ANSYS Flow over an Ahmed Body - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

1. Start-Up

2. Geometry

3. Mesh

4. Physics Setup

5. Results

6. Verification & Validation

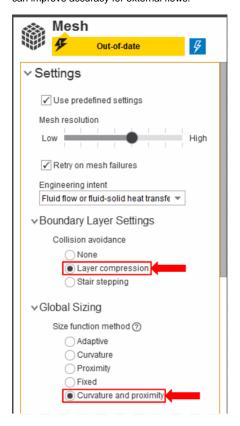
Mesh

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



Set Mesh 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.



AlM 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. AlM 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