# ANSYS Flow through U-Duct - Geometry

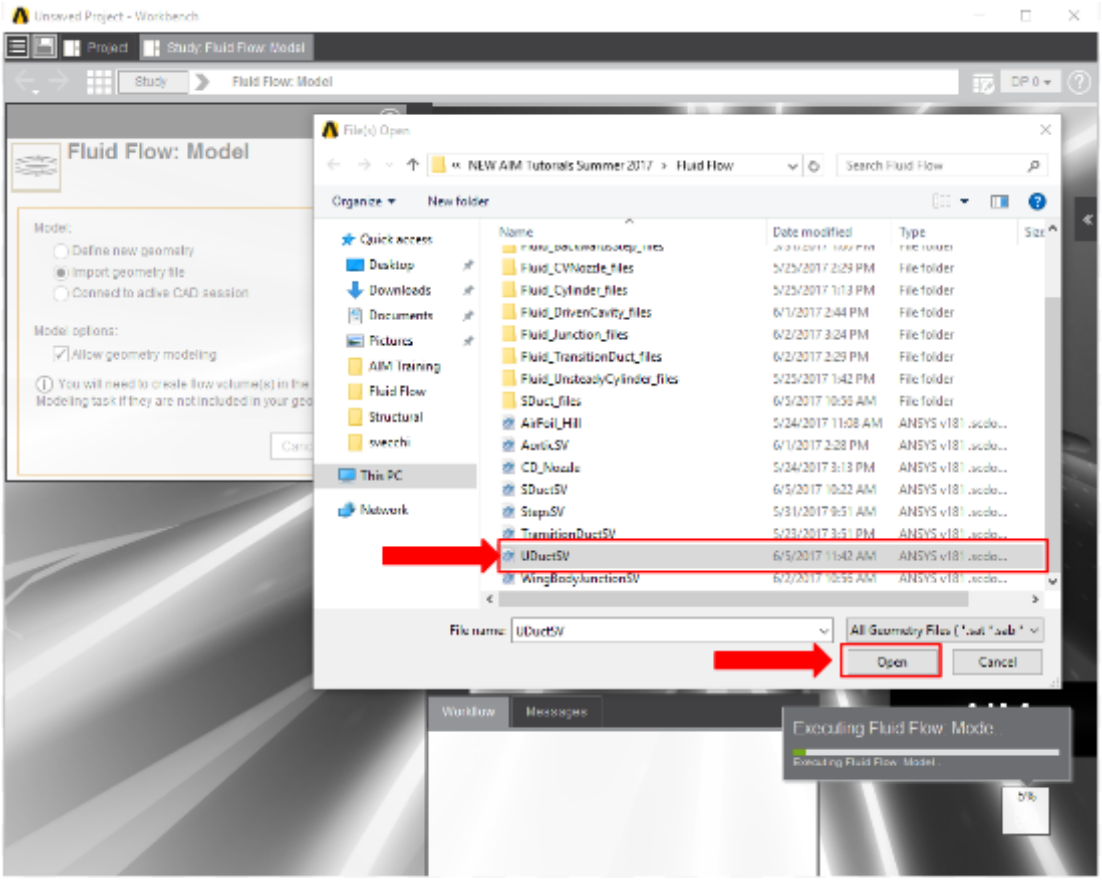
Author(s): Sebastian Vecchi, ANSYS Inc.

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