

ANSYS Transonic Flow over a Wing - Physics Setup

Author(s): Sebastian Vecchi, ANSYS Inc.

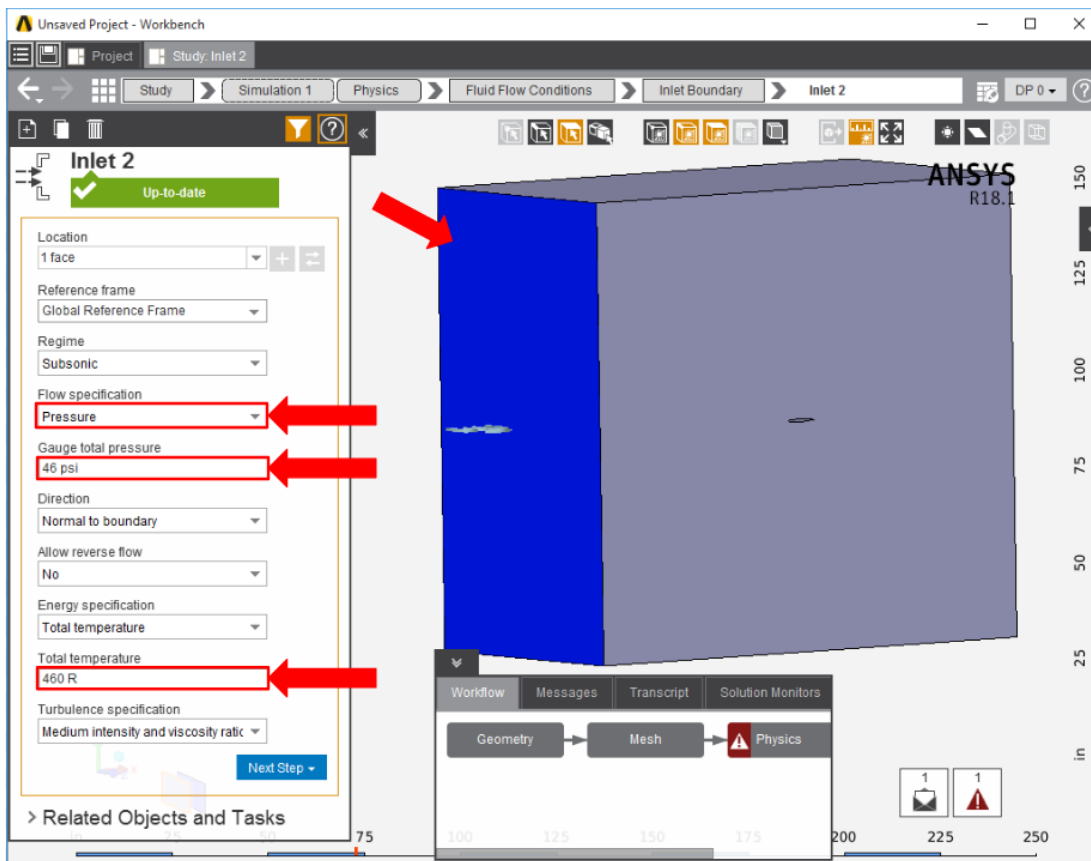
Problem Specification

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

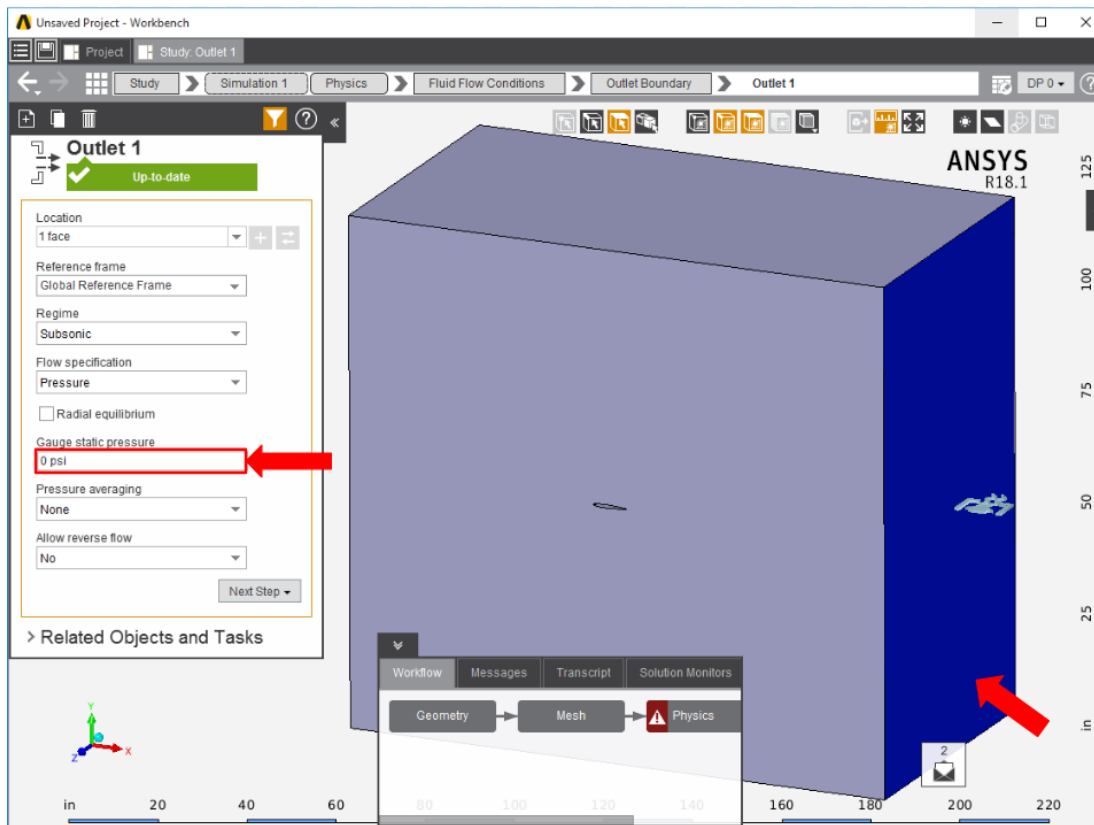
Physics Setup

Boundary Conditions / Forces

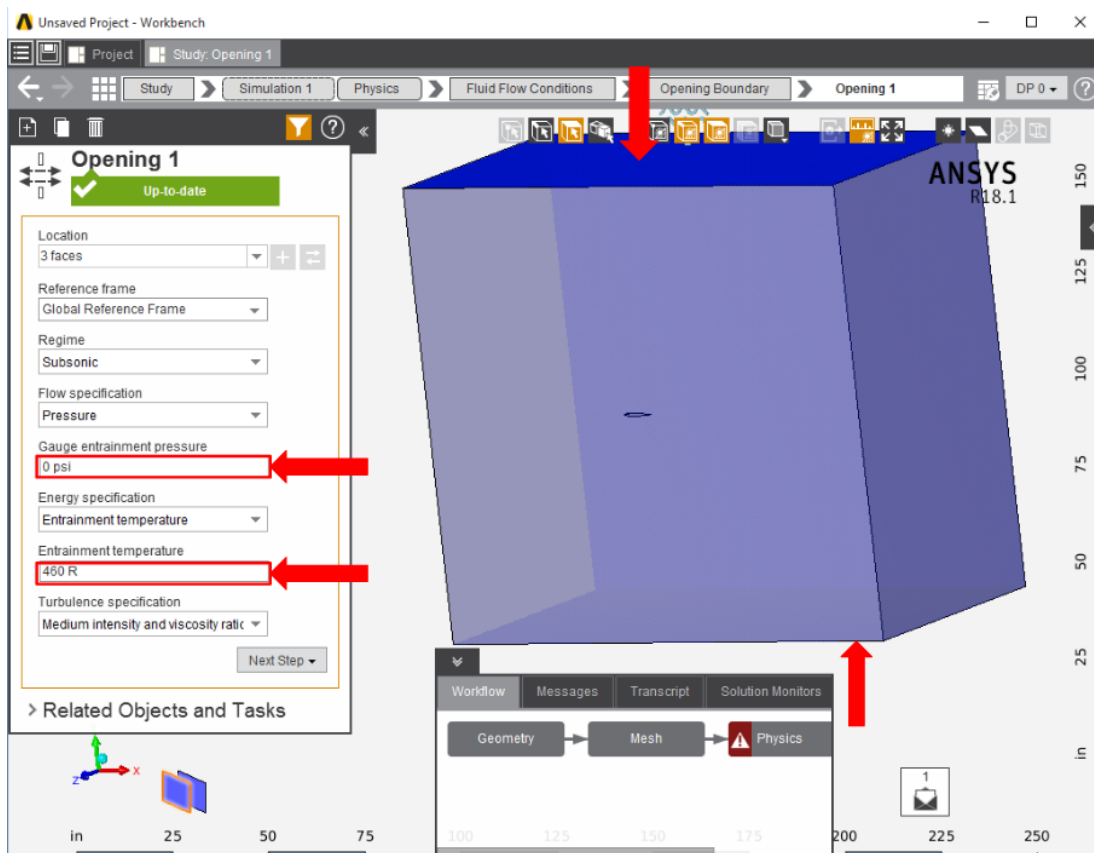
First, the inlet must be defined using the **Fluid Flow Conditions**. In the **Add** drop down menu by **Fluid Flow Conditions**, select **Inlet**. Then, using the face selection tool, define an inlet at the face in front of the leading (rounded) edge of the wing. Enter 11336.56 [in s⁻¹] for the **Velocity magnitude**. The **Total temperature**, as defined in the problem specification, is 460 [R].



In the same **Add** menu, select **Outlet** to define an outlet on the face behind the wing. Change the **Gauge static pressure** to 46 [psi].



Create openings for the sides of the flow volume by selecting **Opening** in the **Add** drop down menu. Select the top, bottom and far side faces of the flow volume. Input 46 [psi] for the **Gauge entrainment pressure** and 460 [R] for the **Entrainment temperature**.



Add a **Symmetry** condition from the **Add** drop down menu to the side of the flow volume that the wing is coincident with.

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 on it.

[Go to Step 5: Solution/Results](#)

[Go to all ANSYS AIM Learning Modules](#)