Lab-1

RISA 2D

1.1 Introduction

The educational version of the software RISA-2D, developed by RISA Technologies for the textbook *Fundamentals of Structural Analysis*, is an interactive computer program for the analysis of two-dimensional structures such as continuous beams, trusses, and frames. This program has been developed to make the definition, solution and modification of 2-dimensional problem data as fast and easy as possible. RISA-2D Educational has full graphical modeling capability allowing you to draw and edit your model on the screen. The analysis results can also be displayed graphically. A help file is also provided for a more detailed description of the program features. The numbers of joints and members are respectively limited to 50 in this educational version.

1.2 Principle of working

The program solution is based on the widely accepted linear elastic direct stiffness method. First, the stiffness of each element of the structure is calculated. These stiffnesses are then combined to produce the model's global structure stiffness matrix. Next, the global matrix is solved for the applied loads to calculate joint deflections that are then used to calculate the individual element forces and deflections.

1.3 Truss Analysis

The 2-dimensional truss to be analyzed is shown in Figure 1-1. It is indeterminate to the first degree. Assume the area of each member is 1.2 in^2 and the Young's modulus is 29,000 ksi. A step-by-step analysis procedure is provided below.

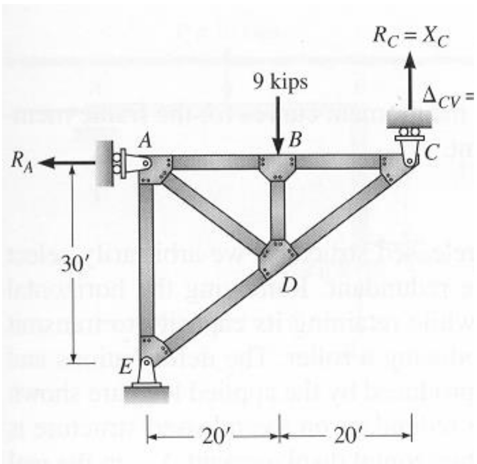
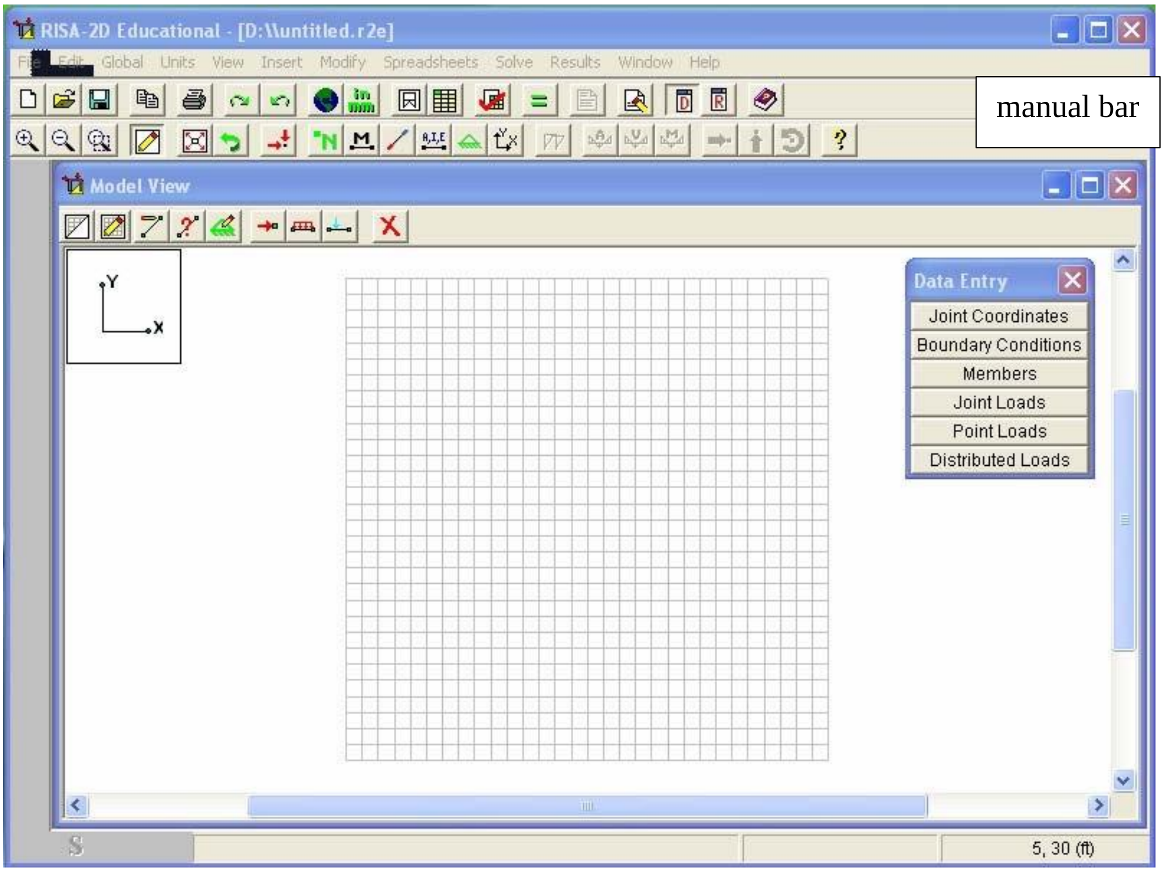


Figure 1-1: 2-Dimensional Truss Example

1. Start the RISA-2D Educational program. Figure 2 shows that a manual bar will appear at the top of the window. In addition to a **Data Entry** toolbar, a global XY coordinate system and a set of grid lines in the **Model View** window will also appear.



2.

Figure 1-2: Interface of RISA-2D

3. If you have created an input file previously, click **File** from the manual bar and select **Open** to open the input file. Otherwise, you can go to the next step to create a new model.
4. Click **Global** from the manual bar and enter the information for **Model Title** and **Designer** in the **Global Parameters** window (see Figure 3). The program can provide internal forces (moment, shear, axial force) at a number of equally spaced sections along a member. The default number of sections is 5, which is useful when you analyze continuous beams or frames. For truss analysis, however, the only internal member force is axial load, and the axial load is constant along a truss member. Set the **Number of Sections** to 2 so that the internal forces at both ends of the member will be provided. Click **OK** once you have completed the information.

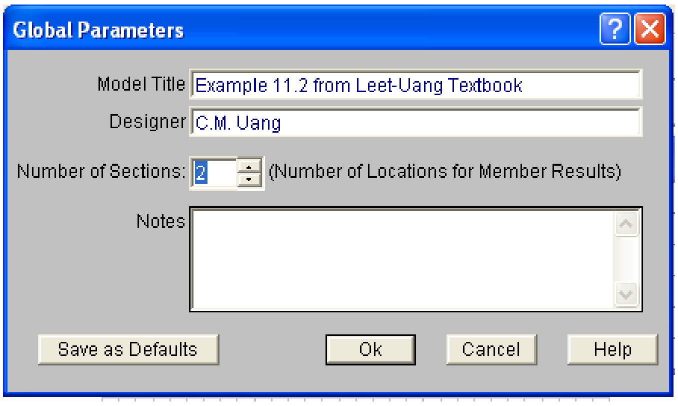


Figure 1-3: Global parameters

5. Click **Units** from the manual bar. One option you can choose in the **Units Selection** window is **Use CONSISTENT units**. This is the method we usually use for hand calculations. That is, all the physical quantities like length, section properties (*A* and *I*), material properties (*E*), loads, reactions, member forces, and deformations are expressed in terms of consistent units (e.g., kips and inches). For practical applications, the program provides a more convenient way of handling unit conversions internally by allowing the designer to choose either the **Standard Imperial** or **Standard Metric** units. We choose **Standard Imperial** in this example. Clock **OK** once you have selected the units.

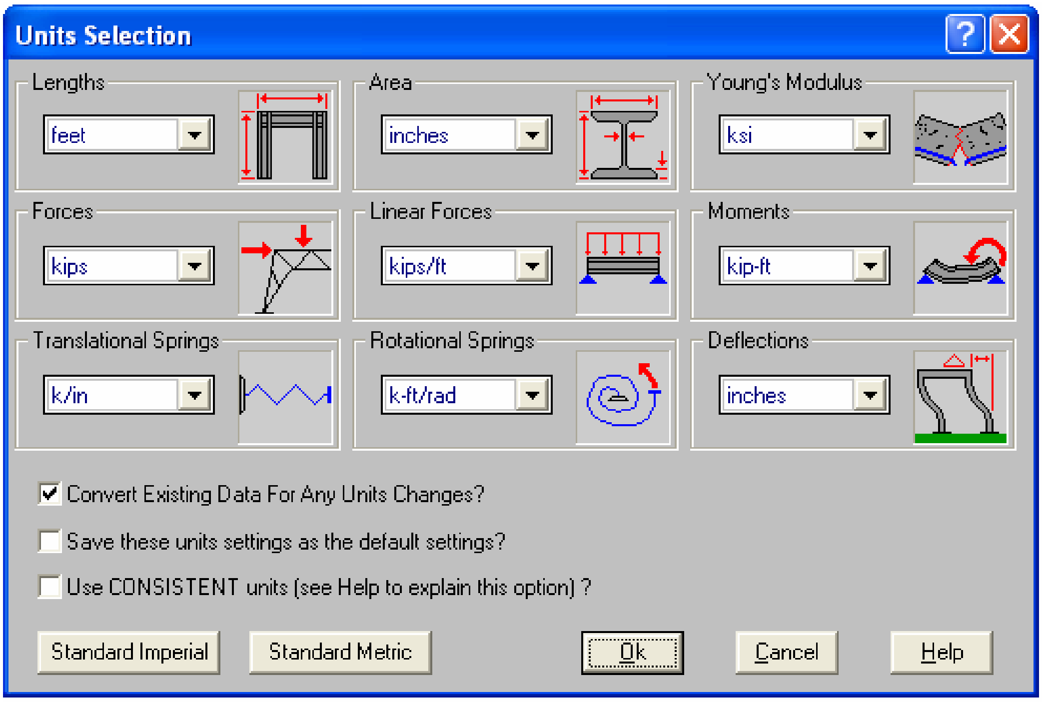


Figure 1-4: Unit Selection

6. Click **Modify** from the manual bar and select **Grid**. A **Define Drawing Grid** window will show that the program assigns, by default, (0, 0) as the coordinates for the origin (see Figure 5). Furthermore, the program assigns 30 grids with a unit length (1 ft) as the increment in each direction (see Figure 2). Considering the overall dimensions of the structure in Figure 1, 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. Save the **Grid Settings** and provide a description (e.g., Drawing Grid 1) for this drawing grid. If you open an existing file, it is necessary to **Retrieve** this grid settings in order to show the grids you previously defined. Click **OK** to complete this step. You will see the new drawing grid (see Figure 11).

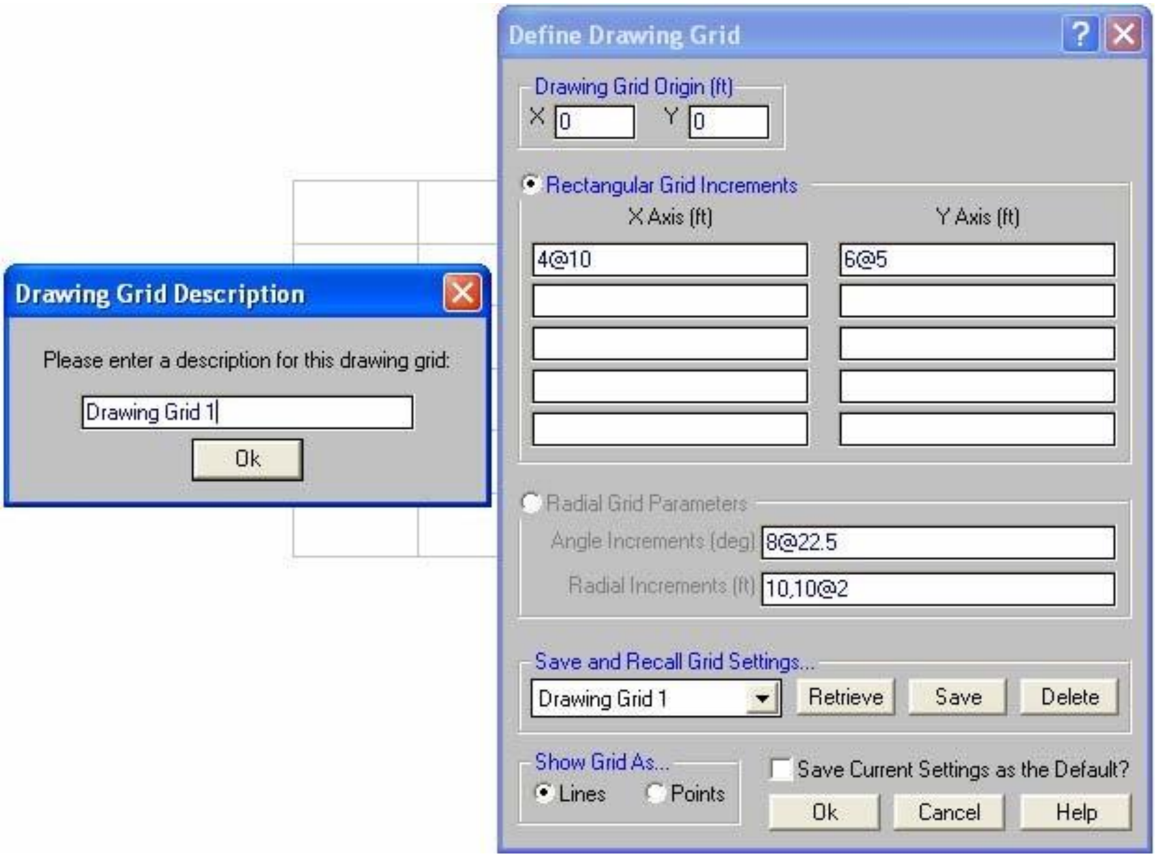


Figure 1-5: Grid System

7. The next step is to provide data for the structure. If the **Data Entry** toolbar in Figure 2 disappears for some reason, click **Spreadsheet** from the manual bar and select **Data Entry Buttons Toolbar** to activate it (see Figure 6).

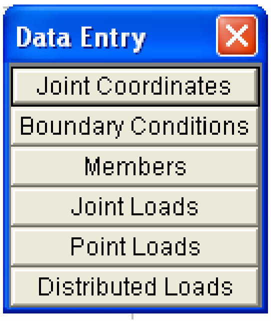


Figure 1-6: Data Entry Bar

8. Click **Joint Coordinates** from the **Data Entry** toolbar to define each joint and its coordinates. (Step 14 shows a more convenient way to specify joint coordinates graphically.) Follow the instruction in the **Joint Coordinates** window to define each joint (see Figure 7). 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). Click **Edit** from the manual bar or right click the mouse and select **Insert Line** to add additional joints. Upon completing the joint data, you can click **View** from the manual bar and select **Joint Labels** to check the joint input graphically (see Figure 11).

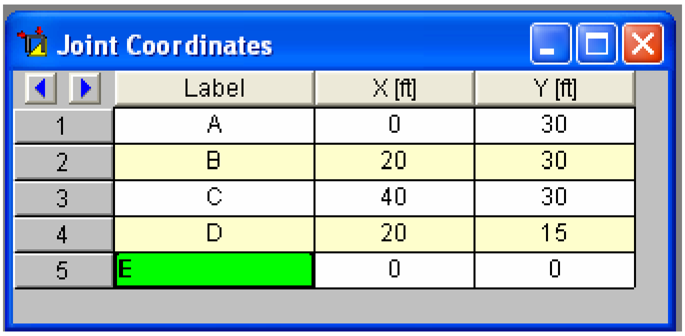


Figure 1-7: Joint Coordinates

9. Click **Boundary Conditions** from the **Data Entry** toolbar to specify the support condition. For this example, joint A is supported by a vertical roller. Click the field for X for a red arrow. Clicking on the arrow will allow you to define whether that direction is free to move, fixed, or supported by a spring. We specify joint A as **Fixed** because it cannot move in the horizontal (or X) direction. Click **Edit** from the manual bar and select **Insert Line** to add another two entries for the support condition for joints C and E (see Figure 8). Clicking **View** from the manual bar and selecting , the program will show

graphically the boundary condition of the structure (see Figure 11). A horizontal green line at joint A means that the joint cannot move in the horizontal direction.

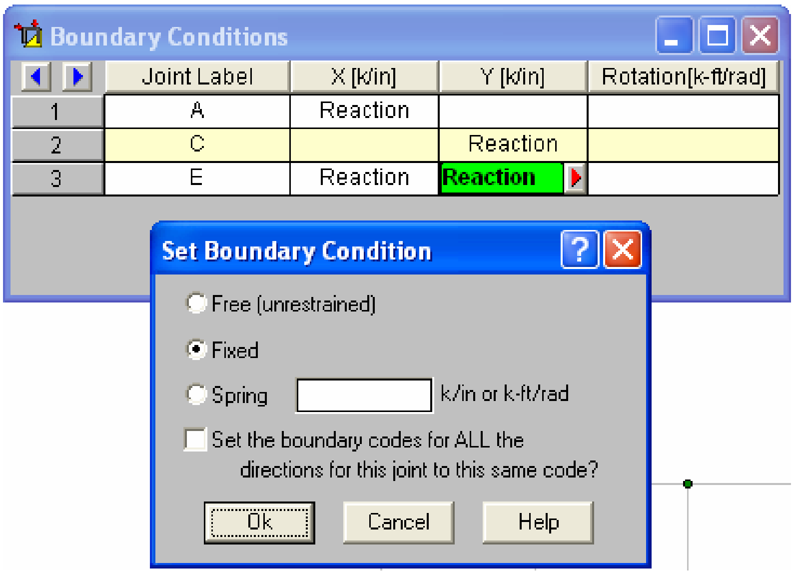


Figure 1-8: Boundary Conditions

10. Member information is provided in this step. A total of 7 truss members exist in the structure. Click **Members** in the **Data Entry** toolbar to specify member data, which include the member label, joint labels at both ends (I for near joint and J for far joint), area, and Young's modulus (see Figure 9). You can ignore the field of moment of inertia by using the default value because it is not needed for truss analysis. The length of each member will be computed by the computer program automatically. Since members in a truss are pin-connected at both ends, it is necessary to "release" the moment at both ends of the member (that is, zero moment at member ends). This can be achieved by clicking the field of **I Release** (and **J Release**). Clicking on the red arrow will then bring up the **Set Member Release Codes** window (see Figure 9), from which you can specify that both ends are **Pinned**. In the **Model View** window, the program will insert an open circle near the member end to indicate that moment has been released (see Figure 11). Also see Step 14 for graphic input of members.

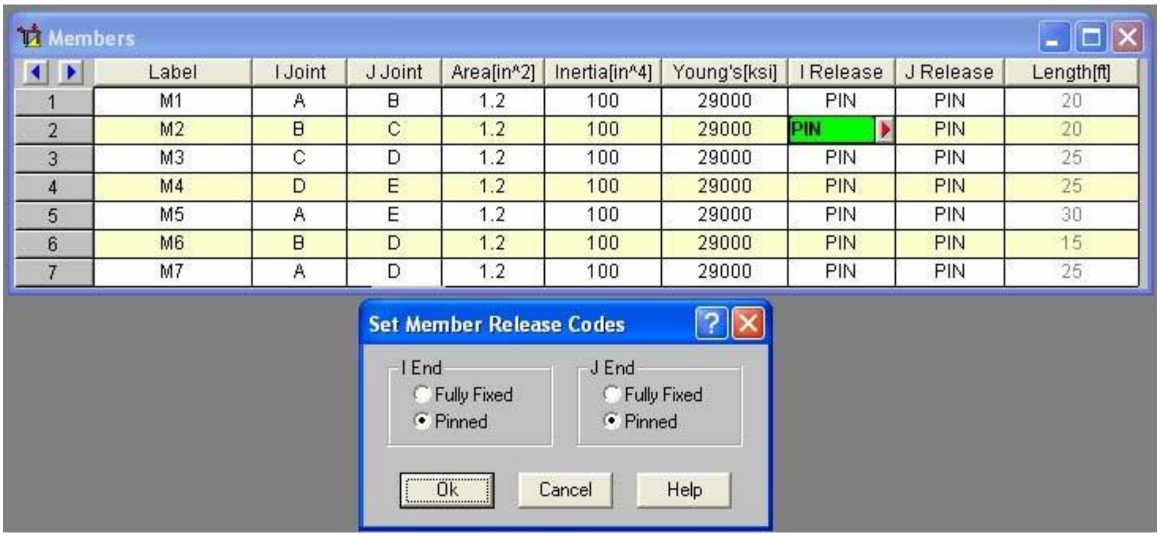


Figure 1-9: Members

11. Joint loads are specified in this step. Only a 9-kip vertical load is applied at joint B. Click **Joint Loads** from the **Data Entry** toolbar. Specify the joint label in the first column of the **Joint Loads and Enforced Displacements** spreadsheet (see Figure 10). Specify **L (Load)** in the second column. The direction of load, which is in the **Y** direction for vertical load, is specified in the third column. The magnitude of the vertical load is specified in the fourth column. Because the vertical load acts in the downward direction, which is in the negative Y direction, the magnitude of the joint load is –9. [You can specify **D (Displacement)** in the second column for problems that involve support settlements.] Also see Step 14 for graphic input of loads.

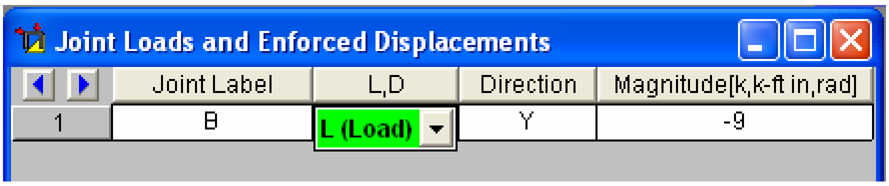


Figure 1-10: Joint Loads and Displacements

12. The last two entries (**Point Loads** and **Distributed Loads**) in the **Data Entry** toolbar are used to specify loads that act on a member. These two entries are not needed in this example because the truss, by definition, can only carry joint loads. The data entry is now complete. You can check the geometry, the boundary condition, as well as the labels of joints and members graphically by clicking **View** from the manual bar. From the **View** drop-down manual, you can select whatever information including the applied load for display (see Figure 11).

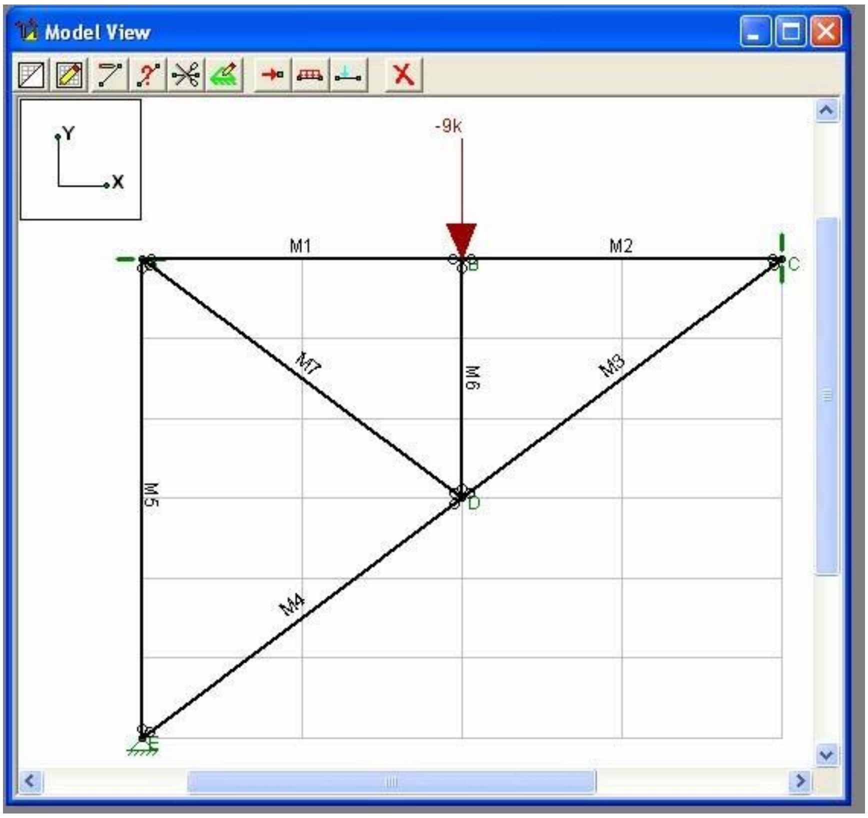


Figure 1-11: Model View

13. Now click **Solve** (or click the  icon) from the manual bar. The program will perform the structural analysis. A **Results** toolbar will appear if the analysis is successful (see Figure 12). (Clicking **Results** from the manual bar and select **Results Button Toolbar** can also activate this toolbar.) If the data entry is incomplete or the structure is unstable, the program will issue an error message. A **Joint Reactions** spreadsheet summarizing all the reaction forces will also appear in the window. The last row represents the summation of all reaction forces in the X and Y directions, respectively, which can be used to check global equilibrium. Figure 12 shows that the sum of horizontal reactions is equal to zero. In the vertical direction, the summation of the vertical reaction forces (9 kips) is also in equilibrium with the downward external load (-9 kips).

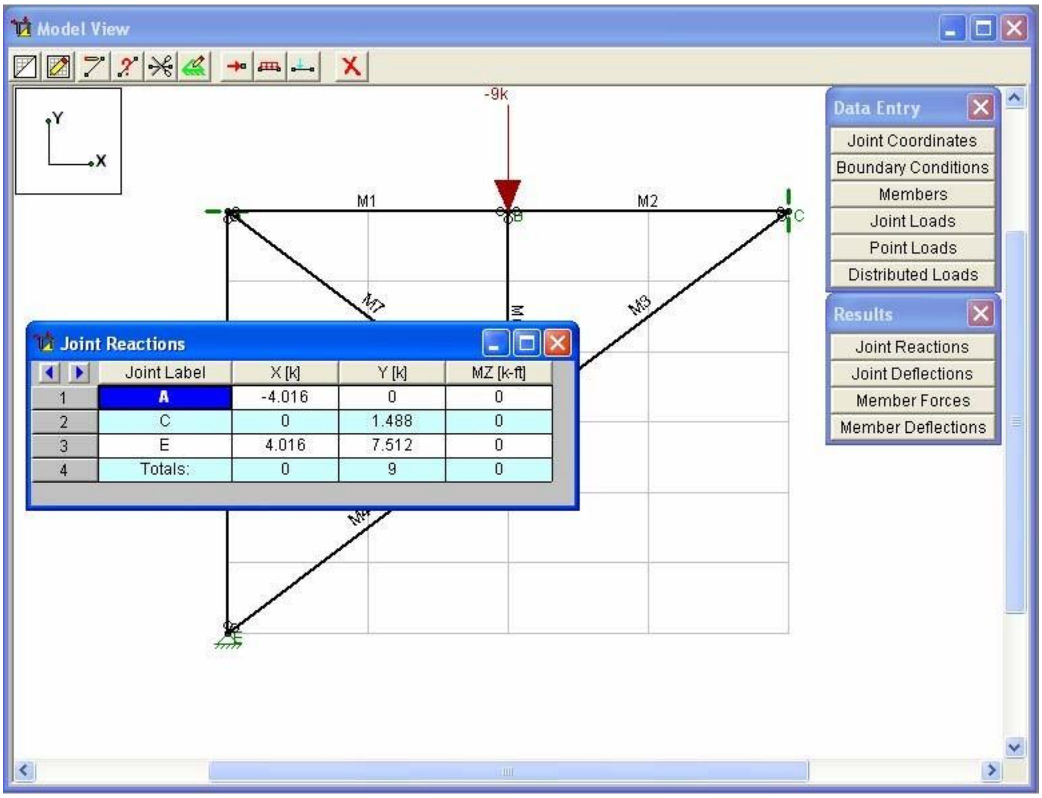


Figure 1-12: Model View and Results

14. The joint deflection information can be viewed by clicking **Joint Deflections** from the **Results** toolbar (see Figure 13). Clicking **Member Forces** from the **Results** toolbar gives a summary of internal forces in all members (see Figure 14). These are the member forces calculated along each member. The number of sections for which forces are reported is controlled by the **Number of Sections** specified in the **Global Parameter** window (see Figure 3). The number of member segments is the **Number of Sections** minus 1. The length of each segment is the same. For example, if you specify 5 sections, the member is divided into 4 equal pieces, and the forces are reported for each piece (see Figure 15). 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, shear and moment. As can be seen in Figure 15, 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.

Joint Deflections

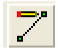
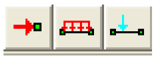
	Joint Label	X [in]	Y [in]	Rotation [rad]
1	A	0	-.006	0
2	B	-.002	-.019	0
3	C	-.003	0	0
4	D	.004	-.014	0
5	E	0	0	0

Figure 1-13: Joint Deflection

Member Section Forces

	Member Label	Sec	Axial[k]	Shear[k]	Moment[k-ft]
1	M1	1	1.984	0	0
2		2	1.984	0	0
3	M2	1	1.984	0	0
4		2	1.984	0	0
5	M3	1	-2.48	0	0
6		2	-2.48	0	0
7	M4	1	5.02	0	0
8		2	5.02	0	0
9	M5	1	4.5	0	0
10		2	4.5	0	0
11	M6	1	9	0	0
12		2	9	0	0
13	M7	1	-7.5	0	0
14		2	-7.5	0	0

Figure 1-14: Member Forces

15. Note that creating the model and specifying loading as described in step (6) through step (11) can also be performed graphically. Figure 16 shows the icons that can be used for this purpose. For example, clicking the third  icon ( ) allows you to specify both the joints and members. The support conditions can be specified by and the loadings can be  specified by clicking ( ) either one of 

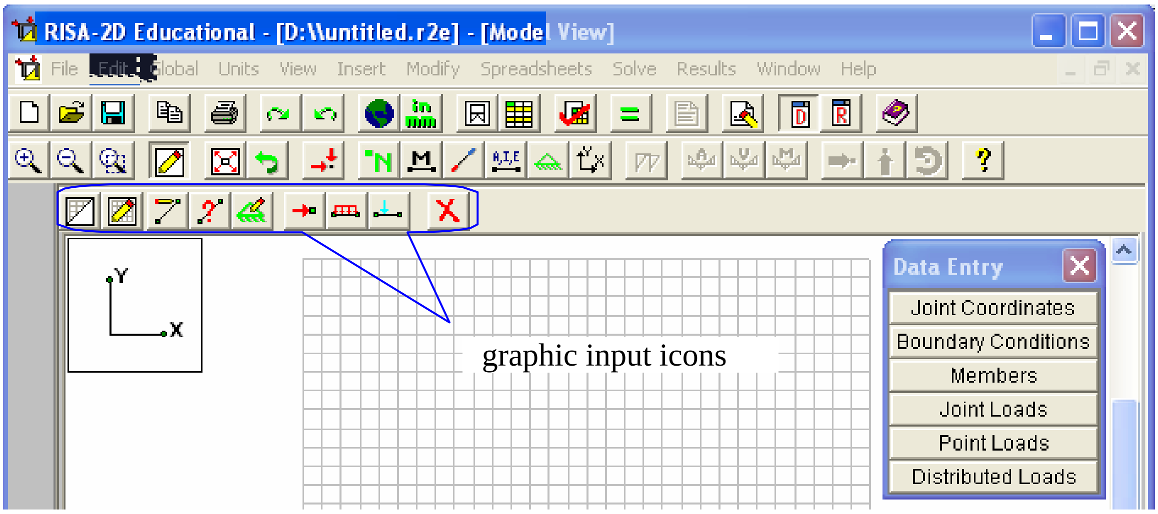
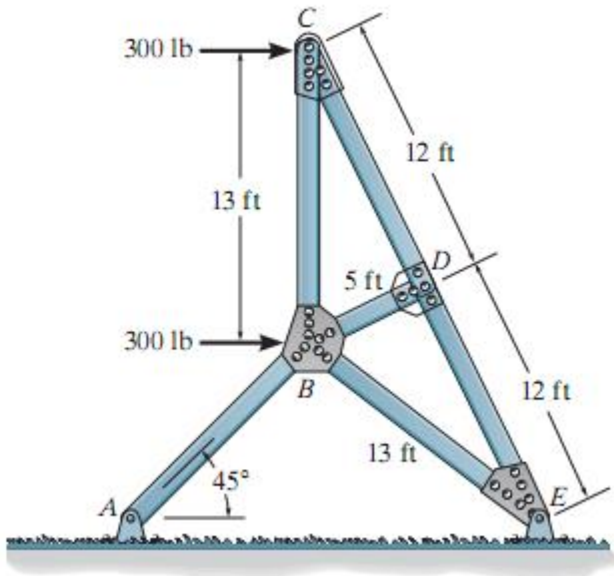


Figure 1-15: Graphic input Tool Bars

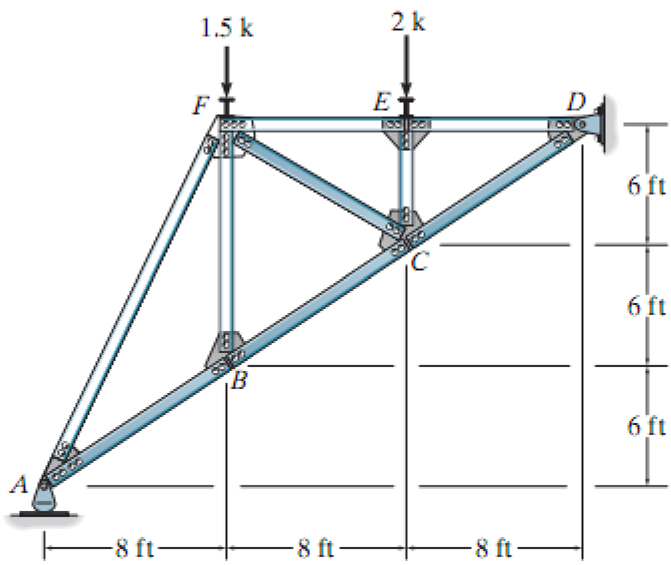
Lab -2

Develop the following Truss models in RISA 2D and Evaluate the member forces, Reactions and deflection.

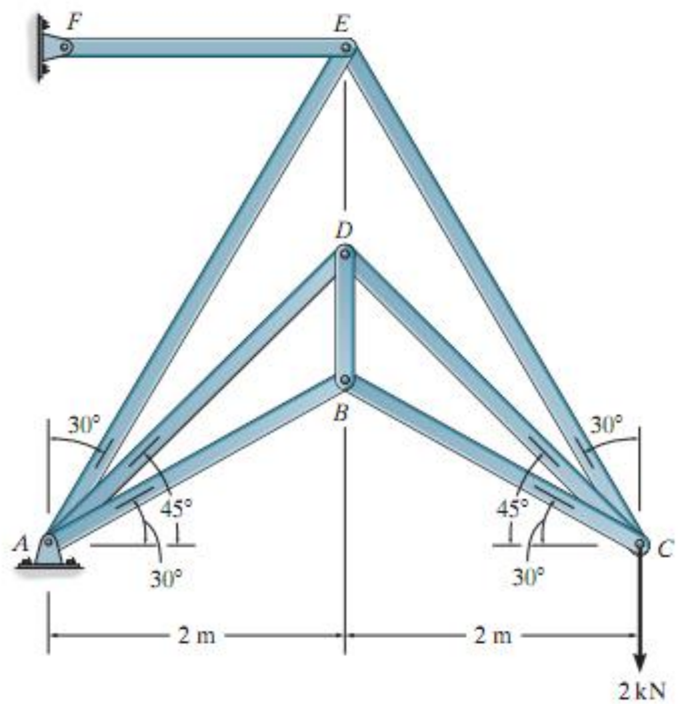
Task-1:



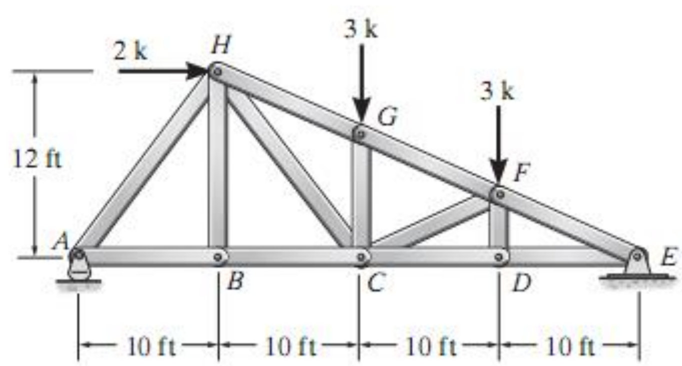
Task-2



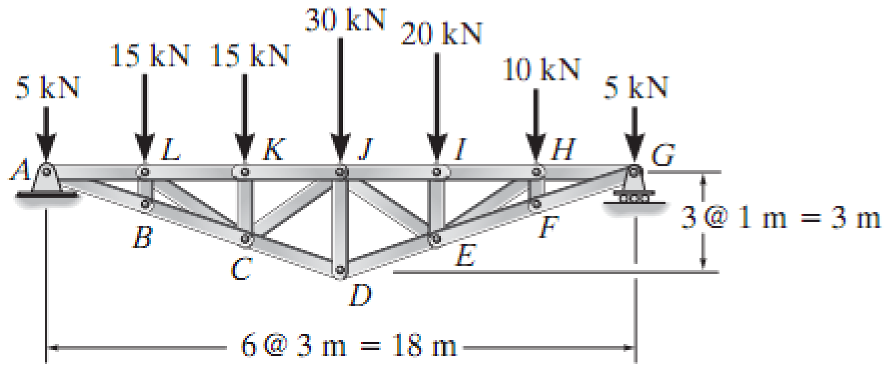
Task-3



Task-4



Task-5


30 kN
20 kN
15 kN
15 kN
10 kN
5 kN
5 kN
A
B
C
D
E
F
G
H
I
J
K
L
3 @ 1 m = 3 m
6 @ 3 m = 18 m