AIAA 81-0185R

Two-Dimensional Analysis of a Scramjet Inlet Flowfield

Ajay Kumar*
NASA Langley Research Center, Hampton, Va.

Abstract

A COMPUTER code has been developed to analyze the flow in a supersonic combustion ramjet (scramjet) inlet using the two-dimensional Navier-Stokes equations. A numerical coordinate transformation is used to generate boundary-fitted curvilinear coordinates. Turbulence is modeled by a two-layer eddy viscosity model. The governing equations are solved by an explicit, finite-difference method. The code can analyze both inviscid and viscous flow with no strut, one strut, or multiple struts in the flowfield. The application of the two-dimensional analysis in the preliminary parametric design studies of the scramjet inlet is discussed briefly and results are presented for several inlet configurations.

Contents

A comprehensive research program is underway at the NASA Langley Research Center to develop an airframe-integrated, hydrogen-fueled scramjet engine for hypersonic propulsion.[1] The basic concept is shown in Fig. 1. The engine system is divided into several identical modules, one of which is shown in the figure with a sidewall removed. A cross-sectional view is also shown in the figure. The module has a fixed geometry inlet with wedge-shaped sidewalls. Sweep of these sidewalls, in combination with a recess in the cowl, allows spillage to occur efficiently with a fixed geometry inlet, thus allowing it to start over a range of Mach numbers. Inlet compression is completed by three wedge-shaped struts located at the minimum-area section which also provide locations for fuel injection.

The flow in a scramjet inlet is highly three dimensional, possibly turbulent, and has complex shock-expansion wave interactions. It also involves strong shock-boundary layer interactions which may result in separated regions. To analyze such flows, it is necessary to use the full Navier-Stokes equations with proper turbulence modeling. Due to the complex nature of the flowfield, most of the scramjet inlet design has been based on experimental studies with little analytical work to support it. The purpose of the present investigation is to develop a two-dimensional (2-D) numerical code to analyze scramjet inlet flowfield using the full Navier-Stokes equations in conservative law form.[2] The turbulence is modeled by a two-layer eddy viscosity model.[3] In order to facilitate the treatment of a general inlet geometry with embedded bodies, a numerical coordinate transformation is used which generates a set of boundary-fitted curvilinear coordinates. It transforms the physical domain into a rectangular computational domain with uniform mesh spacing. The embedded bodies in the flowfield are transformed into slits. The explicit, time-dependent, finite-difference method of MacCormack[4] is used to solve the governing equations. The details of the governing equations, numerical transformation, and the method of solution are given in Ref. 2. The code, in its present form, can analyze inviscid, laminar, and turbulent flows with no strut, one strut, or multiple struts in the flowfield.

In order to verify the code, several model problems are solved in Ref. 2. The results of these model problems show that the present code predicts the complex supersonic flow very well. The code was then used in a quasi-three-dimensional sense to analyze actual scramjet inlet configuations of the type shown in Fig. 1. If the shock waves in the inlet do not detach and if the end effects (cowl plate and top surface of the inlet) are neglected, the velocity component parallel to the sweep should remain unchanged. The flow disturbances should occur in a plane normal to the sweep. The flow can be solved using the 2-D code in the normal plane with proper Mach number component. The solution in the plane of the cowl can then be obtained by projecting the 2-D solution and properly superimposing the constant velocity component. This gives the solution for the three-dimensional inlet under the assumptions stated earlier and can be used to obtain the inlet performance parameters such as spillage, etc.

The preceding approach is verified by analyzing a three-strut inlet for which experimental results are available. The details of the inlet configuration and test conditions are given in Ref. 5. Figure 2 shows a comparison of the sidewall pressure distribution in a plane parallel to the cowl located at about half of the inlet height. This particular plane is chosen so that the cowl shock does not affect the flow up to the point of comparison in the axial direction (the present analysis cannot account for cowl shock disturbances). It is seen that the present calculations are in very good agreement with the measured values.

The type of agreement obtained for the three-strut inlet gives credibility to the code in its use as a tool for parametric studies in inlet design. In the present study, this is done by analyzing one and two strut inlets. The sidewalls of these inlets are swept at 33 deg. Realistic flow conditions listed in Table 1 are used in the analysis. Here, M_∞ is the flight Mach number, M_I, P_I, T_I are the flow conditions at the face of the inlet, and M_{IN} is the Mach number normal to the sweep.

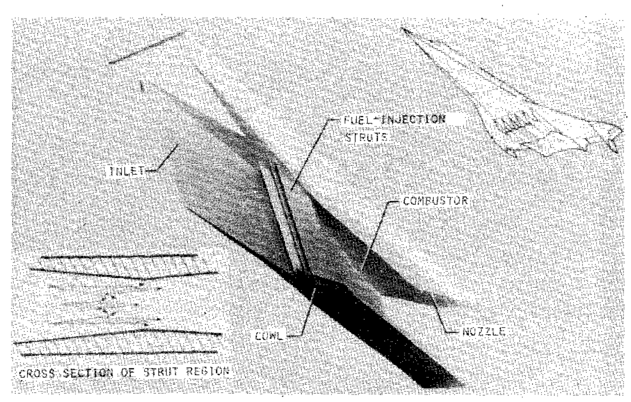

Fig. 1 Scramjet engine module and its cross section.

Presented as Paper 81-0185 at the AIAA 19th Aerospace Sciences Meeting, St. Louis, Mo., Jan. 12-15, 1981; submitted Feb. 9, 1981; synoptic received July 24, 1981. This paper is declared a work of the U.S. Government and therefore is in the public domain. Full paper available from AIAA Library, 555 W. 57th Street, New York, N.Y., 10019. Price: Microfiche, $3.00; hard copy, $7.00. **Remittance must accompany order.**

*Aerospace Engineer, Computational Methods Branch, High-Speed Aerodynamics Division. Member AIAA.

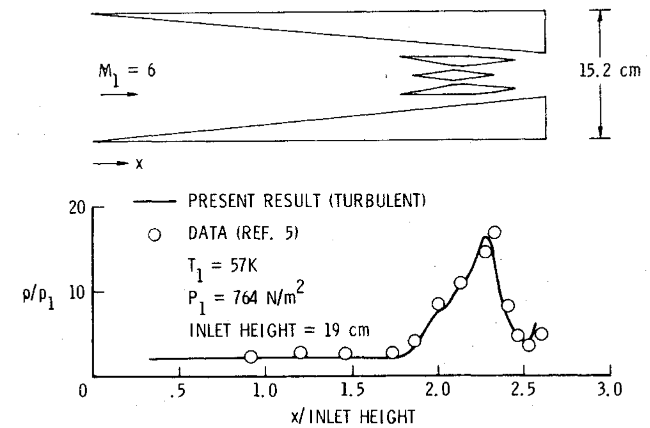

Fig. 2 Comparison of sidewall pressure distribution for three-strut inlet.

Table 1 Flow conditions

M_∞	M_I	M_{IN}	P_I, atm	T_I, K
7	6.0	5.032	0.035	335
6	5.18	4.34	0.045	329
5	4.29	3.60	0.064	328
4	3.43	2.88	0.095	322

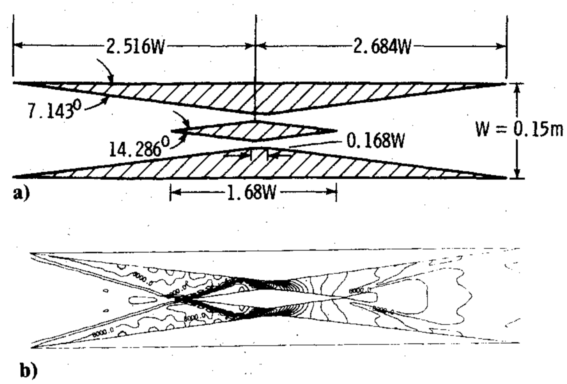

Fig. 3 One-strut inlet: a) geometry in a plane normal to sidewall sweep; b) pressure contours for inviscid flow at $M_\infty = 7$.

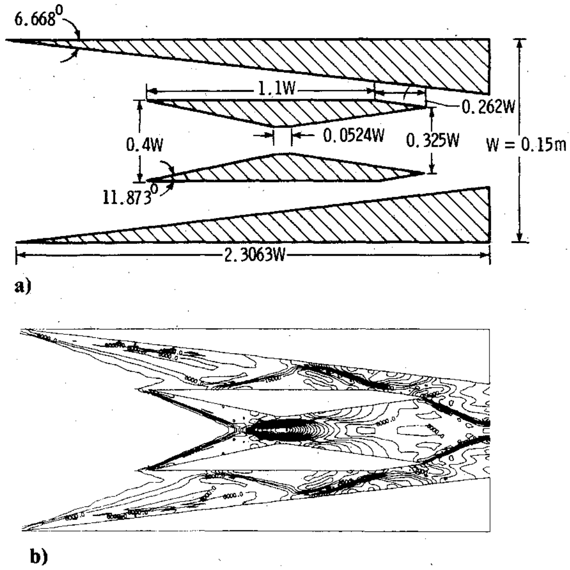

Fig. 4 Two-strut inlet: a) geometry in a plane normal to sidewall sweep; b) pressure contours for laminar flow at $M_\infty = 7$.

Figure 3a shows the geometry of the single strut inlet and Fig. 3b shows the pressure contours for the inviscid flow at $M_\infty = 7$. It is seen from the contour plots that the shock waves from the sidewall and strut leading edges coalesce to form a stronger shock. For the viscous flow, this strong shock causes a large separated region on the sidewall which results in the aerodynamic choking of the flow at all the Mach numbers considered here except at the highest Mach number for the turbulent flow. Solutions could be obtained for the inviscid flow at the highest three Mach numbers, but at the lowest Mach number, no solution could be obtained due to the choking of the flow caused by the shock wave detachment in the inlet.

To eliminate the problem of shock wave coalescence in the one-strut inlet, a two-strut inlet is considered in which the strut surface, on which the sidewall shock strikes, is made parallel to the oncoming flow so that no shock is produced by this surface, thus avoiding the possibility of shock wave coalescence. Figure 4a shows the geometry of the two-strut inlet and Fig. 4b shows the pressure contours for the laminar flow at $M_\infty = 7$. For this inlet, solutions could be obtained at all the Mach numbers for the inviscid flow and at the highest three Mach numbers for the viscous flow. No viscous flow solution could be obtained at the lowest Mach number due to the choking of the flow caused by large boundary-layer separation. This analysis clearly shows that the two-strut inlet performs better than the one-strut inlet.

All the preceding calculations were made on CDC-CYBER-203 vector processing computer using a grid size of 51×51. The solution marches about 30 time-steps per second for the inviscid flow and 20 time-steps per second for the viscous flow. A typical solution is obtained in 2-5 min depending upon the number of time-steps required for convergence.

The preceding study shows that the present 2-D code can be used in preliminary inlet design to study the effects of various parameters. It can be used to modify or eliminate those designs which are not expected to perform well and, thus, can help in reducing the experimental testing required in inlet design.

References

[1] Jones, R. A. and Huber, P. W., "Toward Scramjet Aircraft," *Astronautics & Aeronautics*, Vol. 16, Feb. 1978, pp. 38-49.

[2] Kumar, A., "Numerical Analysis of the Scramjet Inlet Flow Field Using Two-Dimensional Navier-Stokes Equations," AIAA Paper 81-0185, Jan. 1981.

[3] Baldwin, B. S. and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, Jan. 1978.

[4] MacCormack, R. W., "The Effect of Viscosity in Hypervelocity Impact Cratering," AIAA Paper 69-354, May 1969.

[5] Trexler, C. A., "Design and Performance at a Local Mach Number of 6 of an Inlet for an Integrated Scramjet Concept," NASA TN D-7944, 1975.