

ANSYS Compressible Flow over a Wing-Body Junction - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc.

Problem Specification

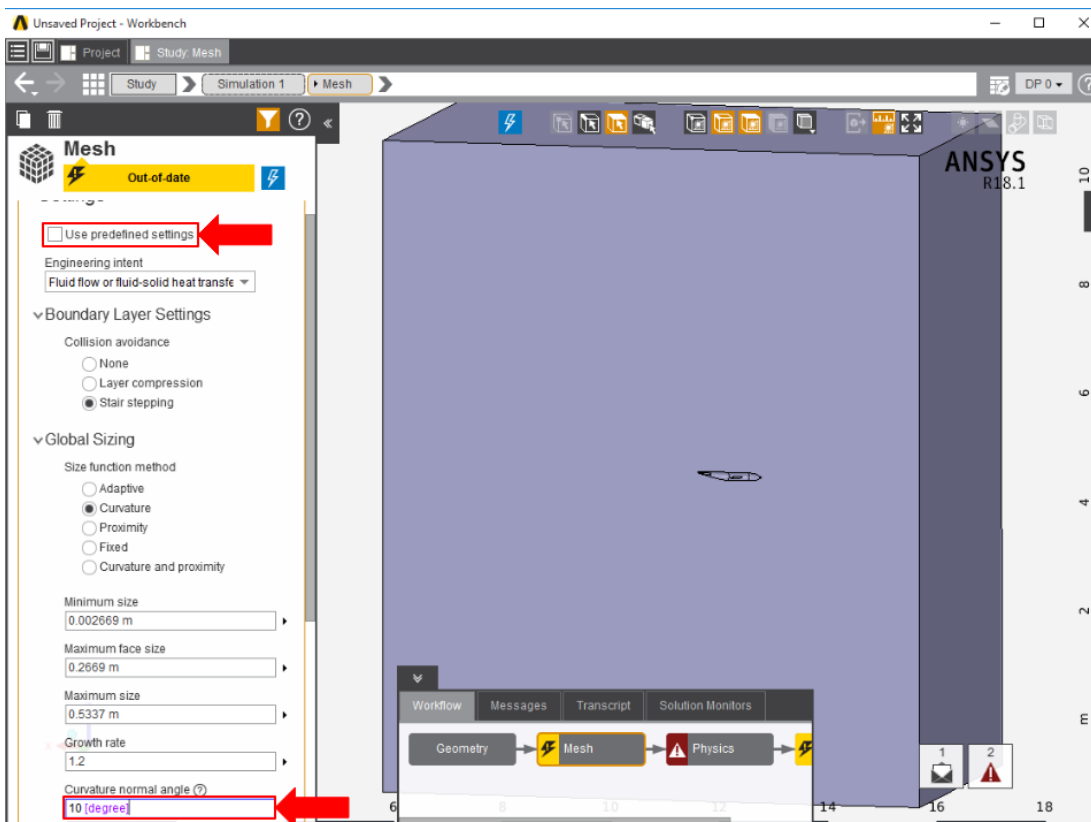
1. Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Solution/Results

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](#)