ANSYS - Truss Step 2

Step 2: Specify element type and constants

Enter the Preprocessor module

Main Menu > Preprocessor

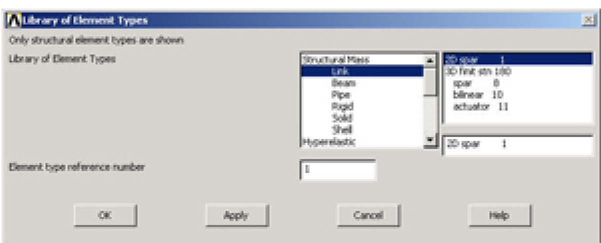
This is the module where you create the geometry, specify appropriate displacement constraints and loading, and mesh the geometry. We are more or less going to march down this menu to set up the problem. You will find yourself negotiating through a series of menus as you work off the ***Main Menu***.

Specify Element Type

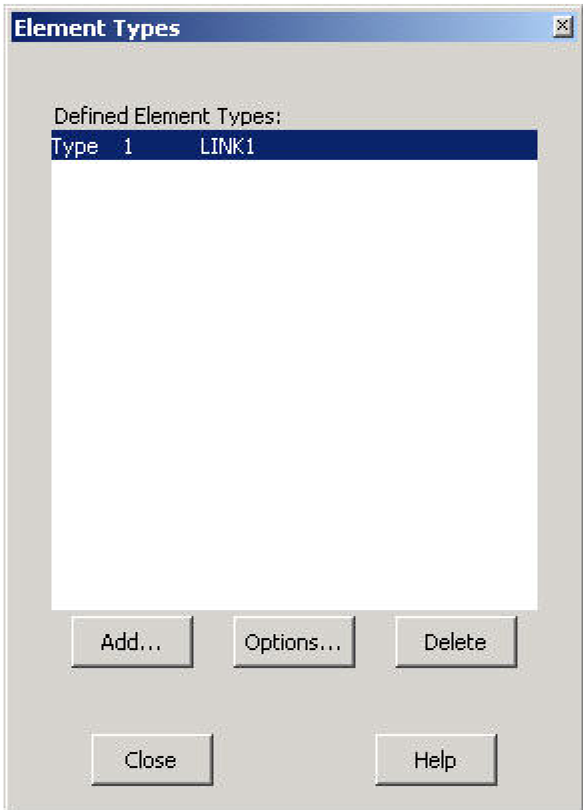
ANSYS provides a large selection of element types to solve linear and non-linear problems in structural mechanics, heat transfer, fluid mechanics and electro-magnetics. We next select the appropriate element type for our problem from this large list. Consider this as equivalent to rifling through a sizable chest, picking out one or more entities and placing them on a table for later use (in step 5, in our case). Select

Main Menu > Preprocessor> Element Type > Add/Edit/Delete > Add...

Pick ***Structural Link*** in the left field and ***2D spar 1*** in the right field. Click **OK** to select this element.



Now you will see the following in the Element Types dialog box:



LINK1 appears as the only defined element type in the *Element Types* dialog box. To view the help pages for this element type, click on ***Help*** in the ***Element Types*** dialog box. This brings up the Help window. Click on ***Search*** in the left pane and type in LINK1. (If the left pane is hidden, click the Show button in the toolbar). The first search result is the help page for *the LINK1* element. Bring up this help page by double-clicking on the search result. Note that this is a two-dimensional spar element that supports uniaxial tension and compression but not bending, so it is appropriate for modeling a truss structure. There are two degrees of freedom at each of its two nodes: translations in the nodal x and y directions. The "1" in the element name is the internal reference number for this element type in ANSYS' list of available element types.

Before proceeding, let's take a quick peek at the pictorial summary of the element types available in ANSYS. Search for "pictorial summary" and double-click on the search result titled *3.2 Pictorial Summary*. Click on the link to *Link Elements*. You will see our own humble *LINK1* element as well as other link elements in the pictorial summary. Clicking on the *LINK1* link will take you to the help page for the element that we just visited. In general, you need to take the time to understand the element types and pick the appropriate one(s) for your problem. The pictorial summary is a good place to start for identifying the appropriate element type for your problem. Your choice of element type has a significant effect on the speed and accuracy of the solution. Quickly browse through the pictorial summary to get an idea of the wide variety of element types available in ANSYS. This is what allows the software to solve engineering problems from a wide variety of disciplines.

Minimize the Help window.

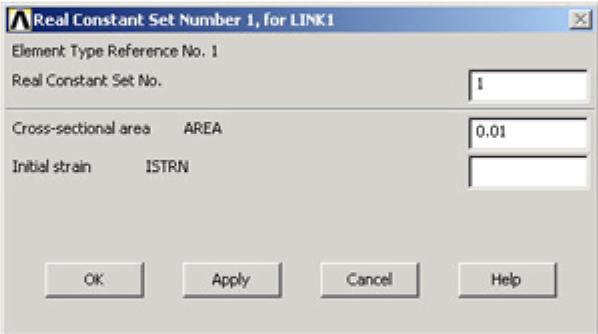
Close the *Element Types* window by clicking ***Close***.

Specify Element Constants

Main Menu > Preprocessor > Real Constants > Add/ Edit/ Delete

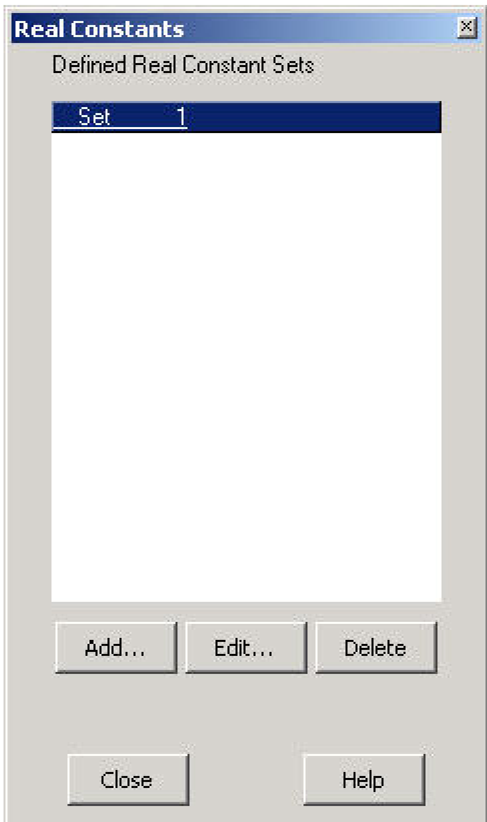
This opens up the *Real Constants* dialog box. Click ***Add...*** This brings up the *Element Type for Real Constants* dialog box with a list of the element types defined in the previous step. We have only one element type, *LINK1*, defined and it's automatically selected. Click ***OK.***

We now enter the constants needed for the *LINK1* element. For ***AREA***, enter `0.01` which is the ***Cross-sectional area*** of the element. We'll work in SI units. Leave the ***Initial strain*** field blank since it's not applicable to our problem.



It is the responsibility of the ANSYS user to make sure that the values entered are in consistent units. Click ***OK***.

We see in the *Real Constants* menu that the constant set that we just created is "Set 1". So, when we mesh the geometry later on, we'll use the reference no. 1 to assign this constant set.



Click ***Close***.

Go to Step 3: Specify material properties

See and rate the complete Learning Module

Go to all ANSYS Learning Modules