

Compressible Flow in a Nozzle - Geometry

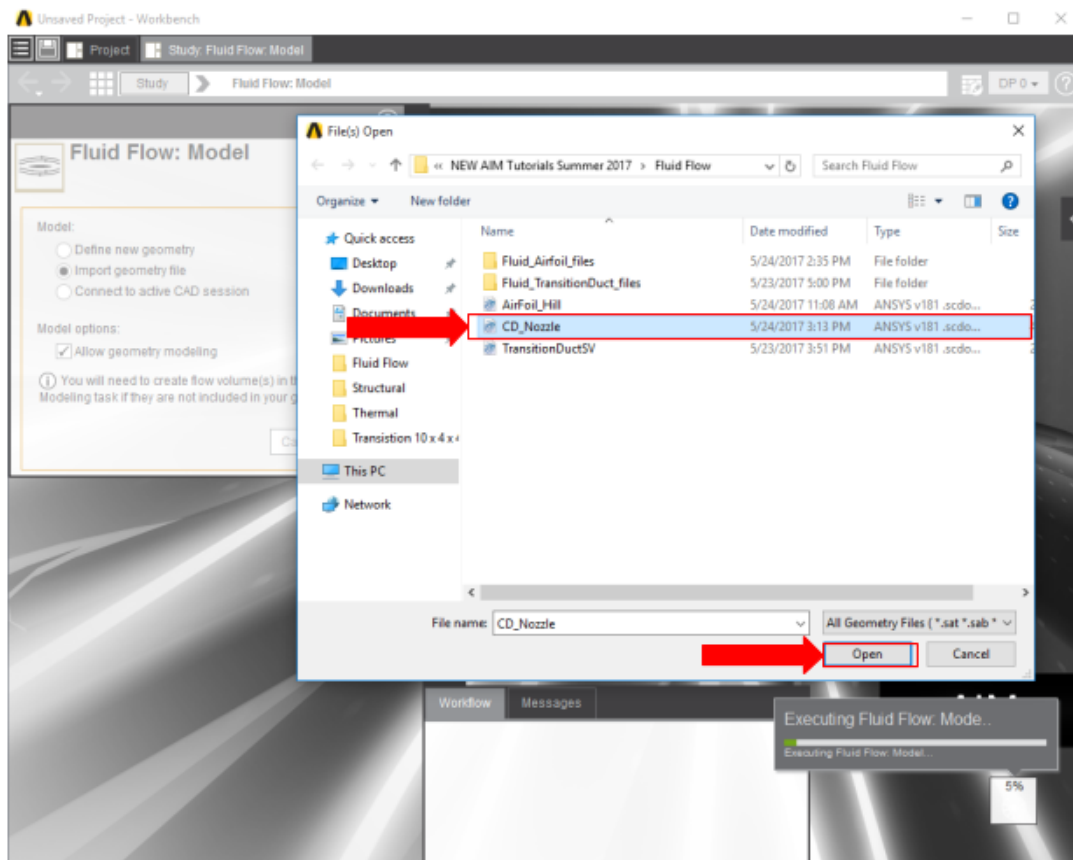
Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

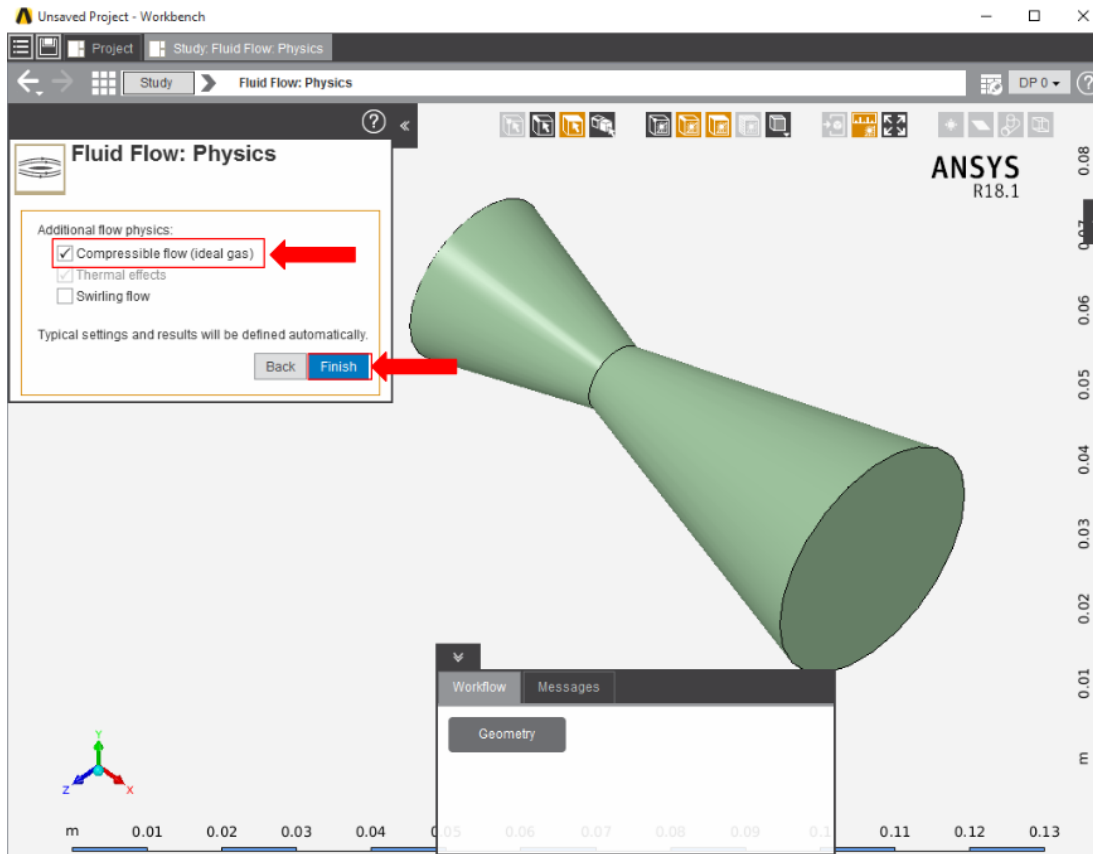
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Results
6. Verification & Validation

Import Geometry

Select and open the geometry file, which can be downloaded from [here](#).



Once successfully imported, enable the **Compressible flow (ideal gas)** option and press **Finish**.



[Go to Step 3: Mesh](#)

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