Compressible Flow Over an Airfoil - Geometry

Author(s): Sebastian Vecchi, ANSYS Inc.

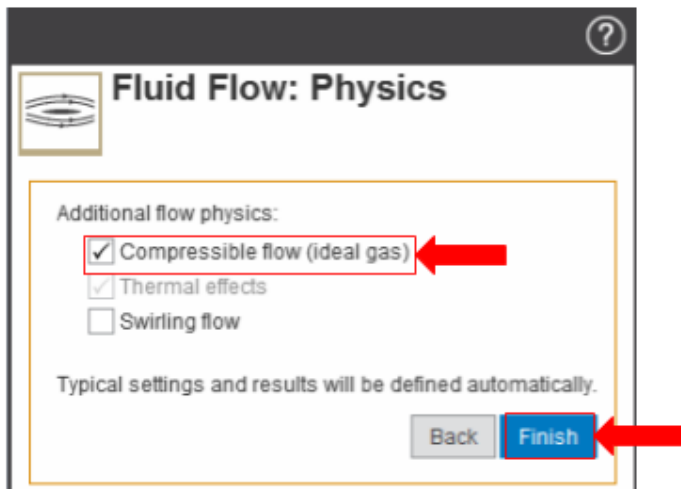## Geometry

Import Geometry

For this problem, we are going to import the geometry from a CAD package. Download the part file here, then select and open the file in AIM.

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


Repair

The imported geometry contains only the curved surfaces of the airfoil. We need to close the ends of the airfoil to create a solid body representing it. Select the **Geometry** task in the **Workflow** and press **Edit Geometry**. After the Model Editor opens, select the **Repair** tab and then **Missing Faces** in the **Solidify** section. The ends of the airfoil will be highlighted in red. Press the green check mark to create the missing faces and form the solid.

Resize

Since the airfoil that was imported is about 5m long, shortening it will likely speed up our mesh generation and calculations down the line. Use the **Pull** tool, in the **Edit** section of the **Design** tab, to shorten it to a more manageable size. Click to select the face on one end and drag it 2.4m toward the middle; repeat for the other end.

Rotate

The simulation that will be used as validation indicates that the angle of attack is 1.25 degrees. Press the **Move** tool and then, using the **Select Component** tool, choose the airfoil. Press the blue, curved arc and rotate the airfoil so that the input box appears, then input -1.25deg to rotate it precisely.
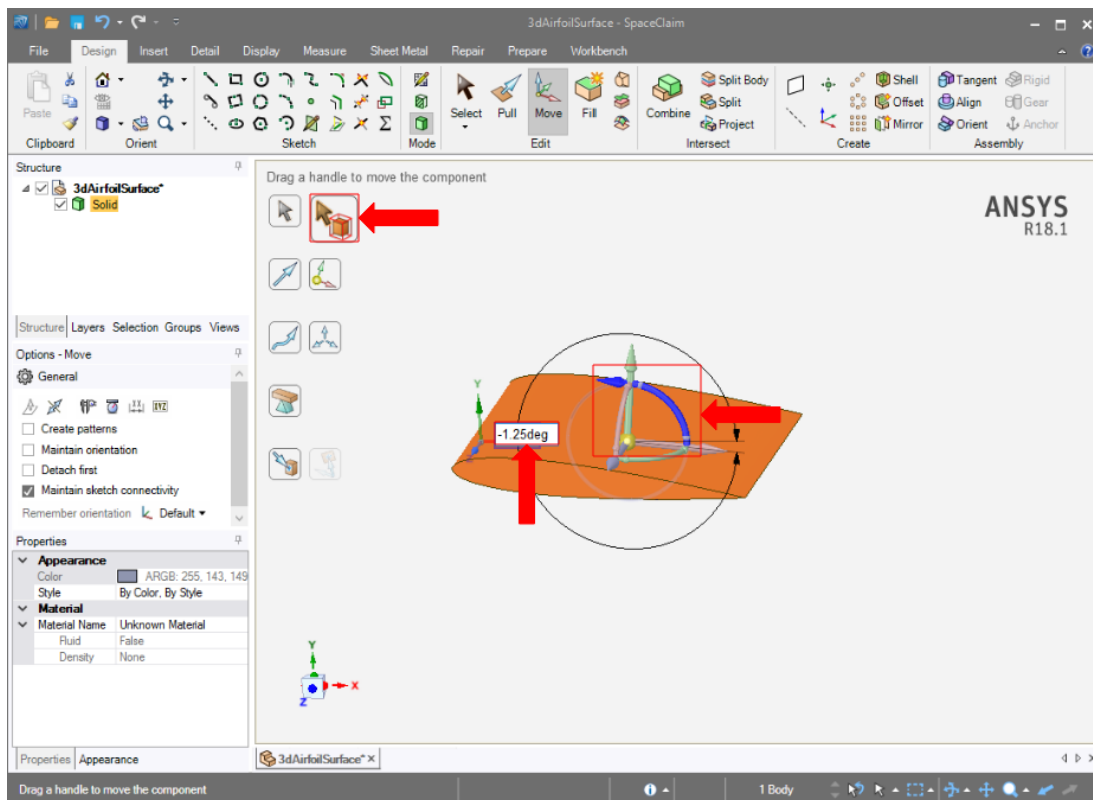
Enclose

In order to create an area around the airfoil where air will move, an enclosure must be made around it. The **Enclosure** tool can be found in the **Analysis** section of the toolbar under the **Prepare** tab. Activate the **Enclosure** tool, then select the airfoil body. A box will appear around the airfoil. Enter 500% into the **Default cushion** box at left. Select the distance from the airfoil end to the enclosure and set it to 0m. Use the picture below for guidance. Press the green checkmark and the enclosure will be generated.
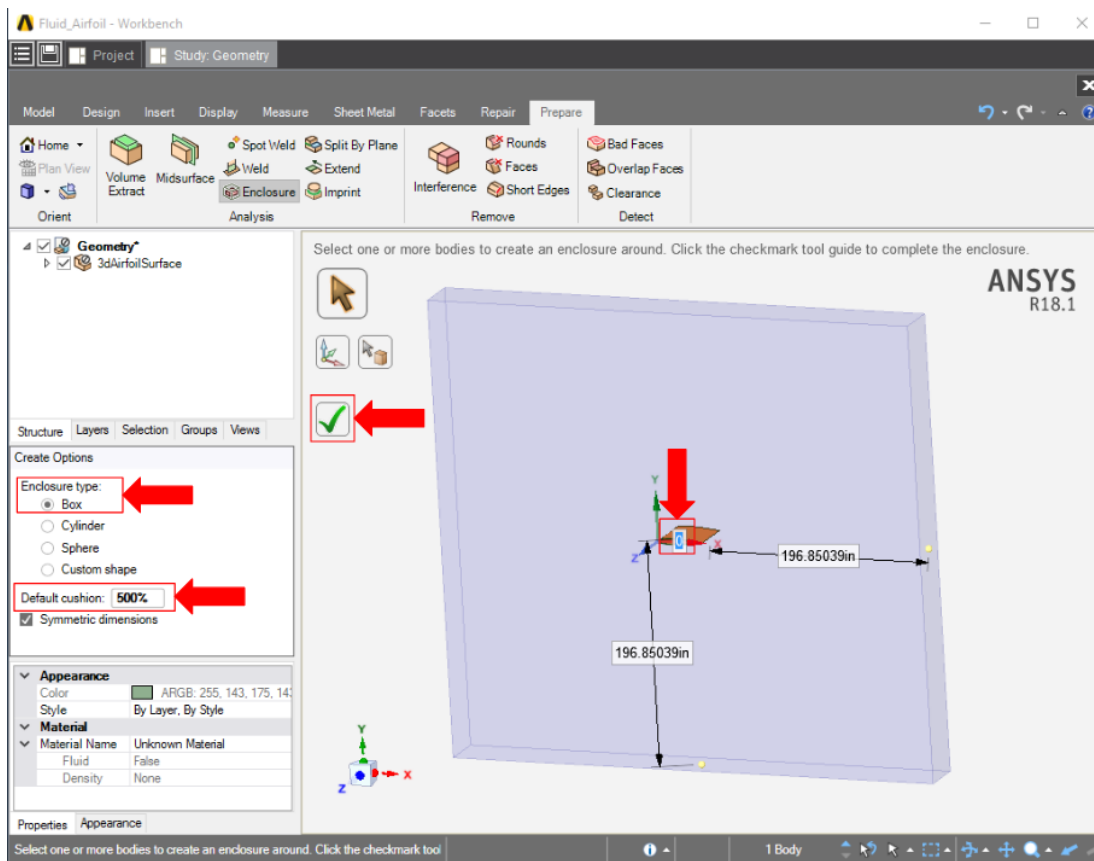
Suppress

Now that the geometry of the flow volume has been created, we can suppress the airfoil from the physics calculation. Right click on the **Solid** in the **Geometry** tree and select **Suppress for Physics**.
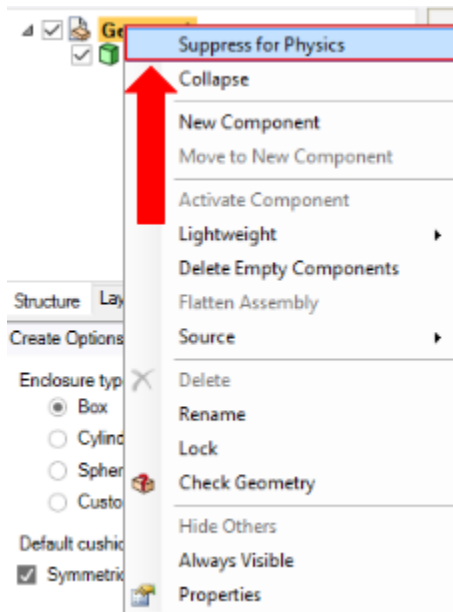
Go to Step 3: Mesh

Go to all ANSYS AIM Learning Modules