

# Compressible Flow in a Nozzle - Physics Set-Up

Author(s): Sebastian Vecchi, ANSYS Inc.

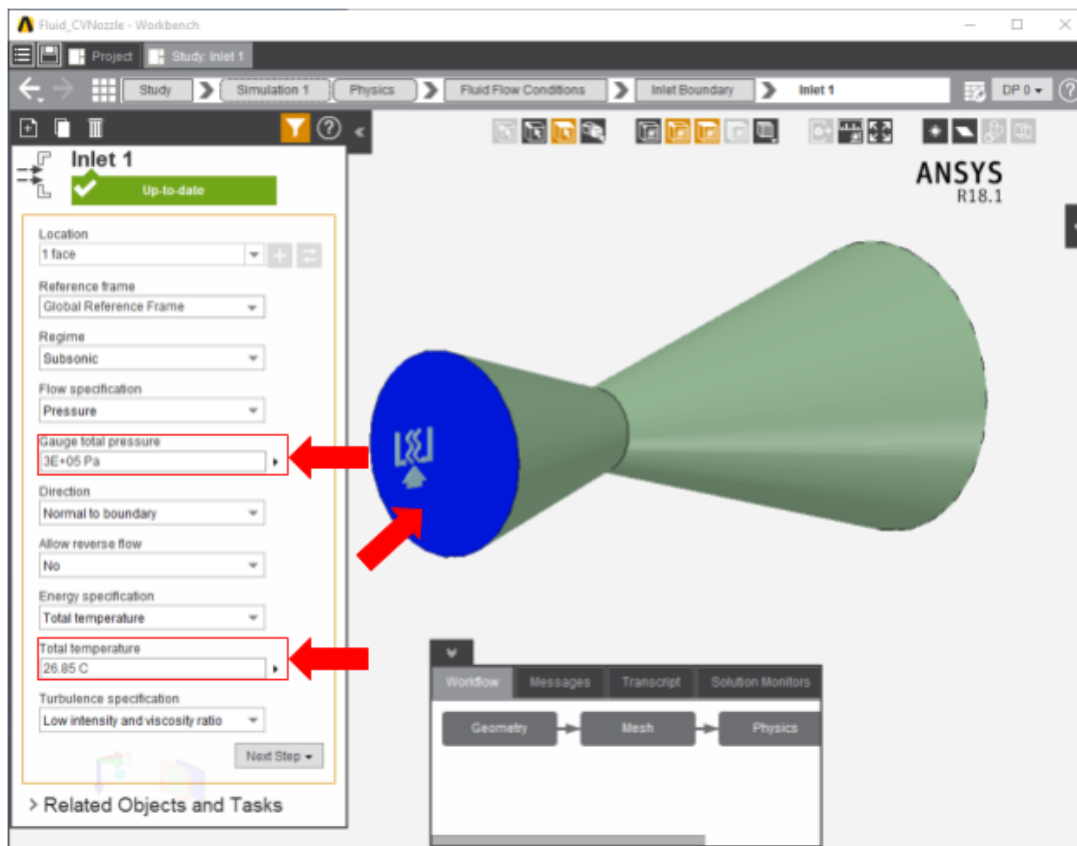
## Problem Specification

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

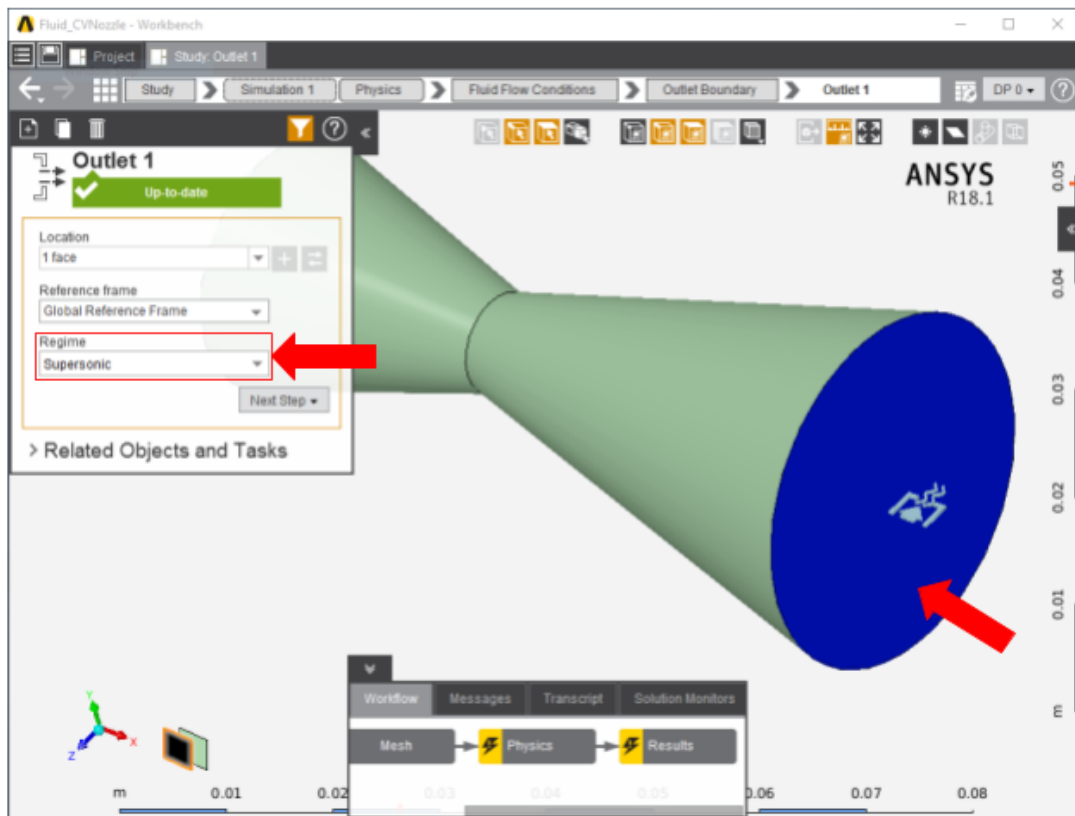
## Physics Set-Up

### Boundary Conditions / Forces

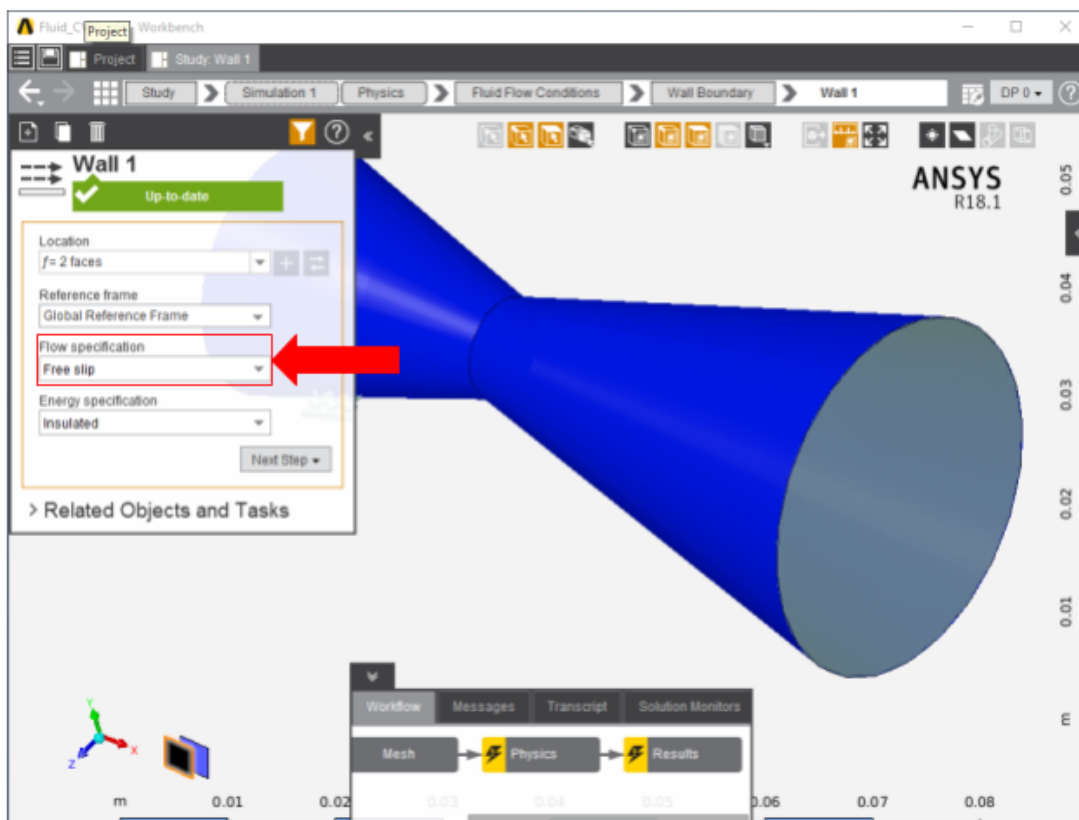
Select the **Physics** task in the **Workflow**. First, the inlet must be defined in the **Fluid Flow Conditions**. In the **Add** drop down menu by **Fluid Flow Conditions**, select **Inlet**. Then, using the **Face selection** tool, define an inlet at the small end of the nozzle. Change **Flow specification** to **Pressure**, input 3 [bar] as the **Gauge total pressure**, and 300 [K] as the **Total temperature**.



Once the inlet is defined, the outlet is next. In the same **Add** menu, define an **Outlet** at the big end of the duct. Change the **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 the wall faces once the option is selected; AIM selects every face that doesn't already have a constraint on it. [Change the Flow specification to Free slip.](#)



**[Go to Step 5: Results](#)**

[Go to all ANSYS AIM Learning Modules](#)