# **ANSYS Fluid Flow over a Bluff Body - Physics Setup**

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

1. Pre-Analysis & Start-Up

2. Geometry

3. Mesh

4. Physics Setup

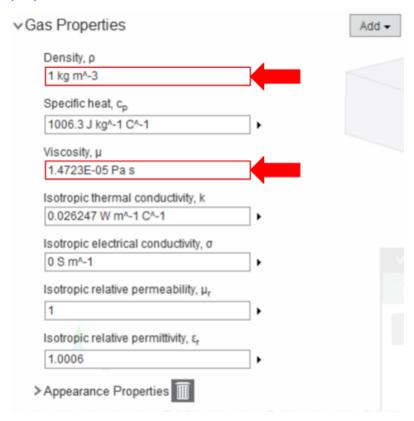
5. Results

6. Verification & Validation

## Physics Set-Up

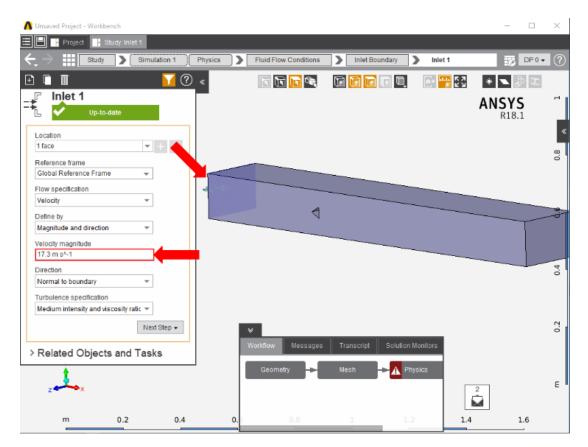
#### Material

The dynamic viscosity must be modified in order to reach the target reynolds number of 47,000 at the inlet. Select **Material Assignments** in the **Physics** panel and select **Air** near the bottom of the panel. Under the **Gas Properties** section, change the **Density** to 1 [kg m^-3] and the **Viscosity** to 1.47234e-5 [Pa s].

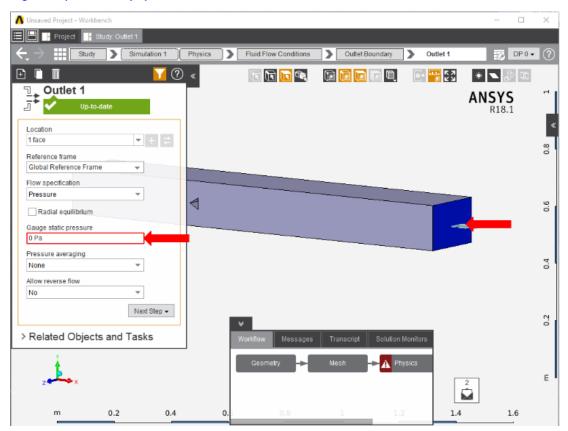


## **Boundary Conditions / Forces**

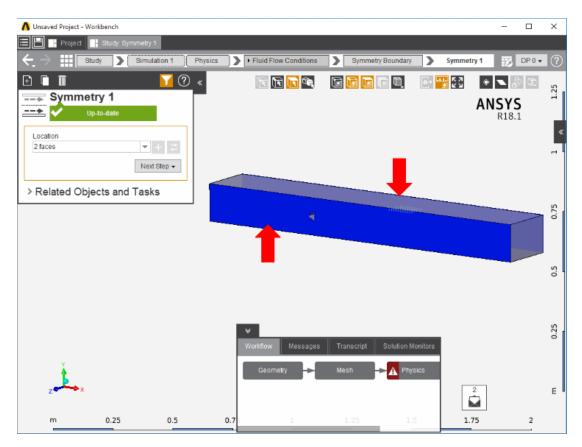
First, the inlet must be defined using the **Fluid Flow Conditions**. In the **Add** drop down menu by **Fluid Flow Conditions**, select Inlet. Then, using the face selection tool, define the inlet as the end face of the volume closest to the triangular flameholder. Make sure to input the **Velocity magnitude** as 17.3 [m s^1].



Once the inlet is defined, the outlet is next. In the same drop down menu, choose **Outlet** to define an outlet at the other end of the flow volume. Assign a **G** auge static pressure of 0 [Pa].



Add a **Symmetry** condition from the **Add** drop down menu to the front and rear 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. Al M 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