

NASA/MSFC's Calculation for Test Case 1a of ATAC-FSDC Workshop on After-body and Nozzle Flows

November 15-16, 2006
Noordwijk, Netherlands

Joseph H. Ruf
Thermal Analysis and Combustion Fluids Branch
Marshall Space Flight Center, Huntsville, AL. 35812.

Introduction

Mr. Ruf of NASA/MSFC executed the CHEM computational fluid dynamics (CFD) code to provide a prediction of the test case 1a for the ATAC-FSDC Workshop on After-body and Nozzle Flows. CHEM is used extensively at MSFC for a wide variety of fluid dynamic problems. These problems include; injector element flows, nozzle flows, feed line flows, turbomachinery flows, solid rocket motor internal flows, plume-vehicle flow interactions, etc.

Approach

CHEM is a 3D, finite volume, density based CFD code developed by Dr. E. Luke of Mississippi State University with funding from MSFC. For test case 1a CHEM was run on an axisymmetric domain, steady state. The k-omega turbulence model with a compressibility correction of the Sarkar form was implemented. The compressibility correction implemented was not the standard Sarkar form. In previous work it had been calibrated for cold flow nozzle separation locations.

The grid for this analysis used structured cells inside the nozzle. Unstructured cells were used in the farfield. The structured cells were converted to unstructured cells prior to running CHEM. Figures 1 and 2 show several images of the finest grid implemented.

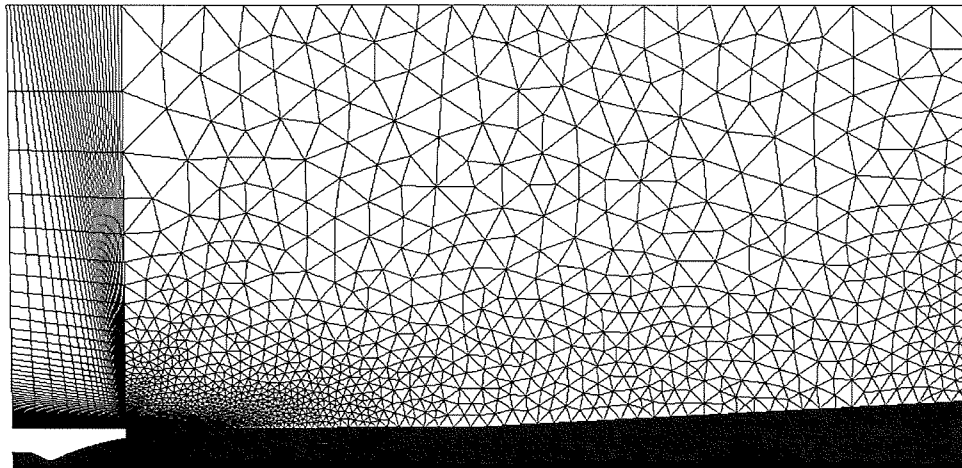


Figure 1. The grid for the full domain.

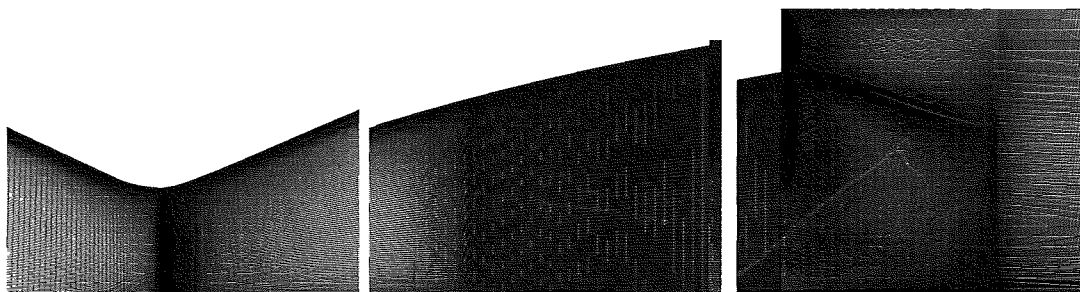


Figure 2. Details of the finest grid.

Multiple grid densities were run. The dimensions of the grids in the nozzle and the total number of nodes in the entire domain are provided in Table 1. Grid independence was obtained with the fine grid. Other grid densities were implemented to ensure grid independence. They are not discussed here for the sake of brevity. For this analysis, the number of cells for this calculation was not a concern. No attempt was made to develop a grid with more efficient use of cells.

Table 1. Grid Information.

	Nodes in Nozzle	Total Nodes
Coarse	111x351	129,572
Medium	181x501	229,104
Fine	181x801	314,404

The spacing of the first nodes off the wall at the throat and nozzle exit were $1e-7m$ and $1e-6m$, respectively. The y^+ of the first cell off the nozzle was less than 1.0. The results presented below are from the fine grid.

The boundary conditions were as follows. The nozzle inlet was specified as an isentropic inlet with total pressure of 25bar and total temperature of 300°K. The nozzle walls were set to no-slip adiabatic. The left-hand and top farfield boundaries were set as inflow boundaries with pressure of 1.003bar, temperature of 300°K and Mach 0.05 in the axial direction. The right-hand boundary was specified as an outlet with pressure of 1.0bar. The gas was N_2 and assumed to follow the ideal gas law.

Results

The Mach number contours are shown in figures 3 and 4. The Mach disk is bowed slightly. It is, however, normal to the centerline for a short length. The maximum Mach number in the flowfield is 4.974.

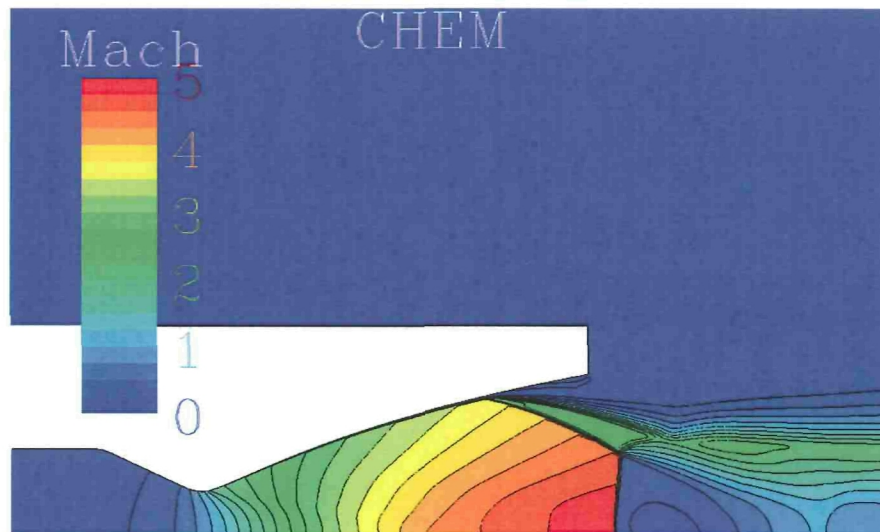


Figure 3. The Mach number contours.

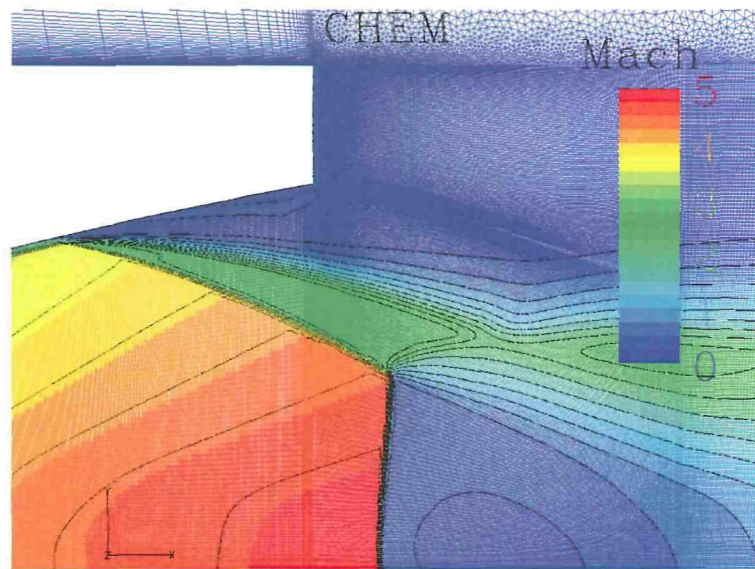


Figure 4. Details of the Mach number contours and grid at the Mach disk.

The nozzle wall pressure is compared to the test data in figures 5 and 6. The calculation appears to have produced a good prediction of the location of the separation. The only minor difference appears to be the slope of the pressure rise after the separation.

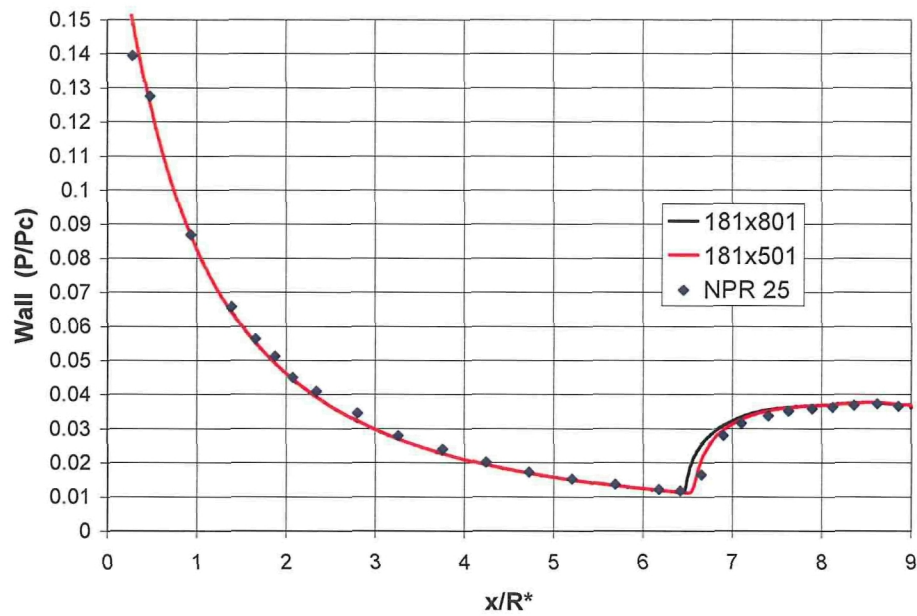


Figure 5. Nozzle wall pressure compared to test data.

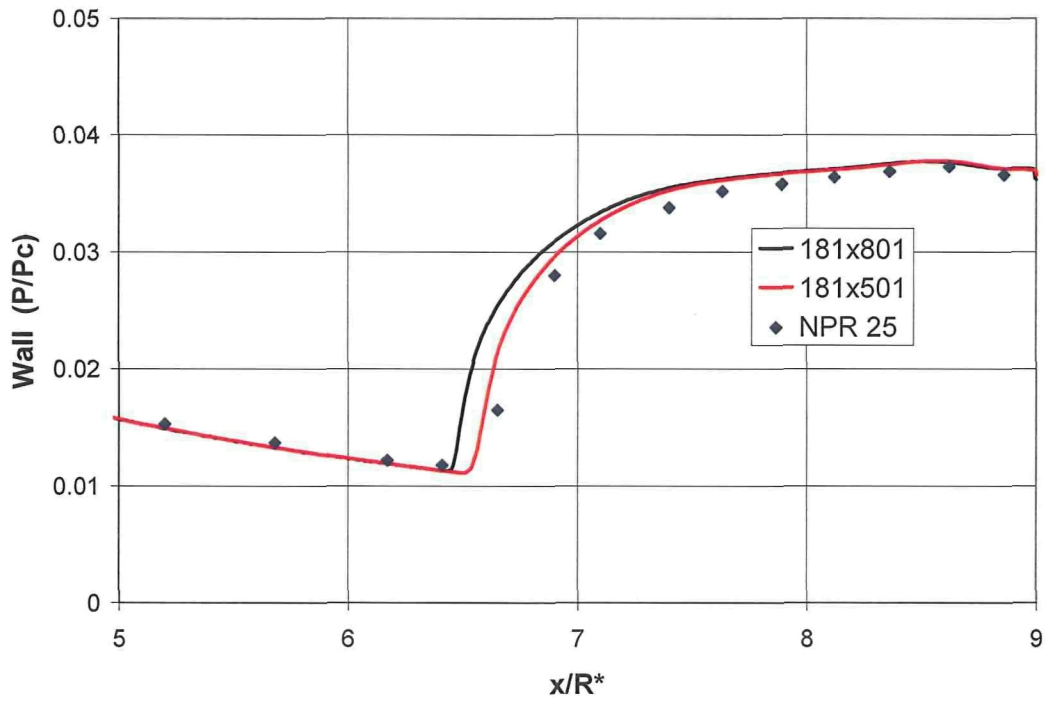


Figure 6. Enlargement of the region of separation.

The pressure along the centerline of the CFD solution is shown in figure 7. The experimental value reported for the axial location of the Mach disk at the centerline is shown, as a dashed line, in figure 7. The Mach disk in the CFD solution, indicated by the sharp pressure rise, was slightly upstream of the experimental location.

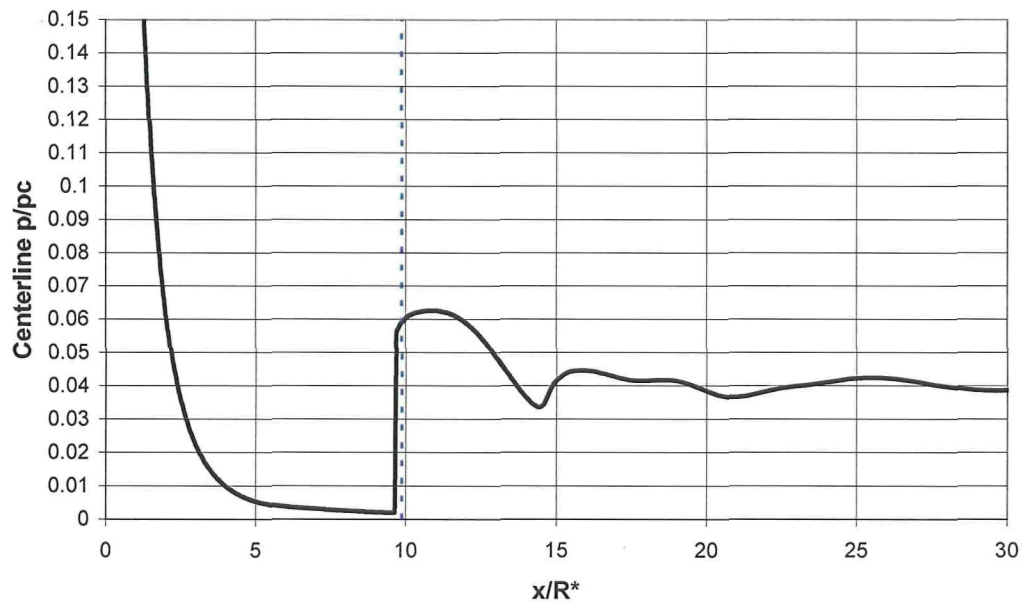


Figure 7. Centerline pressure from CFD and experimental Mach disk location.