Transition Duct - Geometry

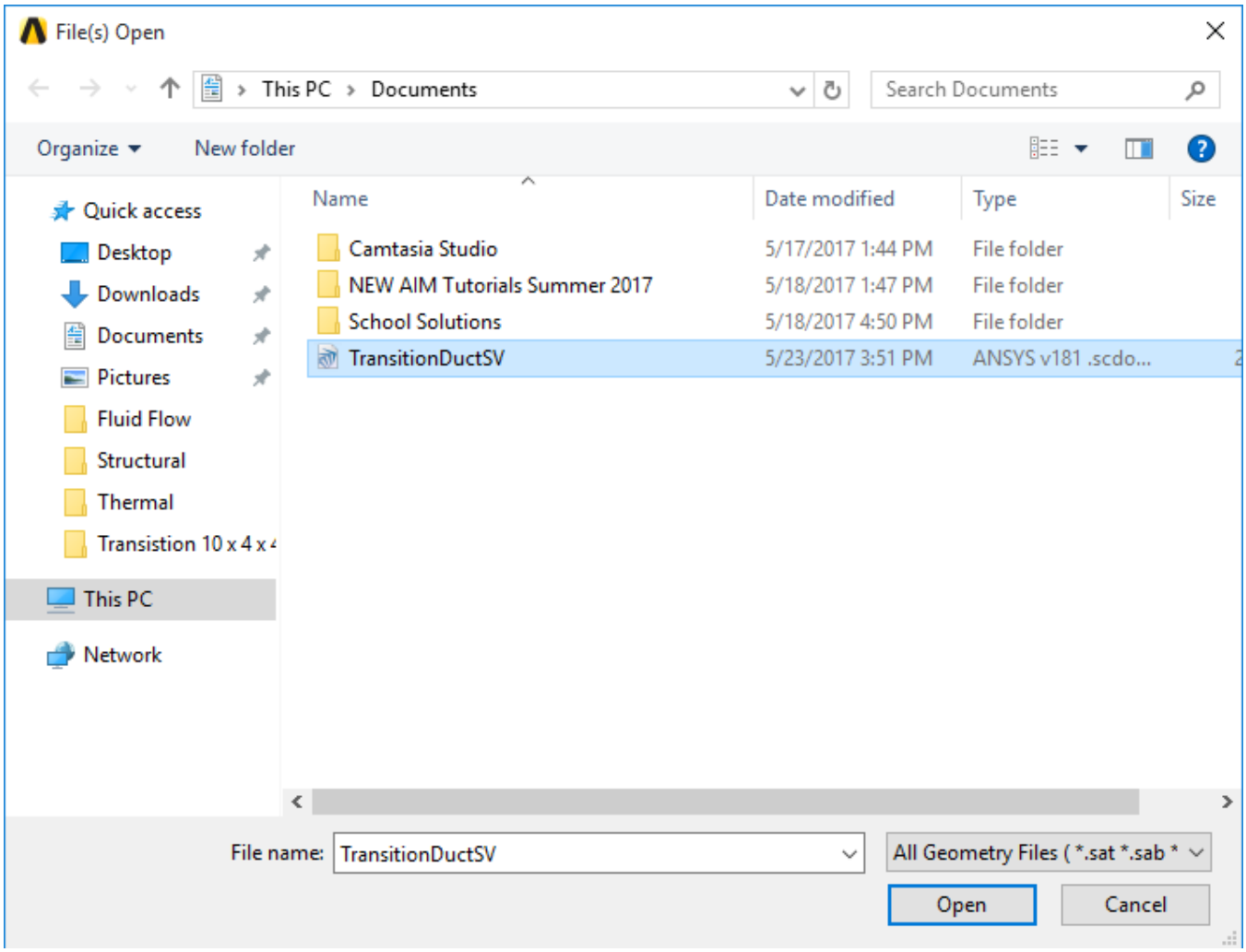
Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification
1. Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Results

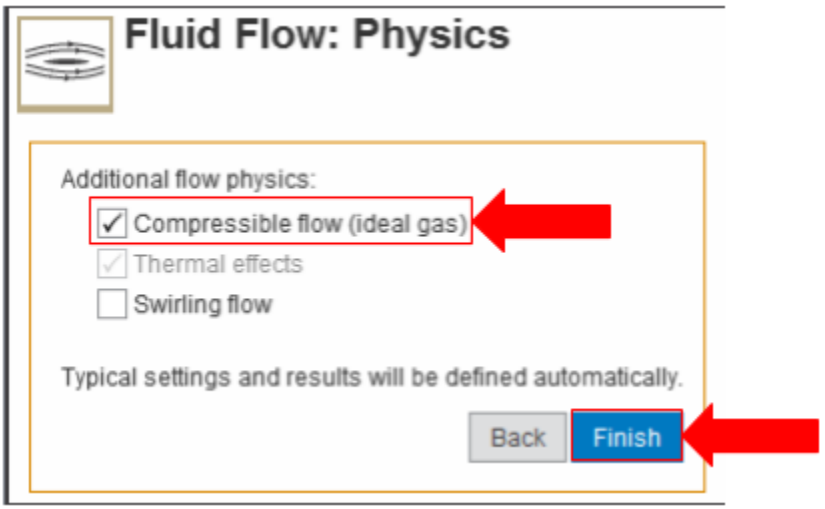
Geometry

Import Geometry

Download the geometry from here and open it.

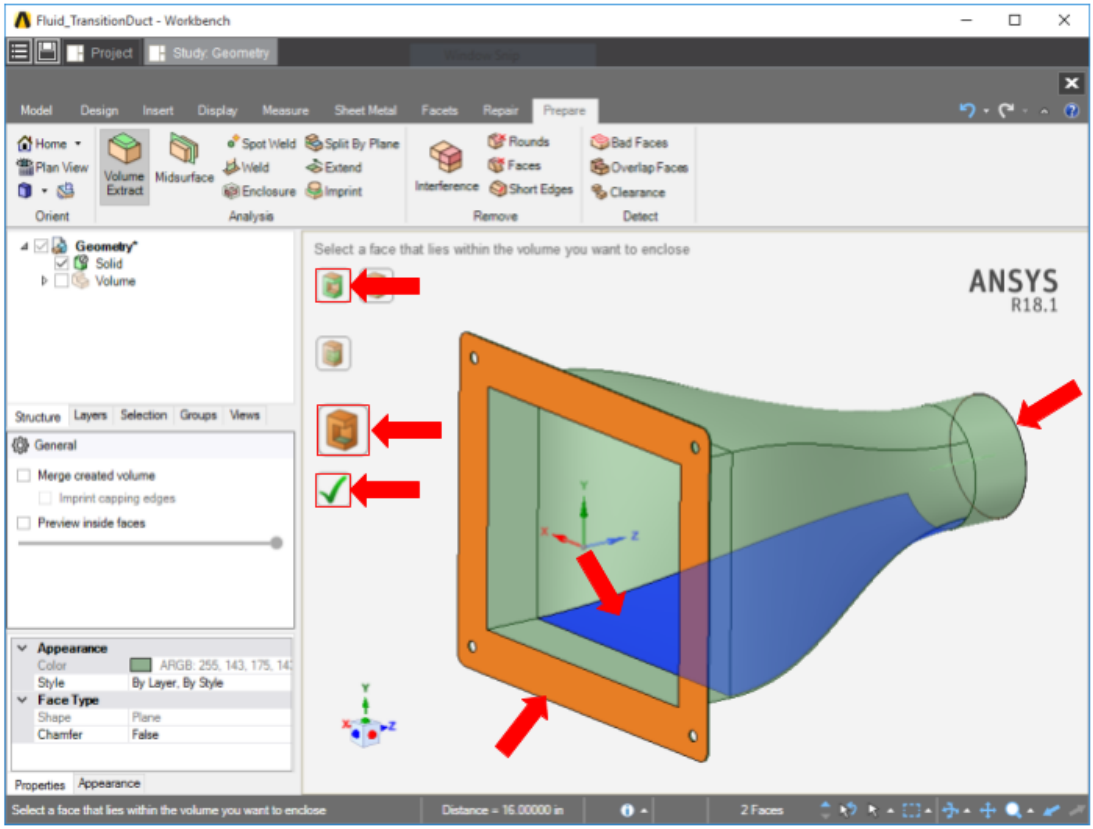


Once successfully imported, enable the **Compressible flow (ideal gas)** option and press **Finish**. In most situations, air behaves closely enough to an ideal gas that it can be modeled as one.



Extracting Volume

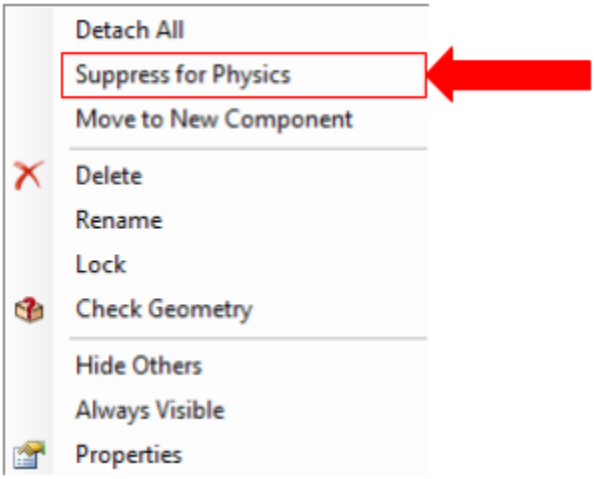
Press **Geometry** in the **Workflow** panel and select **Edit Geometry** in the **Geometry** template. In the **Prepare** tab of the Model Editor, select the **Volume Extract** tool. Use the **Select Faces** tool to select the faces at the opposite ends of the duct. Lastly, use **Select Seed Face** to select a face inside the duct. This can be any face on the inside of the duct. Press the green checkmark and your interior flow volume will be extracted.



Below is what the flow volume will look like when the imported solid is hidden.

Suppress

Now that the geometry of the flow volume has been created, we can suppress the imported solid geometry from the physics calculation. Right click on the **Solid** in the geometry **Structure** tree and select **Suppress for Physics**. Exit the modeler to continue with the simulation.



Go to Step 3: Mesh

Go to all ANSYS AIM Learning Modules