

ANSYS Steady Flow over a Cylinder - Physics Setup

Author(s): Sebastian Vecchi, ANSYS Inc.

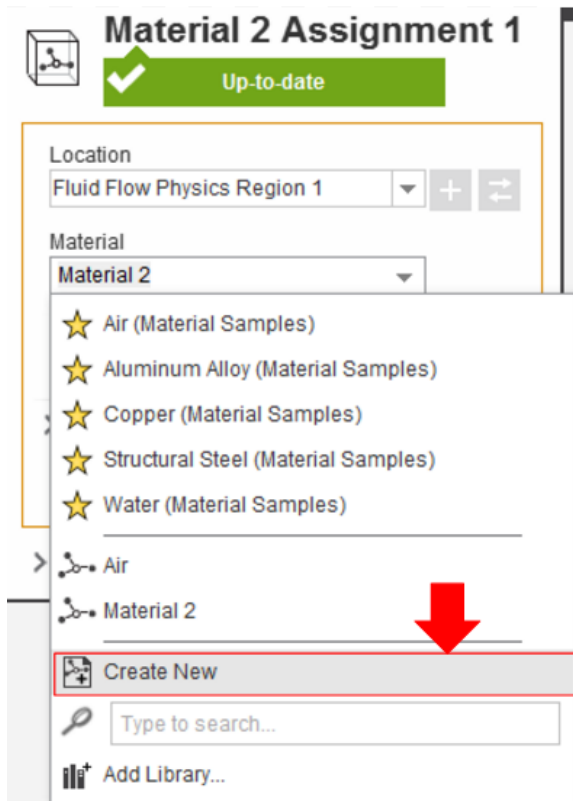
Problem Specification

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 **Ad** 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]

Material 2

Up-to-date

Description

Enter a description.

Default state

None

Solid Properties

Add

▼ Liquid Properties

Add

Density, ρ

1 kg m⁻³

Viscosity, μ

0.05 Pa s

Gas Properties

Add

> Related Objects and Tasks

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.

Fluid_Cylinder - Workbench

Study: Inlet 1

Study Simulation 1 Physics Fluid Flow Conditions Inlet Boundary Inlet 1 DP 0 ?

Inlet 1

Up-to-date

Location

1 face

Reference frame

Global Reference Frame

Flow specification

Velocity

Define by

Magnitude and direction

Velocity magnitude

1 m s⁻¹

Direction

Normal to boundary

Turbulence specification

Medium intensity and viscosity ratio

Next Step

> Related Objects and Tasks

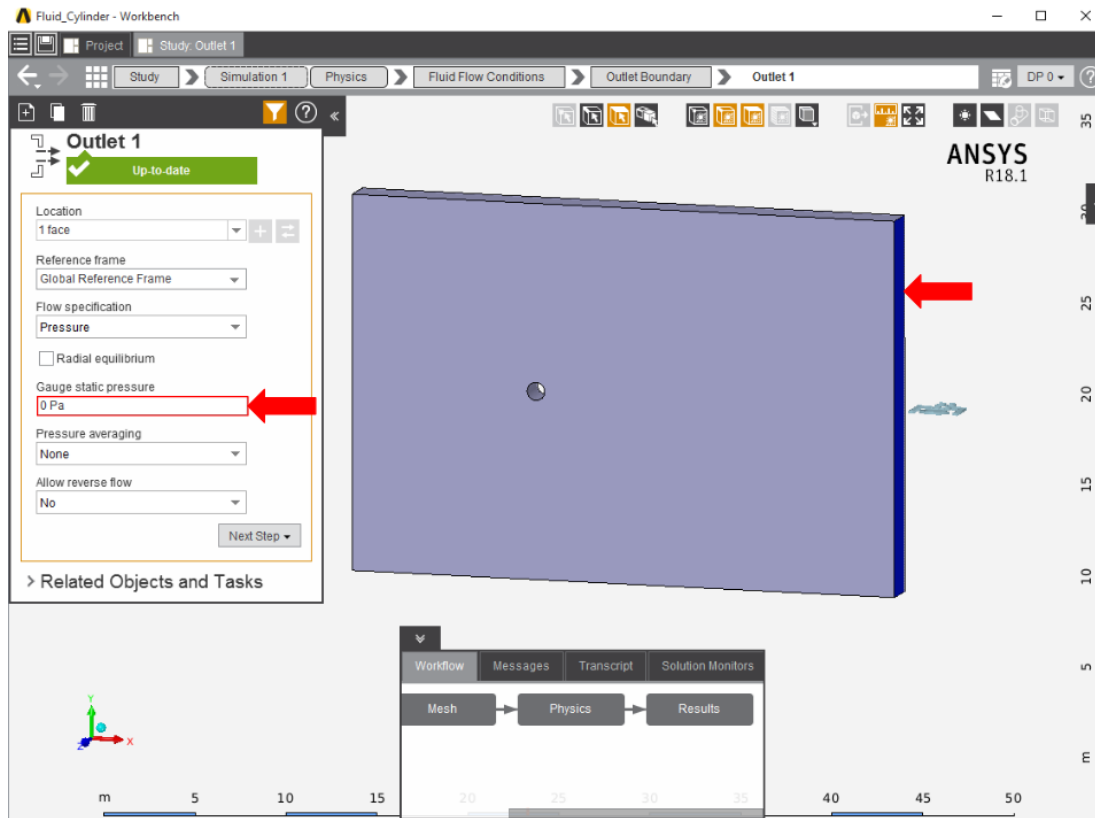
ANSYS R18.1

Workflow Messages Transcript Solution Monitors

Mesh Physics Results

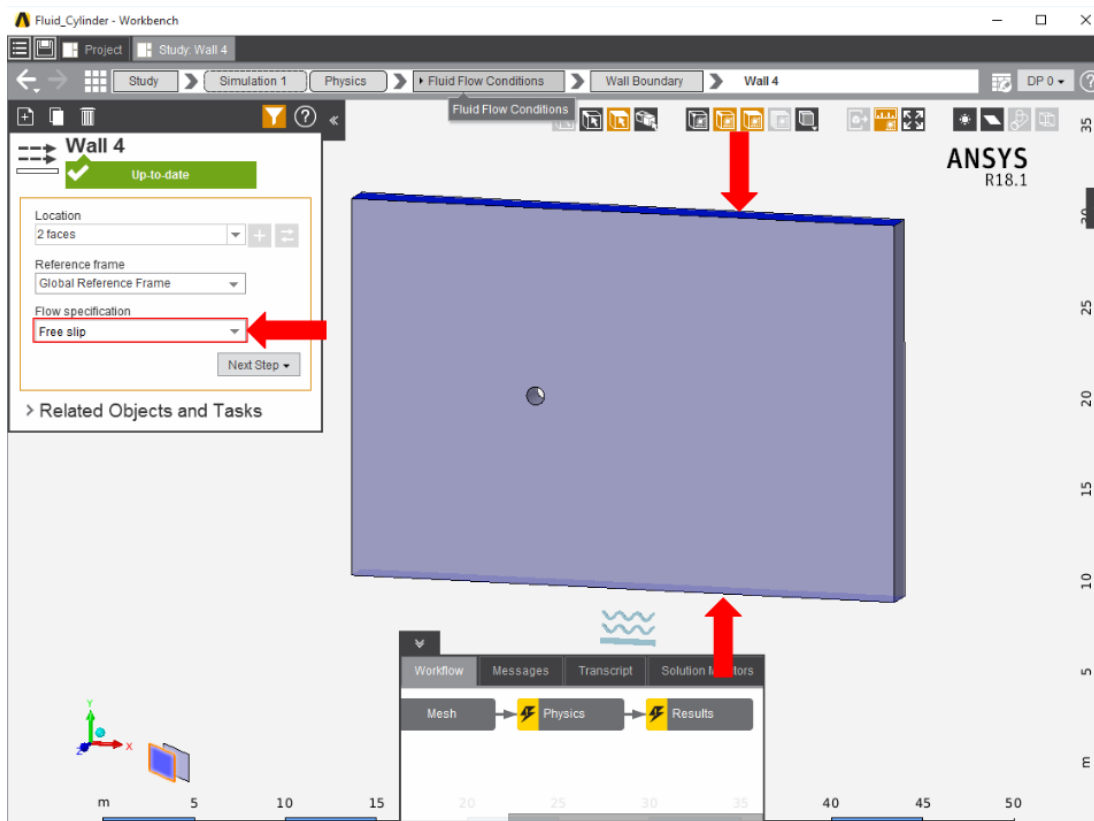
m 5 10 15 20 25 30 35 40 45 50

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** condition 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](#)