

ANSYS - Truss 5 Mesh geometry content

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.

MeshTool

Element Attributes:

Global

▼

Set

☐ Smart Size

◀

▶

Fine6Coarse

Size Controls:

Global

Set

Clear

Areas

Set

Clear

Lines

Set

Clear

Copy

Flip

Layer

Set

Clear

Keypts

Set

Clear

Mesh:

Lines

▼

Shape:

☒ Radio1

☐ Hex/wedge

☒ Free

☐ Mapped

☐ Sweep

3 or 4 sided

▼

Mesh

Clear

Refine at:

Elements

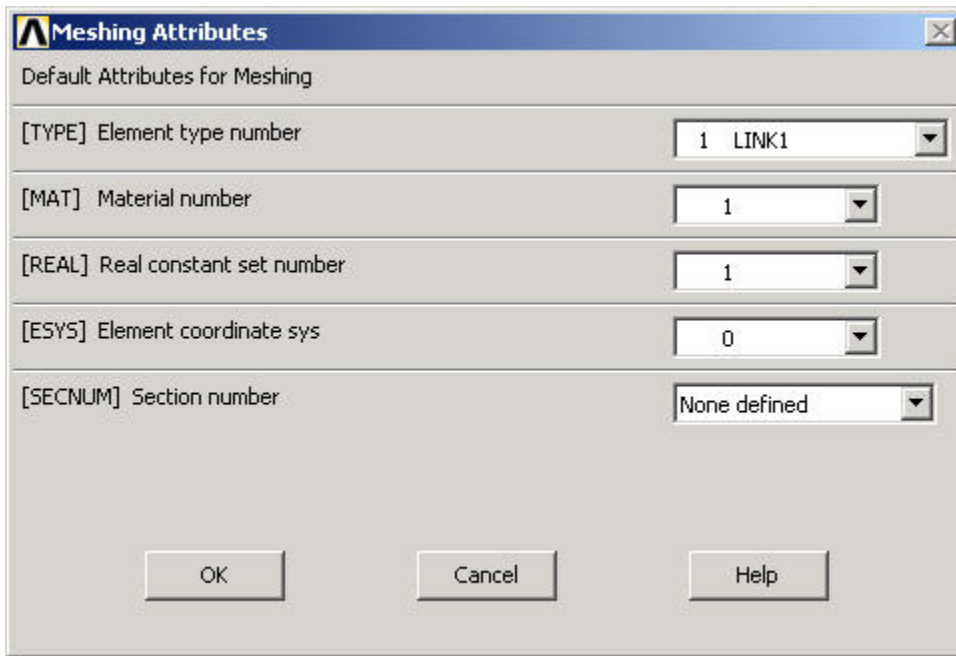
▼

Refine

Close

Help

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.



Meshing Attributes

Default Attributes for Meshing

[TYPE] Element type number: 1 LINK1

[MAT] Material number: 1

[REAL] Real constant set number: 1

[ESYS] Element coordinate sys: 0

[SECNUM] Section number: None defined

OK Cancel Help

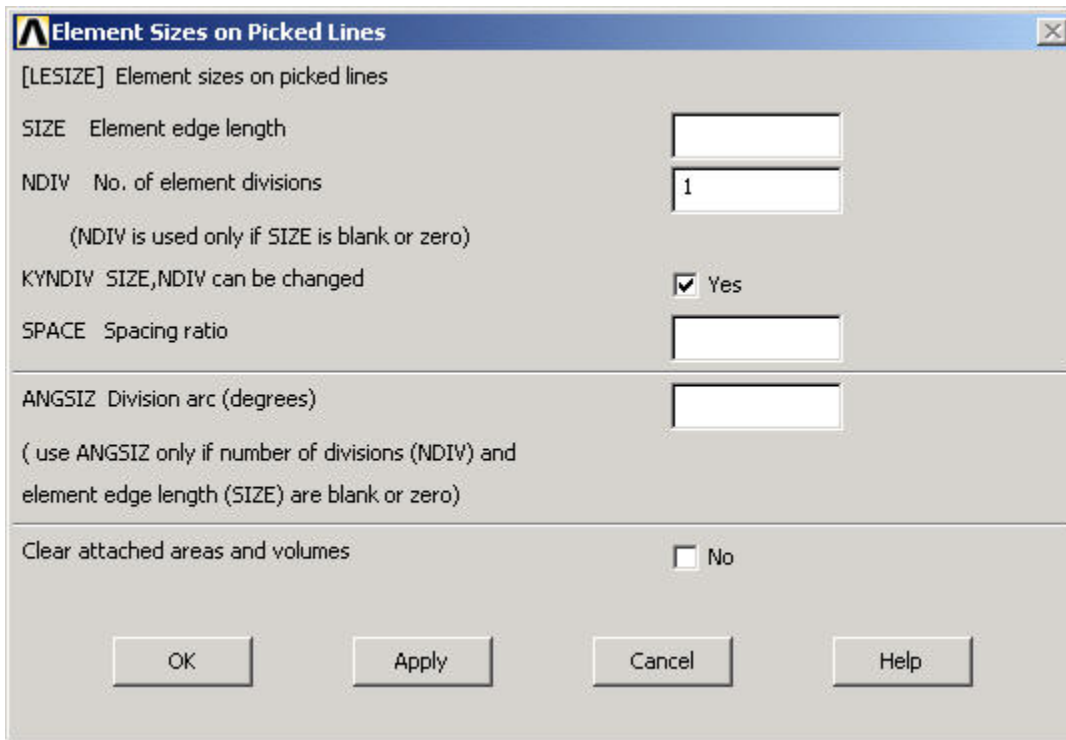
Click OK. ANSYS now knows what element type (and associated constants) and material type to use for the mesh.

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.



Element Sizes on Picked Lines

[LESIZE] Element sizes on picked lines

SIZE Element edge length:

NDIV No. of element divisions: 1

(NDIV is used only if SIZE is blank or zero)

KYNDIV SIZE,NDIV can be changed: ☒ Yes

SPACE Spacing ratio:

ANGSIZ Division arc (degrees):

(use ANGSIZ only if number of divisions (NDIV) and element edge length (SIZE) are blank or zero)

Clear attached areas and volumes: ☐ No

OK Apply Cancel Help

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.

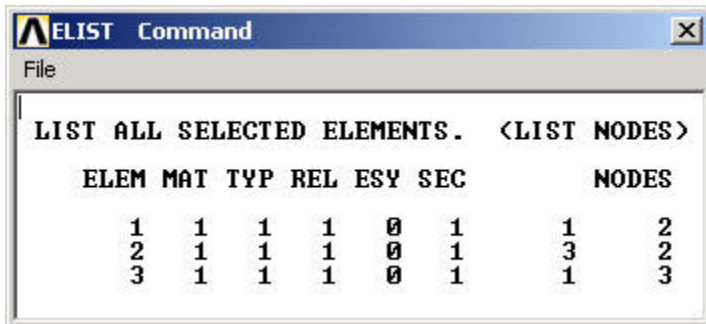
This brings up the pick menu. Since we want to mesh all lines, click on Pick All. The lines have been meshed. This is reported in the *Output Window* (usually hiding behind the *Graphics Window*):

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

Close the *MeshTool*.

View Element List

Utility Menu > List > Elements > Nodes + Attributes



The screenshot shows a window titled "ELIST Command" with a menu bar containing "File". The main area displays the command "LIST ALL SELECTED ELEMENTS. <LIST NODES>" and a table of element data.

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.

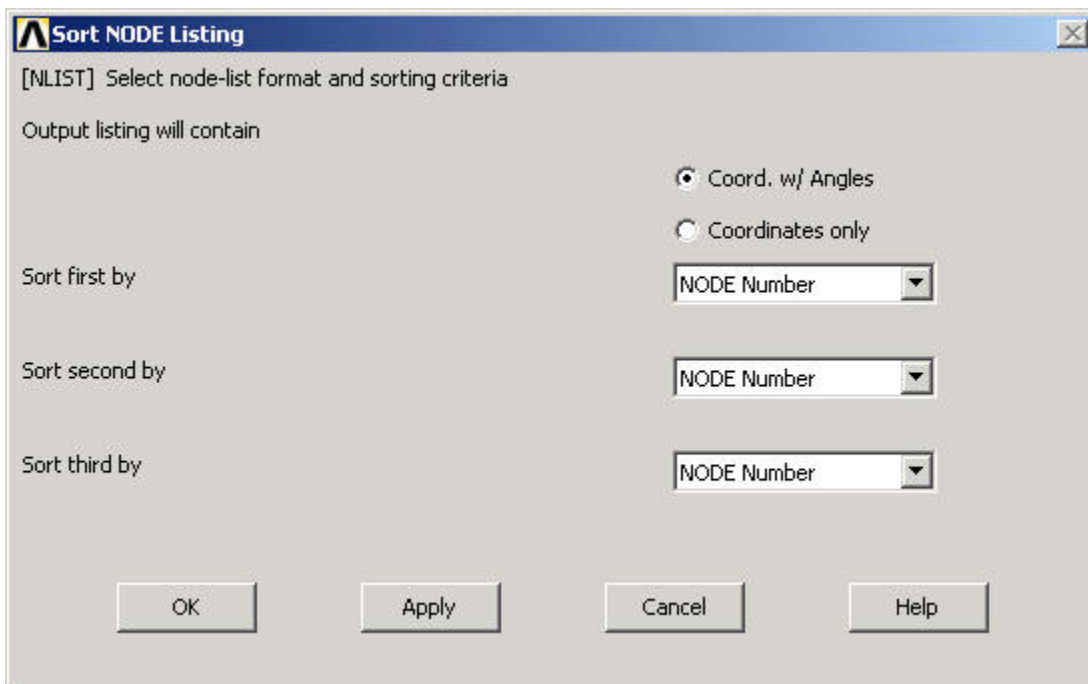
Close the window listing the elements.

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.



The screenshot shows a dialog box titled "Sort NODE Listing" with a close button (X) in the top right corner. The main text reads "[NLIST] Select node-list format and sorting criteria". Below this, it says "Output listing will contain". There are two radio buttons: "Coord. w/ Angles" (selected) and "Coordinates only". Below these are three rows for sorting criteria, each with a label and a dropdown menu:

- Sort first by: NODE Number
- Sort second by: NODE Number
- Sort third by: NODE Number

At the bottom, there are four buttons: OK, Apply, Cancel, and Help.

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	2.8000	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