ANSYS Flow over an Ahmed Body - Physics Setup

Author(s): Sebastian Vecchi, ANSYS Inc.

Physics Setup

Boundary Conditions / Forces

First, the inlet must be defined using the **Fluid Flow Conditions**. In the **Add** drop down menu by **Fluid Flow Conditions**, select Inlet. Then, using the face selection tool, define an **Inlet** at the face upstream of the Ahmed body. The front of the body is the taller end with the rounded edges and corners. Make sure to input the **Velocity magnitude** as 40 m/s.
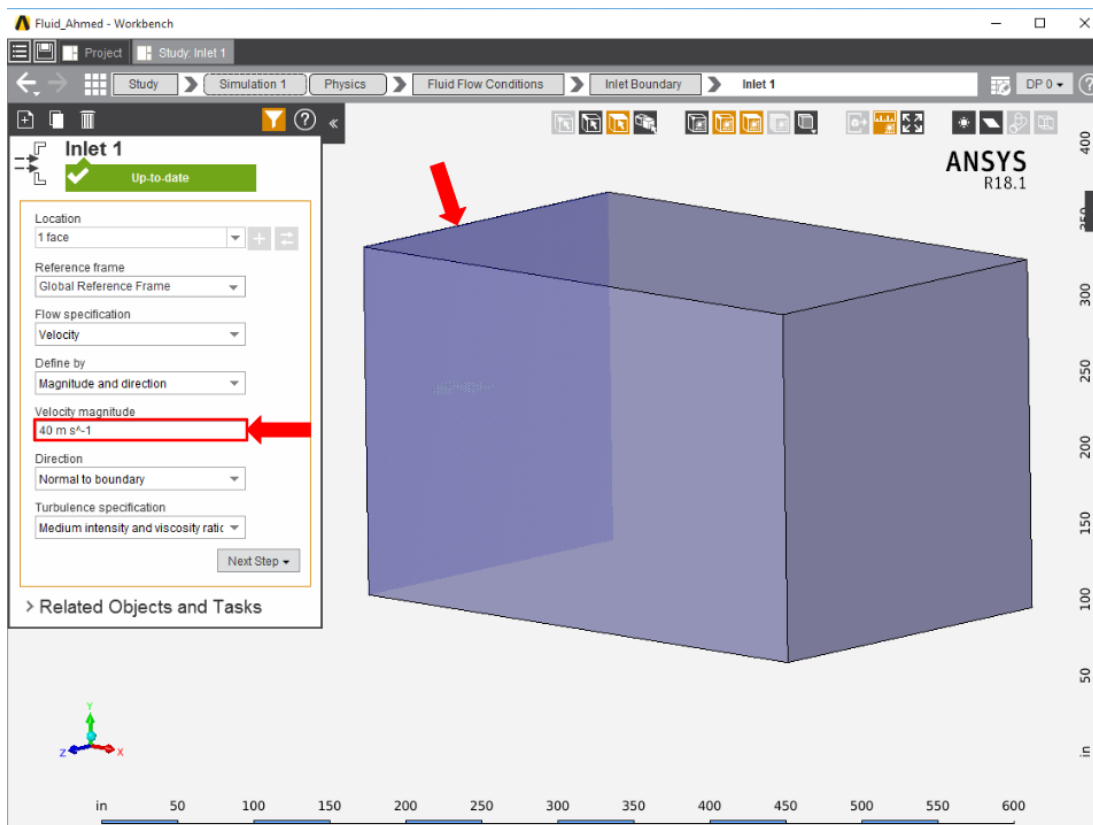

Once the inlet is defined, the outlet is next. In the same **Add** menu, use the **Outlet** condition to define an outlet downstream of the body. Assign a **Gauge static pressure** of 0 psi.
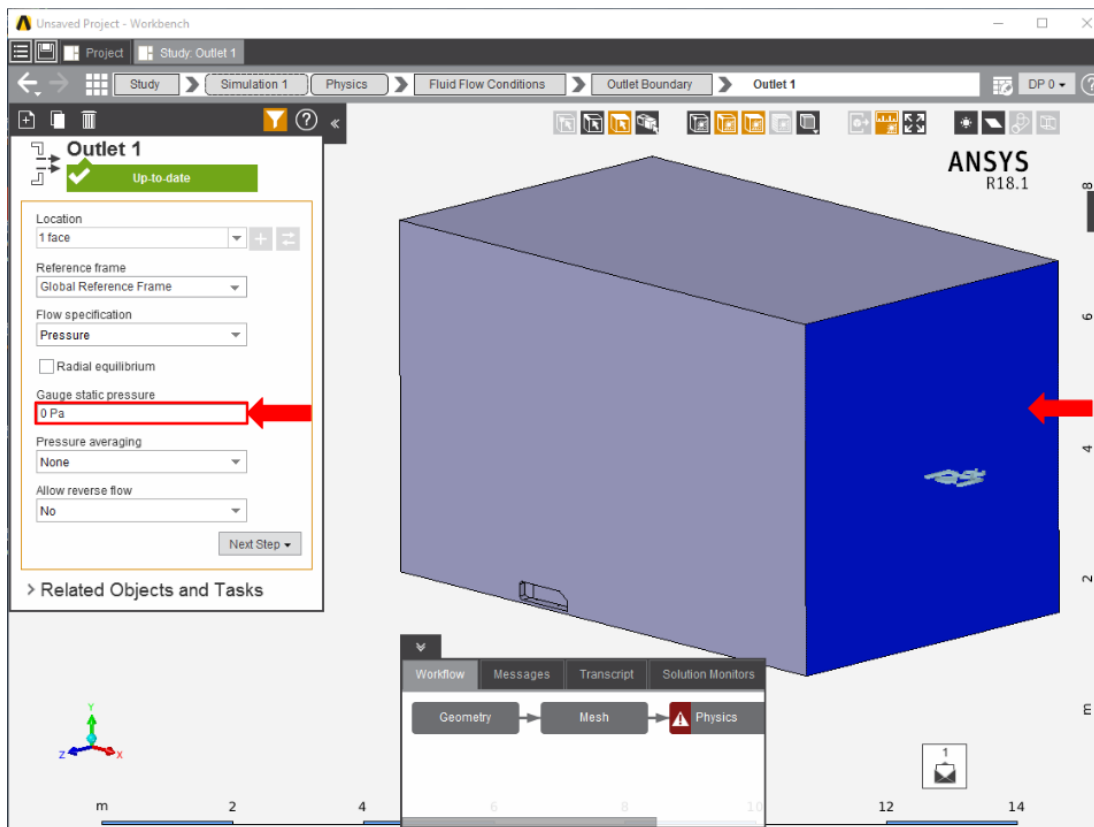
Create openings for the sides of the flow volume by selecting **Opening** in the **Add** drop down menu. Select the top face of the flow volume, and the side face away from the Ahmed body, then input 0 Pa for the **Gauge entrainment pressure**.
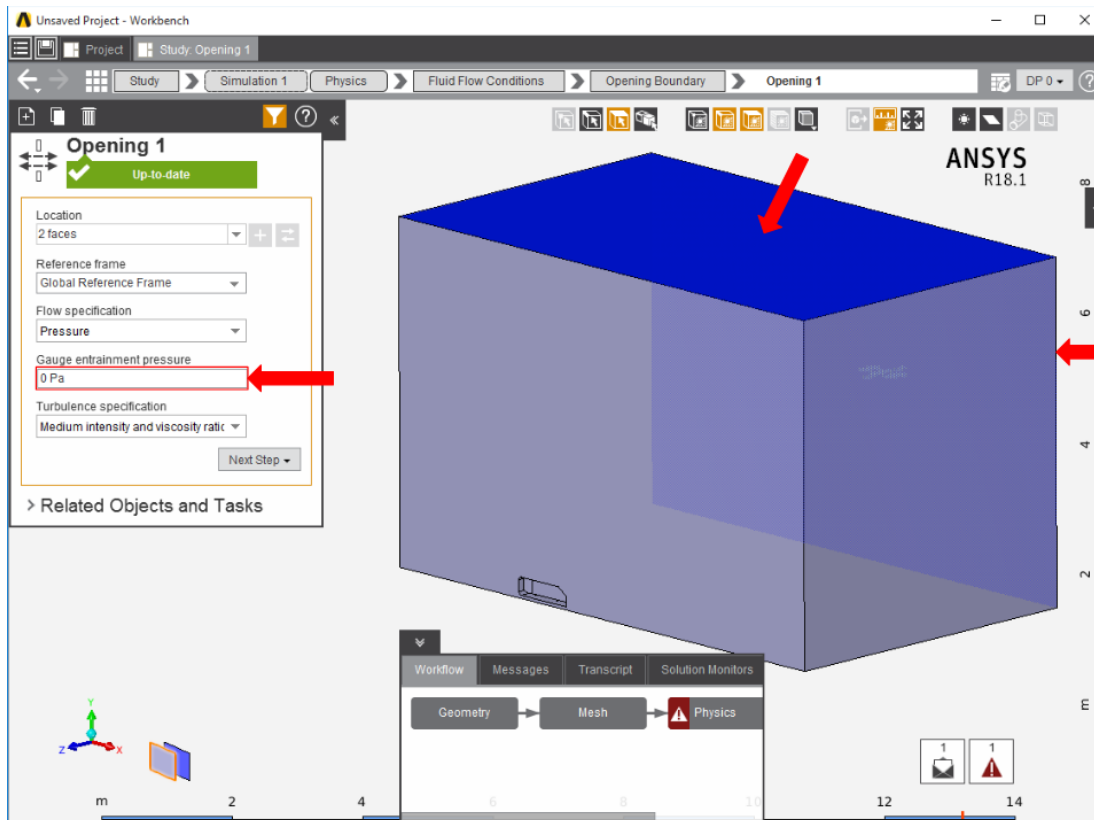
Add a **Symmetry** condition from the **Add** drop down menu to the face of the flow volume which passes through the Ahmed body.
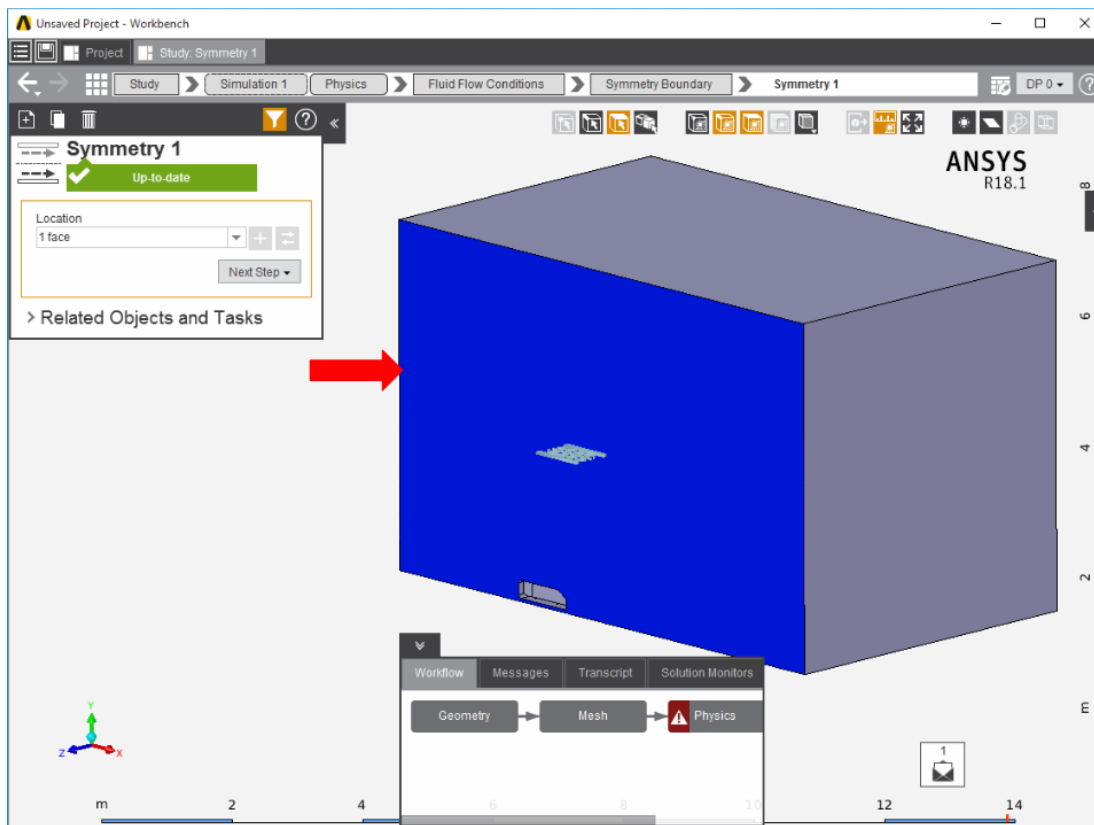
A **Wall** condition needs to be added for the remaining faces of the flow volume. In the **Add** menu, select **Wall**. AIM will automatically select all faces not already assigned.

Press **Solve Physics** in the **Physics** panel to run the calculations, then move on to the next step.

Go to Step 5: Results

Go to all ANSYS AIM Learning Modules