

ANSYS Flow through U-Duct - Geometry

Author(s): Sebastian Vecchi, ANSYS Inc.

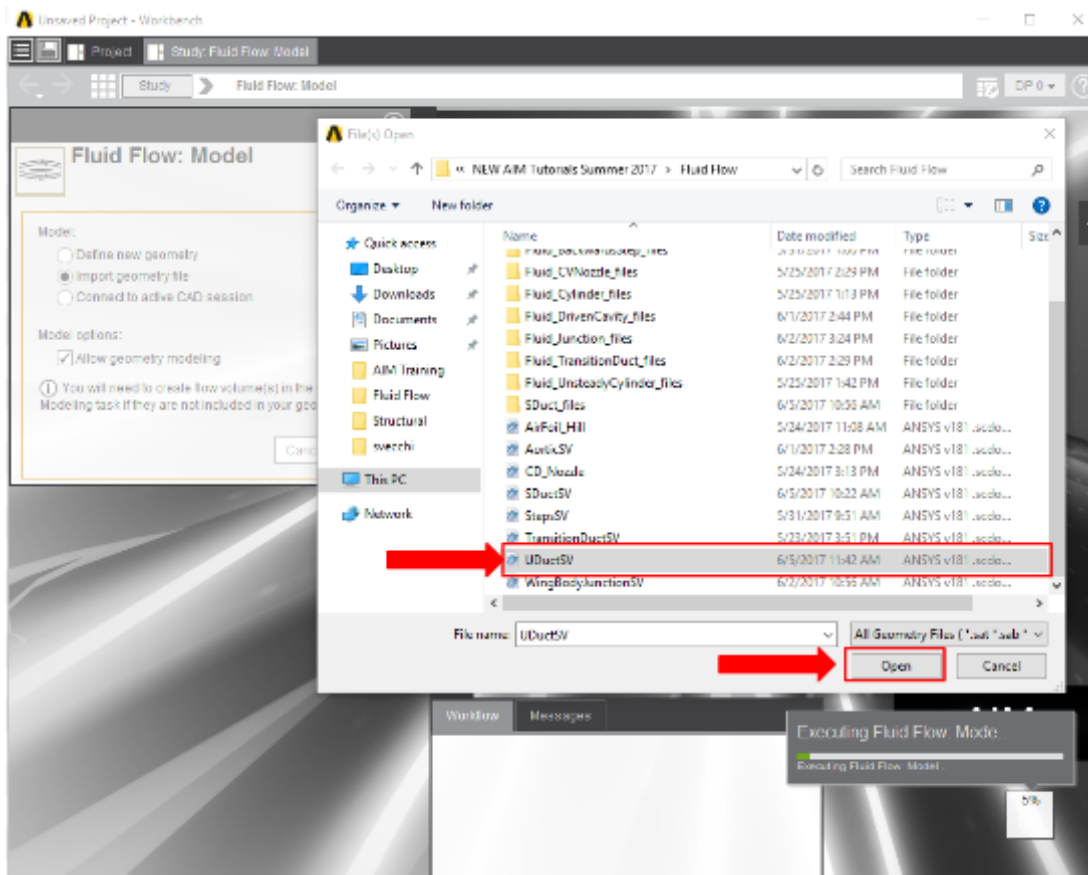
Problem Specification

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

Geometry

Import Geometry

Download the geometry file [here](#). Select and open the appropriate model file.



Once successfully imported press **Finish**.

Go to Step 3: Mesh

Go to all ANSYS AIM Learning Modules