Compressible Flow Over an Airfoil - Physics Set-Up

Author(s): Sebastian Vecchi, ANSYS Inc.

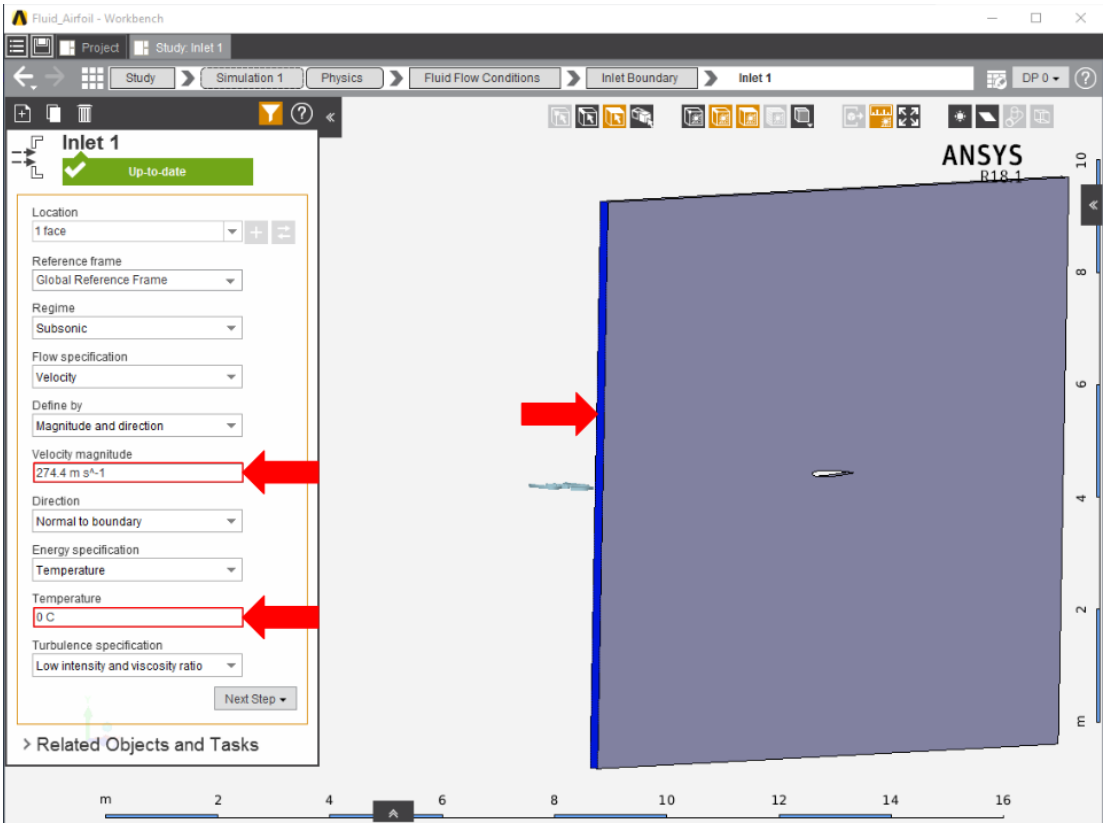
Problem Specification
1. Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Results
6. Verification & Validation

Exercise

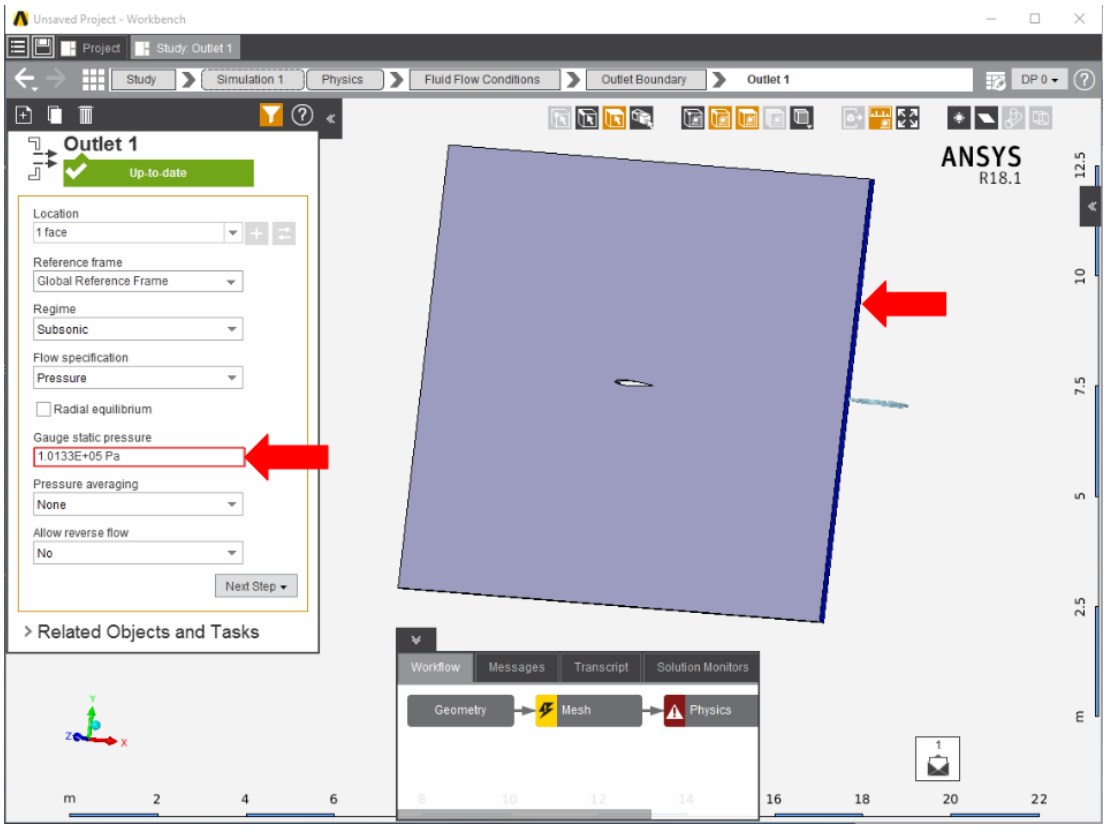
Physics Set-Up

Boundary Conditions / Forces

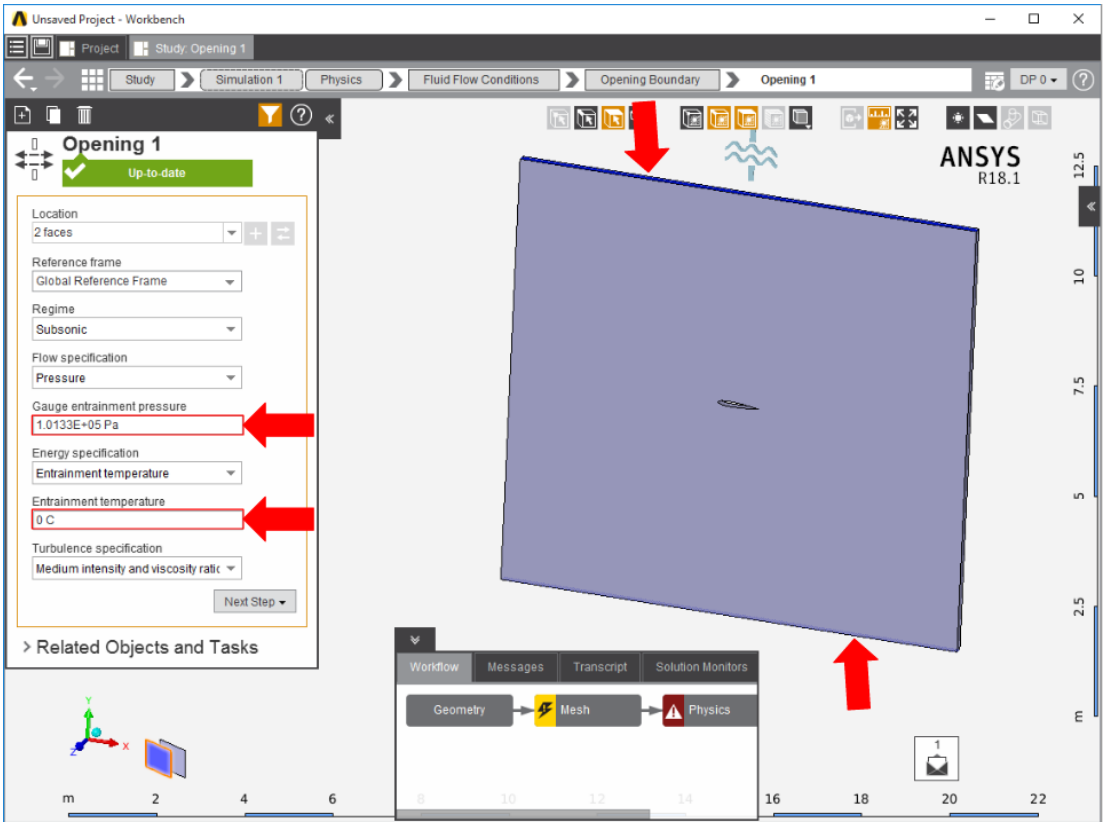
Select the **Physics** task in the **Workflow**. First, the inlet must be defined within the **Fluid Flow Conditions**. In the **Add** drop down menu by **Fluid Flow Conditions**, select **Inlet**. Then, using the **Face selection** tool, define an inlet at the rectangular face by the rounded end of the airfoil. Make sure to input the **Velocity magnitude** as 987.84 [km hr^-1] and a **Temperature** of 0 degrees Celsius.



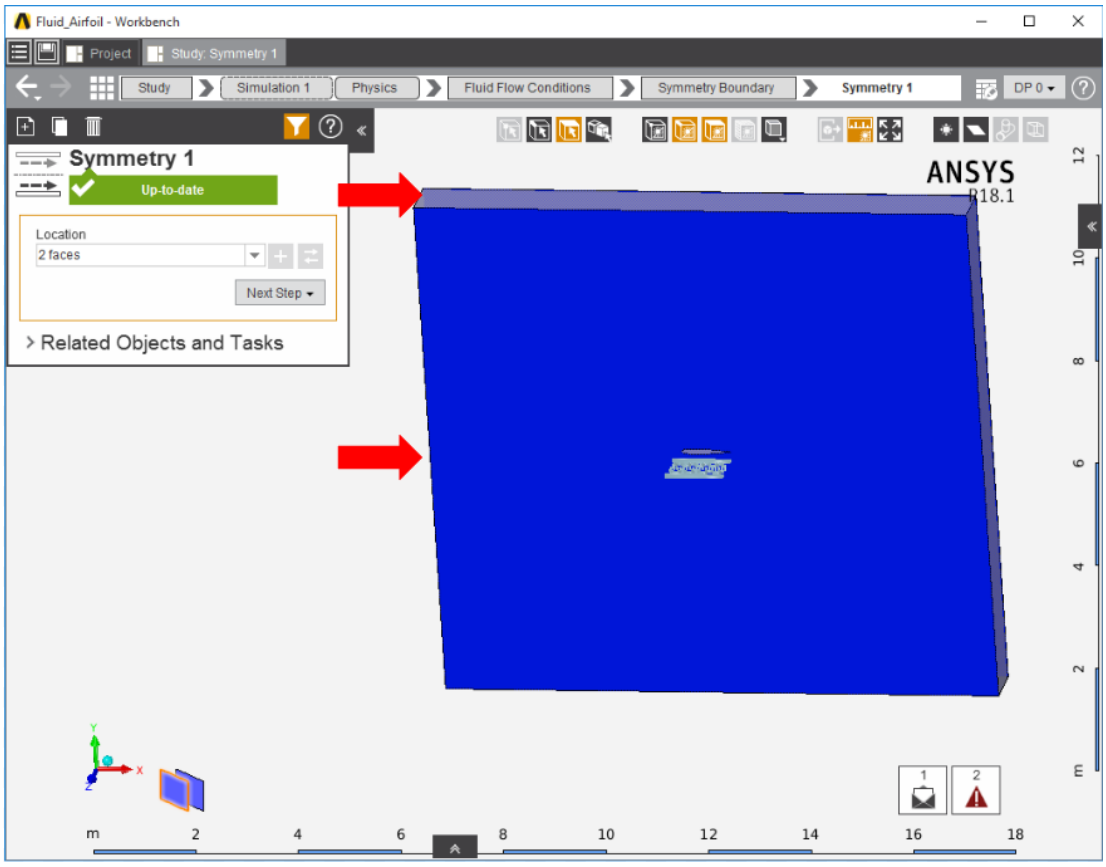
Once the inlet is defined, the outlet is next. In the same **Add** menu, choose **Outlet** to define an outlet downstream of the airfoil. Assign a **Gauge static pressure** of 1 [atm], or 1.0133E+05 [Pa].



Create openings above and below the airfoil by selecting **Opening** in the **Add** drop down menu. Select the top and bottom faces of the flow volume. Input 1 [atm], or 1.0133E+05 [Pa], for the **Gauge entrainment pressure** and 0 C **Entrainment temperature**.

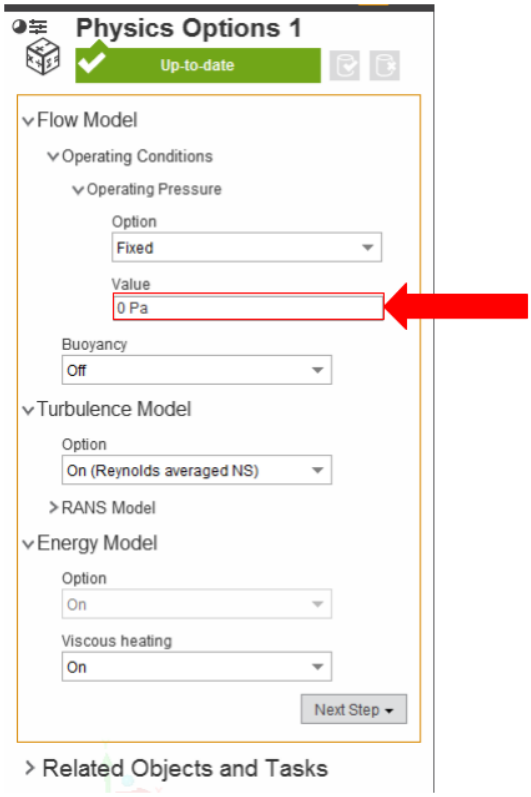


Add a **Symmetry** condition, from the **Add** drop down menu, to the two large sides of the flow volume.



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. Most of the time, AIM will automatically create the walls once the option is selected; AIM selects every face that doesn't already have a constraint on it.

Lastly, the operating pressure must be set to zero. Press on **Physics Options** in the main **Physics** panel, expand **Operating Conditions** under **Flow Model**, and then expand **Operating Pressure**. Change **Value** to 0 [Pa].



Go to Step 5: Results

Go to all ANSYS AIM Learning Modules