ANSYS Compressible Flow over a Wing-Body Junction - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc.

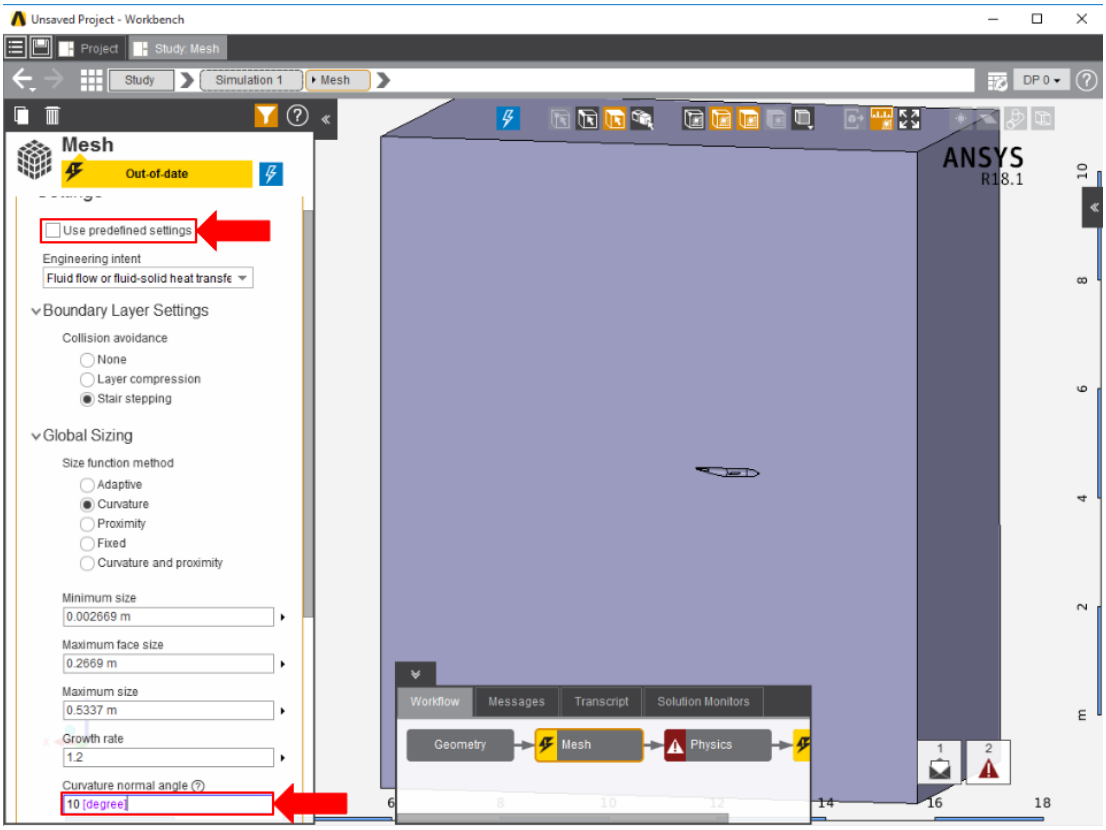
Problem Specification
1. Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Solution/Results

Mesh

Once you have exited the modeling window, initiate the meshing process by clicking on **Mesh** in the workflow. If desired, In the **Choose display mode** menu near the top of the model window, select **Translucent display** to view what is going on inside the enclosed volume.

Set Mesh Size & Controls

Drag the **Mesh resolution** slider to the second highest setting. Uncheck the **Use predefined settings** button and input 10 [degree] for the **Curvature normal angle** under **Global Sizing**.



AIM will prompt you to fix the boundary layer before generating the mesh. Click on **Boundary Layer** under **Mesh Controls**. Select all of the faces of the airplane body.

Generate Mesh

Return to the **Mesh** panel, then click **Generate Mesh** under **Output** or at the top of the screen by the status window for Mesh. AIM will detect that you are ready to generate the mesh and highlight the buttons in blue.

Go to Step 4: Physics Setup

Go to all ANSYS AIM Learning Modules