

# ANSYS Flow Through U-Duct - Physics Set-Up

Author(s): Sebastian Vecchi, ANSYS Inc.

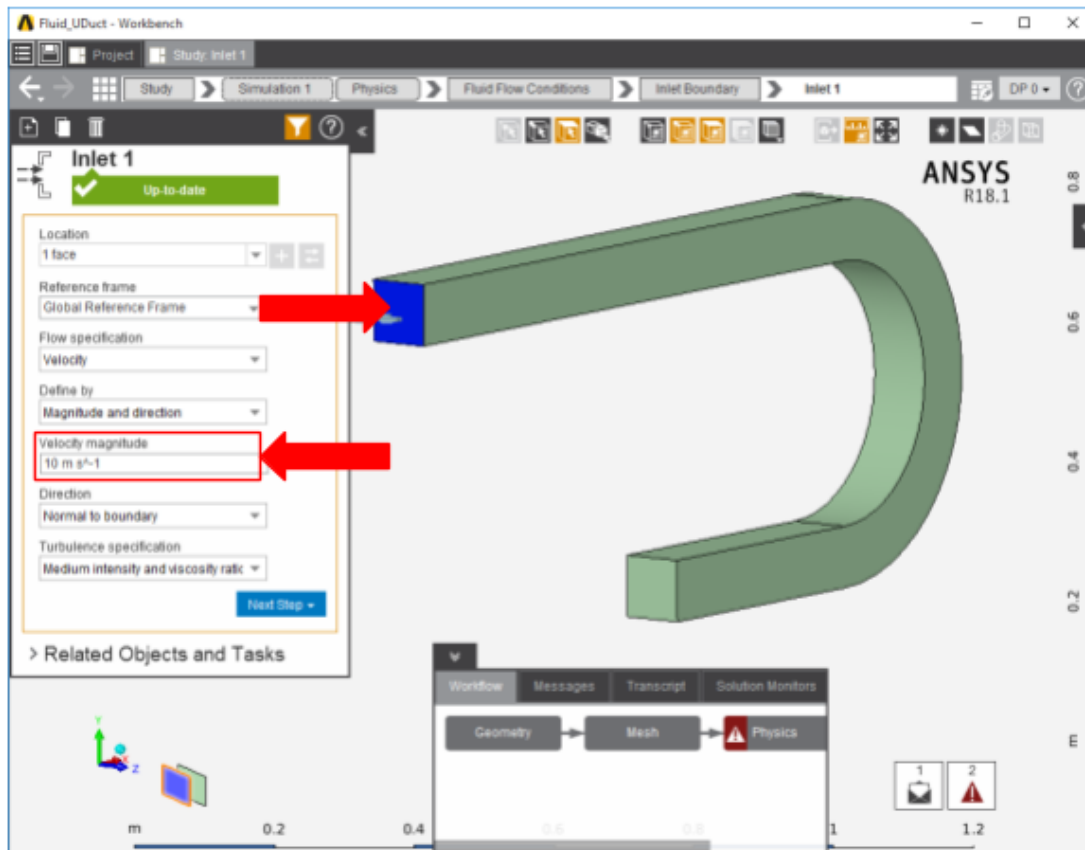
## Problem Specification

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

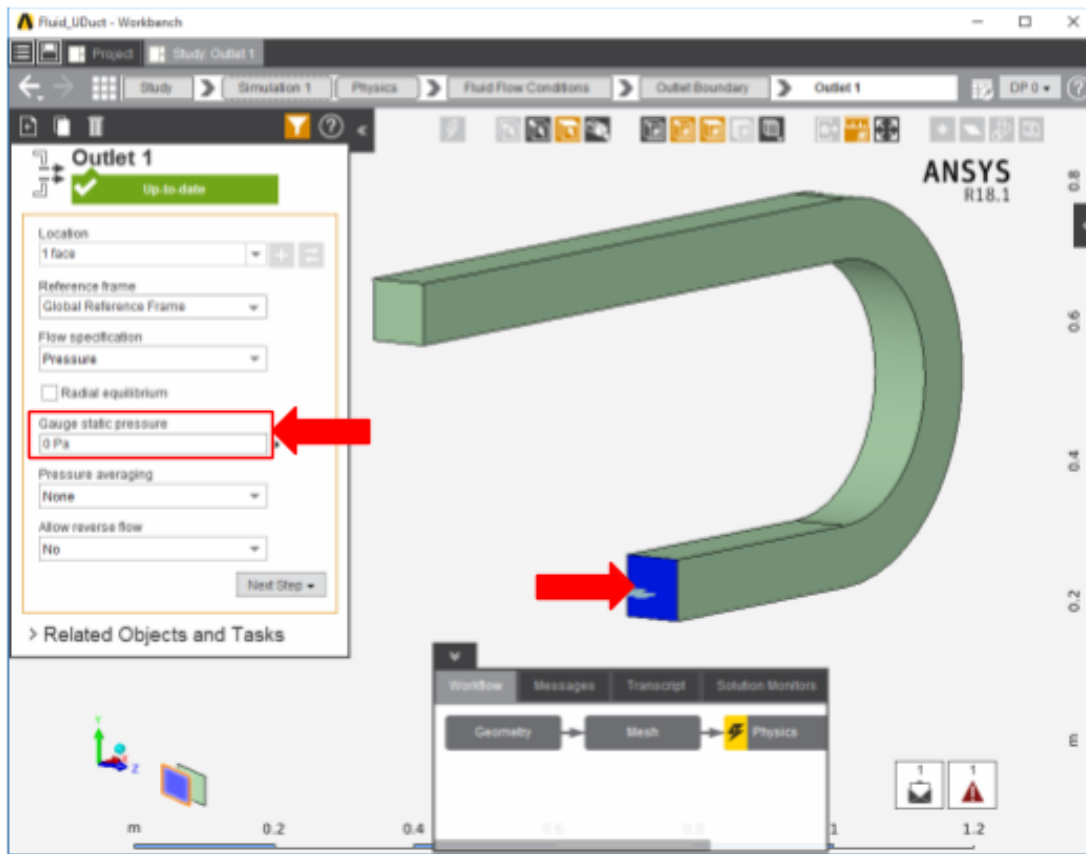
## Physics Set-Up

### 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 low end of the duct. Make sure to input the **velocity magnitude** as 10 [m s<sup>-1</sup>].



Once the inlet is defined, the outlet is next. In the same menu, use the outlet tool to define an outlet at the small end of the duct. Assign a **Gauge Static Pressure** of 0 psi.



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 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](#)