

AIM Stepped Shaft in Axial Tension - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc.

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Results](#)

[6. Verification & Validation](#)

Mesh

Close the **Model Editor**, then initiate the meshing process by clicking on **Mesh** in the workflow.

Set Mesh Size

Uncheck the **Use predefined settings** to access a greater depth of control over the mesh. Under **Global Sizing**, change the **Size function method** to **Curvature**. Input 0.05 [in] as the **Minimum size** and set the **Curvature normal angle** to 5 [degrees].



Mesh



Attention required



Settings

☐ Use predefined settings

Engineering intent

Structural, thermal or electric cond

Boundary Layer Settings

Collision avoidance

- ☐ None
- ☐ Layer compression
- ☒ Stair stepping

Global Sizing

Size function method ?

- ☐ Adaptive
- ☒ Curvature
- ☐ Proximity
- ☐ Fixed
- ☐ Curvature and proximity

Minimum size

0.05 in

Maximum face size

0.4884 in

Maximum size

0.9767 in

Growth rate

1.2

Curvature normal angle

5 [degree]

Under **Objects** select the **Add** drop down menu next to **Mesh Controls**, add an **Element Shape control**. Use the **Body Selection** tool to add the entire shaft volume as the **Location** and change the **Shape** to **Hexahedrons**.



Element Shape 1



Up-to-date

Location

1 volume



Shape

Hexahedrons



Midside nodes

Use engineering intent



Next Step ▾

> Related Objects and Tasks

Under **Objects**, select the **Add** drop down menu next to **Size Controls** and add a **Face Sizing**. As the **Location** select either of the flat sides of the stepped shaft and input 0.2 [in] as the **Element size**.



Face Sizing 1



Up-to-date

Location

1 face



Element size

0.2 in

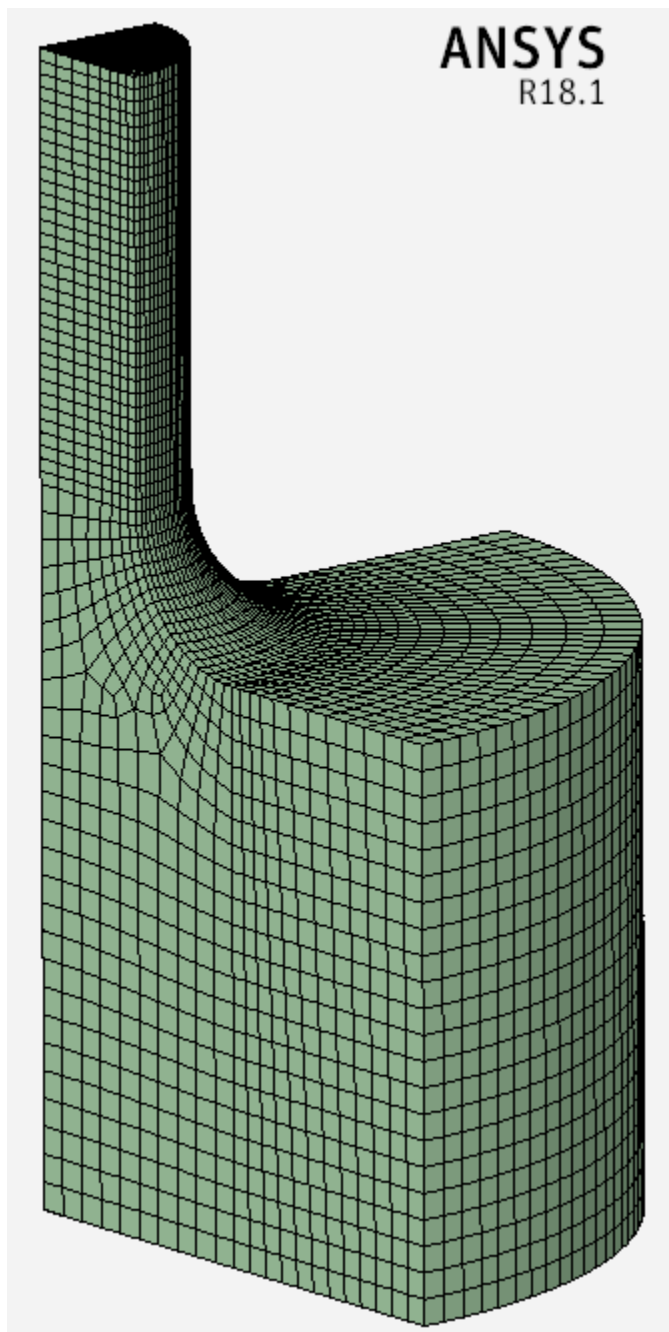


Next Step ▾

> Related Objects and Tasks

Generate Mesh

Click **Generate Mesh** under **Output** or at the top of the screen by the status window for Mesh. AIM should detect you are ready to generate the mesh and highlight the buttons in blue. Below is an example of what the mesh should look like.



[Go to Step 4: Physics Set-Up](#)

[Go to all ANSYS AIM Learning Modules](#)