# **ANSYS - Truss Step 5**

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

# 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 Siz	e				
Fine	6	Foarse			
	<u> </u>	Coarse			
Size Controls:					
Global	Set	Clear			
Areas	Set	Clear			
Lines	Set	Clear			
	Сору	Flip			
Layer	Set				
Keypts	Set	Clear			
Mesh: Lines 💌					
Shape: 📀 Radio1 C Hex/Wedge					
€ Free C Mapped C 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.

Default Attributes for Meshing	
[TYPE] Element type number	1 LINKI 💌
[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 *Elemen t 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.

Clement Sizes on Picked Lines		X
[LESIZE] Element sizes on picked lines		
SIZE Element edge length		
NDIV No. of element divisions	1	
(NDTV is used only if SIZE is blank or zero)		
KYNDIV SIZE,NDIV can be changed	🔽 Yes	
SPACE Spacing ratio		
ANGSEZ Division arc (degrees)		
( use ANGSEZ only if number of divisions (NDIV) and		
element edge length (SIZE) are blank or zero)		
Clear attached areas and volumes	IT No	
CK Anthe I	Carrol Hub	1

### **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

File							
LIST ALL	SEI	LECTI	ED EI	LEME	NTS.	(LIST	NODES>
ELEM I	MAT	TYP	REL	ESY	SEC	1	NODES
1	1	1	1	Ø	1	1	2
2	1	1	1	Ø	1	3	2
3	1	1	1	Ø	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.

Sort NODE Listing	×
[NLIST] Select node-list format and sorting criteria	
Output listing will contain	
	Goord. w/ Angles
	C Coordinates only
Sort first by	NODE Number
Sort second by	NCDE Number
Sort third by	NODE Number
Ск Асріу	Cancel Help

My list of nodes looks like this:



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