

A Numerical Simulation of a Normal Sonic Jet into a Hypersonic Cross-flow

Damon K. Jeffries, Ramesh Krishnamurthy, and Suresh Chandra
NASA Center for Aerospace Research, College of Engineering
North Carolina A & T State University, Greensboro NC 27411

Abstract

This study involves numerical modeling of a normal sonic jet injection into a hypersonic cross-flow. The numerical code used for simulation is GASP (General Aerodynamic Simulation Program.) First the numerical predictions are compared with well established solutions for compressible laminar flow. Then comparisons are made with non-injection test case measurements of surface pressure distributions. Good agreement with the measurements is observed. Currently comparisons are underway with the injection case. All the experimental data were generated at the Southampton University Light Piston Isentropic Compression Tube.

Introduction

A jet in cross-flow (JICF) consists of a jet exhausting at a large angle into a freestream flow. It is a flow field which is relevant to a wide variety of technologies and applications. When the primary importance is the mixing of the jet with the cross-flow, then an extensive application could involve heat transfer, gas turbines, and fuel injection. The JICF also plays a crucial role in technologies such as vehicle attitude control, the hover in-ground effect (HIGE) of some military aircraft, and the vertical and/or short takeoff/landing (V/STOL) aircraft. The lift forces created by certain complicated flows are applied to V/STOL aircraft technologies. The jet induced force created by the JICF flow field can be applied to vehicle control, such as orbital flight maneuvering and re-entry of the Space Shuttle or potentially for high velocity and/or high altitude atmospheric vehicles.

One area of interest results from a transverse jet being directed into a hypersonic, laminar crossflow. The ensuing interaction of these two flows causes a change in surface pressure in the vicinity of the injector. When this modified pressure distribution is integrated a force is produced which can be several times larger than the nominal jet thrust. An application of this jet induced force is vehicle attitude control. When encountering situations where conventional aerodynamic surfaces cannot function properly, the system (of a jet induced force) is particularly advantageous. It is widely believed that this may be due to the considerable aerodynamic heating effects associated with large flight speeds or possibly due to the low density of the surrounding medium. Also Vehicle Attitude Control is still possible from the thrust of a jet induced force, even when the external flow is so rarefied that the interaction force is negligible. Orbital Flight Maneuvering and re-entry of the Space Shuttle are well known examples of the control jet application, but other potential uses of this flow field can be applied to the high velocity and/or high altitude atmospheric vehicles [1].

The 72nd AGARD Fluid Dynamics Panel Meeting and Symposium on Computational and Experimental Assessment of Jets in Cross Flow [2] was the first meeting since 1981 where the primary theme was Jets in Cross Flow. The conference concluded with the general understanding that there is a need for improved prediction methods for the JICF problem. Investigations into transient flow features of a supersonic jet in a low speed cross flow [3], the separated flow generated by a vectored jet in a crossflow [4], and scalar mixing in the subsonic

jet in cross-flow [5] further define characteristics of the JICF flow field. Despite these efforts and others there are few results available for **laminar** hypersonic flows, a combination which will be encountered by **re-entry** and high **altitude** vehicles over some portion of their flight path. The experimental problem being simulated results from a series of nominally two dimensional **experiments**[1,6]. Interaction force data was obtained in a **laminar**, hypersonic **freestream** flow. To gain more knowledge about the influence of the jet within this part of the flow field, detailed measurements of the separated region have been made with various **injectant** species. Heat transfer and oil flow visualizations have also been used to elucidate the separation and reattachment process upstream and downstream of the jet.

Analysis

The experimental problem being modeled examines the interaction between a **two-dimensional**, Normal Sonic Jet of Nitrogen (or Methane) and a two-dimensional, Hypersonic Cross-Flow of Nitrogen over a flat plate. Performed in the Southampton University Light Piston Isentropic Compression Tube, a free stream of nitrogen at Mach 6.69 flows over an Isothermal Flat Plate with a sharp leading edge. This **freestream** interacts with a gas injected at room temperature via a normal slot jet located 0.0745 m from the leading edge of the plate (Figure 1).

The experimental problem was modeled and analyzed by the General Aerodynamic Simulation Program (GASP), version 2.2. The code GASP solves the full Reynolds-averaged, compressible form of the **Navier-Stokes**, energy and species conservation equations. The code can be run in explicit, or implicit, space marching or elliptic modes. The governing equations are discretized using a finite volume approach and can be solved for one, two, and three dimensional models. The code can also utilize several thermodynamic, turbulent, and chemistry models [7]. In GASP the experimental problem was modeled using input decks which contain all of the relevant fluid flow information. The problem simulation and solution was performed on a grid which was generated by a FORTRAN program created outside of GASP. The numerical model consists of a two-dimensional, viscous, fully **laminar** flow with three different regions in the solution process: a upstream region, that is space marched; an injection region, that is globally iterated; and a downstream region, this is space marched. It is necessary to setup the solution process in this way because the injection into the flow will cause **re-circulation** of the flow in an area (injection region) surrounding the jet. This type of flow can only be captured by using a global iteration process. The flow upstream and downstream of this injection and or region can be simulated using a space marching solution scheme.

The inlet of the flow is supersonic, and therefore, the velocities, static temperature, static pressure, and species concentration are fixed values calculated from the conditions specified in [1]. The plate has the no-slip condition applied to it and is held at a constant temperature of 300K. This condition is appropriate for the high velocity and small test time encountered by the flow. The other boundaries of this two-dimensional control volume are setup to be extrapolated from the interior to first order accuracy. For the **inviscid** fluxes, the van Leer flux vector splitting with first order spatial accuracy is used. This **inviscid** flux calculation is used in conjunction with the **MIN-MOD** limiting algorithm. Once this more robust set-up creates an initial solution, the problem solution is completed by using Roe's flux difference splitting with **Harten** correction calculation with third order spatial accuracy. And the **MIN-MOD** limiting algorithm is used once again to control strong oscillations. The thermodynamics and chemistry options were set up to define the flow as an ideal, non-reacting, mixture of two species.

The computational simulation is performed with a two-dimensional grid. There are 201 grid points in the x-direction and 130 in the y-direction. An exponential stretching formula was

used to create appropriate clustering in the leading-edge region, the region downstream of injection and at the plate surface to capture all flow characteristics. And in the injection region the grid is uniformly spaced with very small increments to capture large flow variations. The grid was validated by comparing the **Blasius** solution curve from results of the present numerical model with this grid to the exact solution in [8].

Results and Discussions

All of the numerical simulations were performed on a **Cray T-90** platform. In the results reported here, only injection of nitrogen is considered.

The main focus of this research is to produce a valid numerical model of the experimental problem of a normal sonic jet injected into a hypersonic cross-flow. The process of modeling the experimental problem was divided into two main parts. Part one consists of the verification and validation of a non-injection case of the problem, which is a simpler form of the problem. And in part two, the verification and validation of the more complicated injection case is considered.

The non-injection case, which is a modified version of the experimental problem where the normal sonic jet is turned off, was set-up and analyzed first. The non-injection case was set up to be space marched across the entire plate in the direction of the flow (x-direction.) This case can be space marched because it will not experience any of there-circulation of flow and other characteristics associated with injection into the **freestream**. The **inviscid** flux in the x-direction was set to a full flux with no splitting with a **fully** upwind second order accurate spatial accuracy. And in the y-direction Roe's flux difference splitting with **Harten** correction was used with a upwind-biased third order accurate setting. A **MIN-MOD** limiting was used. And only thin-layer contributions in the y-direction were considered.

The results showed that the non-injection simulation of the problem was very accurate. This model was verified using Van Driest calculations of the **laminar** compressible boundary layer on an flat plate for velocity and temperature profiles. The velocity and temperature profile behavior was very similar to that of the Van Driest solutions [8]. The model was also validated by comparing the measured surface pressure data [1] with the numerical model (figure 2). This comparison showed that the model predictions were accurate with only a $\pm 2\%$ error from the experimental values. Once the non-injection case was verified and validated to ensure that proper and accurate results were being produced, it was used to develop the injection case model.

The results from the non-injection case was used as an initial solution for simulating the injection case. The injection of the gas into the **freestream** results in **re-circulation** of the flow, therefore, the injection case is treated as an elliptic problem where the global iteration is appropriate. The model was also set up to consider the flow as a **fully laminar**, compressible flow with all the viscous terms in the x- and y-directions. The assumption of a **laminar** flow in our simulations is based on the observation of Ball [9] that the Reynolds number at the separation location was lower than its critical value and that there was a clear lack of evidence of transition in the thermographs record with no-injection.

The validation of the injection case model is currently underway. This case is a more complicated problem to solve than the non-injection case, so mesh sequencing was incorporated to facilitate the solution. Mesh sequencing is the process of converging a solution using a series of progressively finer meshes or grids. Vector plots of the present calculations show that re-circulation of the flow to be occurring. The injection case model is also being validated by comparing **surface** pressure values from the experiment to the surface pressure results from the numerical model (figure 3.) The model shows reasonably good agreement with the

measurements in the region between the leading edge and lip of injector. No measurements were reported for the region downstream of the injector.

Concluding Remarks

The non-injection case of the model shows very good agreement with the experimental measurements and in the injection case of the model good agreement with measurements particularly in the upstream region is observed. Agreement with the measured surface pressure may be improved with a more appropriate model setup. A multi-zone setup will be incorporated to model solve the injection case. This setup will have three zones, an injection region which will be globally iterated and two space marching region upstream and downstream of the injection region. Such a setup would allow for fine-tuning of the various parameters in the numerical model and the procedure is expected to yield better results.

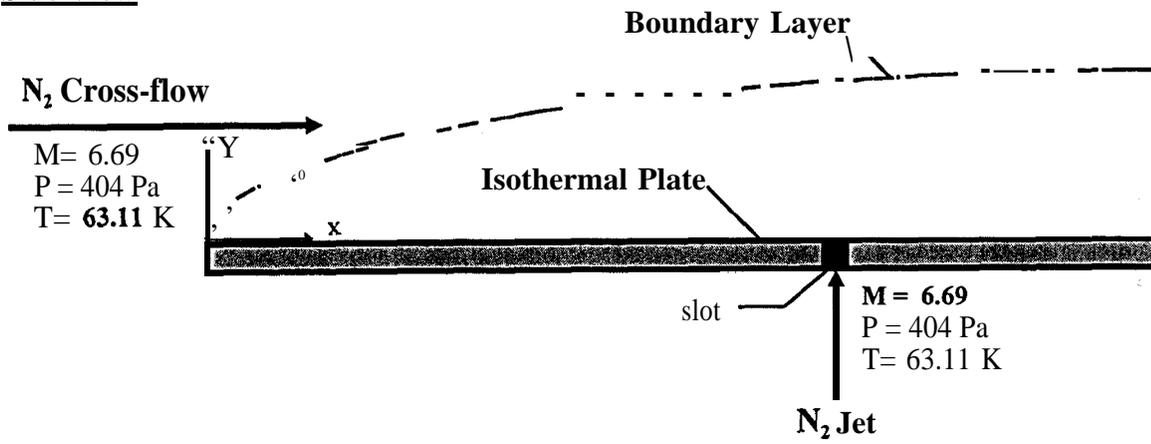
Acknowledgments

The numerical computations reported here were performed on Cray T90 systems and were made possible by the North Carolina Supercomputing Center in Research Triangle Park, NC. Financial support provided by the grant Nagw-2924 from NASA is gratefully acknowledged.

References

- [1] Powrie, H.E.G., Ball, G.J., East, R.A. "Comparison of the Interactions of Two and Three Dimensional Transverse Jets with a Hypersonic Free Stream", AGARD Conference Proceedings 534, November, 1993, pp. 20-1--20-8.
- [2] AGARD Conference Proceedings 534, Computational and Experimental Assessment of Jets in Cross Flow, North Atlantic Treaty Organization, November 1993.
- [3] X. Zhang, D.W. Hurst, and G.M. Lilley, "Transient Flow Features of a Supersonic Jet in a Low Speed Cross Flow", AGARD Conference Proceedings 534, Reference 3, North Atlantic Treaty Organization, November 1993.
- [4] A. Krothapalli and C. Shih, "Separated Flow Generated by a Vecteded Jet in a Crossflow", AGARD Conference Proceedings 534, Reference 5, North Atlantic Treaty Organization, November 1993.
- [5] S.H. Smith, A. Lozano, M.G. Mungal, and R. K. Hanson, "Scalar Mixing in the Subsonic Jet in Crossflow", Reference 6, AGARD Conference Proceedings 534, Reference 5, North Atlantic Treaty Organization, November 1993.
- [6] Powrie, H.E.G., Ball, G.J., and East, R.A., "Experimental Study of a Two Dimensional Control Jet with a Hypersonic Flow", IUTAM Conference on Aerothermochemistry of Spacecraft and Associated Hypersonic Flows, Marseille, September 1992
- [7] McGrory, William D., Slack, David C., Applebaum, Michael P., Walters, Robert W., Gasp version 2.2 User's Manual, AeroSoft, Inc., 1993.
- [8] Anderson, John D., Jr., Computational Fluid Dynamics, McGraw-Hill, Inc., New York, 1995.
- [9] Ball, G.J., personal correspondence, August 5, 1996.

Side View



Top View

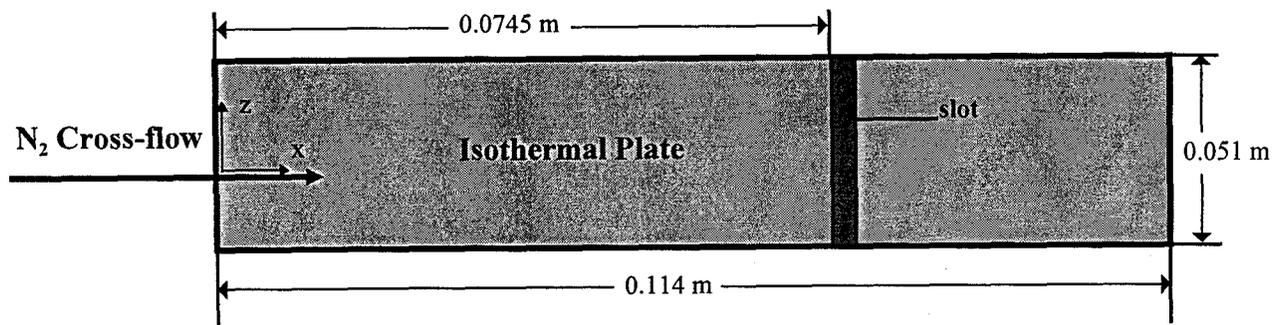


Figure 1. Schematic of experimental problem to be simulated numerically.

Surface Pressure Comparison

(Non-Injection Case)

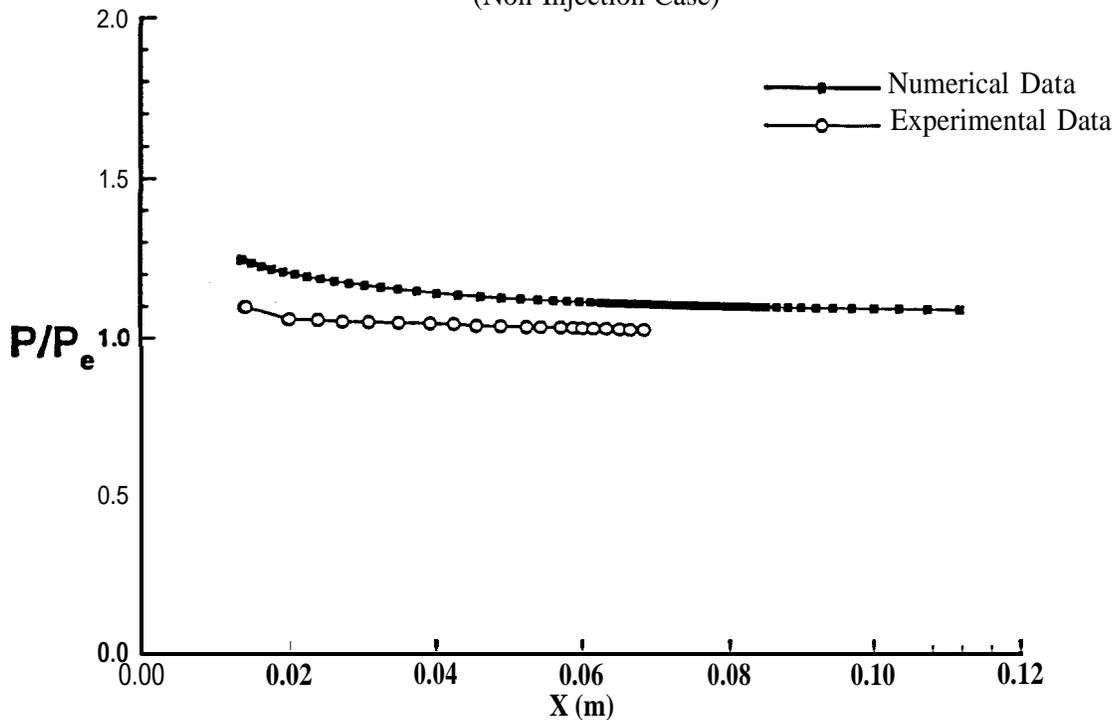


Figure 2. Surface Pressure Comparison Plot

Surface Pressure Comparison

(Injection Case)

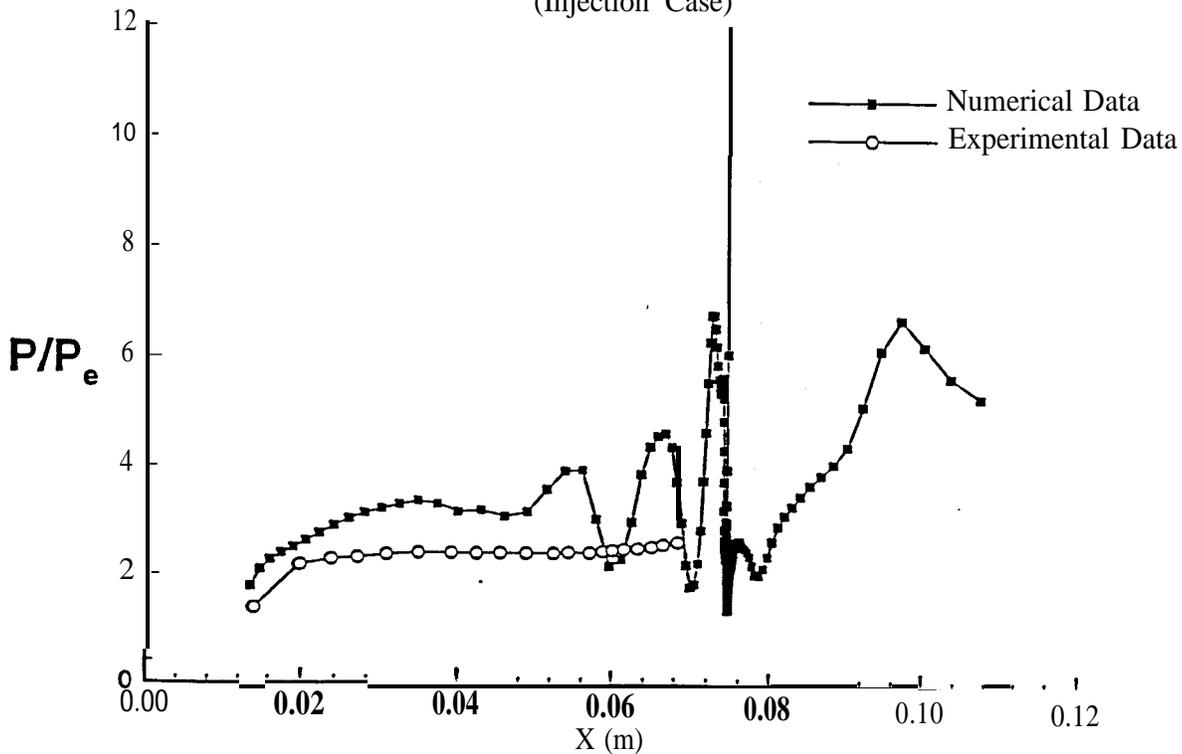


Figure 3. Surface Pressure Comparison Plot (Injection)