

WinTruss Tutorial

By Matt Sutton

WinTruss is a powerful, intuitive, and flexible truss analyzer. This tutorial is written to introduce you to many of the features available on WinTruss. The easiest way to learn a new software tool is to execute it and begin experimenting with its features. That is exactly what we are going to do. In order to experiment with the features included in WinTruss, we will solve Example 8.2 in the Statics textbook (Jong and Roger, 1991). You can find this problem on page 287. If you have not started WinTruss, do so now.

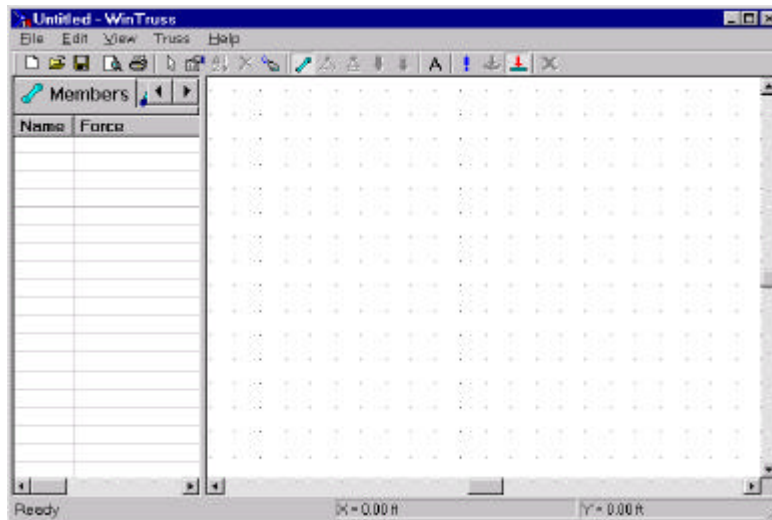


Fig. 1 Start-up window of WinTruss

The start-up window of WinTruss is shown in Fig. 1. You will find it is easy to analyze trusses with WinTruss after having familiarized yourself with the following five major steps.

Step 1. Setting up the Environment

The first step in solving any truss with WinTruss is to initialize the software environment. This primarily involves setting key parameters that will enable both you and the software to work together in solving a given truss problem. The physical units, drawing aids, and default properties are set in the Options dialog under Edit on the menu. To open this dialog, click on “Edit” and select “Options...” like the screen shown in Fig. 2. Over the next few pages we will examine these property sheets in more detail, and we will highlight the properties which are most commonly adjusted for different trusses.

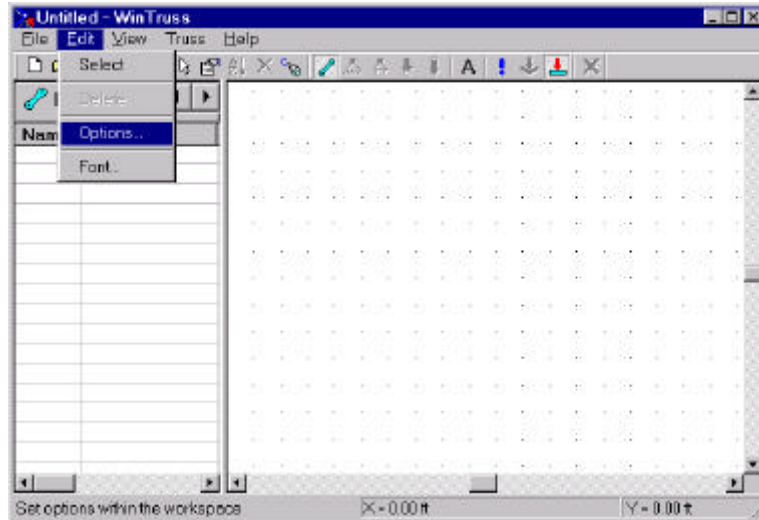


Fig. 2 Opening “Options” dialog box in “Edit” on the menu

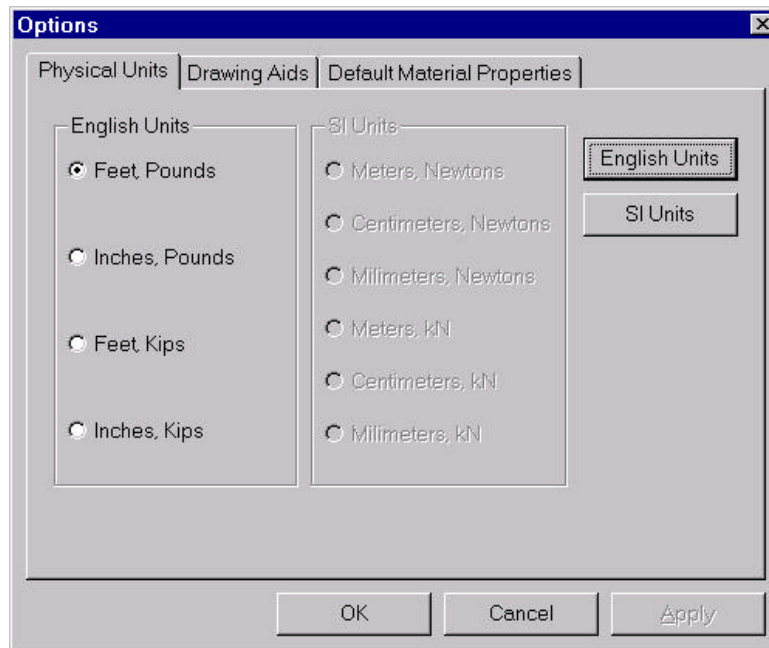


Fig. 3 Physical units in “Options” dialog

The first property sheet shown in Fig. 3 allows you to adjust the units in which the applied forces and truss dimensions will be specified. For solving the truss in Example 8.2 in the textbook, we need to change the default from “English Units” (i.e., the U.S. Customary System of Units) to the “SI Units.” To accomplish this, press the “SI Units” button. This will dim the English Units side and activate the SI Units side. Once the SI Units side is activated, click to select the “Meters, kN” option. Your dialog would look like that shown in Fig. 4.

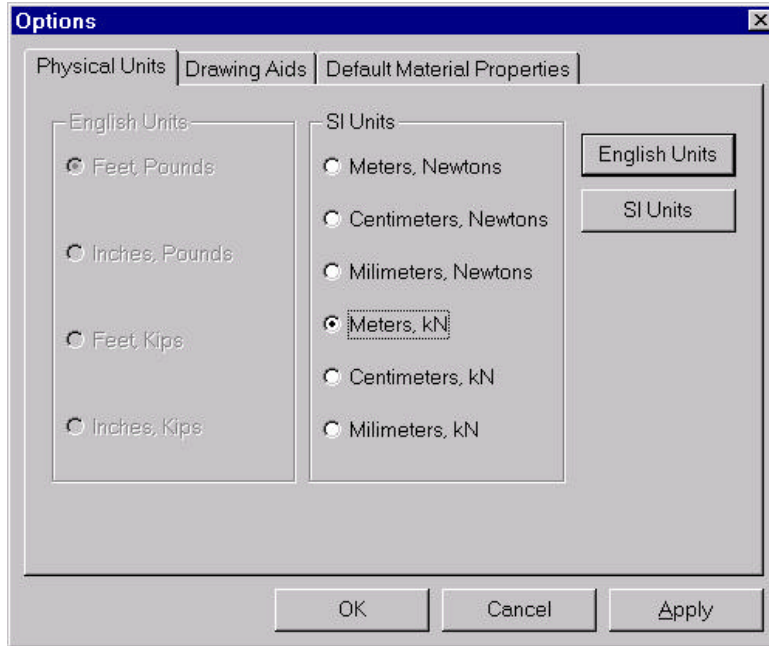


Fig. 4 Physical units with the “SI Units” side activated

The next property sheet is the “Drawing Aids” sheet as shown in Fig. 5. Here, you can specify how the schematic of a truss is scaled on the screen. You can specify attributes (e.g., grid resolution and grid scale) that will enable the truss schematic to be drawn to scale and still fit on the screen.

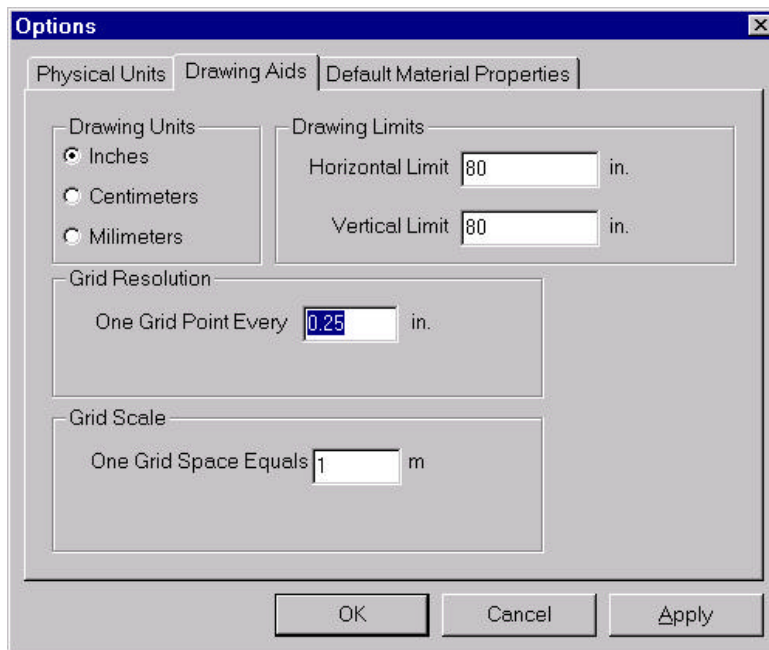


Fig. 5 Specifying “Drawing Aids” in the “Options” dialog

WinTruss allows you to draw a truss on a virtual sheet of graph paper. The “Drawing Units” grouping allows you to specify the units in which the size of your piece of paper is measured. Furthermore, the “Drawing Limits” grouping allows you to specify the actual dimensions of your piece of paper. By default, WinTruss creates for you a virtual sheet of paper which is 80 in. by 80 in. This should be more than enough room for you to sketch most of your trusses. Because of the size of the virtual sheet, scroll bars are provided on the graphics pane. You have a space of 80 in. by 80 in. to draw, and that amount of space is far more than what a usual monitor can display at any one time.

The next grouping is “Grid Resolution.” Since WinTruss allows drawing to occur on a virtual sheet of graph paper, we may adjust the coarseness of the grid spacing in order to maximize the use of the available screen space. As a default, WinTruss chooses to space the grid on 0.20-in. intervals. This is generally acceptable for most problems. However, you may choose to reduce the grid spacing in order to gain finer control, or increase the grid spacing in order to use more screen space.

The “Grid Scale” attribute allows you to specify the conversion factor from grid spacing to physical dimensions. The dimensions of the truss in Example 8.2 of the textbook are given in meters. We previously specified them on the last property sheet when we selected the “Meters, kN” option. The “Grid Scale” attribute allows us to specify how many meters one grid space equals. As you can see, this value by default is one; but it can be changed to any real number. For this example problem, let’s change this value to 0.5 as shown in Fig. 6.

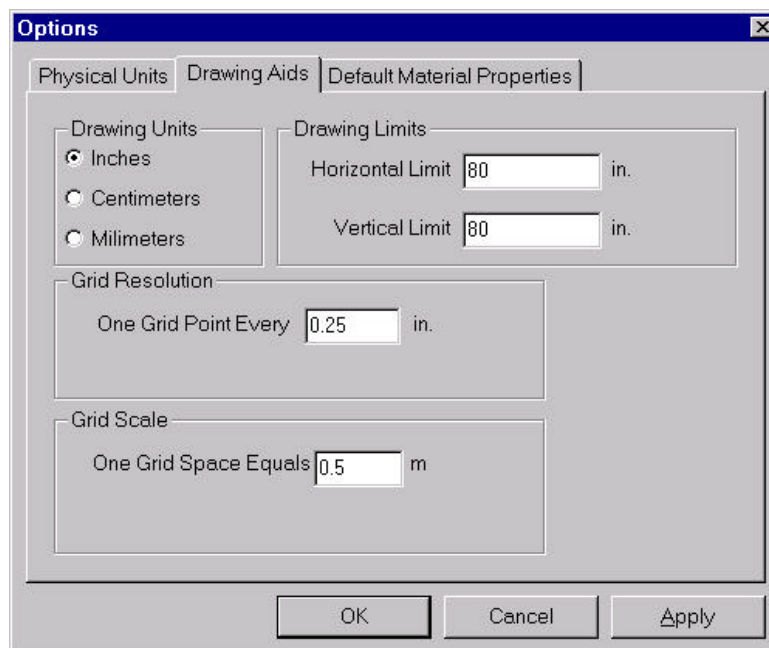


Fig. 6 Adjusting “Grid Scale” in the “Options” dialog

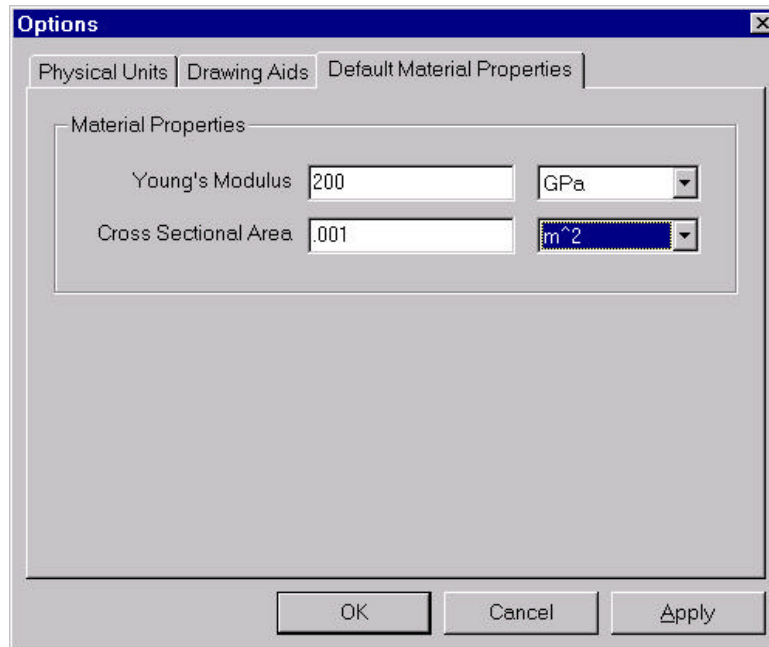


Fig. 7 Adjusting the “Default Material Properties” in the “Options” dialog

WinTruss is capable of solving a statically indeterminate truss as well as providing a view of the deflected configuration of the truss analyzed, beyond the scope of Statics. Thus, we may click on the “Default Material Properties” tab to adjust the values. Here, we will simply change the units for Young’s modulus and the cross-sectional area to appropriate SI units. Let’s set Young’s Modulus to 200 GPa and the cross-sectional area to 0.001m^2 as shown in Fig. 7. When you are finished, press the “OK” button to accept the changes and close the dialog. The screen should go back to appear as shown in Fig. 1.

Step 2. Sketching the Truss

Now that we have set up the environment, we will begin to sketch out the schematic of the truss. WinTruss has several features that aid in sketching a truss schematic. First is the *Grid/Snap* function that implements *dimension-as-you-go* technology. Because we have specified the grid spacing and grid scale, WinTruss will use the grid to dimension the schematic for us as we sketch it. You should see that the toolbar button representing a truss member is already depressed. If for some reason it is not, depress it now.

The first member *A-B* of the truss may start in the lower left-hand corner of the graphics pane and proceed up and to the right. In sketching it, you press and hold down the left mouse button at the origin as the beginning joint (Joint *A*). Continue to hold the left mouse button down as you drag the truss member up and to the right. Drag the mouse out until the status bar panes read $X = 3.00\text{ m}$ and $Y = 4.00\text{ m}$, which are the coordinates of the ending joint (Joint *B*). Once the ending joint is reached, release the mouse button.

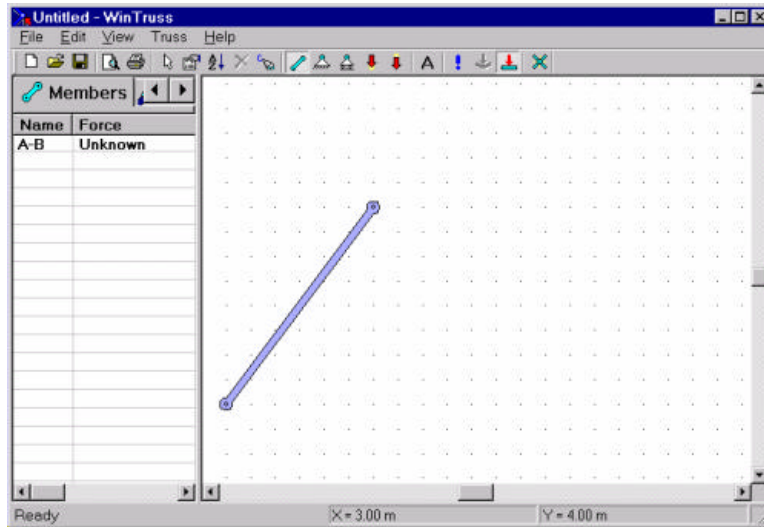


Fig. 8 Sketching the first member of the truss

The window with the first member sketched should resemble the picture shown in Fig. 8. Next, place the mouse cursor over the ending joint (Joint *B*) of the first member on the screen. Press and hold down the left mouse button on the ending joint of the first member, and drag the mouse out until the status panes read $X = 6.00$ m and $Y = 4.00$ m, which are the coordinates of the ending joint of a second member. Repeat this process for a third member. The status panes should read $X = 9.00$ m and $Y = 0.00$ m when you are finished, and the screen should resemble the one shown in Fig. 9.

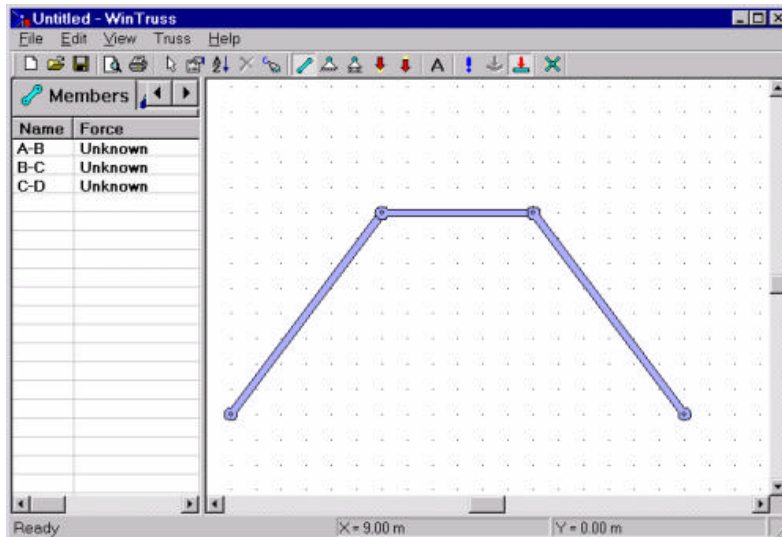


Fig. 9 Three members of the truss sketched on WinTruss

Next, draw a fourth member from the ending joint of the preceding third member all the way to the far left joint (Joint *A*). Your window should look like the one shown in Fig. 10.

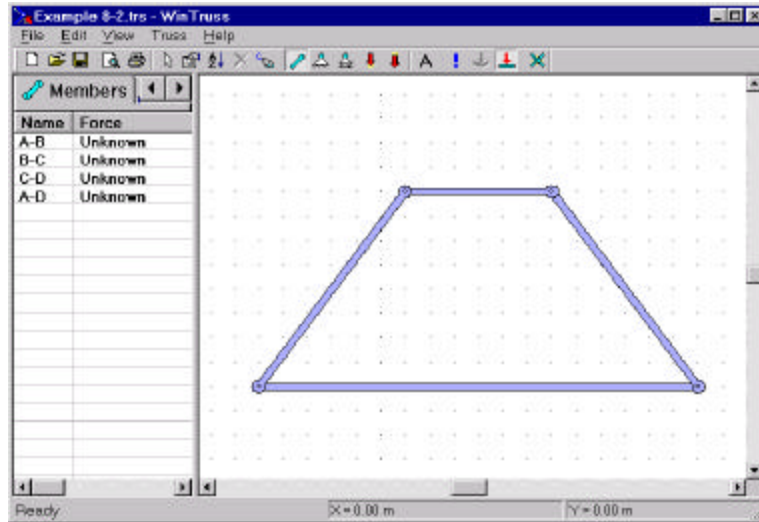


Fig. 10 Four members of the truss sketched on WinTruss

We still need to sketch in the vertical members and the central cross member. To do so, start by moving the mouse pointer over the top left joint (Joint *B*). Press and hold down the left mouse button and drag the new member vertically down until the mouse pointer rests on top of the horizontal bottom member. Upon release of the mouse button, WinTruss will automatically create a joint at that spot for you. Repeat this process for the right vertical member, and then sketch in the central cross member using the technique mentioned above. Your window should now resemble the one shown in Fig. 11.

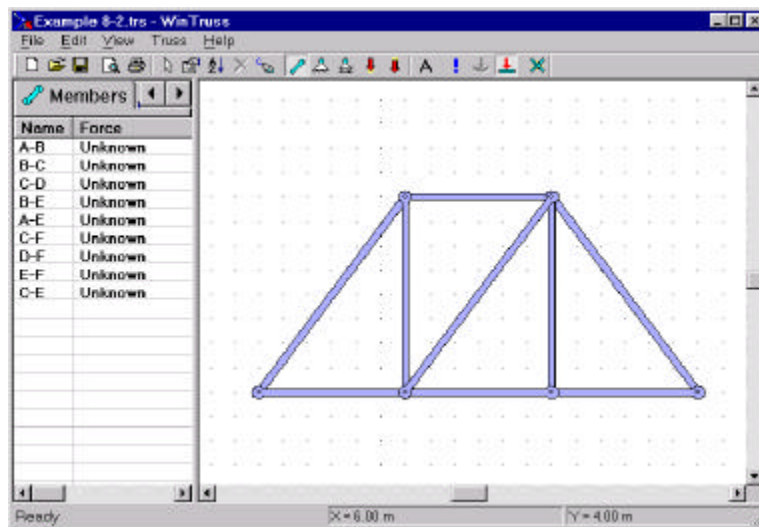


Fig. 11 Complete sketch of all members of the truss on WinTruss

To view the labels that have been automatically assigned for the joints by WinTruss, press the “Label” button, which has the letter “A” on it, on the toolbar. Depending on the order in which you sketched the truss schematic, the labels generated automatically, as shown in

Fig. 12, by WinTruss may not match the labels given in the original problem. Nevertheless, we can change the labels later.

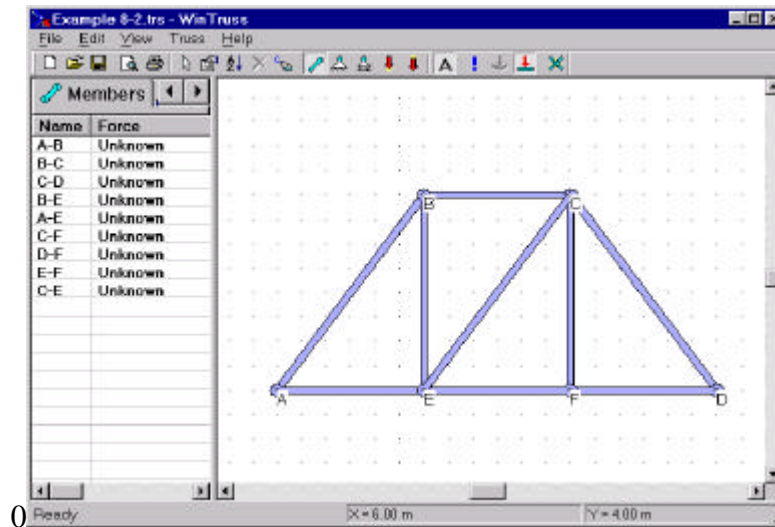


Fig. 12 Labels of the joints initially generated by WinTruss

The positions of the labels generated by WinTruss are always slightly below and to the right of the centers of the joints. Let's first move the joint labels to better locations. To do so, press the "Select" button on the toolbar, where the "Select" tool looks like an arrow pointer on the toolbar. Once we are in "Select" mode, we can left-click on the center of a joint label and, while holding down the left mouse button, drag each label individually to a new location, as shown in Fig. 13. Fonts may be changed under "Edit" on the menu bar.

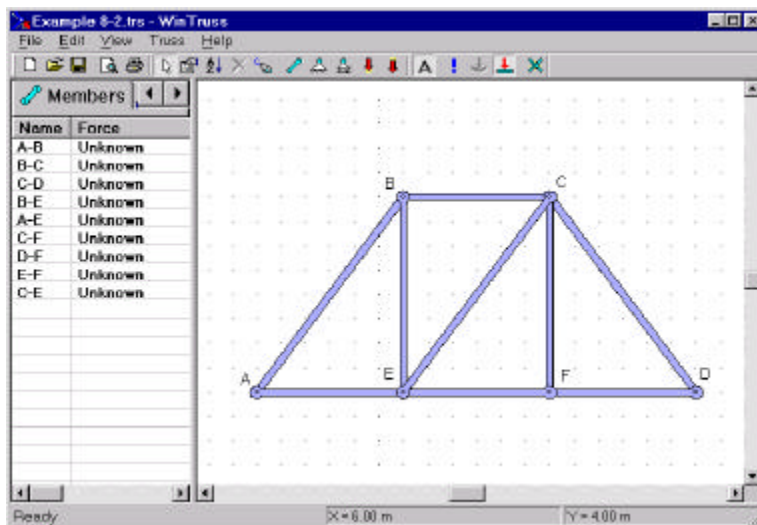


Fig. 13 Labels of joints dragged to better positions

As mentioned earlier, the default joint labels created by WinTruss may not match the labels given to the joints in the problem. Let's change them so that they match. To do so, move the mouse pointer over the joint whose label you wish to change, then right click on the joint. This action will bring up a *context menu* for the joint. Select the "Properties" item from the context menu. In our example, we need to change the joint currently labeled "C" to "D". So, position the mouse over the joint currently labeled "C" and right click to bring up the context menu. Select "Properties" from the menu. You should see a dialog box that looks like the one shown in Fig. 14.

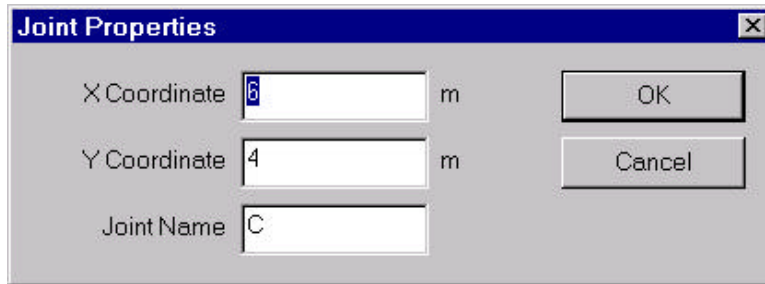


Fig. 14 Dialog box for "Joint Properties" under context menu

In the "Joint Properties", tab down until the "Joint Name" field is highlighted; then change the name from "C" to "D". Press OK to accept the change. Similarly, change the current labels "E", "F", and "D" for the joints along the bottom of the truss to "C", "E", and "F", respectively. Your window should now resemble the one shown in Fig. 15.

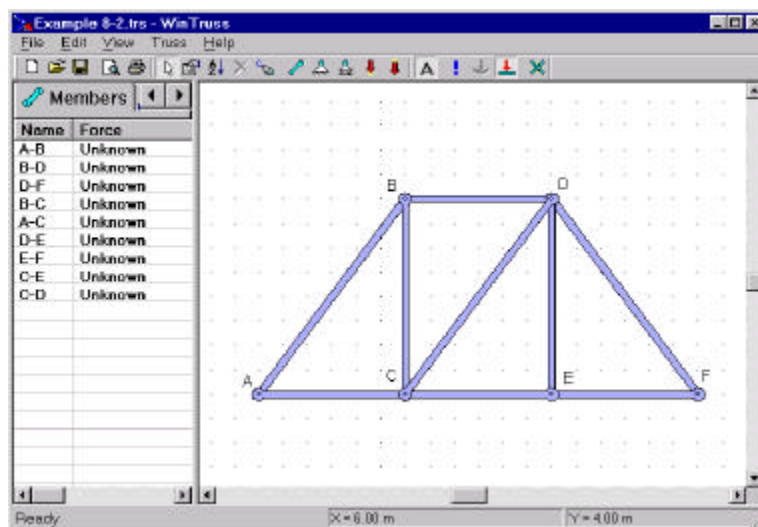


Fig. 15 Labels of joints changed to match those in the given problem

The “Sort” button for sorting the names of the truss members has the letters “A” above the letter “Z” and a downward arrow on the right side. The members in the output pane can be sorted alphabetically by clicking on the “Sort” button on the tool bar. Upon clicking on this “Sort” button, we set the list of truss members on the output pane in ascending order as shown in Fig. 16.

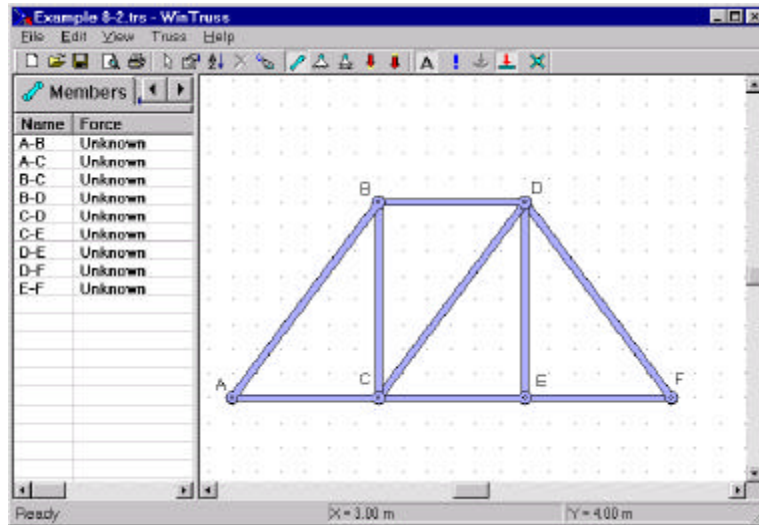


Fig. 16 List of truss members sorted alphabetically on WinTruss

Step 3. Adding the Supports

Now that the members of the truss have been sketched, you may proceed to add the supports to the truss. We need to add a hinge support to joint “A” and an inclined roller support to joint “F”.

Let’s add the hinge support first. Move the mouse to select the “Hinge Support” tool from the toolbar. Then move the cursor over the joint “A”. A box will surround the joint indicating that you may add a hinge support to the joint. Press the left mouse button down and hold it down while the cursor is over the joint “A”. Drag the mouse downward and from side to side until you have the support foundation perfectly horizontal. When you have it where you want it, release the mouse button. If you happen to release the mouse button while the hinge support is at an angle, it will not affect the validity of the solution.

If you don’t like how the hinge support looks though, you can delete the support and add a new one. To delete the hinge support (or other truss item), you need to use the “Select” tool from the toolbar to first click on the hinge support to select it. Then, press the “Delete” tool (a red X button on the toolbar) to delete the selected item. Once you have the *hinge support* drawn successfully, your window should resemble the one shown in Fig. 17.

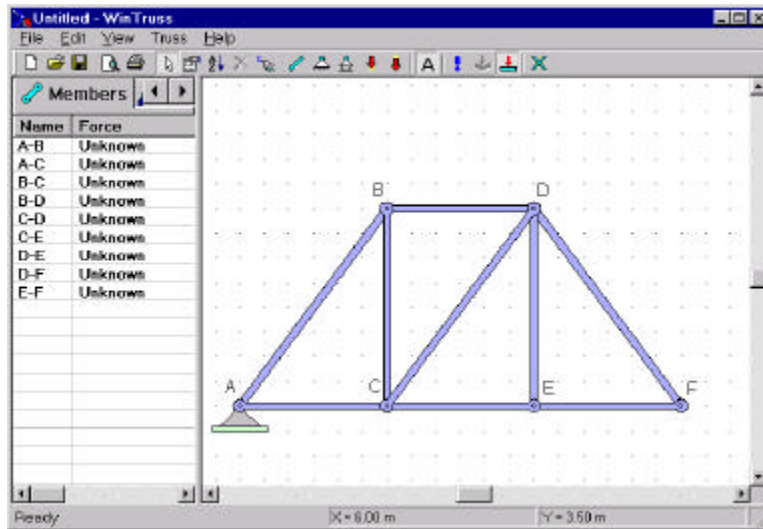


Fig. 17 Addition of a hinge support to the truss

Next, you proceed to add the roller support resting on an inclined surface. You choose the “Roller Support” tool from the toolbar. To add a roller support to joint “F”, repeat the same process you used for adding the hinge support to joint “A”. Try to draw the roller support at an angle similar to the schematic shown in Example 8.2 in the Statics textbook, but don’t be overly concerned with the precise orientation of the inclined surface now. Once you have drawn the *roller support* at joint “F”, your window should resemble the one shown in Fig. 18.

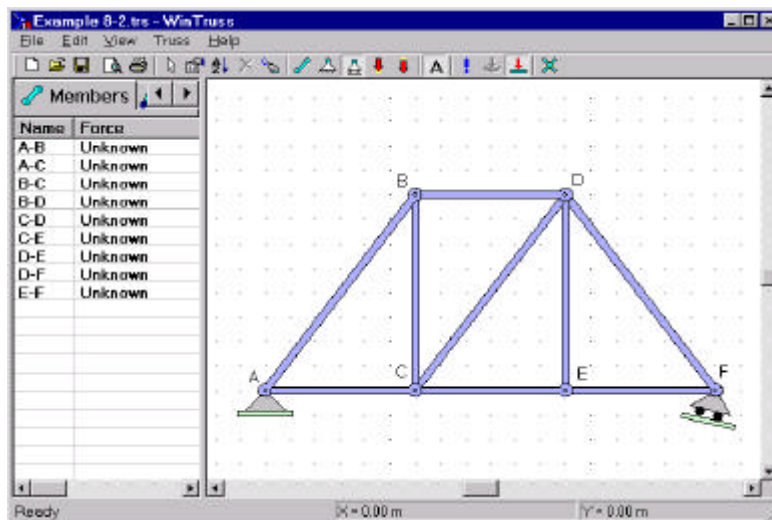


Fig. 18 Addition of a roller support resting on an inclined surface

Now that we have sketched the roller support in Fig. 18, we need to tell WinTruss the precise *directional angle* of a line normal to the surface of the incline on which the roller support rests. Move the mouse cursor over the roller support and right click the mouse. A context menu will pop up prompting us for action. Select properties from the context menu in order to bring up the “Roller Support Angle” dialog box, as shown in Fig. 19.

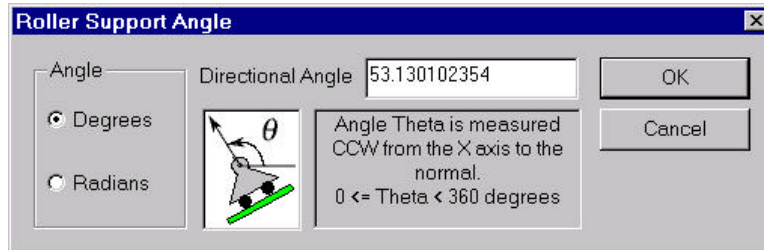


Fig. 19 Specifying the value of the *directional angle* of the normal to the incline on which the roller support rests

The dialog box in Fig. 19 allows you to specify an accurate value of the directional angle of the normal to the incline on which the roller support rests. The angle can be specified in either degrees or radians by selecting the appropriate option in the “Angle” group. The default setting is degrees.

Using a calculator, we find that the directional angle of the normal to the incline is

$$\mathbf{q} = \tan^{-1}\left(\frac{4}{3}\right) = 53.130\ 102\ 354\ ^\circ$$

In the “Directional Angle” edit box in Fig. 19, type in the number 53.130 102 354, or whatever precise number your calculator can give you. It is recommended that more significant digits be input to the computer in specifying the value of the directional angle, even though the final answers are usually reported to only three or four significant digits. Press “OK” to input the entered value and close the dialog box. The computer will redraw the roller support with an accurate orientation the next time.

Step 4. Adding Forces

We need to add the external loads to the truss schematic. Select the “Force (Tail on Joint)” tool from the toolbar. Move the mouse cursor until it rests on top of joint “E”. You will see a box surrounding the joint signifying that you may add a force to that joint. Press and hold the left mouse button down. Drag the cursor downward while keeping the left button depressed. Just as with the roller support, you do not need to be overly concerned with making the force perfectly vertical. Your screen should resemble the one shown in Fig.20.

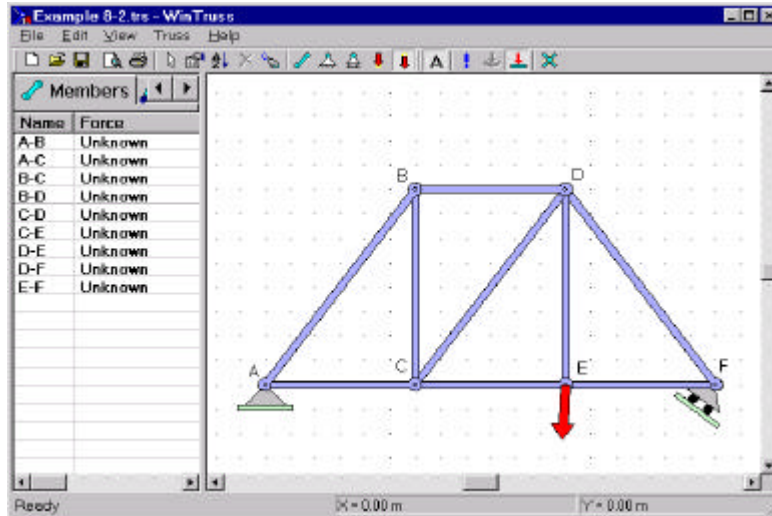


Fig. 20 Indicating the force acting on the truss for WinTruss

We know that the force acting on the truss at joint E is a downward force of 120 kN. We need to input this force. In Cartesian coordinate system, this force is

$$\mathbf{F} = 0 \mathbf{i} - 120 \mathbf{j} \text{ kN}$$

Move the mouse cursor somewhere over the force and right click to bring up a context menu. Select the “Properties” option to open the “Force Component” dialog. The dialog resembles the one shown in Fig. 21, where we keep the value of zero for the Horizontal Component, but type in the value -120 in the “Vertical Component” edit box.

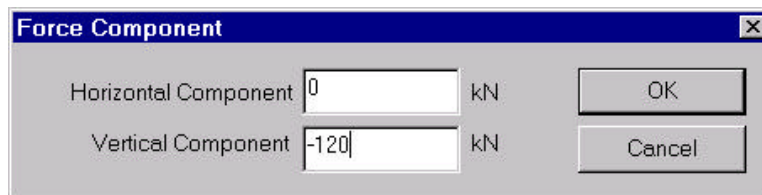


Fig. 21 Dialog box for “Force Component” under context menu

Pressing the “OK” button in Fig. 21 to input the indicated values, we see that WinTruss automatically realign the force so that it is perfectly vertical as shown in Fig. 22. WinTruss makes no attempt to scale the length of the force to match the magnitude of the force. Any attempt by the software to do so would not be practical and could result in the adjusted force covering over some part of the truss schematic that you wish to leave uncovered. It is therefore up to you to draw the force as long as you wish it to be.

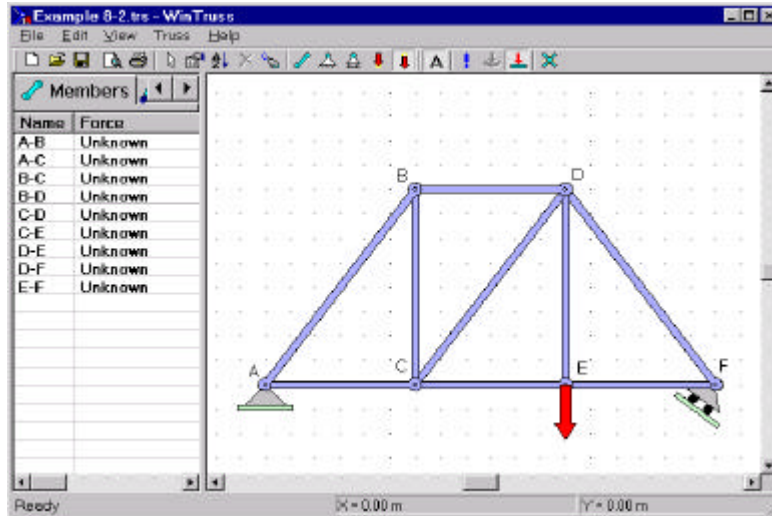


Fig. 22 The complete schematic of the truss in WinTruss

Step 5. Solve

Now that the schematic of the truss, and corresponding parameters, have been entered into WinTruss, we may let WinTruss solve for the forces and reactions. A pedagogical design to monitor the computer usage by students is a special dialog box shown in Fig.23. Pressing the “Solve” tool in the toolbar menu, which looks like a red exclamation mark, will initiate the solver within WinTruss. This will bring up a dialog asking for User Name, User ID, Course ID, and Problem ID as shown in Fig.23.

User Name and ID

User Name: John A. Doe

User ID: 123 - 45 - 6789

Course ID: MEEG 2003 Statics

Problem ID: Example 8.2

Check for Ill-Conditioned System

Solve

Cancel

Fig. 23 Dialog box for user name, user ID, course ID, and problem ID

The proper information in the various fields of the dialog box in Fig.23 should be filled out. Then, press the “Solve” button on the toolbar. WinTruss will record the time of submittal of solution and then solve the problem for you. WinTruss will display its results in the output pane as seen on Fig. 24.

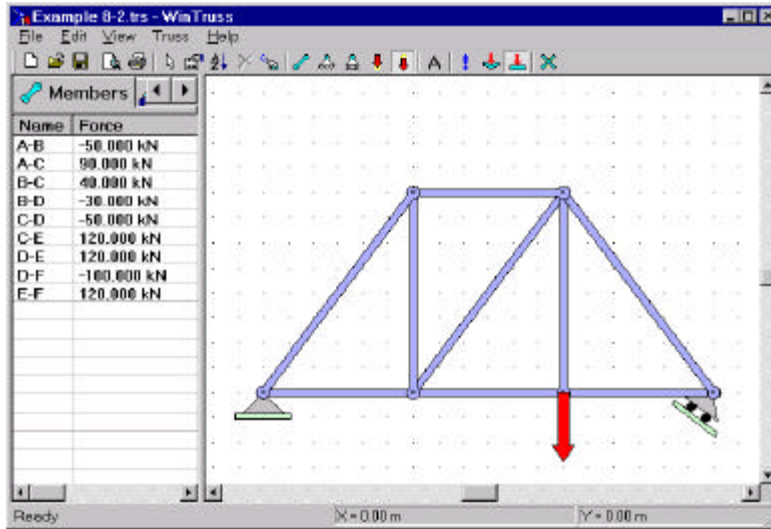


Fig. 24 Display of axial forces in the output pane for all truss members

The coordinates of the various joints of the truss may be viewed by using the right arrow button at the top of the output pane to scroll the output tab over. Once the “Joints” tab is visible, click on it to display the “Joints” output page. Your window will resemble the one shown in Fig. 25.

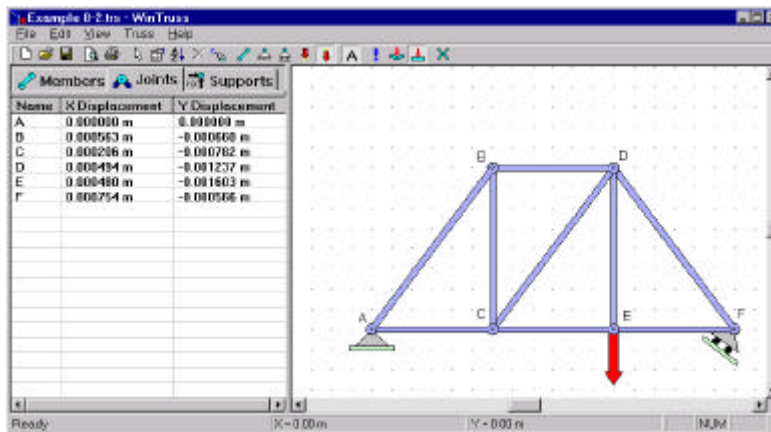


Fig. 25 Display of the “Joints” output page

Similarly, the reactions at the supports can be viewed by using the right arrow button at the top of the output pane to scroll the output tab over. Once the “Supports” tab is visible click on it to display the “Supports” output page. Your window will resemble the one shown in Fig. 26.

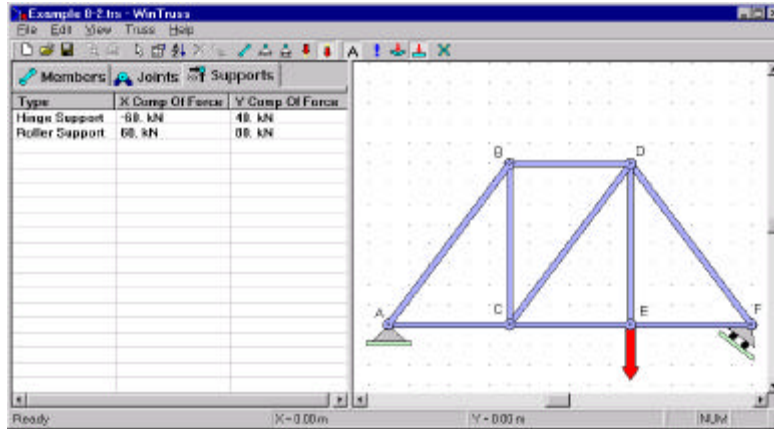


Fig. 26 Display of the “Supports” output page

You may use the scroll bar directly above the “Ready” prompt in the status pane to scroll left or right in the output window. You may press the “Print Preview” button to view a representation of what the printed output will look like. Finally, to obtain a printed hard copy of the output containing the appropriate solution time stamp, simply press the “Print” button (with a printer icon) on the toolbar.

This concludes the tutorial on how to use WinTruss. Best regards in you future endeavors involving WinTruss.