## **Compressible Flow Over an Airfoil - Mesh**

Author(s): Sebastian Vecchi, ANSYS Inc. Problem Specification 1. Start-Up 2. Geometry 3. Mesh 4. Physics Setup 5. Results 6. Verification & Validation Exercise

## <u>Mesh</u>

Close the modeling window, then initiate the meshing process by clicking on Mesh in the Workflow.

## Set Mesh Controls

Under Boundary Layer Settings, change Collision avoidance to use the Layer compression setting.



AIM will prompt you to fix the boundary layer before generating the mesh. Under **Objects**, click on **Mesh Controls**, then **Boundary Layer**. Select the upper and lower faces of the airfoil inside of the flow volume.



## Generate Mesh

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 what the mesh will look like.

Go to Step 4: Physics Set-Up

Go to all ANSYS AIM Learning Modules