

ANSYS Flow in a S-Duct - Geometry

Author(s): Sebastian Vecchi, ANSYS Inc.

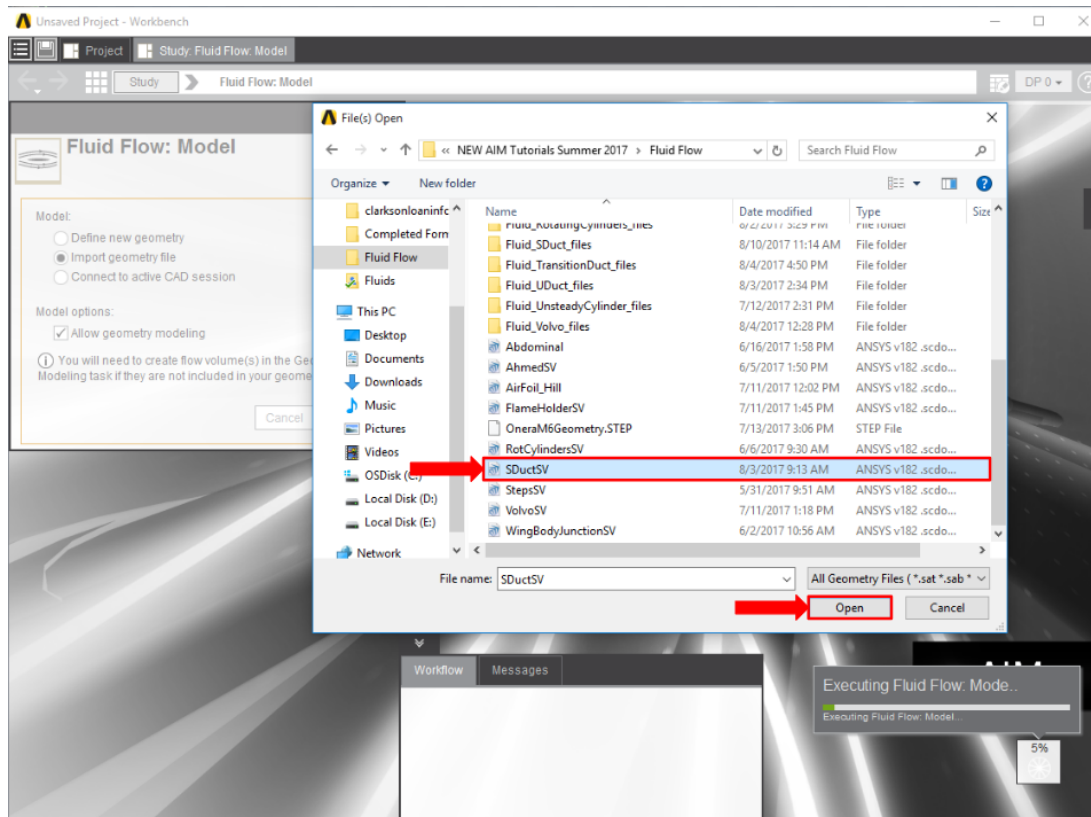
Problem Specification

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

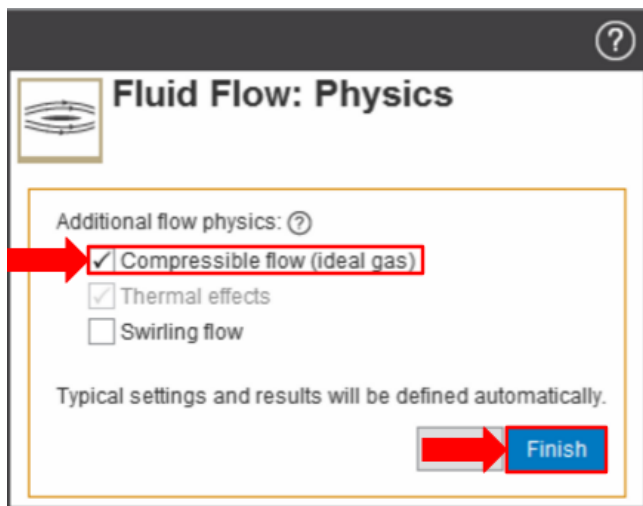
Geometry

Import Geometry

Download [this geometry file](#). Select and open the file.



Once successfully imported, press **Finish**.



[Go to Step 3: Mesh](#)

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