Compressible Flow Over an Airfoil - Mesh
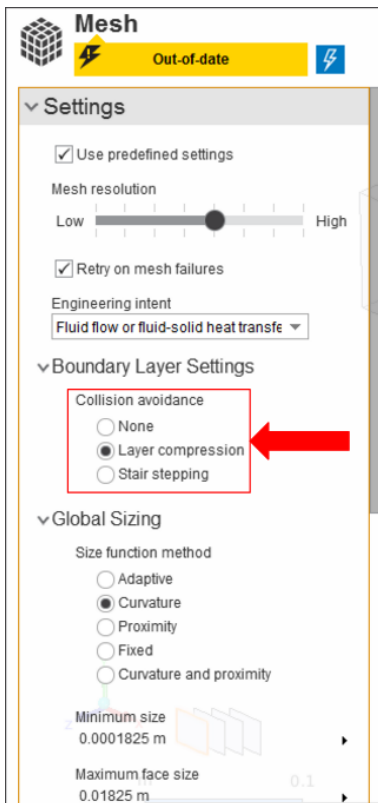
Author(s): Sebastian Vecchi, ANSYS Inc.

Mesh

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.
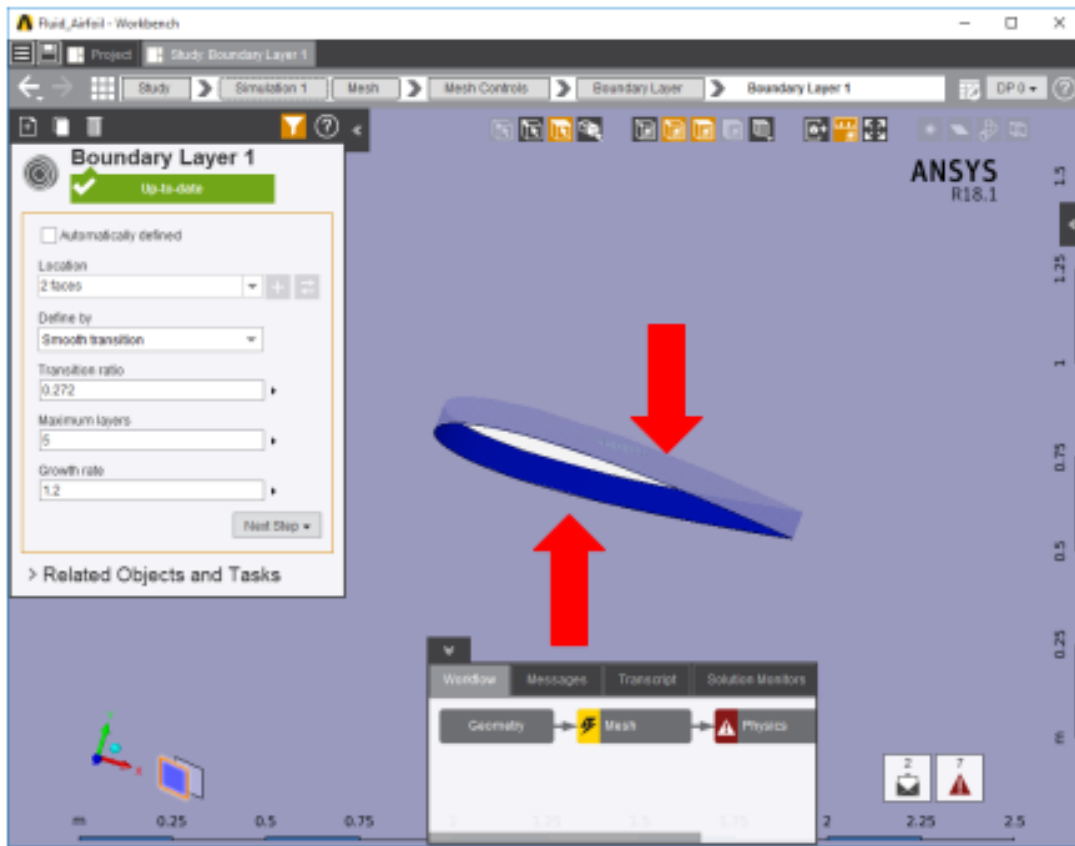
Generate Mesh

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