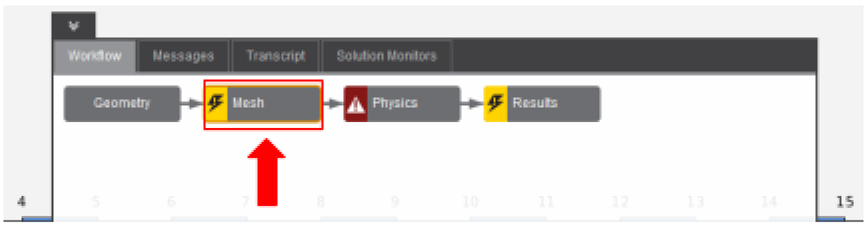
ANSYS Flow Through U-Duct - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc.

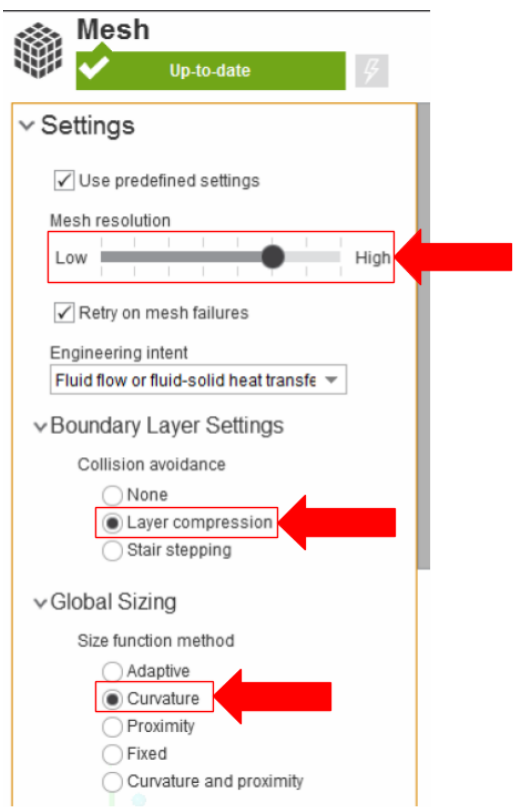
Mesh

Once you have imported the geometry file initiate the meshing process by clicking on **Mesh** in the workflow.



Set Mesh Size and Boundary Layer

In the **Engineering Intent** drop down menu select **Fluid flow or fluid-solid heat transfer** and use a medium-high **mesh resolution** setting on the slider. This will automatically adjust the mesh settings for our needs. Ensure that under **Global Sizing**, the **curvature option** is being used for the **size function method**.. In the **boundary layer settings** under **collision avoidance** select **layer compression**.



AIM will prompt the user to fix the boundary layer before generating the mesh, click on **boundary layer** under **Mesh Controls**. Select every face except for the future inlet and outlet faces.

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