

# ANSYS Flow over an Ahmed Body - Geometry

Author(s): Sebastian Vecchi, ANSYS Inc.

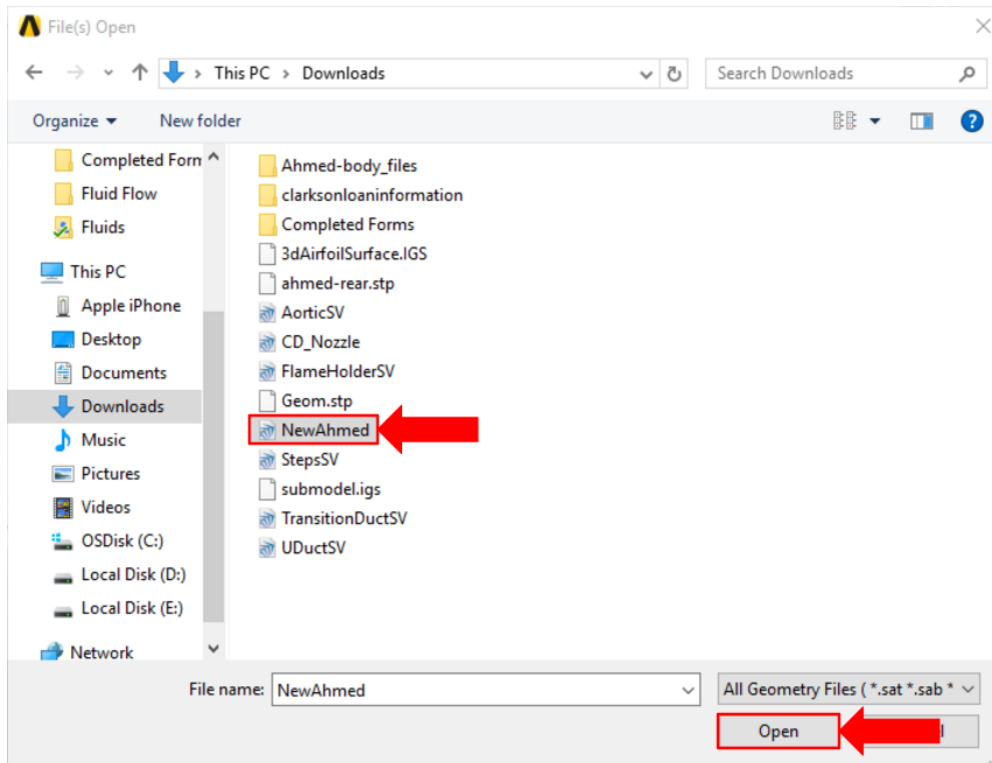
## Problem Specification

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

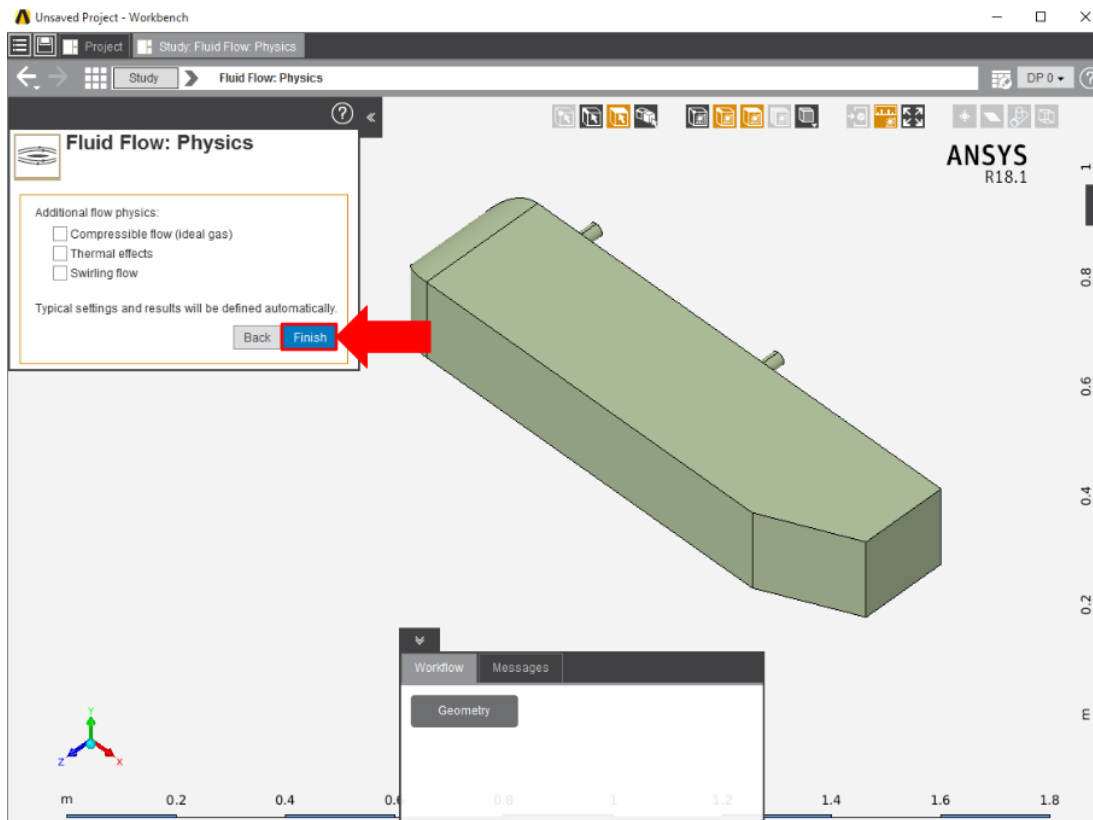
## Geometry

### Import Geometry

For this problem, we are going to import the geometry into ANSYS from a CAD package. Download the part file "NewAhmed.scdoc" from [here](#). Select and open the file.

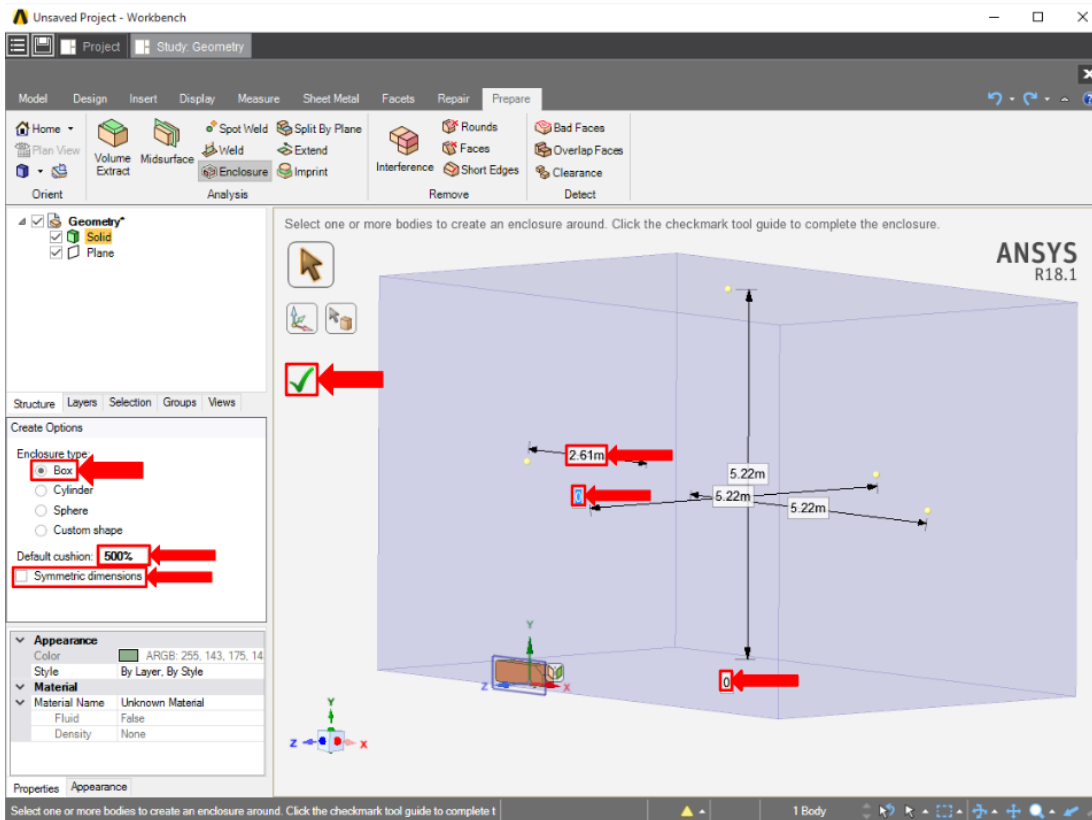


Once successfully imported, [press Finish](#).



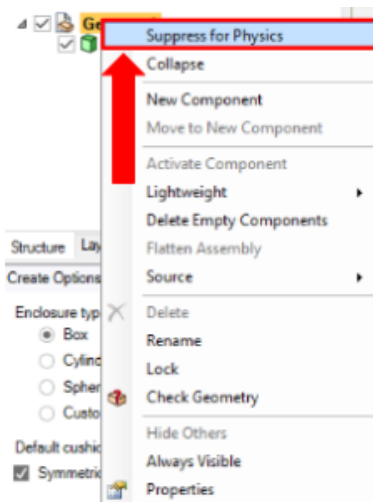
## Enclose

A volume needs to be created to around the Ahmed body using the enclosure feature, to represent the fluid flow area. 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 Ahmed body and a box will appear around the geometry. Input 500% into the **Default cushion box**. Uncheck the **Symmetric dimensions** box. Change the distance from the body to the bottom of the enclosure and distance from the midplane to the enclosure to 0m, then the distance from the front of the body to the enclosure to 2.61m. 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 Ahmed body from the physics calculation. [Right click the solid in the geometry tree and select \*\*Suppress for Physics\*\*.](#)



[Go to Step 3: Mesh](#)

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