ANSYS Fluid Flow over a Bluff Body - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc.

- **Problem Specification**
- 1. Start-Up
- 2. Geometry
- 3. Mesh
- Physics Setup
 Solution/Results
- 6. Verification & Validation

<u>Mesh</u>

Initiate the meshing process by clicking on Mesh in the workflow.

Set Mesh Size

Under Global Sizing, choose the Proximity option for the Size function method. The Proximity option will automatically refine the mesh between nearby faces. In the Boundary Layer Settings, under Collision avoidance, use the Layer compression setting. The Layer compression setting ensures continuous boundary layers around the corners.



The location of the boundary layer will need to be specified, so click **Boundary Layer** under **Mesh Controls** in the **Objects** section of the **Mesh** panel. Use the **face selection** tool and select all of the faces except the front and back end of the flow volume, as shown below.

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