AFRL-VA-WP-TR-2007-3077

COMPUTATIONAL STUDY OF INLET ACTIVE FLOW CONTROL
Delivery Order 0005

Dr. Sonya T. Smith

Howard University
Department of Mechanical Engineering
Washington, DC

Dr. Angela Scribben and Matthew Goettke

Aerospace Vehicle Integration and Demonstration Branch (AFRL/VAAI)
Aeronautical Sciences Division
Air Vehicles Directorate, Air Force Materiel Command
Air Force Research Laboratory
Wright-Patterson Air Force Base, OH 45433-7542

MAY 2007

Final Report for 06 February 2005 – 11 May 2007

AIR VEHICLES DIRECTORATE
AIR FORCE MATERIEL COMMAND
AIR FORCE RESEARCH LABORATORY
WRIGHT-PATTERSON AIR FORCE BASE, OH 45433-7542

NOTICE AND SIGNATURE PAGE

AFRL-VA-WP-TR-2007-3077 HAS BEEN REVIEWED AND IS APPROVED FOR PUBLICATION IN ACCORDANCE WITH ASSIGNED DISTRIBUTION STATEMENT.

*//Signature//
Angela Scribben, Aerospace Engineer

//Signature//
Michael Zeigler, Chief, Aerospace Vehicle Integration & Demo Branch

//Signature//
Michael Stanek, Tech Advisor, Aeronautical Sciences Division

*Disseminated copies will show "//signature//" stamped or typed above the signature blocks.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. REPORT DATE *(DD-MM-YY)*
May 2007

2. REPORT TYPE
Final

3. DATES COVERED *(From - To)*
02/06/2005 – 05/11/2007

4. TITLE AND SUBTITLE
COMPUTATIONAL STUDY OF INLET ACTIVE FLOW CONTROL
Delivery Order 0005

5a. CONTRACT NUMBER
F33615-03-D-3307-0005

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER
0602201

6. AUTHOR(S)
Dr. Sonya T. Smith (Howard University)
Dr. Angela Scribben and Matthew Goettke (AFRL/VAAI)

5d. PROJECT NUMBER
A0A2

5e. TASK NUMBER

5f. WORK UNIT NUMBER
0B

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Howard University
Department of Mechanical Engineering
Washington, DC

Aerospace Vehicle Integration and Demonstration Branch (AFRL/VAAI)
Aeronautical Sciences Division
Air Vehicles Directorate, Air Force Materiel Command
Air Force Research Laboratory
Wright-Patterson Air Force Base, OH 45433-7542

8. PERFORMING ORGANIZATION REPORT NUMBER

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
Air Vehicles Directorate
Air Force Research Laboratory
Air Force Materiel Command
Wright-Patterson Air Force Base, OH 45433-7542

10. SPONSORING/MONITORING AGENCY ACRONYM(S)
AFRL-VA-WP

11. SPONSORING/MONITORING AGENCY REPORT NUMBER(S)
AFRL-VA-WP-TR-2007-3077

12. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution unlimited.

13. SUPPLEMENTARY NOTES
Report contains color. PAO Case Number: AFRL/WS 07-1534, 27 Jun 2007.

14. ABSTRACT
A study was performed using the Air Vehicles Unstructured Solver (AVUS) to solve internal flow fields for diffusing S-ducts with separated flow. The study examined various boundary conditions, inflow, outflow and initial conditions, and grid sizes. The effort struggled to find an overall setup that agreed well with previously published results on the chosen geometry. In the end, it was discovered that several key issues were the cause. These issues included lack of a refined, structured, boundary layer grid region; high CFL numbers; and improper boundary conditions. The case was re-run with more appropriate conditions and a grid better suited to capture the boundary layer impact. The key issues encountered are documented here as well as the setup for the successful test case. This study demonstrated the criticality of proper boundary conditions, the benefit of initializing the flow field, and the importance of understanding critical solution health parameters such as y+. Additionally, the importance of a well constructed grid has been demonstrated. This is of particular importance for solutions that involve or may potentially involve separated boundary layers. Other parameters that may prove crucial are CFL number and sweeps per iteration.

15. SUBJECT TERMS
s-duct, boundary layer, CFD

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT:	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON (Monitor)
a. REPORT	**b. ABSTRACT**	**c. THIS PAGE**			Angela Scribben
Unclassified	Unclassified	Unclassified	SAR	14	**19b. TELEPHONE NUMBER *(Include Area Code)*** N/A

Introduction

Serpentine ducts have been studied in the past to provide a more flexible arrangement within air vehicles and to shorten the overall propulsion system length. In unmanned aerial vehicles (UAVs) the propulsion system can drive the length of the vehicle [1]. The propulsion system length is driven by the ability to design the shortest inlet duct while still maintaining uniform, high total pressure recovery flow at the duct exit and fitting the inlet duct around other critical components within the aircraft. Any decrease in characteristic length of the propulsion system will lead to a shorter, lighter, and therefore less costly aircraft [1]. Serpentine inlet ducts are being considered as a design solution to shorten the overall propulsion system length and allow the inlet to fit around other critical components of the aircraft.

The research described in this report was conducted to evaluate the ability of the Air Vehicles Unstructured Solver (AVUS) Computational Fluid Dynamics (CFD) code to accurately predict the flow within a serpentine inlet with separated flow. At the inception of this effort AVUS had not been used to solve these types of separated internal flow paths.

The M2129 duct, AGARD Fluid Dynamics Panel Working Group 13 test case 3 [2], was chosen for this evaluation due to the availability of published experimental and CFD results for the duct. AVUS results were compared to CFD work in the literature, specifically focusing on total pressure, static pressure and Mach number contours axially along the centerline.

Inlet Duct Configuration

The M2129 geometry is shown in Figure 1. The duct is a circular cross section diffusing S-duct with an exit to inlet area ratio of 1.41. The height offset is 5.4 inches. The throat diameter is 5.1 inches while the exit diameter is 6.0 inches. The axial length of the duct from the throat to the engine face is 19.3 inches, resulting in an inlet length to diameter ratio of 3.2 [3].

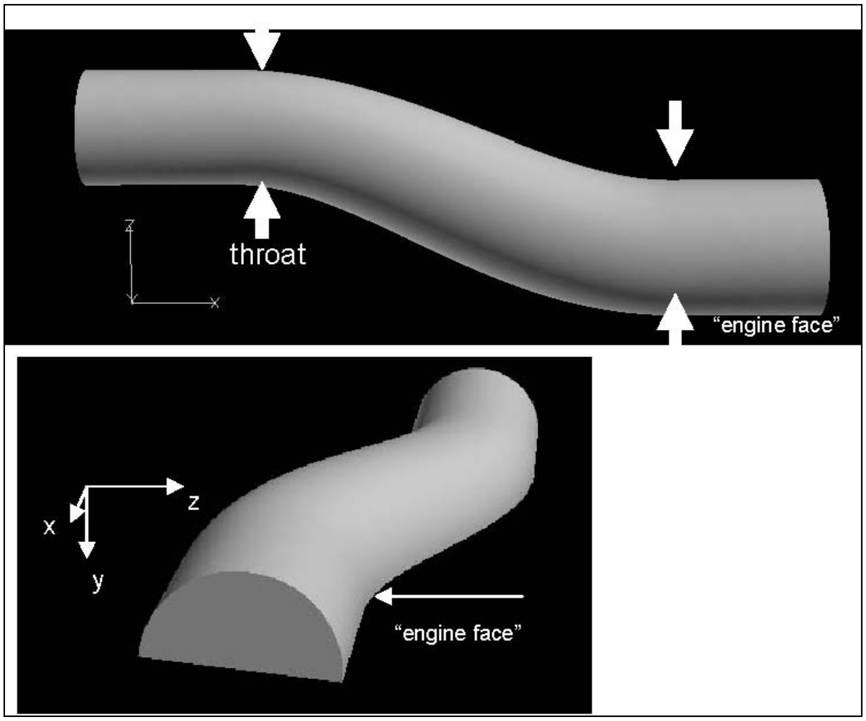


Figure 1: Baseline M2129

Grid Generation

AVUS is an unstructured solver capable of handling both structured and unstructured grids. As a result there are no cell shape limitations. Cells can be tetrahedral, quadrilateral, pyramidal, or any other shape. In this effort three different grids were generated, all of which were unstructured. The Howard University baseline grid and the Howard University refined grid were fully unstructured meshes while the VA baseline gird contained a structured layer along the surface to capture the boundary layer effects. These were compared to published CFD results on the M2129 duct which used an unstructured grid with a structured layer along the surface [3].

When creating a structured mesh, spacing the cells is of utmost importance. Clustering nodes near the surface in order to resolve the boundary layer for a viscous problem is straight forward because the user has control over the number and spacing of cells of each zone or sub-domain. On the other hand, when creating an unstructured mesh, some kind of refined or structured domain must be present in order to resolve the boundary layer.

Figure 2 depicts Howard University's baseline grid. It is a 652,578 cell unstructured grid comprised of only tetrahedral cells, even in the near wall region. The lack of cells near the wall will create problems for a turbulent Navier-Stokes solution due to the inability to resolve the boundary layer. For a CFD solver to resolve the boundary layer a more refined region must be included near the wall. Whether a fully structured sub-domain or prismatic layer of extruded triangles is preferable depends on the surface mesh type of quadrilaterals or triangles respectively.

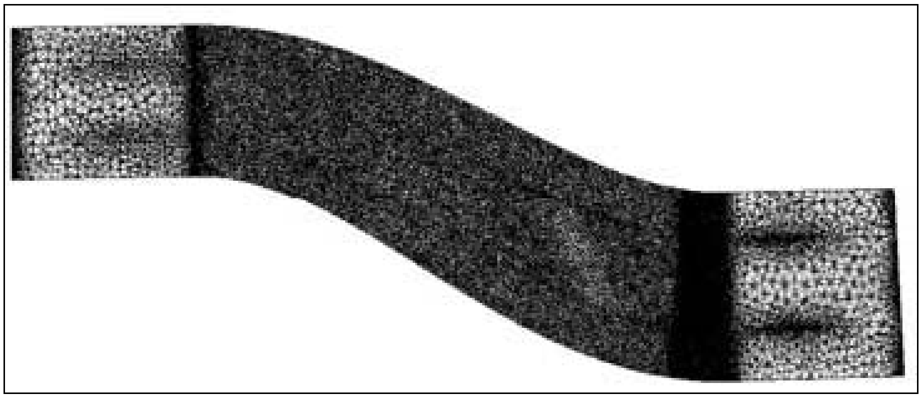


Figure 2: Howard University Baseline Grid File created from Gridgen mesh

Howard University also created a refined grid with 2,130,958 cells. It was created from the same geometry database with the same technique. It is comprised of only tetrahedral cells and no refined boundary layer mesh. Despite the additional cells (and therefore a more highly resolved flowfield in general) a typical flow solver will still have difficulty resolving the boundary layer using this grid because of the lack of a refined mesh in the boundary layer region. This second grid was created in an effort to improve the flow field solution strictly through increased grid points. Changes in the way this grid was run will be discussed in a later section.

Figure 3 is the baseline unstructured grid created by the Air Vehicles directorate (VA). In this grid a closely packed structured mesh is present near the wall. The structured region should encompass the entire boundary layer and cell growth away from the surface should not exceed 20%. In other words, the next cell away from the surface should not be more than 20% larger than the adjacent cell closer to the surface. In the structured or prismatic region this translates directly to height from the surface.

Figures 2 and 3 vary greatly in suitability for use in numerical simulation. The mesh in Figure 2 was created from a geometry file used to create the original structured mesh and included many sub-domain regions. An advantage of unstructured meshes is that these divisions are unnecessary and a smooth volume mesh can be created from the surface mesh as seen in Figure 3. The rapid change in cell size throughout the grid shown in Figure 2 could lead to numerical errors and an incorrect solution; whereas, the grid shown in Figure 3 features smooth gradual changes in cell size throughout the mesh.

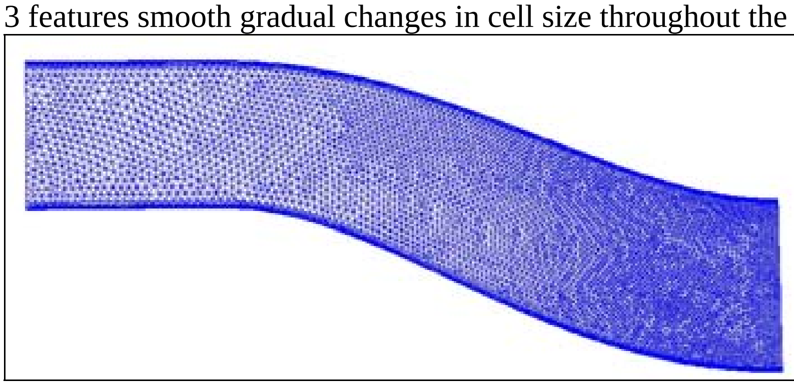


Figure 3: VA Grid File created from Gridgen mesh

Simulation Details and Results

In total, three solutions were obtained from the study, one using the Howard University Baseline grid shown in Figure 2, one using the Howard University refined grid, and one using the VA Baseline grid shown in Figure 3. These solutions were compared to previously published CFD results on the M2129 duct. Table 1 is a summary of the boundary conditions and initial flow conditions, as well as some important input and output parameters. All three cases in the current study were run as steady-state, with ten sweeps per iteration, the Spalart-Allmaras one equation turbulence model, and a CFL number of 10^5. Since AVUS is unconditionally stable, CFL numbers greater than one can be used and still maintain a stable solution. The number of sweeps per iteration was low for a turbulent Navier-Stokes problem. To accurately solve a turbulent N-S case, 16 to 32 sweeps per iteration should be used [4]. The referenced solution used an unstructured grid with 830,000 grid cells and was run at steady-state using the Spalart-Allmaras turbulence model and a CFL number of 50.

Table 1: Summary Boundary Conditions and Output Parameters

Simulation	Inflow BC	Outflow BC	Inflow Conditions	Outflow Conditions	Initial Conditions	y^+	Iterations	Supersonic Cells
Howard Baseline	Source	Farfield	M=0.80 P=101125 Pa T=287.22 K	M=0.80 P=83220 Pa	M=0.8 P=101125 Pa T=287.22 K	1261	2000	248617
	Riemann Invariants	Modified Riemann Invariant						
Howard Baseline 2	Source	Farfield	M=0.487 P=101125 Pa ρ=1.225 kg/m^3 u=165.4 m/s	M=0.80 P=76280 Pa	M=0.8 P=101125 Pa T=287.22 K	760.6	1000	162840
	User-Defined	Modified Riemann Invariant						
VA Baseline	Source	Sink	M=0.65 Pt=101139.2 Pa Tt=292.8 K Mdot=3.4289Kg/s	P=90550 Pa	M=0.25 P=90325 Pa T=269.986 K	2.065	2627	0
	Mass Flow	Static Pressure						
Mohler	Inflow BC	Outflow BC	Pt=101125 Pa Tt=287 K	P=83220 Pa	M=0.78 Pt=101125 Pa Tt=287 K	~0.5		0
	Uniform Total Pressure and Temperature	Uniform Static Pressure						

Figure 4 shows the results from the Howard University Baseline case. It is clearly evident that a normal shock is located near the second turn of the duct. This is due to the improper application of the boundary conditions. Specifying a Mach number at the outlet, when using a Farfield boundary condition, is just an initial condition input since only static pressure is held constant at the exit for the Riemann Invariant method. All other variables are allowed to float for subsonic outflow. At the inflow boundary condition a Source with the Riemann Invariants method was used to the specify flow condition and requires static pressure, static temperature, and Mach number to be specified. The pressure and temperature specified are actually the total conditions at the entrance of the duct. Total and static conditions are the same only if the flow is not moving. In this case the inflow is moving at Mach 0.80. Instead of a uniform total pressure of 101125 Pa, the entrance condition was a uniform total pressure of 154114 Pa and a total temperature of 323.98 K.

As a result the specified inlet static pressure was higher than the exit static pressure, which will create problems in a diffusing duct. In order for the static pressure to decrease the velocity of the flow must increase. Therefore, the inlet conditions adjusted to supersonic flow and the duct acted like a divergent supersonic nozzle and a normal shock formed inside the duct to satisfy continuity and to attain the specified exit static pressure.

The y^+ value for this simulation is extremely high, which is a good indication that the boundary layer is not being resolved. If a refined boundary layer mesh is present near the wall this number would be much lower. When using a one-equation turbulence model the y^+ value should be less than 4. The CFL number is also very high which makes the solution try to converge very quickly. Such an overdriving of the solution could introduce numerical instabilities to the solution and produce erroneous results.

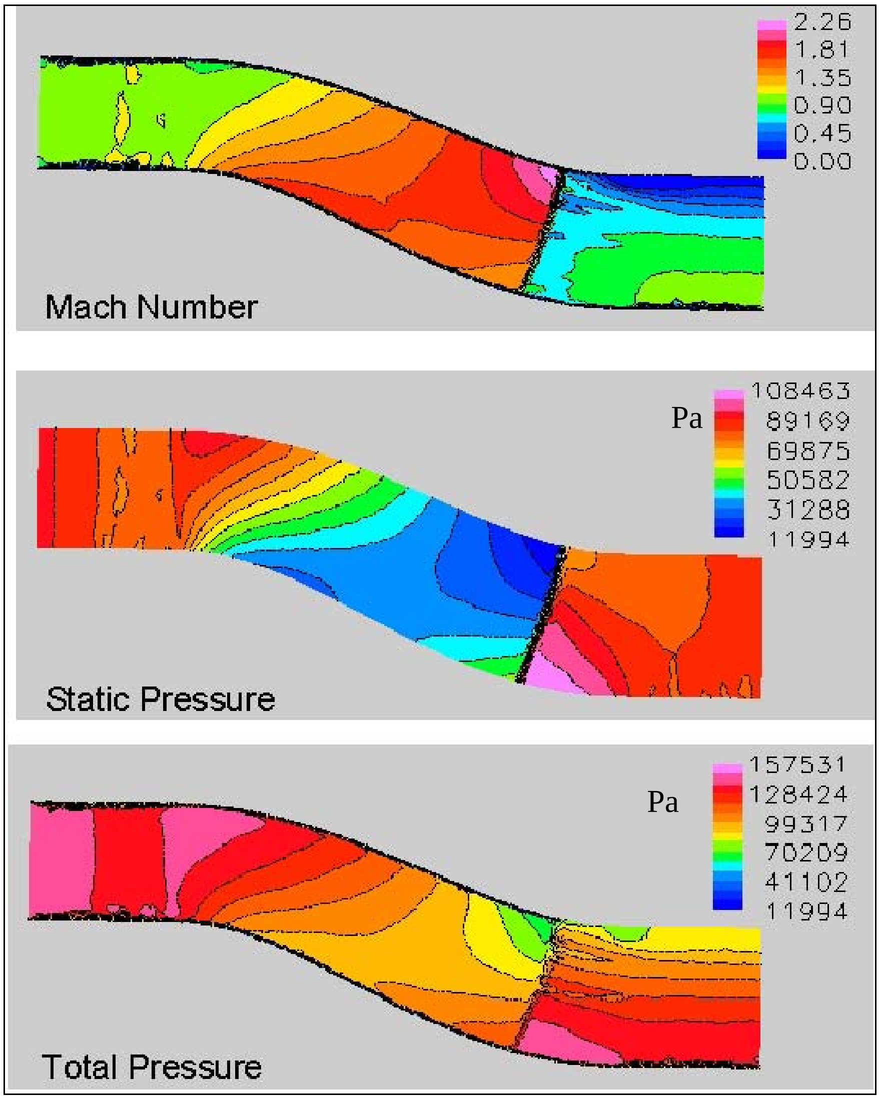
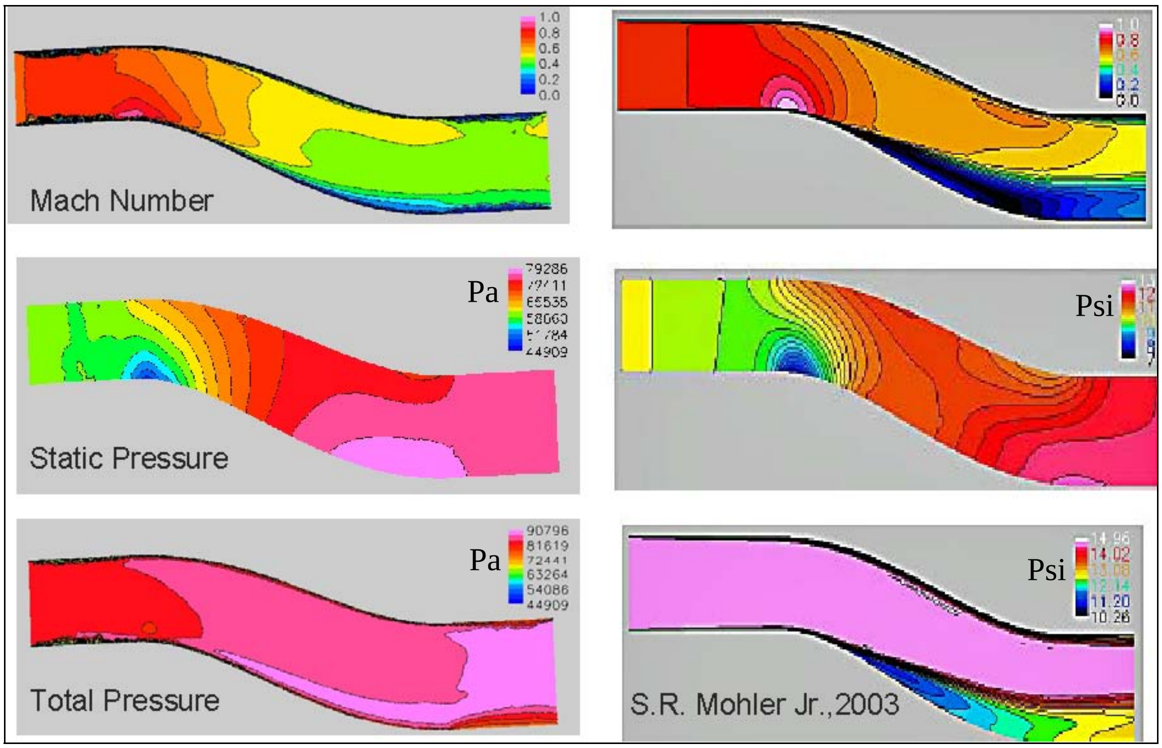


Figure 4: Results from Howard University Baseline Grid

Figure 5 shows the results of the Howard University's refined grid case. The static pressure gradient through the duct is representative of the referenced data although the exit static pressure was 76280 Pa not 83700 Pa, which Mohler ran in his simulations. Although the entrance boundary condition is not ideal, the flow state approached what should have been specified. As the solution progressed, a static pressure near 60 kPa and a Mach of 0.80 developed. The y^+ value for this case is significantly lower than the Baseline simply because of the overall grid refinement. There are three times as many cells; therefore, there are more cells near the wall resulting in a lower y^+ value. Since the flow states are similar, if the grid near the wall had been properly refined with a structured grid layer, the boundary layer probably would have developed and more separation would have been seen on the lower surface resulting in better agreement between the solutions.



a. Howard Results b. Mohler Results

Figure 5: a. Howard Baseline 2 Results, pressure scale is in Pa b. S.R. Mohler Jr. Results, pressure scale is in psi

Figure 6 shows the results from the CFD performed by the Air Vehicles Computational Sciences Branch. These were run on the grid in Figure 3. The only adjustments made to the AVUS input file were the flow initialization and select boundary conditions. The number of iteration sweeps is still lower than what is recommended and the CFL number is still high. As seen in Table 1, the flow field is initialized to a state different than the boundary conditions. The lower initial Mach number allows the boundary condition to propagate through the duct. Initializing the flow in this way can eliminate numerical instabilities and non-physical flow warnings that may otherwise result from the high CFL number. The lower initial Mach number forces the solution to go in the right direction.

AVUS allows for the choice of boundary conditions including Source, Sink, and Farfield. The Sink boundary condition can allow a mass flow rate or a constant static pressure at the patch. A Sink with the static pressure method is the proper outlet boundary condition because it allows the density and velocity to vary. The Sink boundary condition also allows performance analysis on the flow through the patch. Mass flow can be calculated along with pressure recovery and inlet distortion. This is preferable over the Farfield condition because the boundary patch is very near the region of interest, the flow through the duct. Farfield boundary conditions are reserved for flow entering or exiting far from the region of interest. A Source boundary condition was implemented for the intake of the inlet duct. The Mass Flow method for the source boundary condition asks for the total conditions for pressure and temperature at the entrance along with a mass flow. The mass flow rate corresponded to an entrance Mach of 0.65. In the VA case the boundary layer was resolved as seen by the y^+ value of less than 4. The boundary layer developed and separated as would be expected from the geometry.

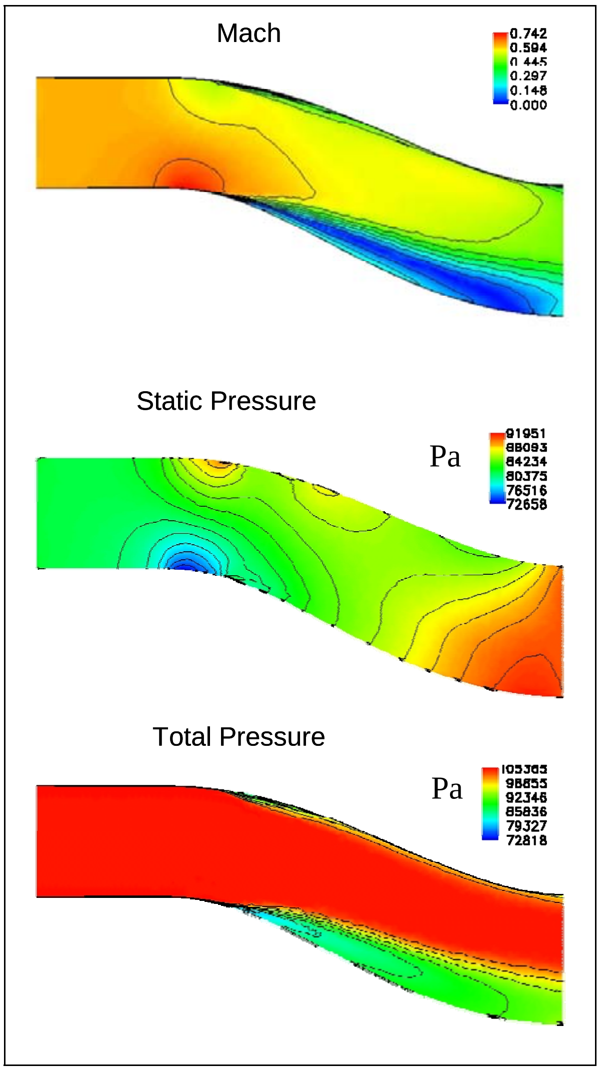


Figure 6: VA Baseline Results

Conclusions

This study demonstrated the ability to use AVUS to solve internal flow fields for diffusing S-ducts with separated flow through the VA baseline case. The study also emphasized some of the missteps along the way that led to inaccurate solutions. These missteps included lack of a refined, structured, boundary layer grid region, high CFL numbers, and improper boundary conditions. It is hoped that in publishing these results others will not repeat these same mistakes. This study demonstrated the criticality of proper boundary conditions, the benefit of initializing the flow field, and the importance of understanding critical solution health parameters such as y+. Additionally, the importance of a well constructed grid has been demonstrated. This is of particular importance for solutions that involve or may potentially involve separated boundary layers. Other parameters that may prove crucial are CFL number and sweeps per iteration.

References

1. Tillman, C.P, R.L. Kimmel, G.A. Addington, and J.H. Myatt. "Flow control Research and Applications at the AFRL'S Air Vehicle Directorate." AIAA Paper 2004-2622.

2. AGARD FTP Working Group 13, "Air Intakes for High Speed Vehicles," AR-270, Sept. 1991.

3. Mohler, S.R. "Wind-US Flow Calculations for the M2129 S-Duct Using Structured and Unstructured Grids," NASA/CR-2003-212736 and AIAA-2004-0425, December 2003, p.14.

4. Air Force Research Laboratory, CFD Research Branch, "Air Vehicles Unstructured Solver (AVUS) User's Manual," Jan 6, 2005.