ANSYS - Truss Step 5

Step 5: Mesh geometry

Each truss member can be represented as a 2D Spar element. We'll use the *MeshTool* to mesh the geometry with this element.

To bring up the *MeshTool*, select

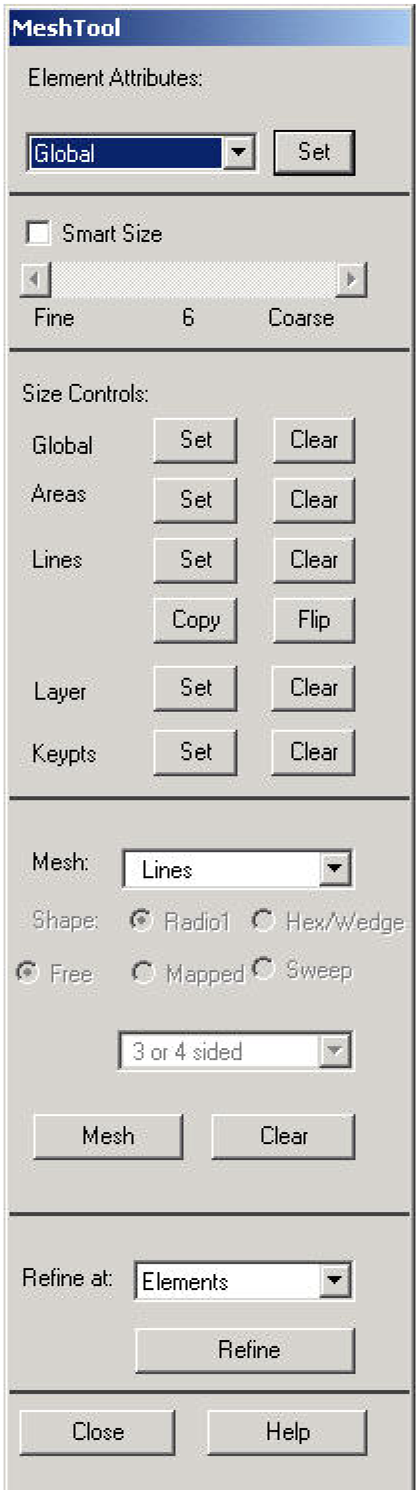
Main Menu > Preprocessor > Meshing > MeshTool

The *MeshTool* is used to control and generate the mesh.

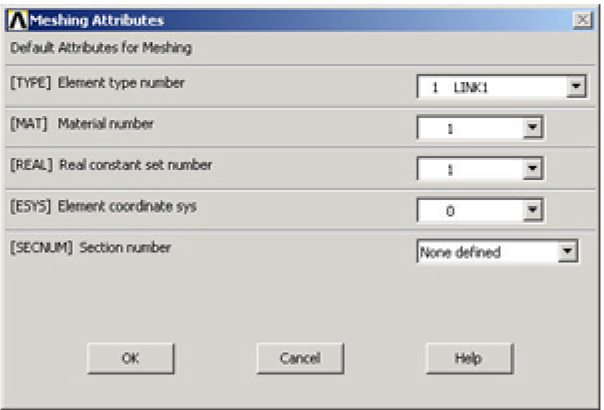
Set Meshing Parameters

We'll now specify the element type, real constant set and material property set to be used in the meshing. Since we have only one of each, we can assign them to the entire geometry using the ***Global*** option under ***Element Attributes***.

Make sure ***Global*** is selected under ***Element Attributes*** and click on ***Set***.



This brings up the *Meshing Attributes* menu. You will see that the correct element type, material number and real constant set are already selected since we have only one of each.



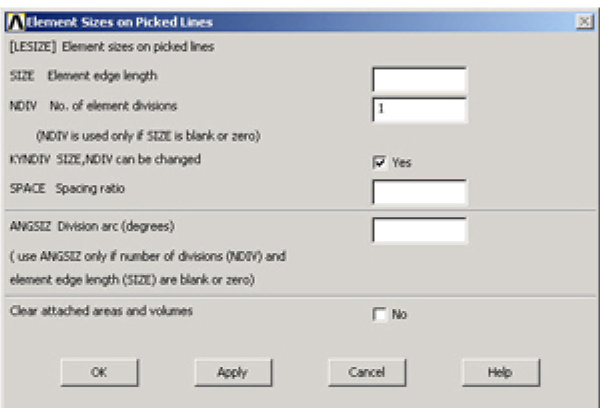
Click ***OK***. ANSYS now knows what element type (and associated constants) and material type to use for the mesh.

Set Mesh Size

Since a *LINK1* element is equivalent to a truss member, we will specify that we want only one element per line. This is a subtle point and also very unusual; in most problems, you want to subdivide your part into many elements.

In the *MeshTool*, under ***Size Controls*** and ***Lines*** , click ***Set***.

In the pick menu that comes up, click ***Pick All*** (since we want the specification of mesh size to apply to all lines in the geometry). This brings up the *Element Sizes on Picked Lines menu*. Specify ***No. of element divisions*** to be 1. Click ***OK***. ANSYS will now use 1 element to mesh each line.



Mesh Lines

In the *MeshTool*, make sure ***Lines*** is selected in the drop-down list next to ***Mesh***. This means the geometry components to be meshed are lines (as opposed to areas or volumes, as we'll see later). Click on the ***Mesh*** button.

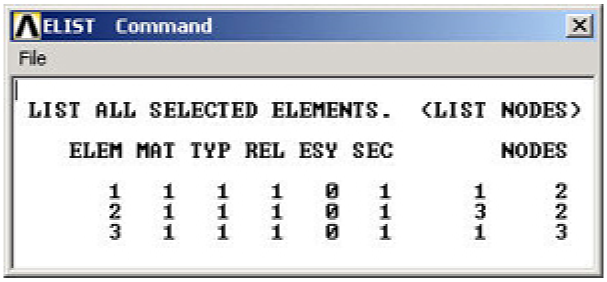
This brings up the pick menu. Since we want to mesh all lines, click on ***Pick All***. The lines have been meshed. This is reported in the *Output Window* (usually hiding behind the *Graphics Window*):

NUMBER OF LINES MESHED = 3
MAXIMUM NODE NUMBER = 3
MAXIMUM ELEMENT NUMBER = 3

Close the *MeshTool*.

View Element List

Utility Menu > List > Elements > Nodes + Attributes


ELIST Command

File

LIST ALL SELECTED ELEMENTS. (LIST NODES)

ELEM	MAT	TYP	REL	ESY	SEC	NODES	
1	1	1	1	0	1	1	2
2	1	1	1	0	1	3	2
3	1	1	1	0	1	1	3

This table says that *Element 1* is of *material type 1* and *element type 1* and is attached to nodes 1 and 2 and so on. In this element list, the order of the two nodes for each element doesn't matter. For example, element 3 can be attached to nodes 2 and 3 or equivalently, nodes 3 and 2. Also, the order of element numbering is not important since it is for internal bookkeeping.

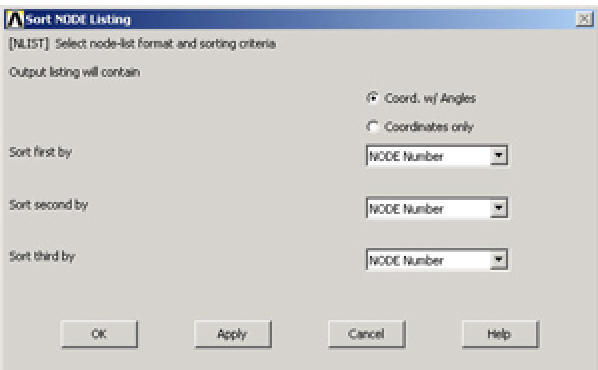
Close the window listing the elements.

View Node Location

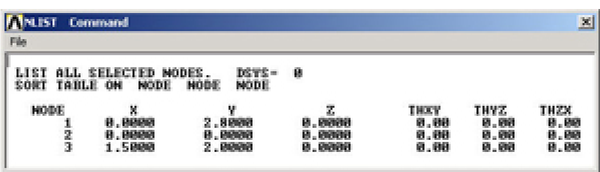
In order to see where the nodes are located, you can look at the list of nodes.

Utility Menu > List > Nodes

In *Sort NODE Listing* menu, click ***OK*** to accept defaults.



My list of nodes looks like this:


NLIST Command

File

LIST ALL SELECTED NODES. DSYS= 0
SORT TABLE ON NODE NODE NODE

NODE	X	Y	Z	THXY	THYZ	THZX
1	0.0000	2.0000	0.0000	0.00	0.00	0.00
2	0.0000	0.0000	0.0000	0.00	0.00	0.00
3	1.5000	2.0000	0.0000	0.00	0.00	0.00

From the node and element lists, one can conclude that in this case:

Node 1 is pin A
Node 2 is pin C
Node 3 is pin B

Element 1 is member AC
Element 2 is member AB
Element 3 is member BC

Your own node and element numbering might be different from this and you would have to account for this while interpreting results in the postprocessing step.

Close the window listing the nodes.

Save your work

Toolbar > SAVE_DB

Go to Step 6: Specify boundary conditions

See and rate the complete Learning Module

Go to all ANSYS Learning Modules