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.

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