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

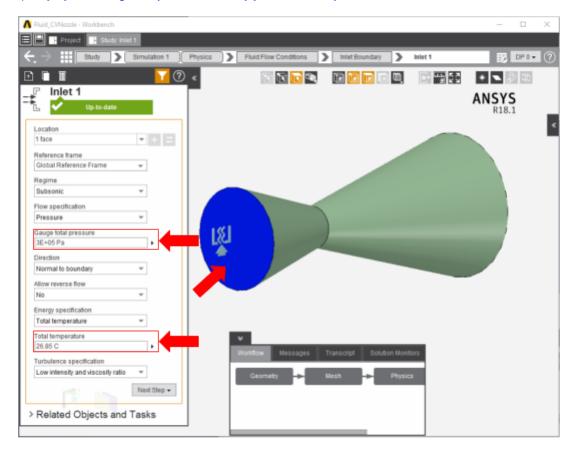
5. Results

6. Verification & Validation

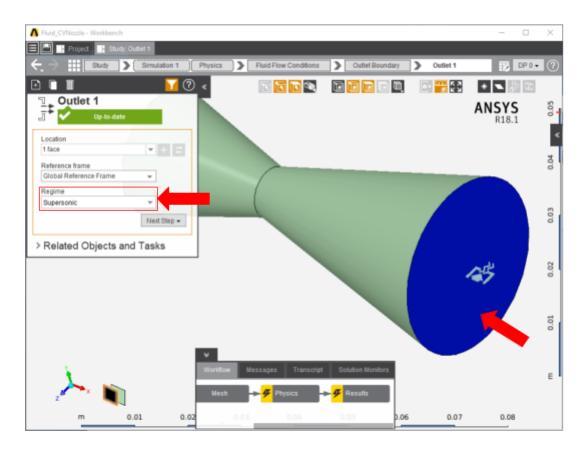
Physics Set-Up

Boundary Conditions / Forces

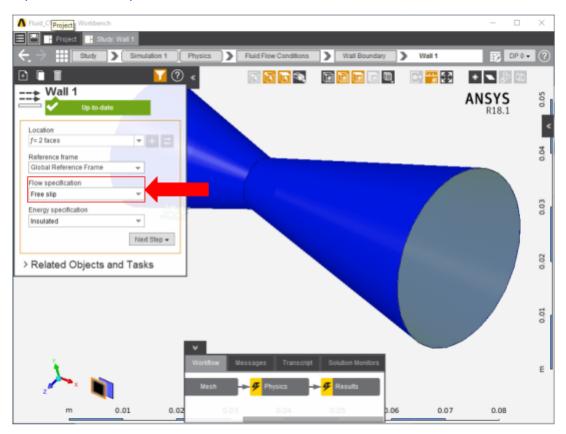
Select the Physics task in the Workflow. First, the inlet must be defined in the Fluid Flow Conditions. In the Add drop down menu by Fluid Flow Conditions, select Inlet. Then, using the Face selection tool, define an inlet at the small end of the nozzle. Change Flow specification to Pressure, input 3 [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