Computational Analyses in Support of Sub-scale Diffuser Testing for the A-3 Facility - Part I: Steady Predictions

Daniel Allgood[1] and Jason Graham[2]
NASA Stennis Space Center, MS 39529, USA

and

Vineet Ahuja[3] and Ashvin Hosangadi[4]
Combustion Research and Flow Technology, Pipersville, PA 18947

Simulation technology can play an important role in rocket engine test facility design and development by assessing risks, providing analysis of dynamic pressure and thermal loads, identifying failure modes and predicting anomalous behavior of critical systems. Advanced numerical tools assume greater significance in supporting testing and design of high altitude testing facilities and plume induced testing environments of high thrust engines because of the greater inter-dependence and synergy in the functioning of the different sub-systems. This is especially true for facilities such as the proposed A-3 facility at NASA SSC because of a challenging operating envelope linked to variable throttle conditions at relatively low chamber pressures. Facility designs in this case will require a complex network of diffuser ducts, steam ejector trains, fast operating valves, cooling water systems and flow diverters that need to be characterized for steady state performance. In this paper, we will demonstrate with the use of CFD analyses's advanced capability to evaluate supersonic diffuser and steam ejector performance in a sub-scale A-3 facility at NASA Stennis Space Center (SSC) where extensive testing was performed. Furthermore, the focus in this paper relates to modeling of critical sub-systems and components used in facilities such as the A-3 facility. The work here will address deficiencies in empirical models and current CFD analyses that are used for design of supersonic diffusers/turning vanes/ejectors as well as analyses for confined plumes and venting processes. The primary areas that will be addressed are: (1) supersonic diffuser performance including analyses of thermal loads (2) accurate shock capturing in the diffuser duct; (3) effect of turning duct on the performance of the facility (4) prediction of mass flow rates and performance classification for steam ejectors (5) comparisons with test data from sub-scale diffuser testing and assessment of confidence levels in CFD based flowpath modeling of the facility. The analyses tools used here expand on the multi-element unstructured CFD which has been tailored and validated for impingement dynamics of dry plumes, complex valve/feed systems, and high pressure propellant delivery systems used in engine and component test stands at NASA SSC.

The analyses performed in the evaluation of the sub-scale diffuser facility explored several important factors that influence modeling and understanding of facility operation such as (a) importance of modeling the facility with Real Gas approximation, (b) approximating the cluster of steam ejector nozzles as a single annular nozzle, (c) existence of mixed subsonic/supersonic flow downstream of the turning duct, and (d) inadequacy of two-equation turbulence models in predicting the correct pressurization in the turning duct and expansion of the second stage steam ejectors. The procedure used for modeling the facility was as follows: (i) The engine, test cell and first stage ejectors were simulated with an axisymmetric approximation (ii) the turning duct, second stage ejectors and the piping downstream of the second stage ejectors were analyzed with a three-dimensional simulation utilizing a half-plane symmetry approximation. The solution i.e. primitive variables such as pressure, velocity components, temperature and turbulence quantities were passed from the first computational domain and specified as a supersonic boundary condition for the second simulation. (iii) The third domain comprised of the exit diffuser and the region in the vicinity of the facility (primary included to get the correct shock structure at the exit of the facility and entrainment characteristics). The first set of simulations comprising the engine, test cell and first stage ejectors was carried out both as a turbulent real gas calculation as well as a turbulent perfect gas calculation. A comparison for the two cases (Real Turbulent and Perfect gas turbulent) of the Mach Number distribution and temperature distributions are shown in Figures 1 and 2 respectively. The Mach Number distribution shows small yet distinct

---

[1] Aerospace Technologist, Stennis Space Center, MS 39529-6000, and AIAA Senior Member.
[2] Jacobs Technology CFD Engineeer, Stennis Space Center, MS 39529-6000, and AIAA Member.
[3] Senior Research Scientist, 6210 Keller's Church Rd., and AIAA Senior Member.
[4] Principal Scientist, 6210 Keller's Church Rd., and AIAA Senior Member.

differences between the two cases such as locations of shocks/shock reflections and a slightly different impingement point on the wall of the diffuser from the expansion at the exit of the nozzle. Similarly the temperature distribution indicates different flow recirculation patterns in the test cell. Both cases capture all the essential flow phenomena such as the shock-boundary layer interaction, plume expansion, expansion of the first stage ejectors, mixing between the engine plume and the first stage ejector flow and pressurization due to the first stage ejectors. The final paper will discuss thermal loads on the walls of the diffuser and cooling mechanisms investigated.
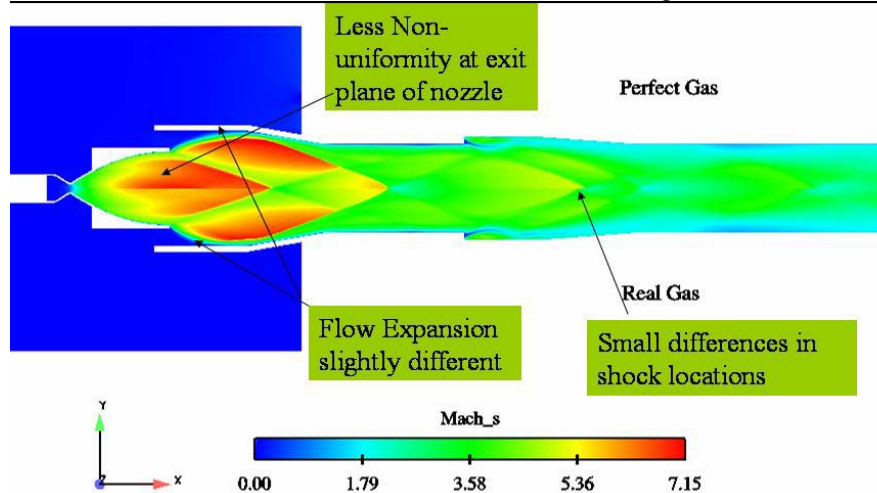

Figure 1. Comparison of Mach Number Distribution for Engine Test Cell, Diffuser and First Stage Ejector simulations – Turbulent Real Gas Simulation (Below) and Turbulent Perfect Gas Simulation (Above).
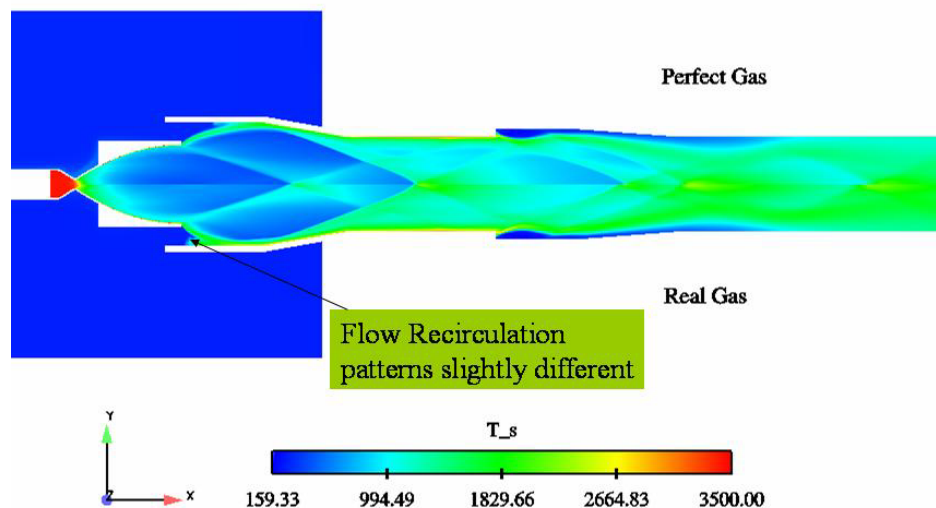

Figure 2. Comparison of Temperature Distribution for Engine Test Cell, Diffuser and First Stage Ejector simulations – Turbulent Real Gas Simulation (Below) and Turbulent Perfect Gas Simulation (Above).

The second series of simulations were performed with the turning duct and the second stage ejectors. These simulations were carried out with and without a two-equation turbulence model. It was found that excessive dissipation in the two-equation turbulence model lead to over-pressurization in the turning duct leading to a predominantly subsonic flow at the exit of the turning duct in the facility. In contrast, the inviscid calculation showed predominantly supersonic flow aft of the turning duct. Figure 3 shows a Mach number comparison between the inviscid calculation and the turbulent calculation. The turbulent flow solution also shows a large recirculation or separated flow region near the inner wall of the duct as the flow exits the turning duct. As the consequence, the expansion and flow dynamics of the second stage ejector is very different in the two cases – in the inviscid case, flow from the second stage ejectors expands significantly more than the turbulent case as indicated by the Mach Number distribution in Figure 3 near the plane of the second stage ejector.

2
American Institute of Aeronautics and Astronautics

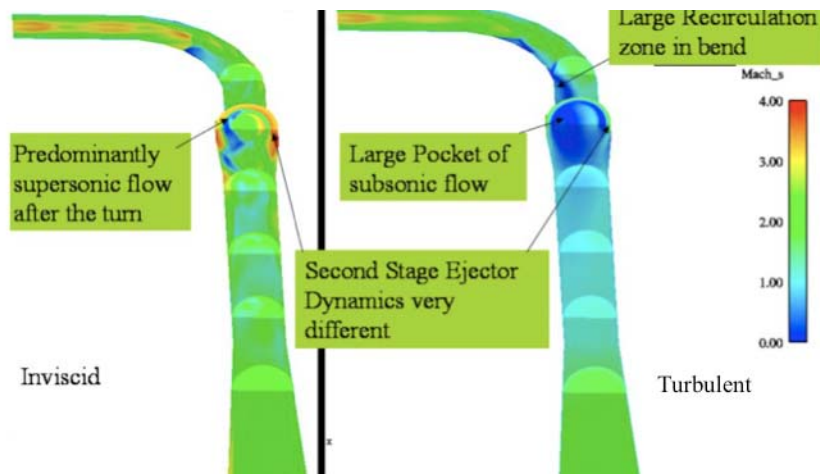RELEASED - Printed documents may be obsolete; validate prior to use.

Figure 3. Comparison of Mach No. Distribution for the second set of simulations – Inviscid Real Gas Simulation (Left) and Turbulent Real Gas Simulation (Right).

A comparison is shown between the measured and predicted pressure distributions along the walls of the facility in Figure 4. The figure shows a large disparity between the measured pressure and the predicted pressure from the two-equation turbulence model in regions near the exit of the turning duct and near the second stage ejectors. The large pressure fluctuations(approx 9 psia) due to the shock structure seen in the sensor data near the exit of the elbow are captured by the inviscid simulation which shows very good agreement with the test data.

The final paper will discuss the modeling framework, analysis procedure, detailed analysis justifying our approach such as the use of a single annular nozzle, steam ejector modeling approximations and improvements to the turbulence modeling framework for more accurate predictions, as well as heat loads on the supersonic diffuser and the turning duct and required cooling mechanisms.
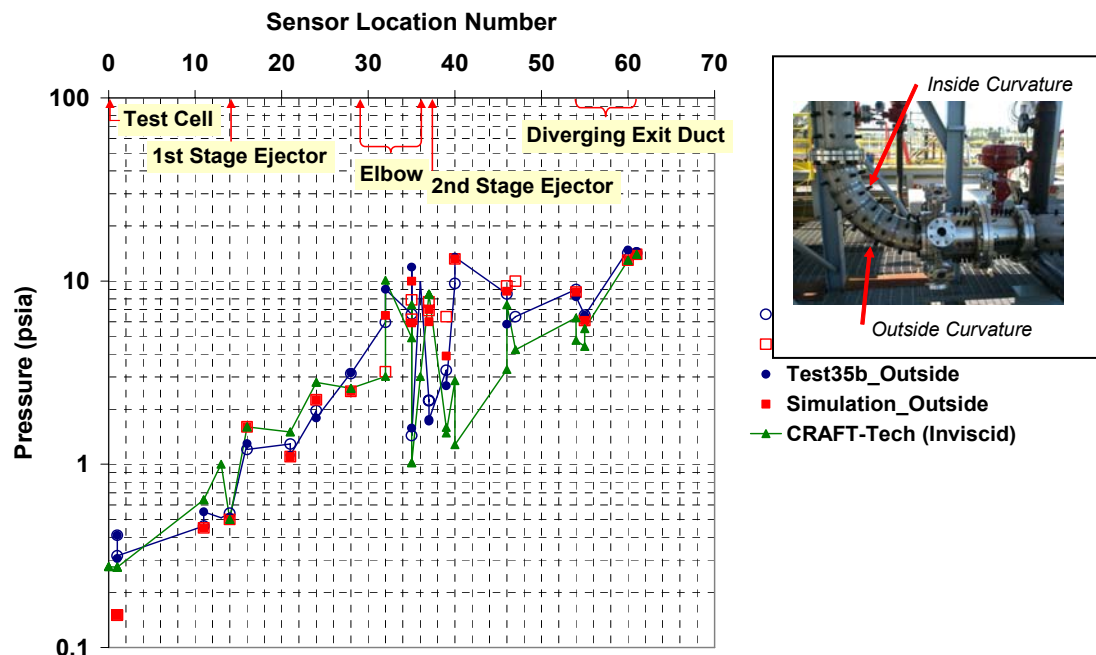

Figure 4. Comparison of Measured Pressure and Predicted Pressure with the Turbulent Real Gas Simulations and Inviscid Real Gas Simulations. The Inviscid Simulation shows marked improvement in the Region of the Second Stage Ejectors and the Turning Duct.

3
American Institute of Aeronautics and Astronautics