

# 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

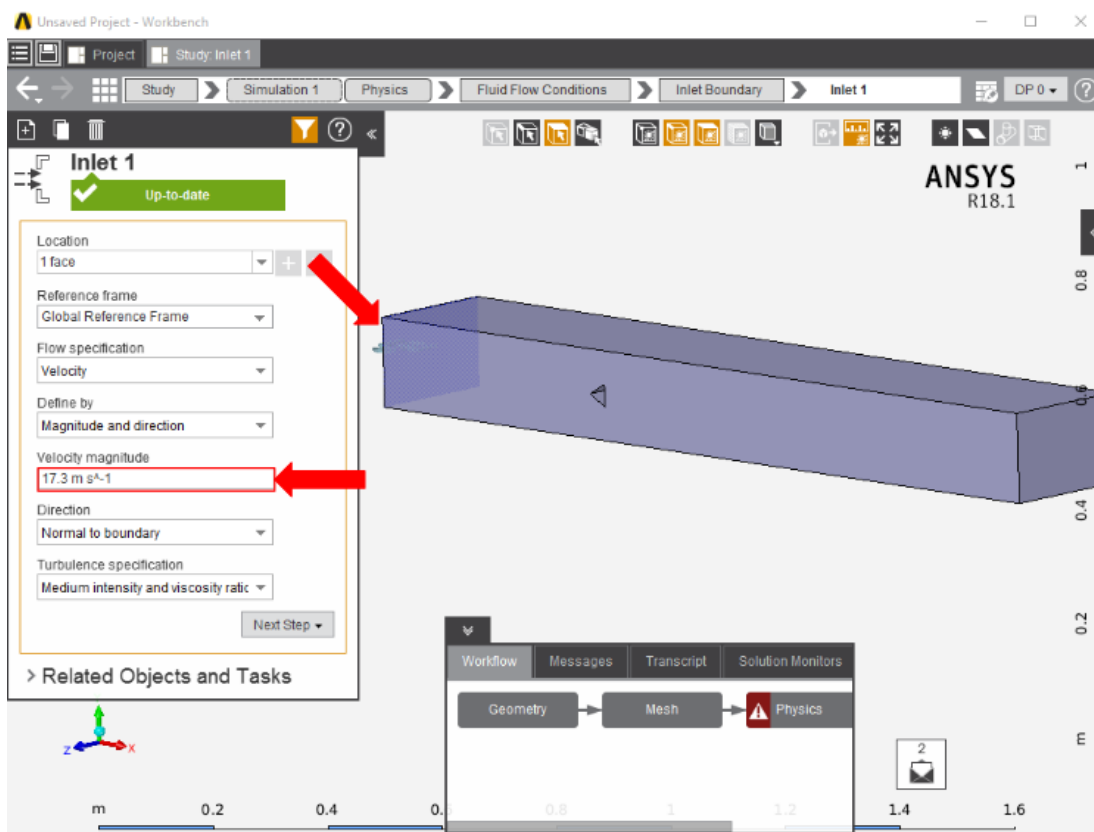
### Material

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<sup>-3</sup>] and the **Viscosity** to 1.47234e-5 [Pa s].

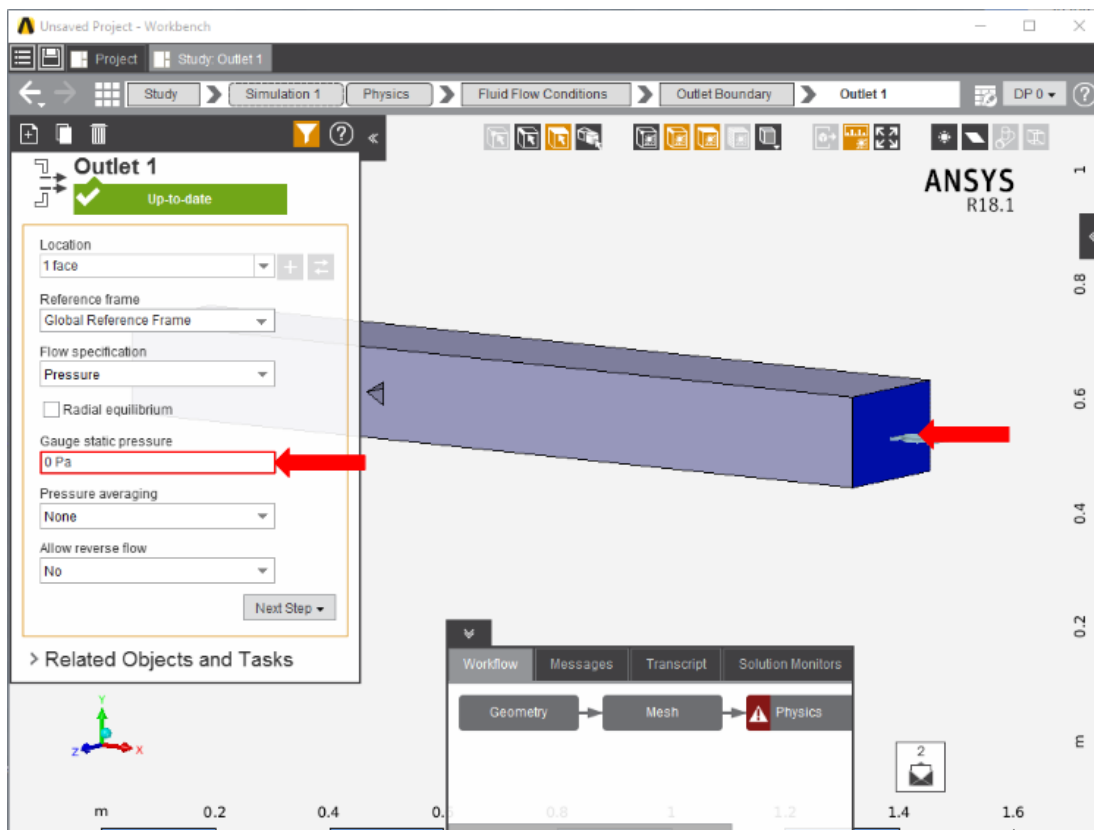
The screenshot shows the 'Gas Properties' panel in ANSYS. The 'Density,  $\rho$ ' field is set to '1 kg m<sup>-3</sup>' and the 'Viscosity,  $\mu$ ' field is set to '1.4723E-05 Pa s'. Both fields are highlighted with red boxes and red arrows. Other fields include 'Specific heat,  $c_p$ ' (1006.3 J kg<sup>-1</sup> C<sup>-1</sup>), 'Isotropic thermal conductivity,  $k$ ' (0.026247 W m<sup>-1</sup> C<sup>-1</sup>), 'Isotropic electrical conductivity,  $\sigma$ ' (0 S m<sup>-1</sup>), 'Isotropic relative permeability,  $\mu_r$ ' (1), and 'Isotropic relative permittivity,  $\epsilon_r$ ' (1.0006). An 'Add' button is at the top right, and 'Appearance Properties' with a trash icon is at the bottom left.

### Boundary Conditions / Forces

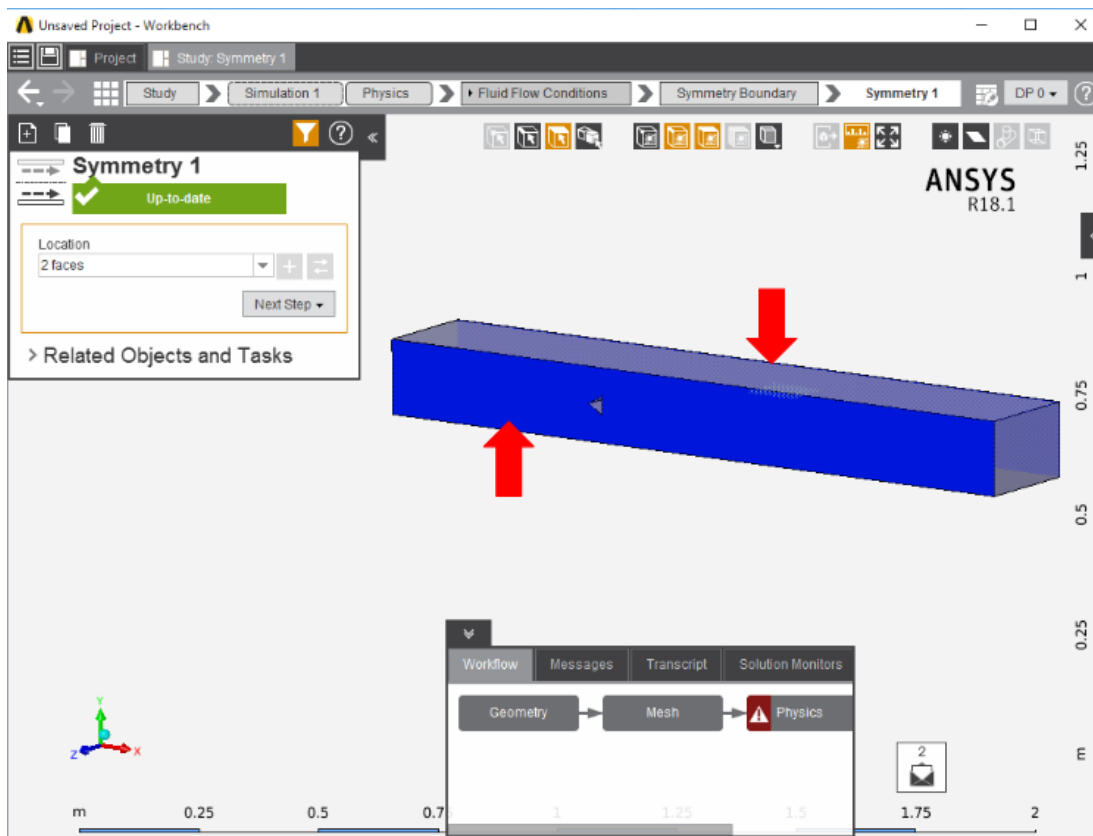
First, the inlet must be defined using the **Fluid Flow Conditions**. In the **Add** drop down menu by **Fluid Flow Conditions**, select Inlet. Then, using the face selection tool, define the inlet as the end face of the volume closest to the triangular flameholder. Make sure to input the **Velocity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



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 **Gauge 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. **Alt 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](#)