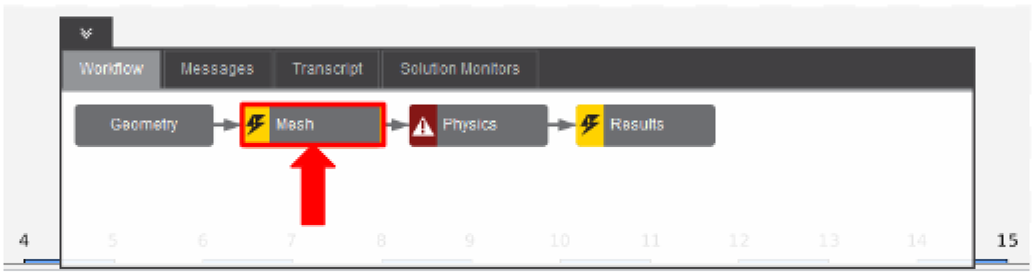
ANSYS Transonic Flow over a Wing - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc.

Mesh

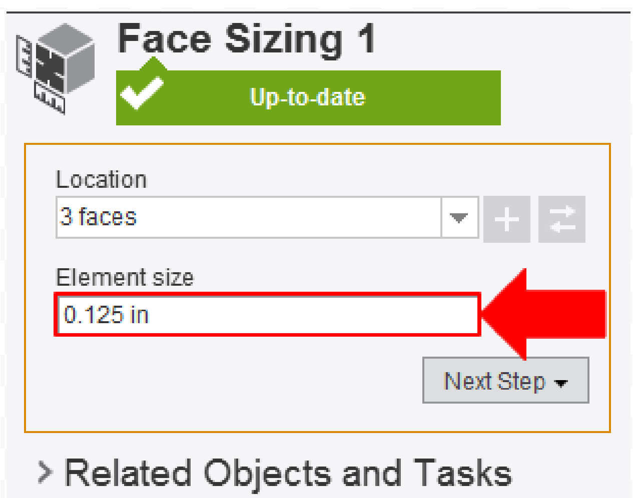
Once you have exited the modeling window, initiate the meshing process by clicking on **Mesh** in the **Workflow**.



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