ANSYS Steady Flow over a Cylinder - Physics Setup

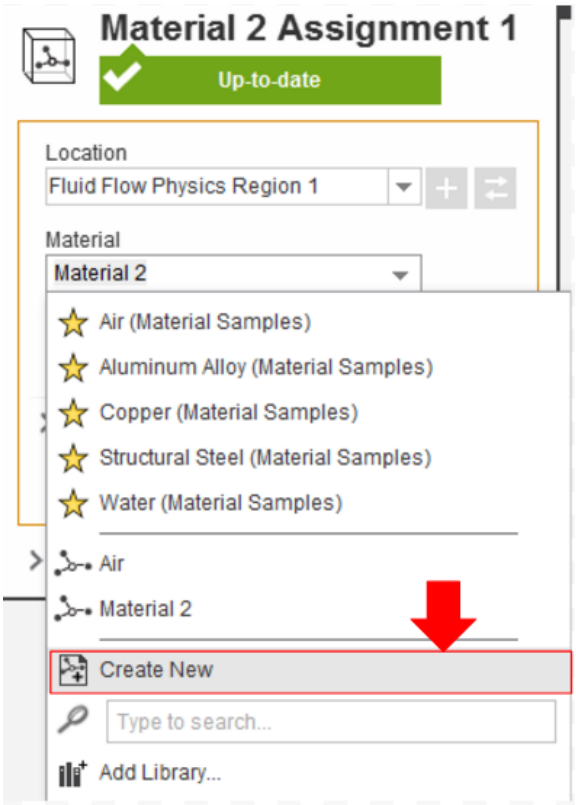
Author(s): Sebastian Vecchi, ANSYS Inc.

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

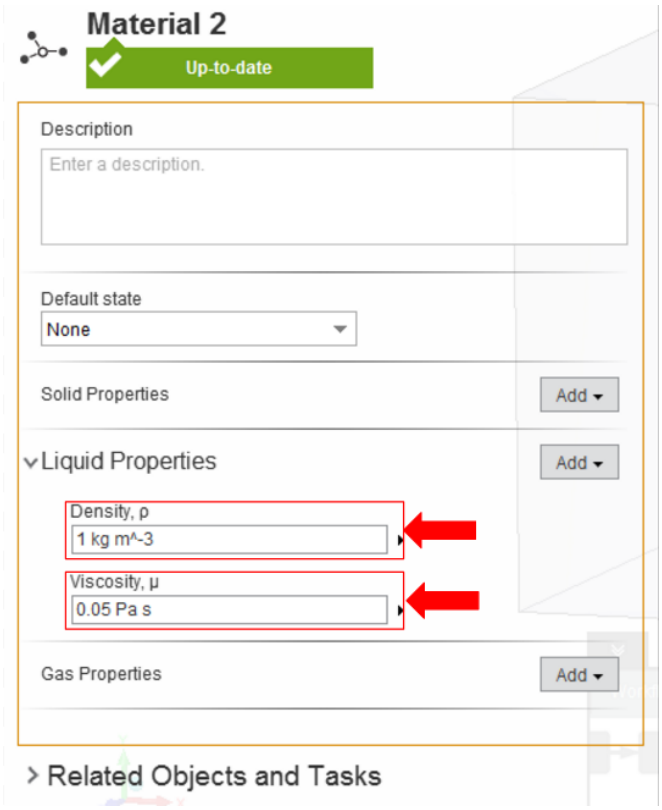
Physics Set-Up

Create New Material

In the problem specification, a density and viscosity are defined for the fluid flow that do not match those of air or water. A new material must be defined with the properties that we wish to have. Select **Material Assignments** and in the **Material** drop down menu choose **Create New**.

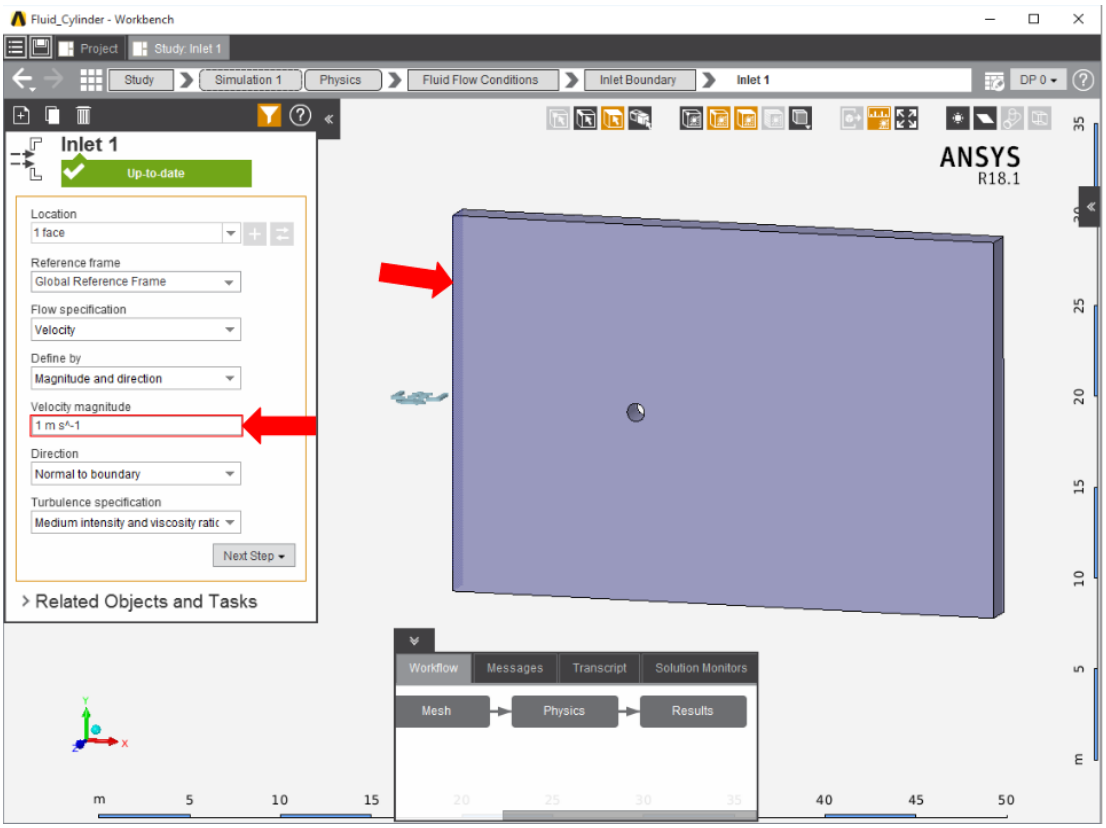


The problem specification did not define whether the fluid was a liquid or gas, but since the density and viscosity were defined, it does not matter. Press **Add** next to either **Liquid** or **Gas Properties**. In the drop down menu, choose **Density** and input 1 [kg m^-3]. Do the same to define the **Viscosity** as 0.05 [kg m^-1 s^-1], or 0.05 [Pa s]

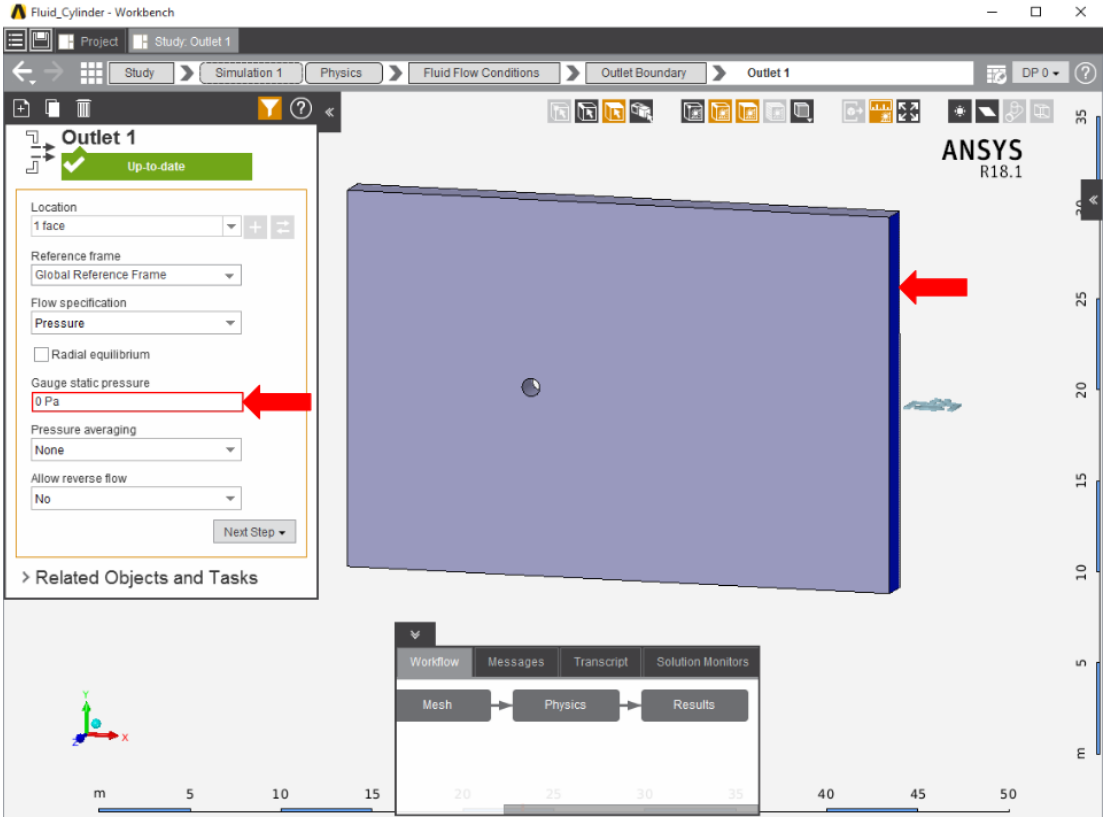


Boundary Conditions / Forces

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 end closest to the cylinder. Make sure to input the **Velocity magnitude** as 1 m/s.

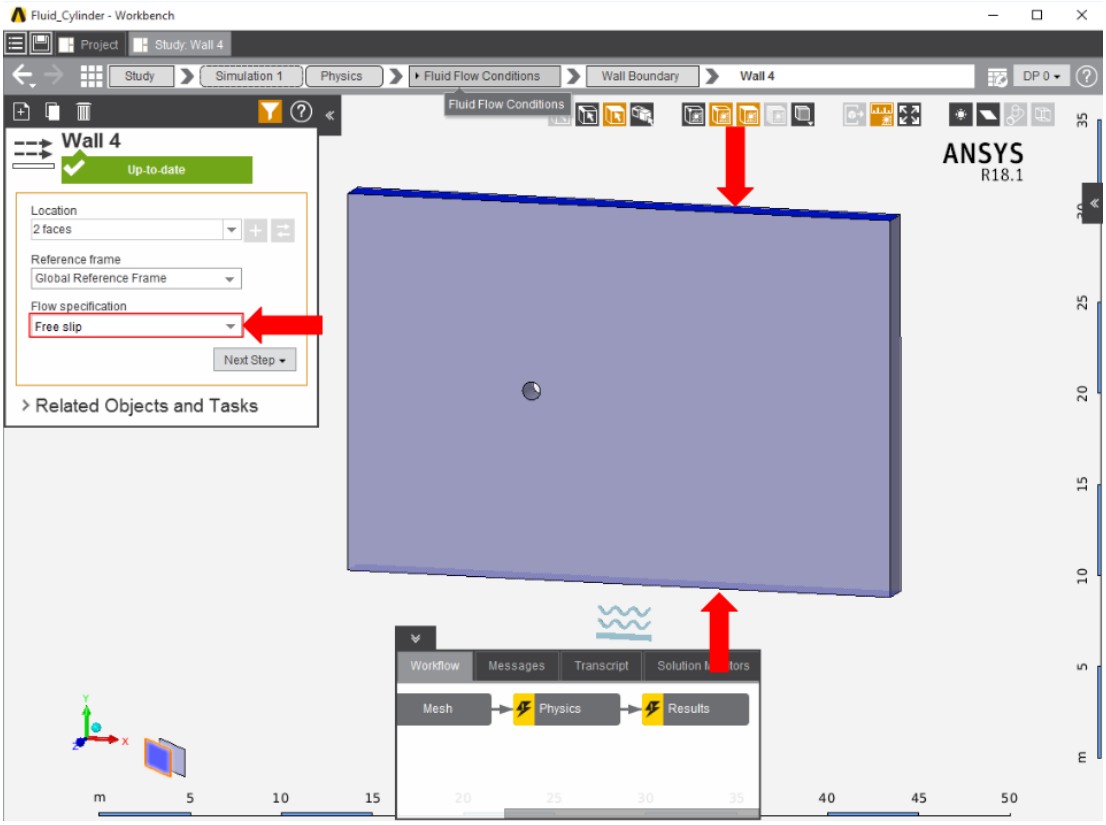


Once the inlet is defined, the outlet is next. In the same menu, use the **Outlet** condition to define an outlet at the end of the enclosure farthest from the cylinder. Assign a **Gauge static pressure** of 0 psi.



Add a **Symmetry** condition from the **Add** drop down menu to the side faces of the flow volume.

A free slip wall condition must be created in order to channel the air inside of the enclosure, much like one would find in a wind tunnel. Select the **Wall** cond ition from the **Add** drop down menu and apply it to the top and bottom of the enclosure, then change the **Option** under **Flow Specification** to **Free slip**.



Next, a **Wall** condition must be added to all surfaces that are not already defined. 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.

Go to Step 5: Results

Go to all ANSYS AIM Learning Modules