

ANSYS Fluid Flow over a Bluff Body - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

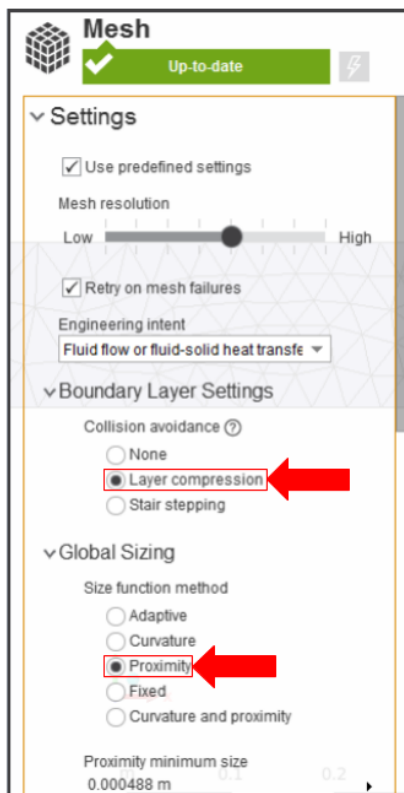
1. Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Solution/Results
6. Verification & Validation

Mesh

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](#)