ANSYS Transonic Flow over a Wing - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc.

- **Problem Specification**
- 1. Startup
- 2. Geometry
- 3. Mesh
- Physics Setup
 Solution/Results
- 5. Solution/Results
- 6. Verification & Validation

<u>Mesh</u>

Once you have exited the modeling window, initiate the meshing process by clicking on Mesh in the Workflow.



Set Mesh Controls

AIM will prompt you to fix the boundary layer before generating the mesh. Click on **Boundary Layer** under **Mesh Controls**. Select the 3 faces of the wing inside the flow volume.

We must create an appropriate mesh to capture the subtle flow features around the wing. A simple way of doing this is to add a Face Sizing control. Under **Objects** in the **Mesh** panel, press the **Add** button next to **Size Controls**, then select **Face Sizing**. Select the 3 faces of the wing as the **Location** and input an **Element size** of 0.125 [in].



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. Below is an example of the mesh.

Go to Step 4: Physics Setup

Go to all ANSYS AIM Learning Modules