

ANSYS - Truss 2 Specify element type and constants content

Step 2: Specify element type and constants

Enter the Preprocessor module

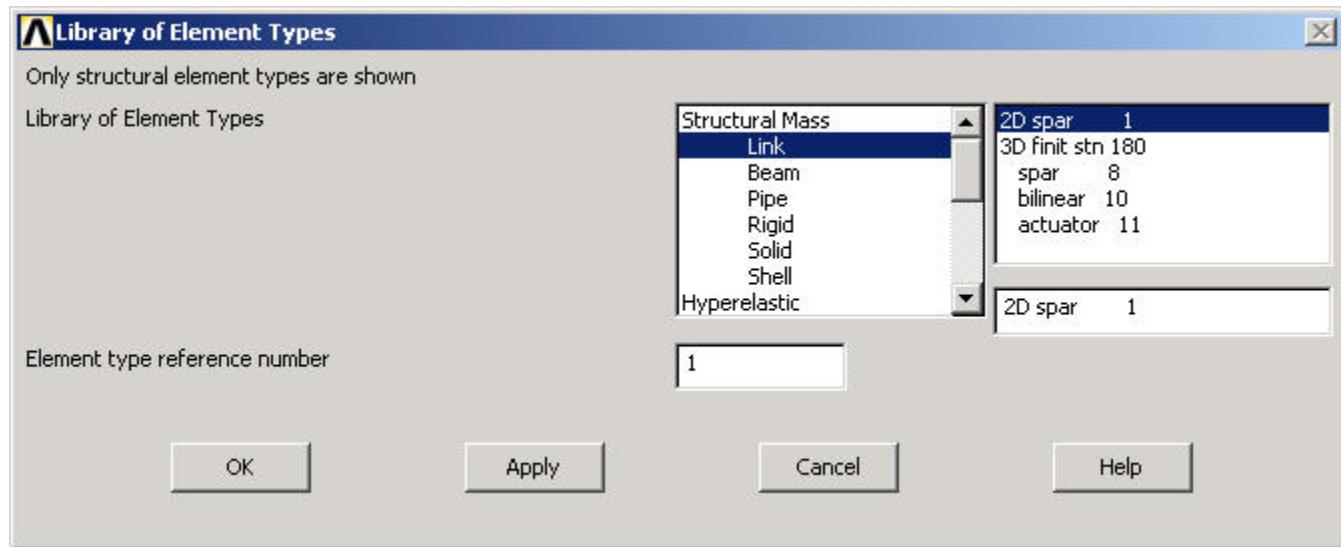
Main Menu > Preprocessor

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*.

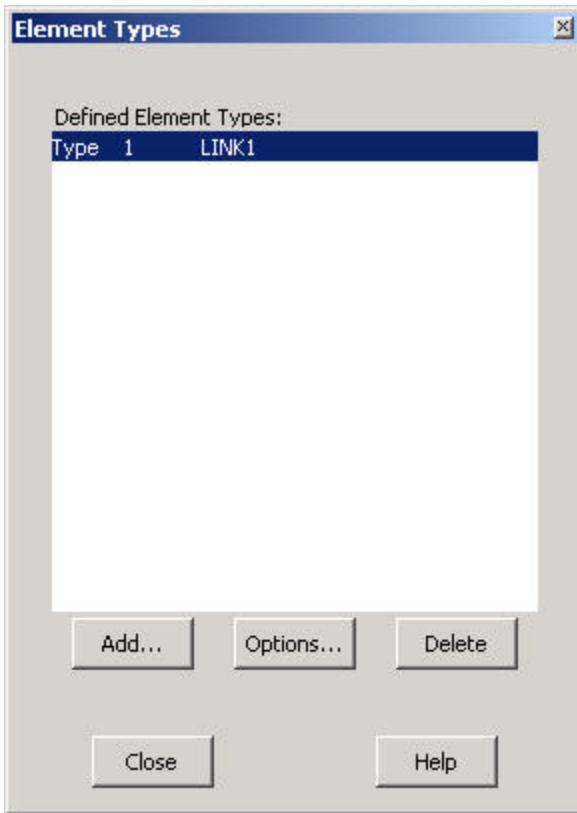
Specify Element Type

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. 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. Click on Release 10.0 Documentation for ANSYS > Element Reference > Element Characteristics > Pictorial Summary in the left part of the Help window. Our own humble *LINK1* element is listed at the top of the pictorial summary. Clicking on the *LINK1* box 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 for your problem. Your choice of element type has a significant effect on the speed and accuracy of the solution.

Minimize the Help window.

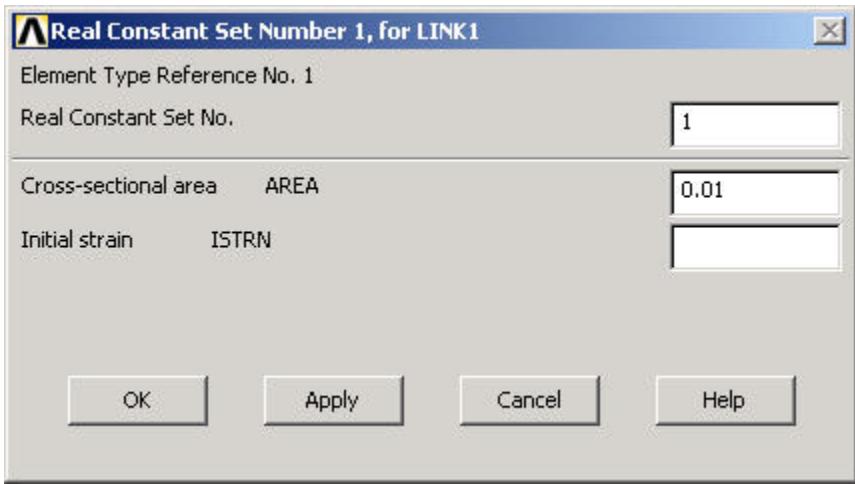
Close the *Element Types* window by clicking Close.

Specify Element Constants

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

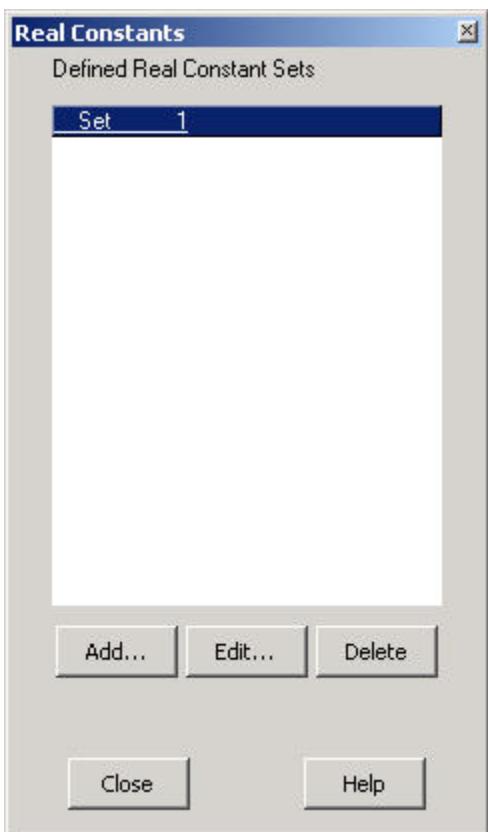
This opens up the *Real Constants* dialog box. Click Add.... This brings up the *Element Type for Real Constants* dialog box with a list of the element types defined in the previous step. We have only one element type, *LINK1*, defined and it's automatically selected. Click **OK**.

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 in the *Real Constants* dialog box.