Forces					
	Member Label	Sec	Axial[k]	Shear[k]	Moment[k-ft]
1	M1	1	6.987	28.718	0
2		2	6.987	10.718	98.588
3		3	6.987	-7.282	107.177
4		4	6.987	-25.282	25.765
5		5	6.987	-43.282	-145.647
6	M2	1	1.148	18.48	-106.073
7		2	1.148	18.48	-13.674
8		3	1.148	-13.52	78.725
9		4	1.148	-13.52	11.123
10		5	1.148	-13.52	-56.478
11	M3	1	61.762	5.839	-18.817
12		2	61.762	5.839	-4.219
13		3	61.762	5.839	10.379
14		4	61.762	5.839	24.976
15		5	61.762	5.839	39.574
16	M4	1	13.52	4.213	-13.237
17		2	13.52	4.213	-2.704
18		3	13.52	4.213	7.828
19		4	13.52	4.213	18.36
20		5	13.52	4.213	28.892
21	M5	1	0	3.065	-27.586
22		2	0	3.065	-20.689
23		3	0	3.065	-13.793
24		4	0	3.065	-6.896
25		5	0	3.065	0

Figure 28

- (7) Analysis results can also be viewed graphically in the **Model View** window by clicking on the icons below the manual bar (see Figure 29). (If this window does not appear, click **View** from the manual bar and select **New View** to create one.) For example, Figure 30 shows the moment diagrams, reactions, and the deflected shape of the structure. Figure 31 depicts the reactions together with the applied loads.

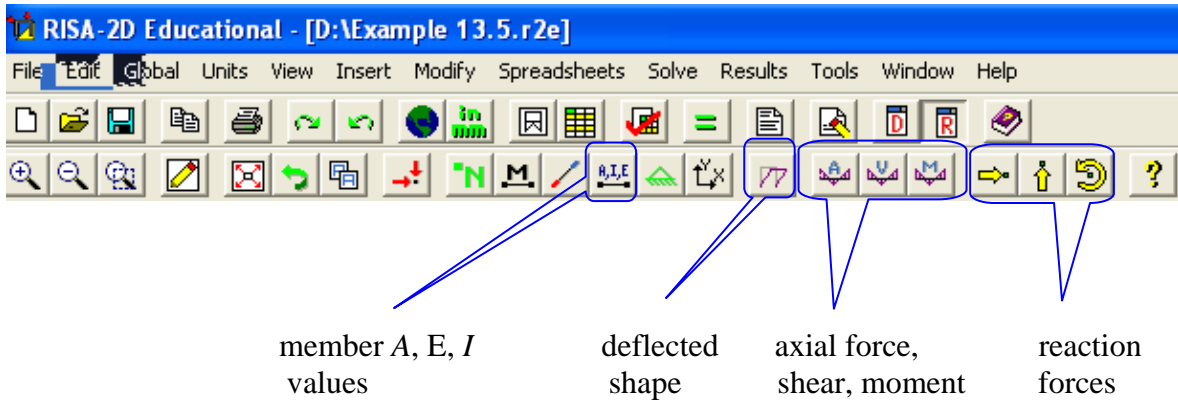


Figure 29

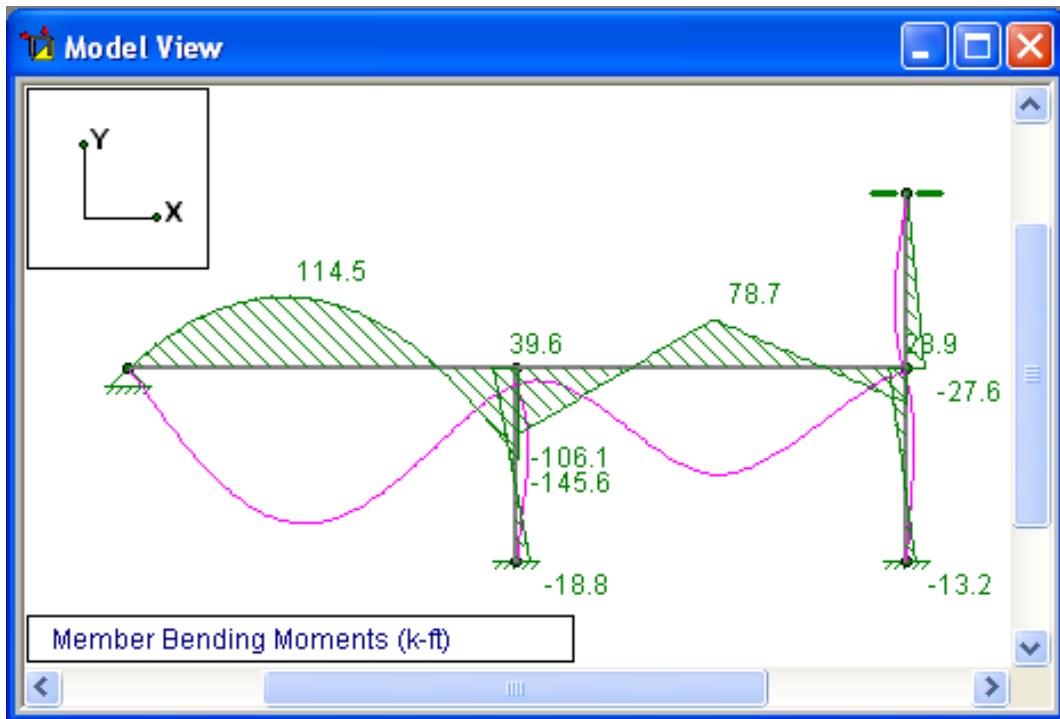


Figure 30

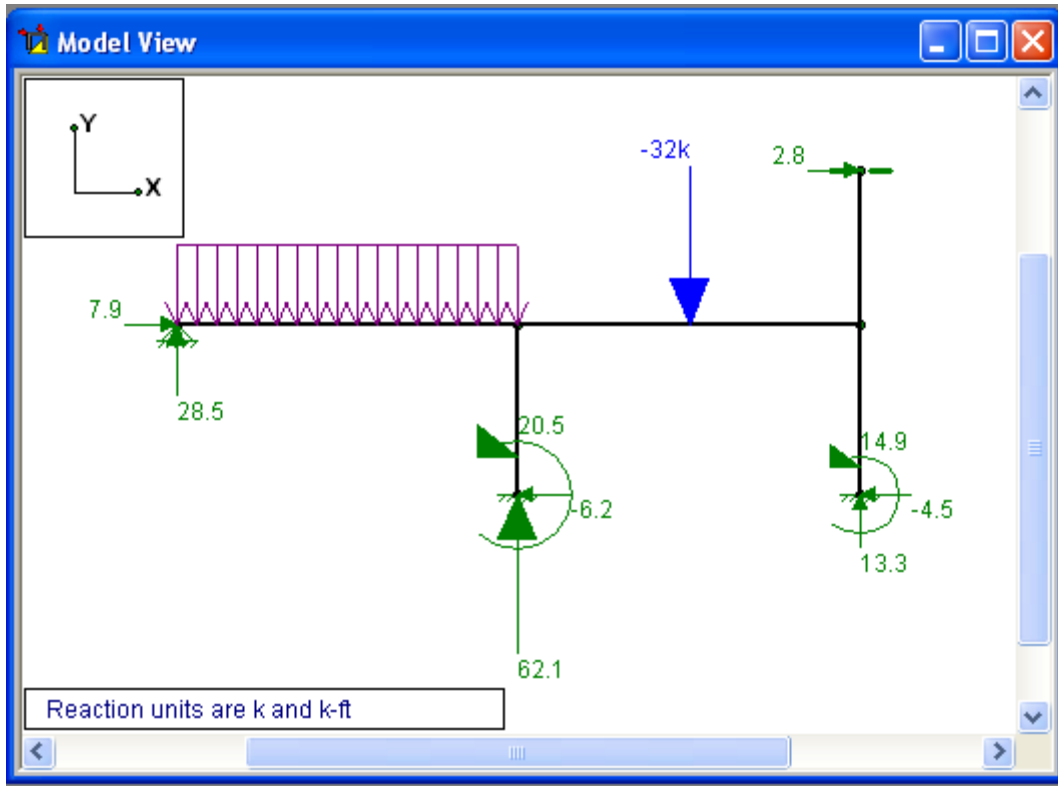


Figure 31