3D Transonic Flow Over a Wing - Numerical Solution

Numerical Solution

In the video we will:

1. Set our solution method and choose options to assist in convergence
2. Create monitors
3. Initialize the solution
4. Check reference values
5. Choose the quantities we want FLUENT to export to CFD Post and solve

Numerical Solution Part 1

Summary of the above video:

1. Solution Methods>Coupled
   a. Pseudo Transient
   b. High Order Term Relaxation
2. Monitors>Residuals
   a. Absolute Convergence Criteria = 1e-3
3. Monitors>Create>Drag
4. Monitors>Create>Lift
5. Solution Initialization
   a. Compute From>Inlet
6. Reference Values
   a. Compute From>Inlet
   b. Reference Area = 1.076 ft2
   c. Reference Length = 1 ft
7. Data File Quantities
   a. Pressure Coefficient
   b. Mach Number

When selecting your reference values to be computed from a boundary, in this case the inlet, Fluent does not compute the reference length or the reference area. This means that we need to set them ourselves for the problem.

Solving may take a long time, depending on your computer. To give you an idea of the time you can expect, times are listed below along with the computer specifications:

<table>
<thead>
<tr>
<th>Computer Specs</th>
<th>Configuration</th>
<th>Run Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>AMD A10-6800K APU quad core 4.10 GHz</td>
<td>Double Precision 4 Parallel Processes</td>
<td>20 Minutes</td>
</tr>
</tbody>
</table>
You should see the solution converge in about 172 iterations. Below a graph of the residuals is given.

Here, the residuals have fallen below 1e-3 for the six variables.

**Go to Step 6: Numerical Results**

**Go to all FLUENT Learning Modules**