ANSYS Compressible Flow over a Wing-Body Junction - Physics Setup

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification
1. Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Solution/Results

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 of the enclosure in front of the airplane. Make sure to input the **Velocity magnitude** as 600 [mi hr^-1] and 0 [F] for the **Temperature**.
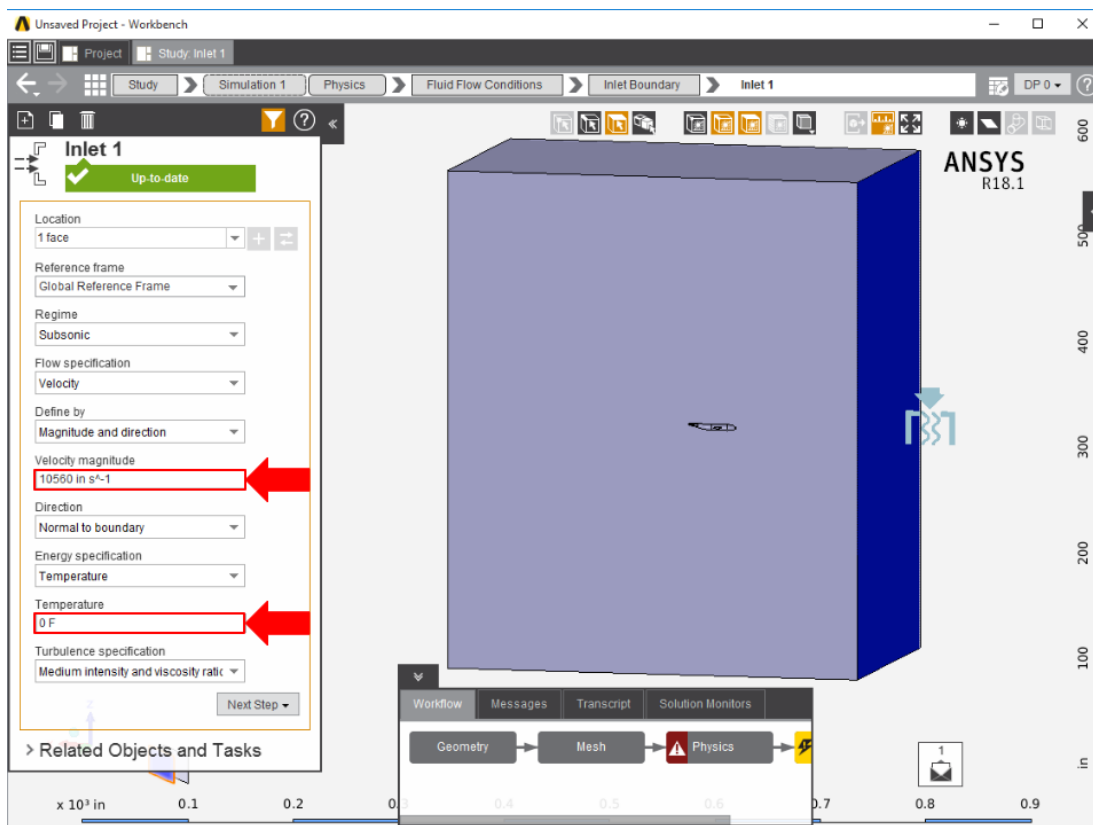
In the same **Add** menu, select **Outlet** to define an outlet at the tail end of the enclosure. Assign a **Gauge static pressure** of 0 [psi].
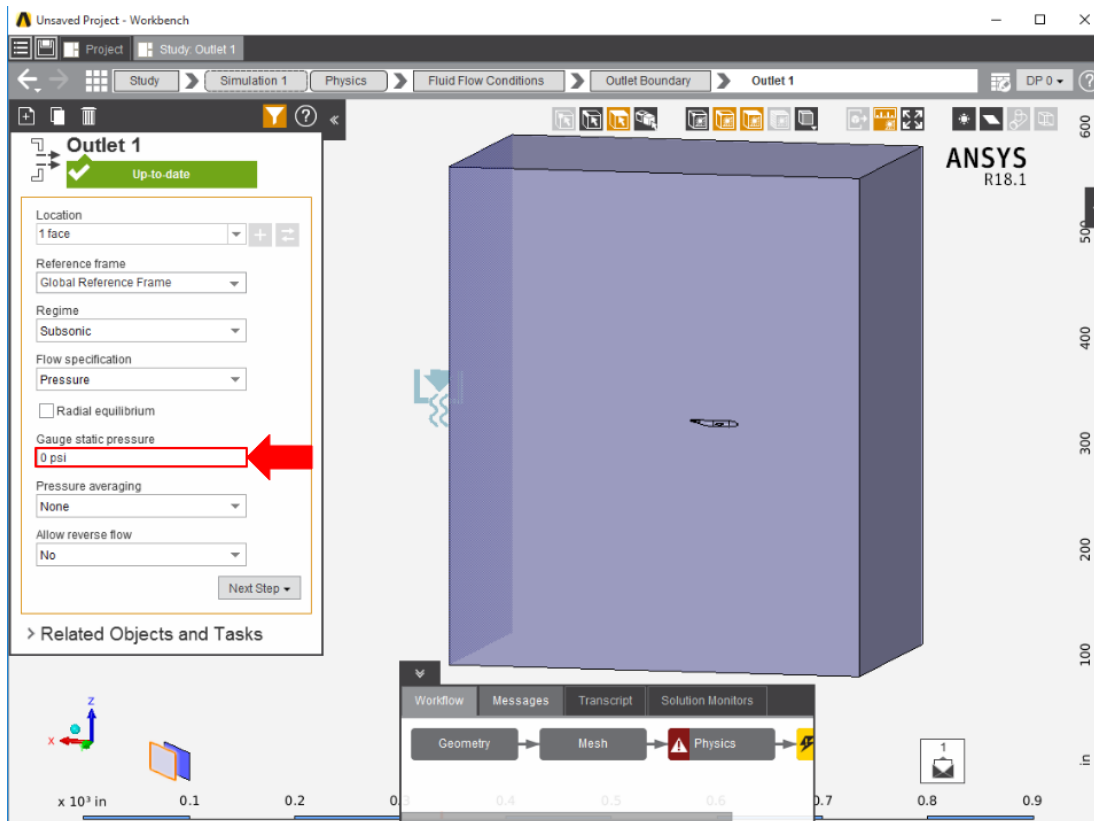
Create openings for the sides of the flow volume by selecting **Opening** in the **Add** drop down menu. Select the top and bottom faces of the enclosure, plus the side face opposite the wing tip. Input 0 [Pa] for the **Gauge entrainment pressure**.
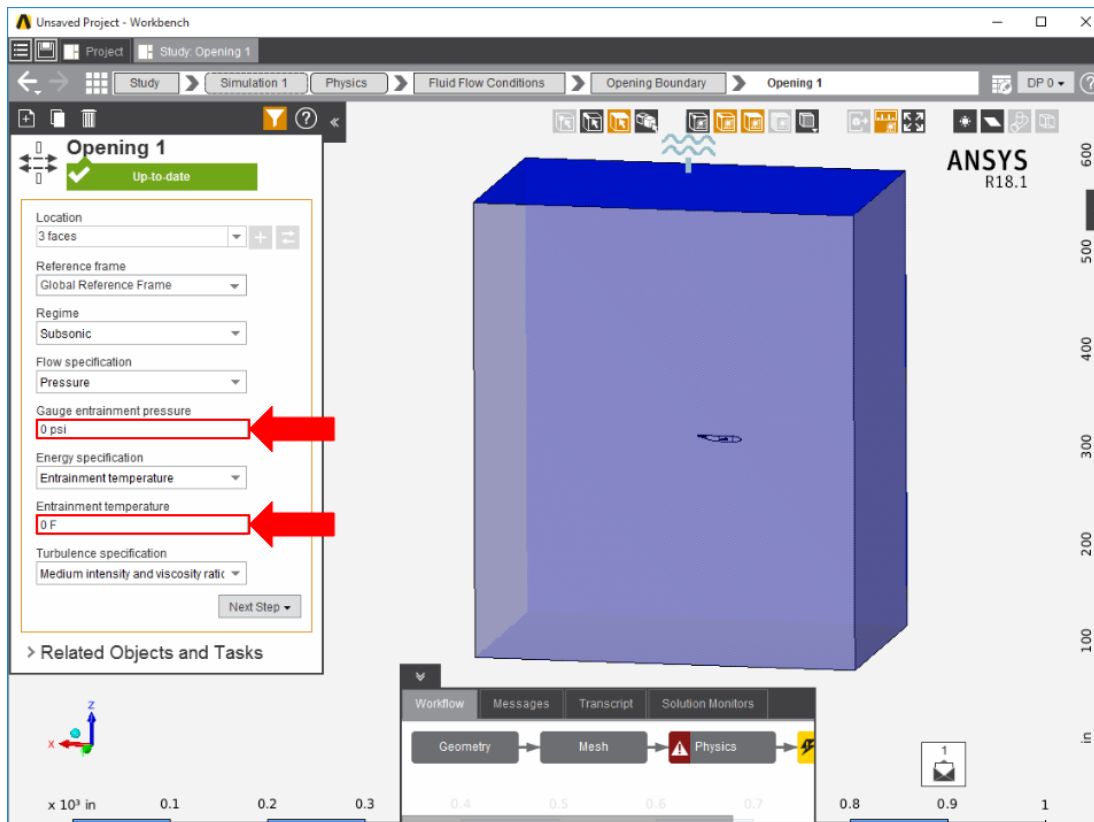
Add a **Symmetry** condition from the **Add** drop down menu to the face coincident with the cut airplane body.

Next, a **Wall** condition must be added to all surfaces that are not already defined. **Wall** can be found in the same **Add** menu as the previous conditions. AIM will automatically select every face that doesn't already have a constraint..

Press **Solve Physics** in the **Physics** panel to run the calculations.

Go to Step 5: Solution/Results

Go to all ANSYS AIM Learning Modules