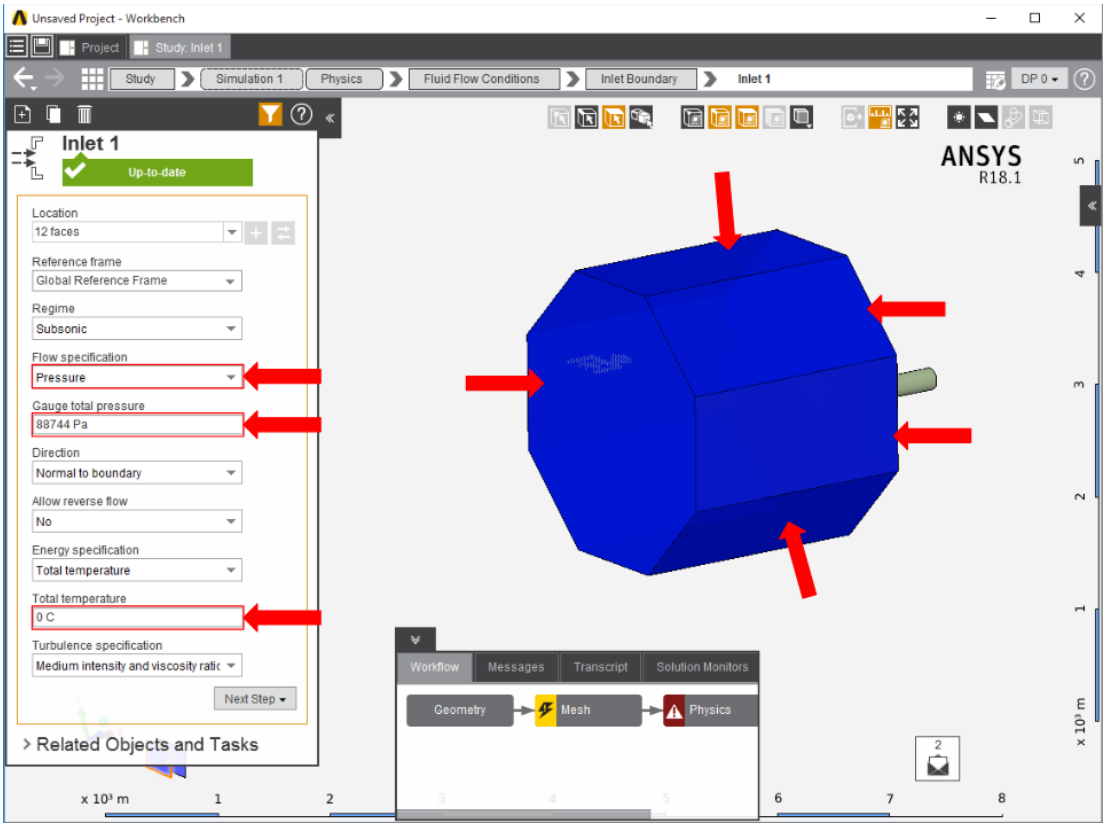
ANSYS Flow in a S-Duct - Physics Setup

Author(s): Sebastian Vecchi, ANSYS Inc.

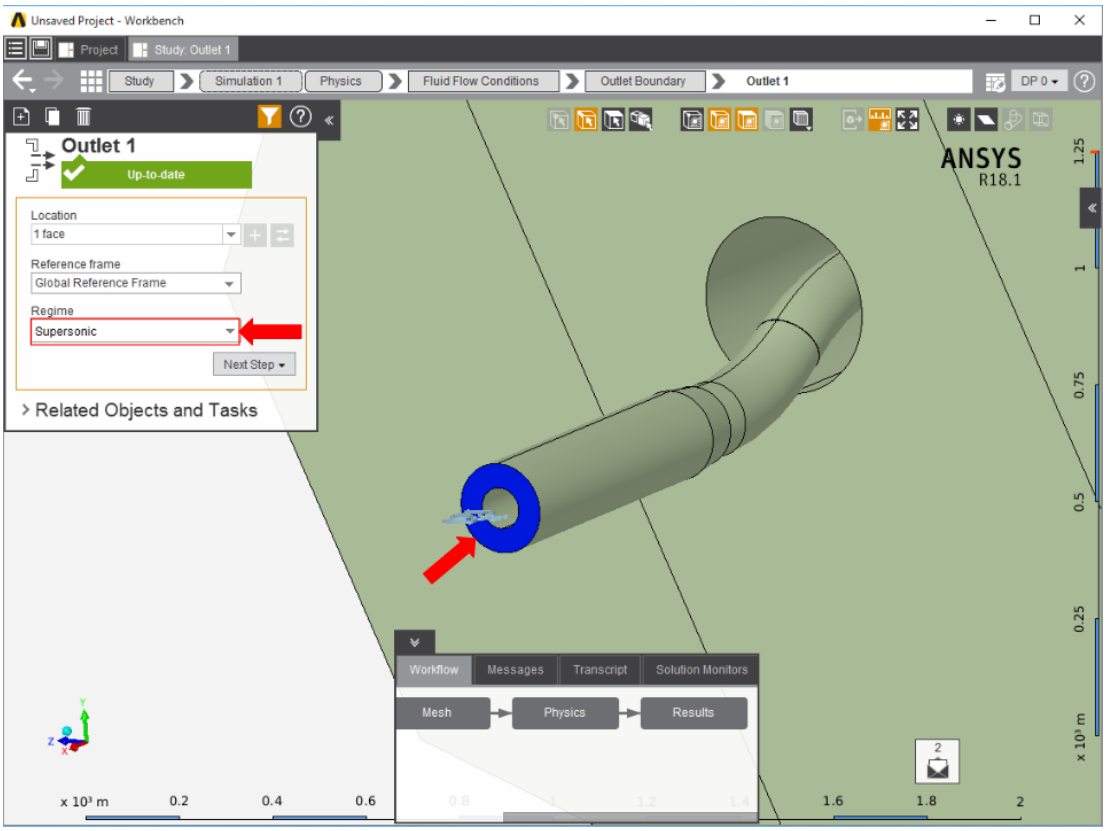
Physics Setup

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 all 12 flat faces of the large volume, as shown below. Change **Flow specification** to **Pressure**, input the **Gauge total pressure** as 88744 [Pa] and 0 [C] as **Total temperature**.



Once the inlet is defined, the outlet is next. In the same **Add** menu, choose **Outlet** to define an outlet at the small, annular end of the duct. Change **Regime** to **Supersonic**.



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: Results

Go to all ANSYS AIM Learning Modules