

# ANSYS Steady Flow over a Cylinder - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc.

[Problem Specification](#)

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

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Results](#)

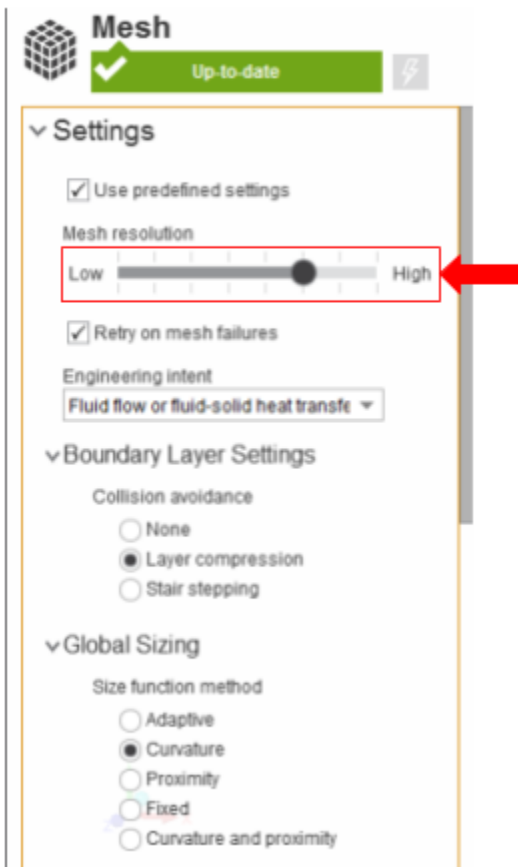
[6. Verification & Validation](#)

## Mesh

Once you have exited the modeling window, initiate the meshing process by clicking on **Mesh** in the workflow.

### Set Mesh Size and Boundary Layer

Use a medium-high setting on the **Mesh resolution** slider. This will automatically adjust the mesh settings for our needs.



Change the view to **Translucent display** using the view dropdown selection in the upper region of the window. AIM will prompt you to fix the boundary layer before generating the mesh. Click on **Boundary Layer** under **Mesh Controls**. The boundary layer will be applied to the face of the cylinder inside of the flow volume.

## Generate Mesh

Return to the **Mesh** panel, then click **Generate Mesh** under **Output** or at the top of the screen by the status window for Mesh. AIM should detect that you are ready to generate the mesh and highlight the buttons in blue.

**Go to Step 4: Physics Set-Up**

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