ANSYS Flow in a S-Duct - Validation

Author(s): Sebastian Vecchi, ANSYS Inc.

Validation

In order to validate the results from the tutorial, they must be compared to a source of credibility. The results will be compared to those found in "Numerical Simulation of Pressure Recovery and Distortion in an Aircraft Engine Intake Serpentine Diffuser" by Saravana Kumar, Balasubramanyam Sasanapuri, Konstantin A. Kourbatski and Angela Lestari.

Below are the Mach number contours on the midplane of the flow volume. The Mach number is at its peak near the intake area of the S-duct, and slowly decreases until the conical nozzle separates the flow with the upper region having the higher Mach number.

A Mach number contour must be created in AIM to compare the simulation results to these. In the **Add** drop down, create a **Contour** on the plane created earlier, then change the **Variable** to **Mach Number**. Below is an example of what the Mach number contour will look like.

The maximum value for the Mach number, calculated to be 0.57, is quite different than the value found in the validation. By refining the mesh, the results can be calculated to a more accurate degree. Go back to the **Mesh** task in the workflow and increase the **Mesh resolution** by one notch. Go back to the **Results** and press **Evaluate Results**. Travel back to the Mach Number contour.

The maximum Mach Number is now less than 5% from the validation maximum, and the contours indicate that the flow behaves similarly. At the inlet, the Mach number is at its highest and it slowly dissipates until it reaches the conical nozzle, where the upper region has the highest Mach profile.

Go to all ANSYS AIM Learning Modules