

# ANSYS Transonic Flow over a Wing - Geometry

Author(s): Sebastian Vecchi, ANSYS Inc.

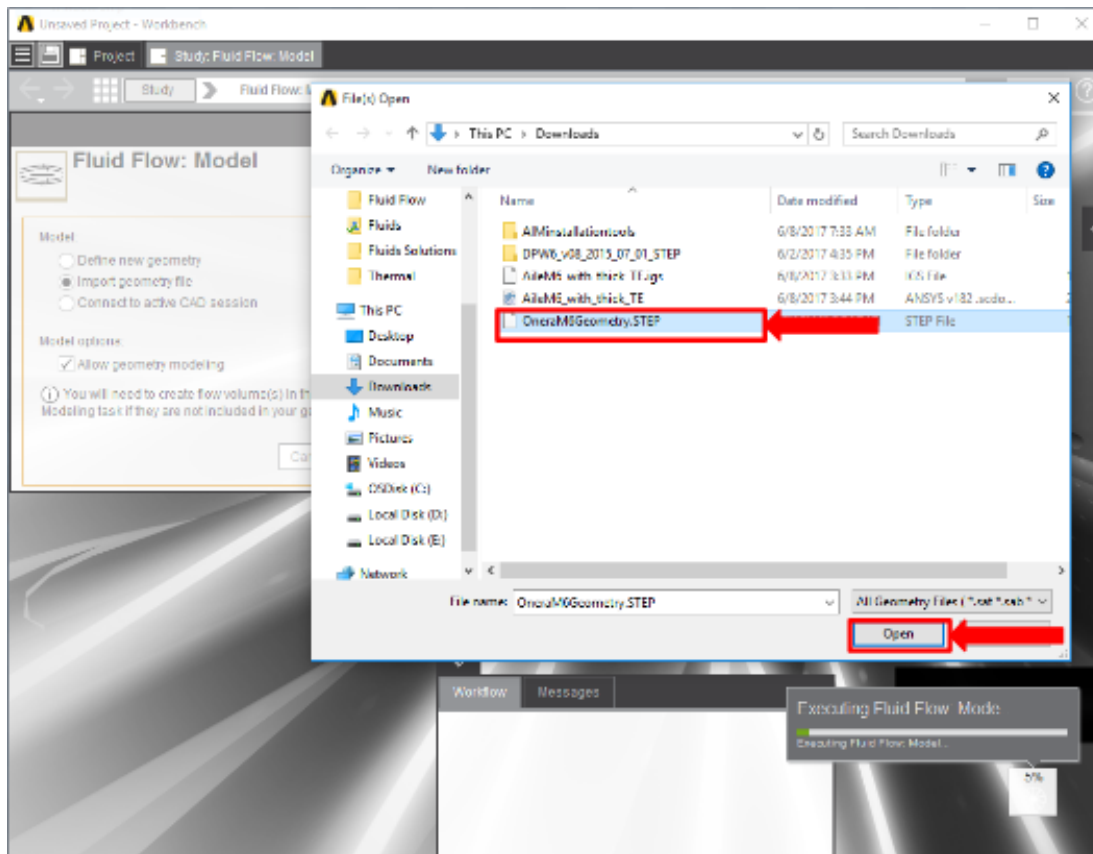
## Problem Specification

1. Startup
2. Geometry
3. Mesh
4. Physics Setup
5. Solution/Results
6. Verification & Validation

## Geometry

### Import Geometry

You will be prompted by the **Fluid Flow** template to either **Define new geometry**, **Import geometry file**, or **Connect to active CAD session**. Select **Import geometry file** and press **Next**. Download the part file "OneraM6Geometry.STEP" from [here](#), then select and **Open** the file.




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



## Fluid Flow: Physics

Additional flow physics:

- ☒ Compressible flow (ideal gas) 
- ☒ Thermal effects
- ☐ Swirling flow
- ☐ Particles (including droplets and bubbles)

Calculation type:

- ☒ Steady/static
- ☐ Time-dependent

Typical settings and results will be defined automatically.



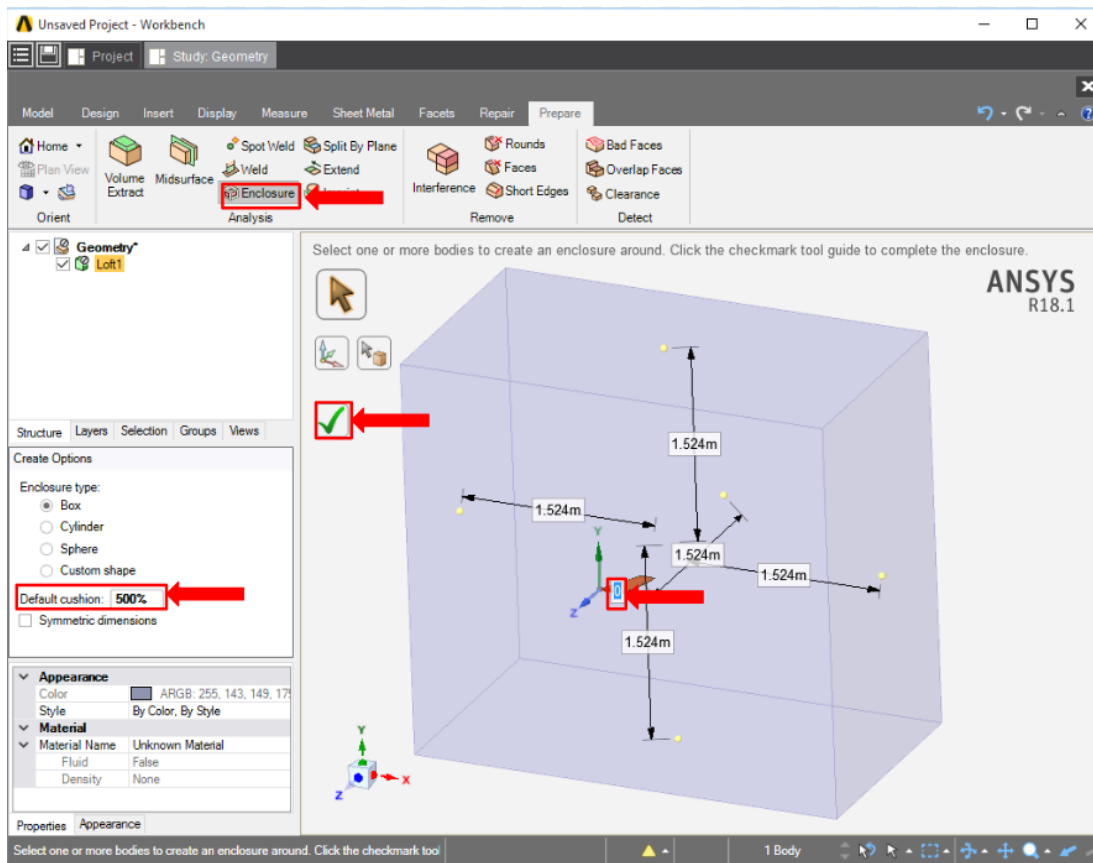
Finish

## Rotate

The simulation that will be used as validation indicates that the angle of attack is 3.06 degrees. Press the **Move** tool and then, using the **Select Component** tool, choose the wing body. Press the blue, curved arc and rotate the wing so that the input box appears, then input -3.06deg to rotate it precisely.

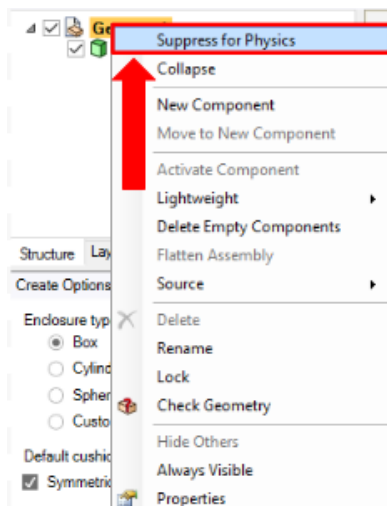
## Enclose

In order to create an area around the wing where air will flow, an enclosure must be made. Press **Geometry** in the **Workflow** tab and select **Edit Geometry** in the **Geometry** panel. The **Enclosure** tool can be found in the **Analysis** section of the toolbar under the **Prepare** tab. Select the **Enclosure** tool. Select the wing body, then a box will appear around the wing. Input 500% into the **Default cushion** box. Uncheck the **Symmetric dimensions** box and select the distance from the larger wing side (the side that would attach to the aircraft) to the enclosure and set it to 0in. Use the picture below for guidance. Press the green checkmark and the enclosure will be generated.



## Suppress

Now that the geometry of the flow volume has been created, we can suppress the wing from the physics calculation. Right click on the solid named **Loft1** in the **Geometry** tree and select **Suppress for Physics**.



[Go to Step 3: Mesh](#)

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