Compressible Flow in a Nozzle - Mesh

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

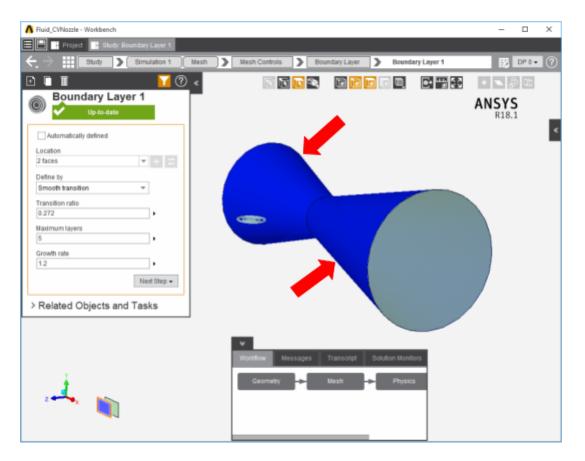
<u>Mesh</u>

Set Mesh Controls

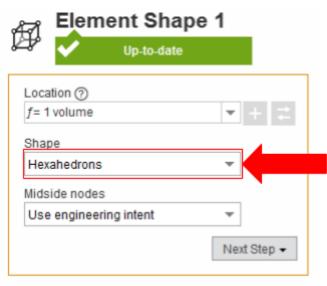
Select the Mesh task in the Workflow. Under Boundary Layer Settings, change Collision avoidance to Layer compression. Under Global Sizing, change the Size function method to Proximity. This will produce continuous boundary layers and refine the mesh in the narrow section of the nozzle.

🚓 Mesh
Up-to-date 🥖
∽ Settings
✓ Use predefined settings
Mesh resolution
Retry on mesh failures
Engineering intent Fluid flow or fluid-solid heat transfe 🔻
✓Boundary Layer Settings
Collision avoidance
None Layer compression Stair stepping
∽ Global Sizing
Size function method
Adaptive
Curvature
Curvature and proximity

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**.



> Related Objects and Tasks

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

Go to all ANSYS AIM Learning Modules