Compressible Flow in a Nozzle - Mesh

Author(s): Sebastian Vecchi, ANSYS Inc.
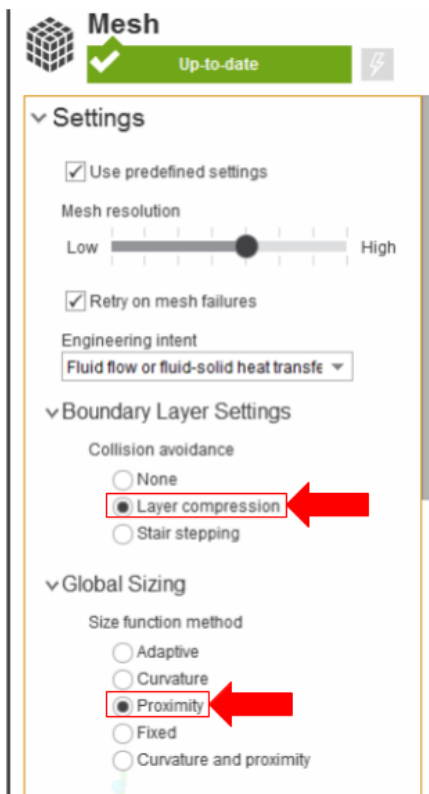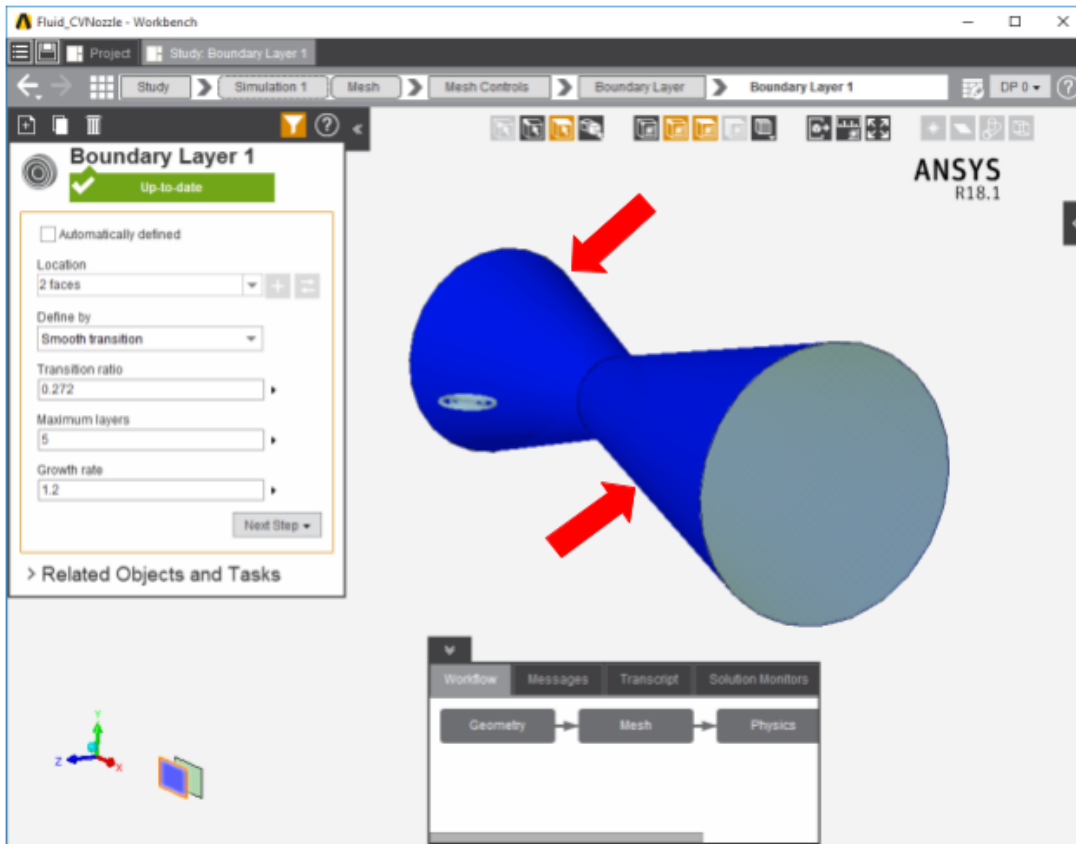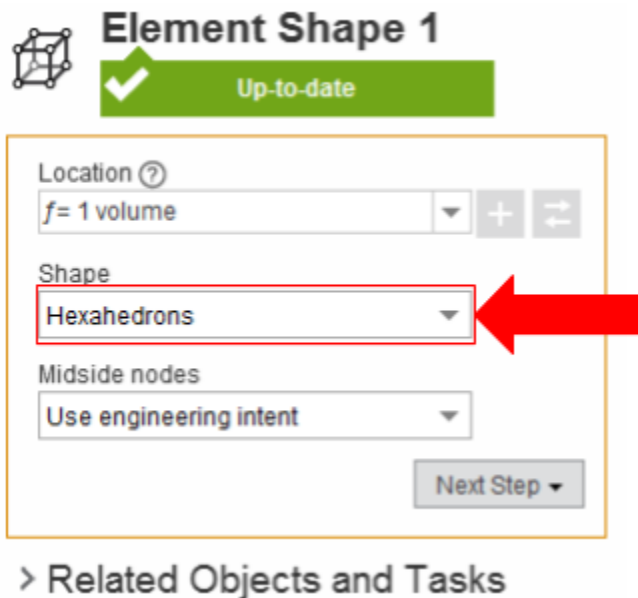
Mesh

Set Mesh Controls

Select the **Mesh** task in the **Workflow**. Under **Boundary Layer Settings**, change **Collision avoidance** to **Layer compression**. Under **Global Sizing**, change the **Size function method** to **Proximity**. This will produce continuous boundary layers and refine the mesh in the narrow section of the nozzle.

AIM will prompt you to fix the boundary layer before generating the mesh. Click on **Boundary Layer** under **Mesh Controls**. Select every face except for the circular inlet and outlet faces. An easy way to do this is drag a square around the shape and then deselect the inlet and outlet using CTRL + click.

We will also change the mesh to use hexahedrons, to more accurately capture the effects in the critical nozzle region and aid convergence of the calculations. Click on **Mesh Controls**, then **Element Shape 1**. Change the **Shape** to **Hexahedrons**.

Generate Mesh

Return to the **Mesh** panel and click **Generate Mesh**. AIM will detect that you are ready to generate the mesh and highlight the buttons in blue. Below, an example of the mesh is shown.

Go to Step 4: Physics Set-Up