ANSYS Compressible Flow over a Wing-Body Junction - Geometry

Author(s): Sebastian Vecchi, ANSYS Inc.
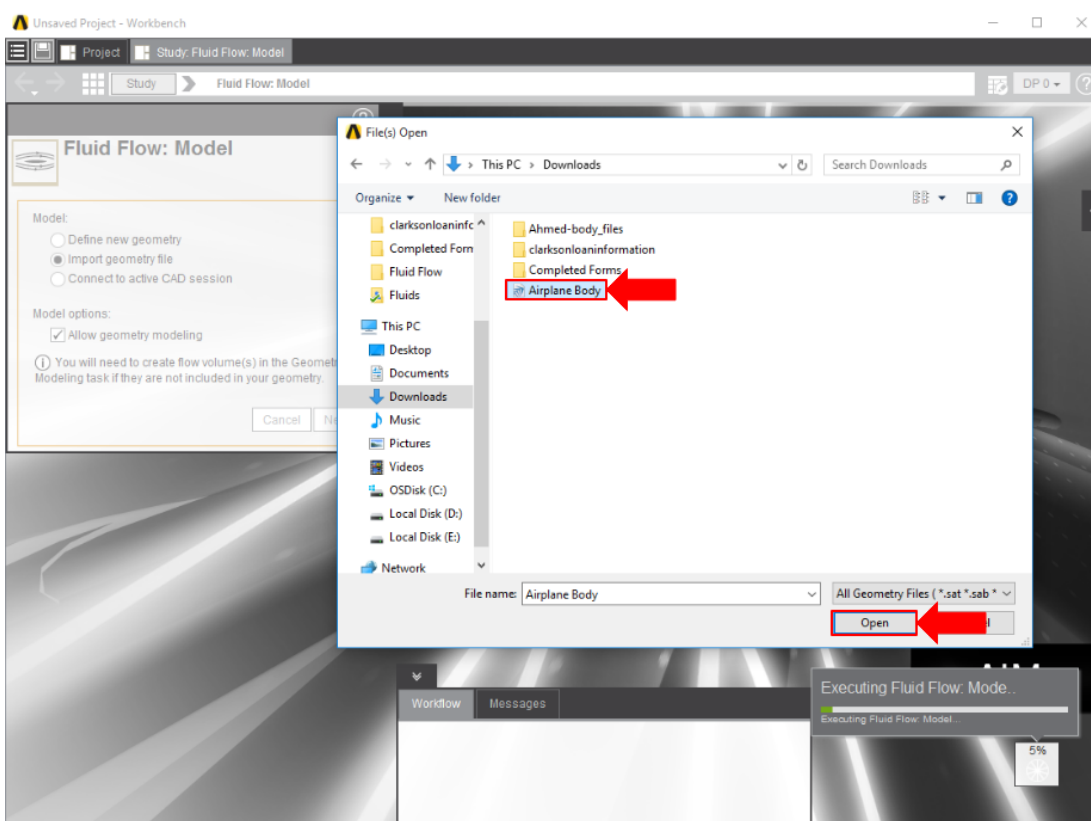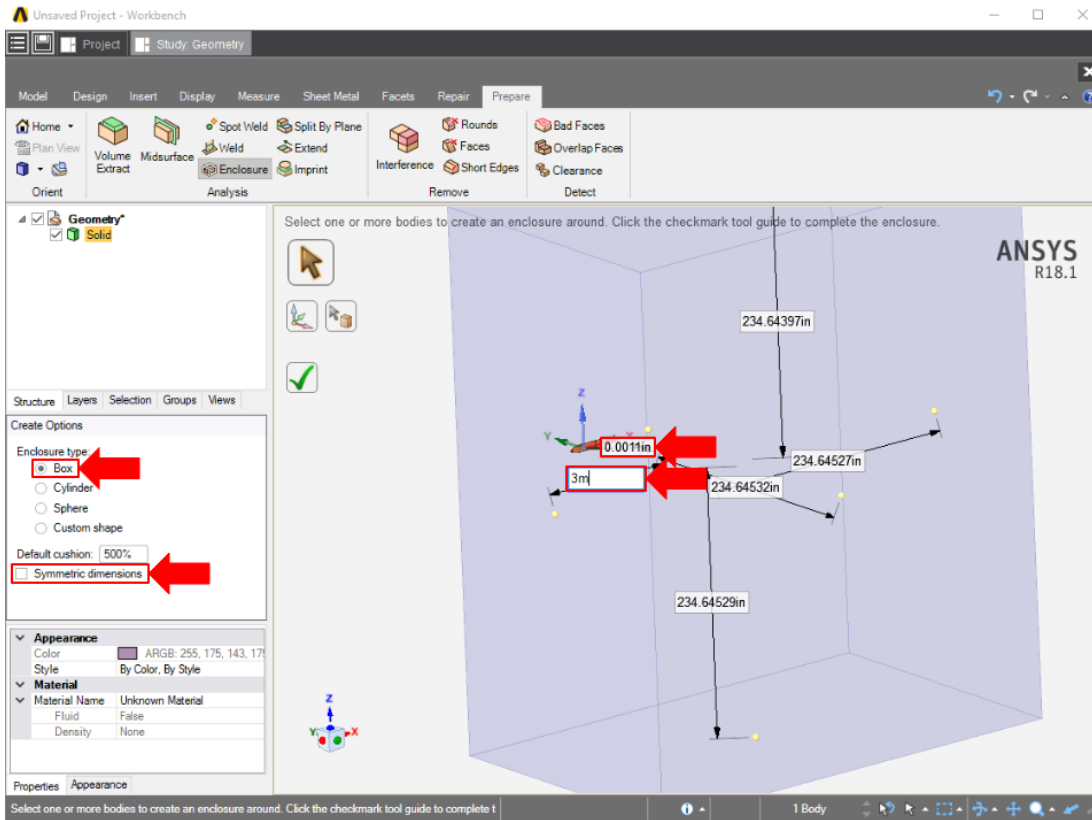
Geometry

Import Geometry

For this problem, we are going to import the geometry into ANSYS from a CAD package. Download the part file "Airplane Body.scdoc" from here. Select and open the CAD file.

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

Enclose

An enclosure must be made around the airfoil to represent the flow region. Select the **Geometry** task in the **Workflow** and press **Edit Geometry**. The **Enclosure** tool can be found in the **Analysis** section of the toolbar under the **Prepare** tab. Select the airplane body and a box will appear around the airplane. Input 500% into the **Default cushion** box. Select the distance from the cut face of the plane to the enclosure and set it to 0in. Then, select the distance from the nose of the plane to the enclosure and set it to 3m. 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 airplan 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