

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](#)

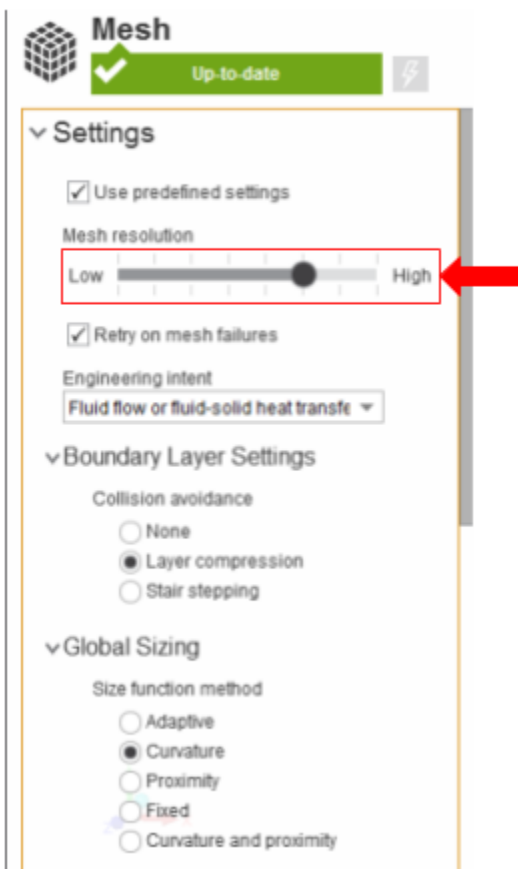
[6. Verification & Validation](#)

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](#)