

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

Author(s): Sebastian Vecchi, ANSYS Inc.

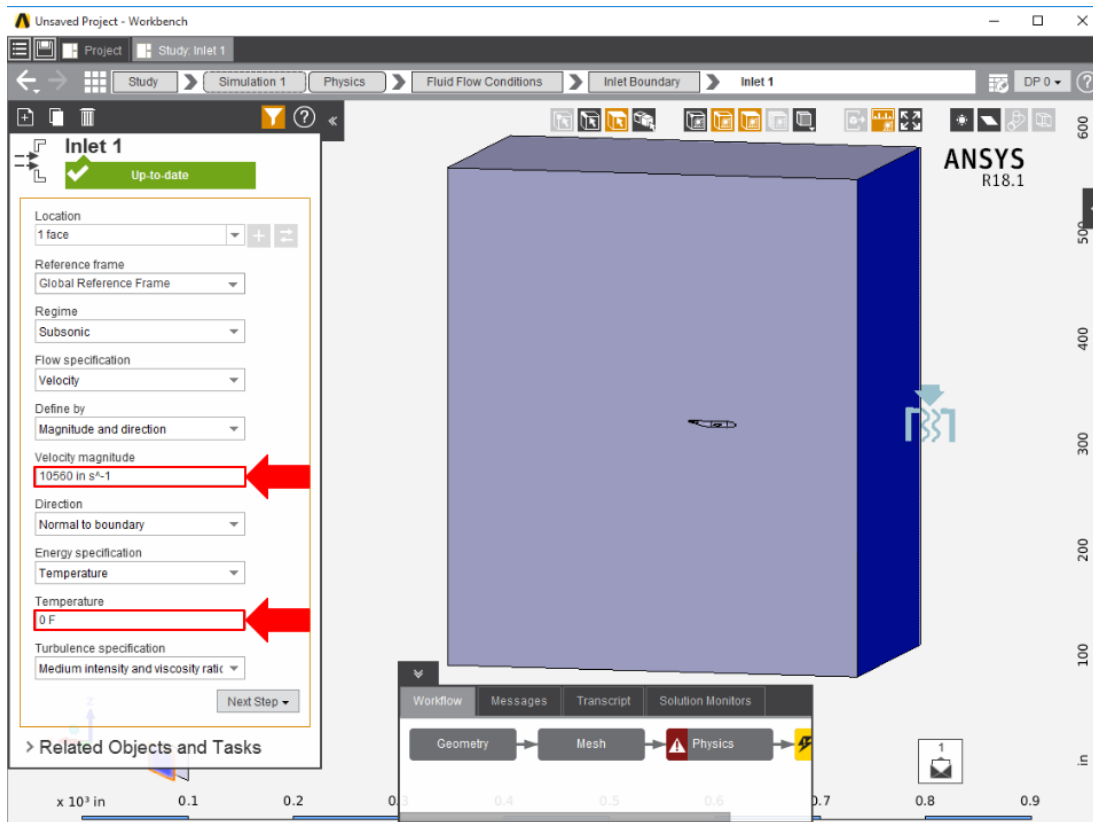
Problem Specification

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

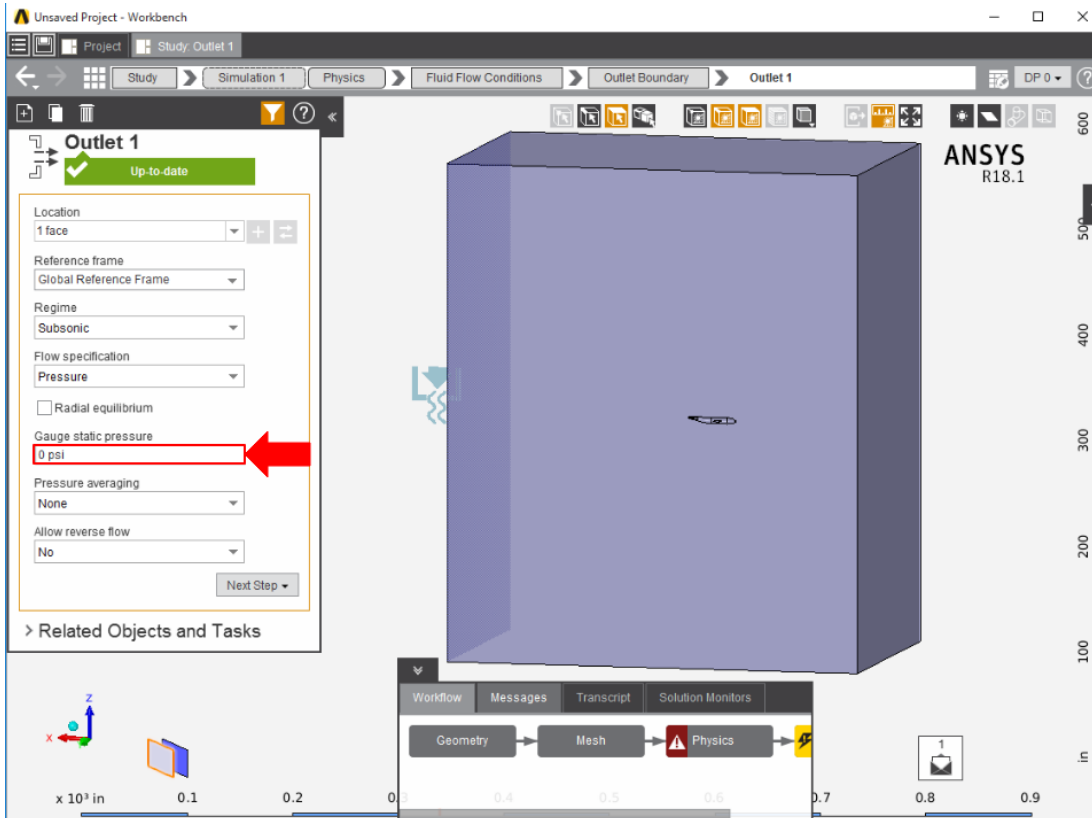
Physics Setup

Boundary Conditions / Forces

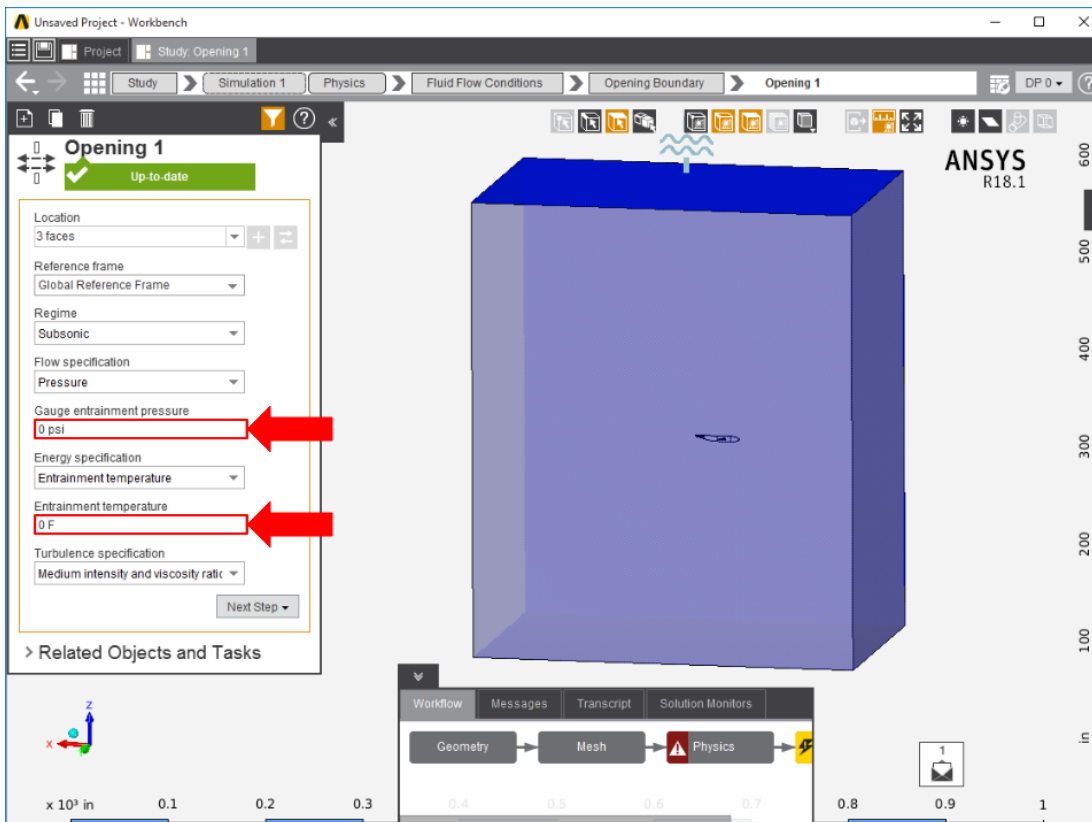
First, the inlet must be defined using the **Fluid Flow Conditions**. In the **Add** drop down menu by **Fluid Flow Conditions**, select **Inlet**. Then, using the **Face selection** tool, define an inlet at the face of the enclosure in front of the airplane. Make sure to input the **Velocity magnitude** 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](#)