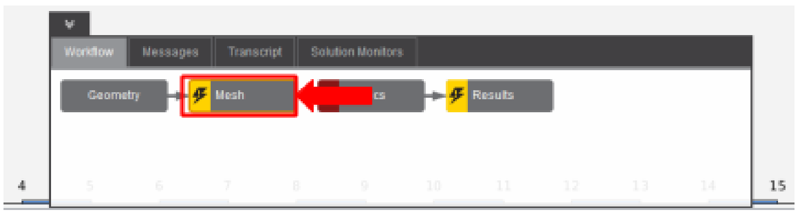
ANSYS Flow over an Ahmed Body - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc.

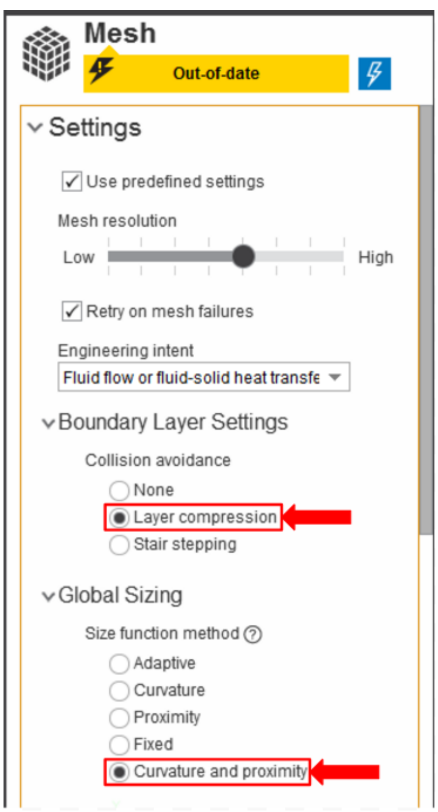
Mesh

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



Set Mesh Controls

Under **Global Sizing**, change the **Size function method** to **Curvature and proximity**. The curvature option will automatically refine the mesh near the curved surfaces of the body. The proximity option will automatically refine the mesh between the bottom of the body and the ground. In the **Boundary Layer Settings**, under **Collision avoidance**, use the **Layer compression** setting. This will ensure continuous boundary layers around the body, which can improve accuracy for external flows.



AIM will prompt you to fix the boundary layer before generating the mesh. Click on **Boundary Layer** under **Mesh Controls**. Select the faces of the volume in contact with the Ahmed body, as shown below. The simplest way to do this is to drag a box around the Ahmed body, from the upper left to the lower right. AIM will select all faces that are completely enclosed.

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 will detect that you are ready to generate the mesh and highlight the buttons in blue.

Go to Step 4: Physics Setup

Go to all ANSYS AIM Learning Modules