NAVIER STOKES SOLVER FOR A SUPERSONIC FLOW OVER A REARWARD-FACING STEP

Stephen Akwaboa*, and Frederick Ferguson**
Center for Aerospace Research
North Carolina A &T State University
Greensboro, North Carolina, 27411
sa020326@ncat.edu

Abstract: In this study, the classical MacCormack explicit unsteady scheme are modified and smartly programmed to serve as the basis for a Navier-Stokes Solver. A two dimensional code capable of solving the perfect gas dynamic equations are developed. Geometry of particular interest used to solve the fluid flow problem is the rearward-facing step. The governing equations are programmed using FORTRAN to solve the 2D planar Navier-Stokes equations. The solution results are visualized using TECPLOT. Supersonic flow over the rearward facing step is strategically solved using a flat plate solution paradigm. Parametric studies performed indicate that the Mach number and the step height affect flow characteristics such as corner expansion, recirculation zone, and the base pressure which are of great importance in the design of SCRAMJET engine. As the Mach number increases, both the base pressure and the recirculation region decrease significantly. Also an increase in step height causes the base pressure and the size of the recirculation region to decrease. Since the size and flow properties of the recirculation zone could strongly affect the ignition and combustion processes of the overall combustor of a SCRAMJET engine, the volume of this recirculation region must therefore be well predicted for SCRAMJET engines.

Nomenclature:

ρ	Density
u	Velocity in the x-direction
v	Velocity in the y-direction
e	Internal energy
T	Temperature
V	Magnitude of velocity vector
q_x	Heat flux in the x-direction
q_y	Heat transfer in the y-direction
p	Pressure
μ	Dynamic (absolute) viscosity
τ_{xx}	Normal stress in the x direction
τ_{yy}	Normal stress in the y direction
τ_{xy}	Shear stress in the y direction exerted in a plane perpendicular to x direction
k	Thermal conductivity
U	Solution vector of Navier-Stokes equation
E_i	Inviscid flux vector in the x-direction
E_v	Viscous flux vector in the x-direction
E	E_i-E_v

* Ph.D. Student, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Student Member
** Director, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Senior Member

F_i	Inviscid flux vector in the y-direction
F_v	Viscous flux vector in the y-direction
Δy	Incremental change in distance along y-direction
Δx	Incremental change in distance along x-direction

1. Introduction:

While computational fluid dynamics (CFD) hold a lot of promise and has led to significant improvements in the art of aircraft design, many of its anticipated gains are yet to be realized. In particular, the aerodynamic design of an aircraft requires a critical understanding of the flow around candidate shapes, since the complexity of the problem itself precludes analytic solutions. However, CFD has not yet fullfilled its promise of readily providing a workable solution. Nonetheless, the careful use of CFD tools can be very useful in two phases of aerodesign analysis, namely, the conceptual and the preliminary phases. Traditionally, designers rely on wind tunnels to determine aerodynamic properties, but the expense of running different configurations quickly becomes prohibitive, both in terms of direct financial cost and time. Notwithstanding these CFD related advantages, the time and expertise required to move from a preliminary aerodesign configuration to a desired candidate is still too long. The key goal, therefore, is to improve the efficiencies of CFD tools thus making aerodesigners more productive.

Various numerical schemes have been used to study flows over different flow geometries, e.g. flat plate, rearward-facing step, and base flows. These studies led to the existence of a large number of experimental and analytical solutions [1-9]. The task of a posteriori evaluation of the quality of the solution to a CFD code problem is an important issue for aerospace designers, researchers, and code developers. Also, the inability of CFD tools to facilitate inexperienced users with limited training is a serious set back. Today, there is a growing family of sophisticated commercial CFD packages using diverse solution methodologies on the market

.

This paper focuses on the development of a new CFD tool and is designed to overcome the aforementioned shortcomings. Further, this tool is designed with the users in mind; it is built to be user-friendly with capabilities to accept any conceptual fluid flow through the smart use of boundary conditions. Also, it is designed to guide users in formulating their concepts by providing workable solutions in a short time. On the whole, it is geared to provide reliable predictions to aerospace concepts before valuable resources are committed to their realization. In this paper , the complete set of Navier-Stokes equations with minimum assumptions is solved numerically. The numerical solution process uses the MacCormack's explicit time marching scheme.

2. Governing Equations:

The equations describing the planar flow a Newtonian fluid are the two dimensional, unsteady Navier-Stokes equations. These can be written in non-dimensional strong conservation form in Cartesian coordinates as:

$$\frac{\partial U}{\partial t} + \frac{\partial}{\partial x}\left(E + E_v\right) + \frac{\partial}{\partial y}\left(F + F_v\right) = 0 \tag{1}$$

where the vector, U, is the solution vector and the vectors, E and F, are the inviscid fluxes in the x and y directions, respectively, and where the viscous fluxes are defined by the vectors, E_v and F_v. The expressions used to defined the flux vectors are described below,

* Ph.D. Student, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Student Member

** Director, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Senior Member

$$E = \begin{pmatrix} \rho u \\ \rho u^2 + \dfrac{1}{\gamma M_\infty^2}\rho T \\ \rho u v \\ \rho u H \end{pmatrix}; \qquad E_v = -\begin{pmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \left(u\tau_{xx} + v\tau_{xy} - q_x\right) \end{pmatrix};$$

$$F = \begin{pmatrix} \rho v \\ \rho u v \\ \rho v^2 + \dfrac{1}{\gamma M_\infty^2}\rho T \\ \rho v H \end{pmatrix}; \qquad F_v = -\begin{pmatrix} 0 \\ \tau_{xy} \\ \tau_{yy} \\ \left(u\tau_{xy} + v\tau_{yy} - q_y\right) \end{pmatrix};$$

The solution vector is defined as:

$$U = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho E_t \end{pmatrix} \qquad (2)$$

The energy terms, E_t and H, along with the viscous terms are also defined as follows:

$$E_t = \frac{1}{\gamma(\gamma-1)M_\infty^2}.T + \frac{u^2+v^2}{2}; \qquad H = \frac{1}{(\gamma-1)M_\infty^2}.T + \frac{u^2+v^2}{2};$$

$$\tau_{xx} = \frac{1}{\mathrm{Re}_\infty}\frac{2}{3}\mu\left(2\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y}\right); \qquad \tau_{yy} = \frac{1}{\mathrm{Re}_\infty}\frac{2}{3}\mu\left(2\frac{\partial v}{\partial y} - \frac{\partial u}{\partial x}\right); \qquad \tau_{xy} = \frac{1}{\mathrm{Re}_\infty}\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right);$$

$$q_x = -\frac{1}{\mathrm{Re}_\infty(\gamma-1)\mathrm{Pr}.M_\infty^2}.k\frac{\partial T}{\partial x}; \qquad q_y = -\frac{1}{\mathrm{Re}_\infty(\gamma-1)\mathrm{Pr}.M_\infty^2}.k\frac{\partial T}{\partial y};$$

* Ph.D. Student, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Student Member
** Director, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Senior Member

The non-dimensionalization process was conducted in accordance with the following expressions:

$$\bar{\rho} = \frac{\rho}{\rho_\infty} \; ; \; \bar{u} = \frac{u}{u_\infty} \; ; \; \bar{v} = \frac{v}{v_\infty} \; ; \; \bar{T} = \frac{T}{T_\infty} \; ; \; \bar{k} = \frac{k}{k_\infty}$$

$$\bar{\mu} = \frac{\mu}{\mu_\infty} = \frac{k}{k_\infty} = \bar{k} \; ; \; \bar{t} = \frac{u_\infty t}{L} \; ; \quad \bar{x} = \frac{x}{L} \; ; \; \bar{y} = \frac{y}{L} \; ; \quad \bar{a} = \frac{a}{u_\infty}$$

3. Solution Methodology:

MacCormak's method, first introduced in 1969, for the past 15 years has been the most popular explicit finite-difference method for solving fluid flow problems, Ref. 10. This method is based on Taylor's series expansion of U vector with respect to time using the predictor-corrector method. The expressions describing the predictor and corrector stages of the MacCormack scheme are defined as follows,

Predictor Step: $$\frac{\partial U^f}{\partial t} = -\left[\frac{(E+E_v)_{i+1,j} - (E+E_v)_{i,j}}{\Delta x}\right] - \left[\frac{(F+F_v)_{i,j+1} - (F+F_v)_{i,j}}{\Delta y}\right]$$

Corrector Step: $$\frac{\partial U^b}{\partial t} = -\left[\frac{(E+E_v)_{i,j} - (E+E_v)_{i-1,j}}{\Delta x}\right] - \left[\frac{(F+F_v)_{i,j} - (F+F_v)_{i,j-1}}{\Delta y}\right]$$

Average Derivative: $$\frac{\partial U^{avg}}{\partial t} = 0.5\left[\frac{\partial U^f}{\partial t} + \frac{\partial U^b}{\partial t}\right]$$

In addition, the process of updating the solution vector is given as follows:

$$U^{new} = U^{old} + \frac{\partial U^{avg}}{\partial t}\Delta t \qquad (2)$$

* Ph.D. Student, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Student Member
** Director, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Senior Member

4. Preliminary Results:

The graphs below show some preliminary results obtained so far. The pressure distribution, shown in Figure 1 depicts that as the flow advances downstream, there is a large static pressure drop at the location of the step. This pressure drop is not ideal for the SCRAMJET engine as it affects the flame-holding ability of the engine. Increasing the Mach number decreases the base pressure significantly. From the design methodology point of view, the gross engine thrust from a SCRAMJET is directly proportional to the total pressure [11]. Hence the supersonic combustor of a hypersonic engine requires a higher combustor inlet Mach number to avoid total pressure loss and achieve a higher thrust. This, however, affects the successful ignition of the engine due to low base pressure and the small size of the recirculation zone.

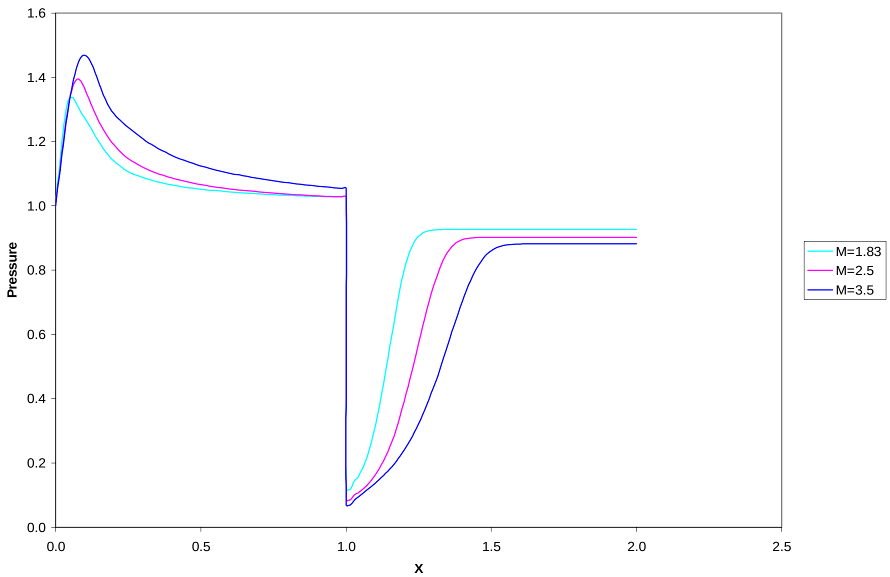
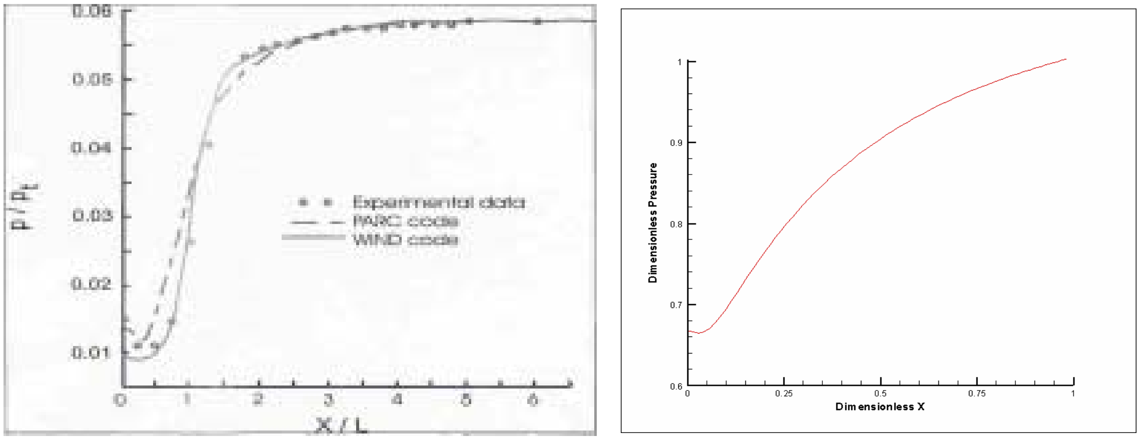


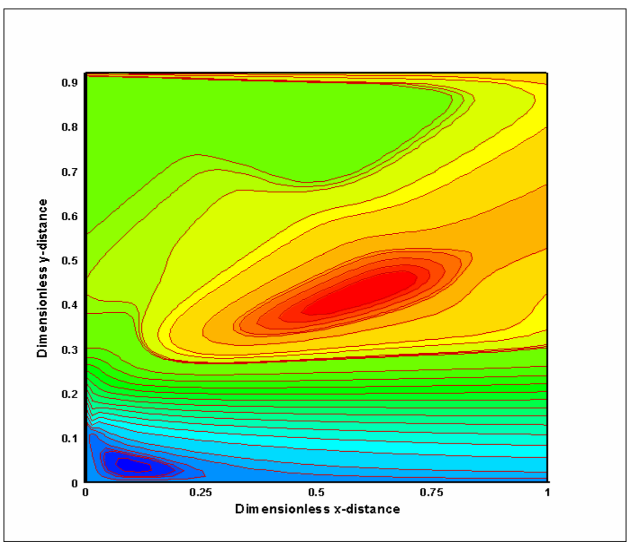
Figure 1. Static Pressure Distribution (M=2.5, Re=1000, sh = $15\Delta y$)

* Ph.D. Student, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Student Member
** Director, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Senior Member



(a) (b)

Fig.2: Pressure Distribution on the surface downstream the step
(a) Previous Study [9], (b) present work



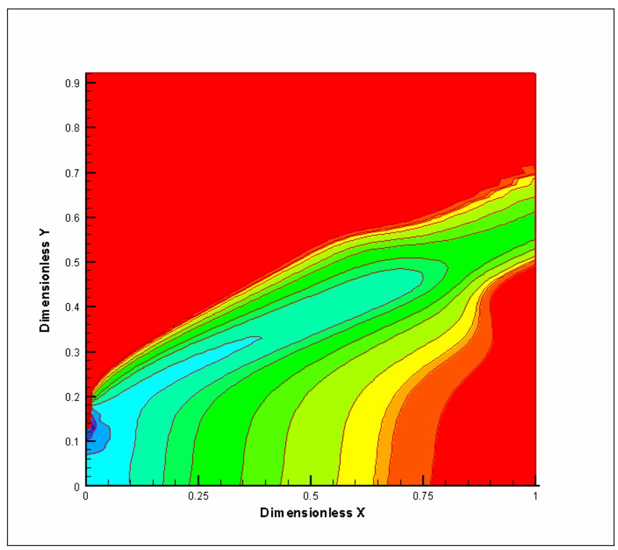


Figure 3: Mach number distribution

Figure 4: Pressure contour downstream of step

* Ph.D. Student, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Student Member
** Director, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Senior Member

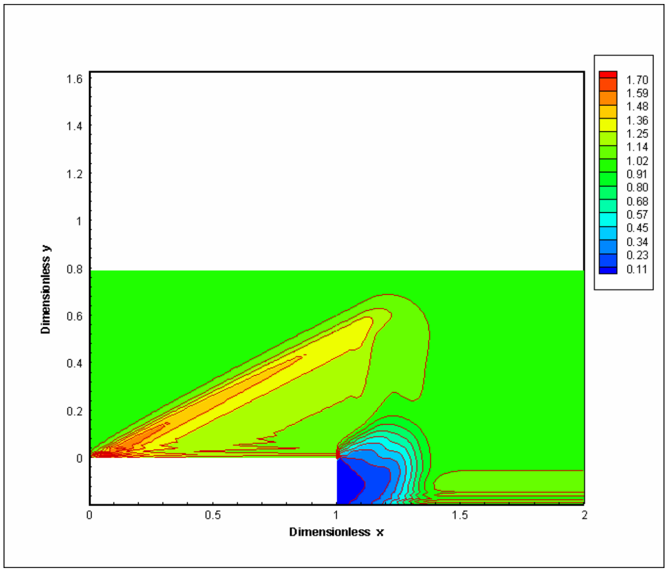


Figure 5: Pressure distribution

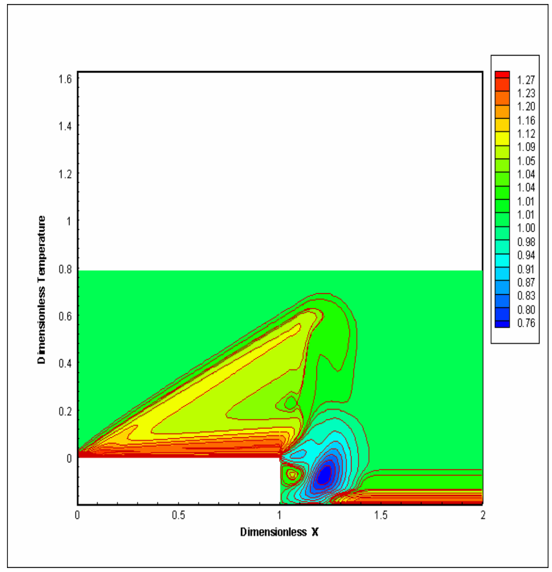


Figure 6: Temperature distribution

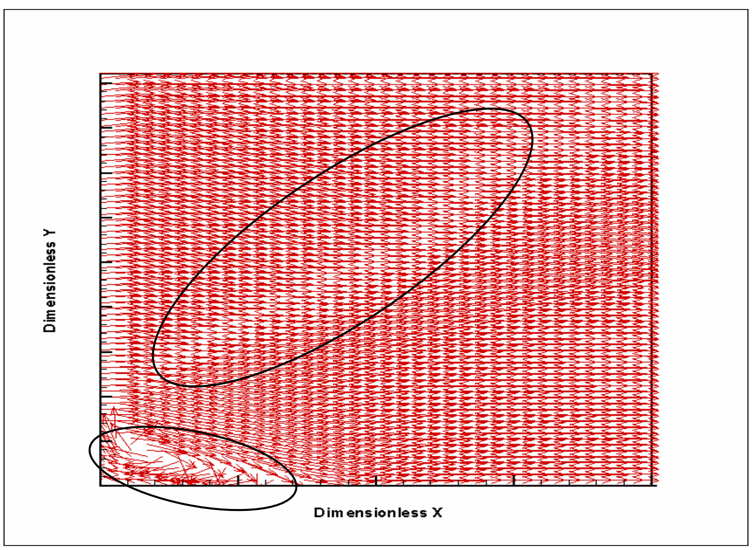


Figure 7: Velocity field vector plot

* Ph.D. Student, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Student Member
** Director, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Senior Member

5. Conclusion:

The study presents an improved numerical method to solve the Navier-stokes equations for a flow over a rearward facing step. The validity of these results is confirmed by established experimental data. For example, as it can be seen in Figures 2 and 6 several characteristics of the flow field, namely, in the expansion-corner, in the free shear layer, the recirculation zones, and the recompression region, are well captured by the numerical scheme.

References

[1] A. S. Yang, W. H. Hsieh, K.K. Kao, "Theoretical Study of supersonic flow separation over a rearward-facing step", Journal of propulsion and power, Vol. 13, No. 2, March-April 1997, pp 324-326.

[2] Eugene S. Love, Carl E. Grigsby, "A new shadow graph taken for the observation of conical flow phenomenon in supersonic flow and preliminary result obtained for a triangular wing", National Advisory Committee for Aeronautics, Technical Notes 2950, 1953.

[3] Engineering Mechanics Research Corporation (EMRC), http://www.emrc.com/webpages/fluid/rocket.htm.

[4] Chang-Shing Tai, Ar-Fu Kao, "Navier-Stoke Solver for hypersonic flow over a slender cone", Journal of Space Craft and Rockets, Vol. 31, No. 2, March-April 1994, pp. 215-222.

[5] M. S. Kim, J. M. Loellbach, K. D. Lee, "Effect of gas model on hypersonic base flow calculations", Journal of Spacecraft and Rockets, Vol. 31, No. 2, Mar-Apr, 1994, pp. 223-230.

[6] Hemdan, Hamdi T., "Hypersonic flows over slender pointed-nose elliptic cones at zero incidence", Acta Astronautica, Vol. 45, No. 1, July 1999, pp 1-10.

[7] METALL-SPEZIALROHR GMBH Company, http://www.msr-special.de/produkte.html.

[8] D. R. Eklund, D. G. Fletcher, R.J. Hartfield Jr., G. B. Northam, C. L. Dancey, "A Comparative-Experimental investigation of Mach 2 flow over a rearward-facing step", Computers and Fluids, Vol. 24, No. 5, 1995, pp 593-608.

[9] NPARC Alliance Validation Archive, http://www.grc.nasa.gov/WWW/wind/valid/backstep/backstep02/backstep02.html

[10] J. D. Anderson, Jr., "Computational Fluid Dynamics-The basis with applications", McGraw-Hill, Inc., 1995.

[11] Klaus A. Hoffmann and Steve T. Chiang, "Computational fluid dynamics for engineers", Engineering Education System, March 1995.

* Ph.D. Student, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Student Member

** Director, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Senior Member

[12] Anderson, John D., " Modern Compressible Flow With Historical Perspective" , 2nd ed.,1990 McGraw-Hill Publishing Company, 1995.

[13] NASA Glenn Research Center: www.grc.nasa.gov/www/k-12/airplane/atmosmet.html.

* Ph.D. Student, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Student Member
** Director, Center for Aerospace Research, NCATSU, Greensboro, NC 27411, AIAA Senior Member