Defence Science Journal, Vol. 54, No. 1, January 2004, pp. 3-16

Numerical Investigation of the Intake Flow Characteristics for a Ramjet Engine with and without Heat Addition in the Combustion Chamber

G. Raja Singh Thangadurai and B.S. Subhash Chandran

Defence Research & Development Laboratory, Hyderabad–500 058

and

V. Babu and T. Sundararajan

Indian Institute of Technology Madras, Chennai–600 036

ABSTRACT

The flow field in the supersonic mixed compression axisymmetric as well as two-dimensional intake have been investigated numerically. The flow characteristics, such as pressure, temperature, and velocity have been obtained. An enlarged chamber with a supersonic convergent-divergent nozzle has been integrated at the air intake exit to simulate the ramjet engine. Suitable heat addition at the inner surface of the wall has enabled the simulation of combustion chamber condition. The well-known modes of operation of the ramjet engine air intake have been predicted by the mathematical simulations. A comparative study for supercritical, critical, and subcritical conditions has been presented. The predicted flow features including shock locations and the static pressure distributions along the cowl, as well as the centre body are in agreement with the available experimental results in the published literature. The effect of thermal input in the combustion chamber on the flow characteristics of the air intake has been studied in detail. The characteristic curve of generic supersonic air intake can be obtained through these numerical studies.

Keywords: Ramjet, air intake, combustion chamber, air-breathing engine, cruise missile, numerical simulation, normal shock, boundary conditions, pressure recovery, grid sensitivity, ramjet engine, flow characteristics, bleed

NOMENCLATURE

D	Cowl inner diameter
e	Internal energy per unit mass
k	Thermal conductivity, turbulent kinetic energy
M	Mach number
p	Pressure
q	Heat flux
R	Gas constant
T	Static temperature
t	Time
v	Velocity
μ	Effective viscosity
ε	Dissipation rate of kinetic energy of turbulence
Φ	Viscous dissipation

Revised 09 January 2003

ρ Density

τ Shear stress

SUBSCRIPT

e At the exit of air intake

0 Stagnation condition

r Radial direction

θ Azimuthal direction

z Axial direction

l Laminar flow

t Turbulent flow

v Constant volume

∞ Free-stream condition

1. INTRODUCTION

Ramjets are the simplest air-breathing engines capable of operating with better efficiency at high supersonic speeds. These do not have any moving part, such as compressor or turbine. Ramjet engines employing liquid fuels have become the mainstay of propulsion systems for supersonic cruise missiles and large air-to-air missiles. Liquid ramjet engines have high propulsion efficiency with the specific impulse of 1000 s. These also provide better controllability at different flight speeds and altitudes. The integrated ramjet, which consists of a liquid fuel ramjet and a solid fuel booster, is the best choice for the propulsion system of supersonic, small volume, medium-and long-range missiles (Fig. 1). Liquid fuel ramjet engines provide a means of satisfying certain performance requirements in supersonic missile propulsion, such as combining long range with a long flight path flexibility for missions requiring major altitude and velocity variations. These ramjet engines also provide high-energy efficiency.

In ramjet engines operating at supersonic speeds, compression of the incoming air is accomplished by ram effect in which the inlet airflow is decelerated in the intake region. The inlets of supersonic missiles are designed in such a way that these offer high-pressure recovery and produce less drag.

Numerical simulation of ramjet components in isolated mode as well as in completely integrated stage is extremely useful for cost and time-saving when the system is under design and development. Computational fluid dynamics (CFD) tools are complementary to wind tunnel and propulsion bench tests and these allow one to analyse and optimise more rapidly and at a lower cost.

A two-dimensional, mixed compression supersonic inlet has been experimentally studied for system performance at the design condition of Mach 3 with boundary layer bleed, by Anderson and Wong[1]. Knight[2] solved the flow fields of two-dimensional high speed inlets of realistic engineering geometry through numerical integration of the time-averaged compressible Navier–Stokes equations using the algorithm of Maccormack in conjunction with a curvilinear body-fitted coordinate system.

Yang and Yu[3] have experimentally investigated the characteristics and dynamics of an abruptly expanded flow in a model combustion chamber. Two-dimensional Navier–Stokes solutions for the flow through various inlet/diffuser configurations

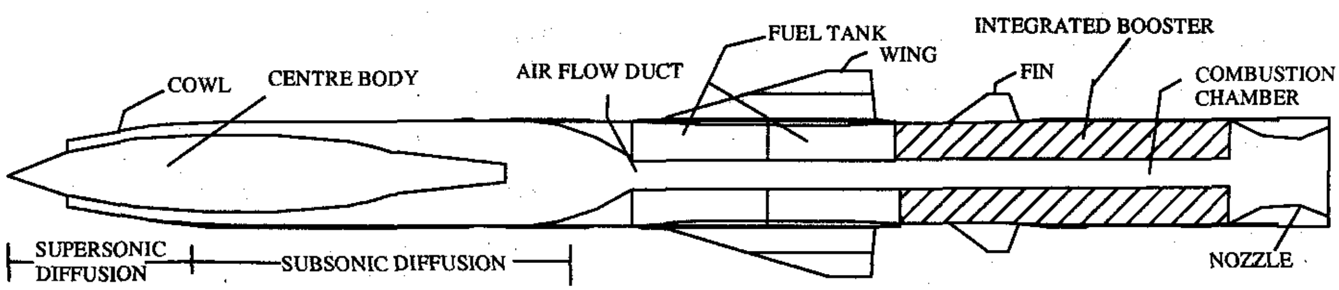


Figure 1. Schematic diagram of a typical ramjet powered flight vehicle

with terminal shocks have been obtained by Talcott and Kumar[4].

Nagarthinam[5] has carried out extensive wind tunnel testing of fixed-geometry axisymmetric air intake and studied the effect of various critical geometric parameters of the intake on intake starting and performance.

Numerical investigations of supersonic inlets with and without bleed, which couple the shock system elements and subsonic diffuser in a single computational space, were carried out by Hunter[6], *et al.* by employing a computational flow plug for the duct outflow boundary. Chan and Liang[7] numerically simulated two-dimensional mixed compression supersonic inlet by solving the time-dependant, compressible Navier-Stokes equations, with the help of Baldwin-Lomax turbulent model and an implicit second-order upwind scheme.

Freskos[8], *et al.* have developed codes with two different approaches for the numerical simulation of flow field around supersonic air intake. Wang Shuo[9], *et al.* have analysed the shock wave structure and flow characteristics of a supersonic inlet, both experimentally and computationally.

Montazel[10], *et al.* developed a Navier-Stokes simulation methodology for supersonic missile inlets and applied this to an isolated generic inlet as well as to a supersonic inlet in the presence of a missile body.

Mittal[11], *et al.* have carried out numerical simulation of a two-dimensional mixed compression supersonic inlet by solving time-dependant compressible Euler equations using a stabilised finite element technique. Starting of the supersonic inlet has been studied by varying the throat area.

An attempt has been made in the present study to numerically simulate the air intake of a ramjet engine in the integrated mode, both with and without heat addition in the combustion chamber.

2. MATHEMATICAL MODEL

The governing equations solved in this simulation study are given below:

Mass balance

$$\frac{\partial \rho}{\partial t}+\frac{1}{r}\frac{\partial}{\partial r}(\rho r v_r)+\frac{\partial}{\partial z}(\rho v_z)=0$$

r-momentum

$$\rho\left\{\frac{\partial v_r}{\partial t}+v_r\frac{\partial v_r}{\partial r}+v_z\frac{\partial v_r}{\partial z}\right\}=$$

$$-\frac{\partial p}{\partial r}+\left[\frac{1}{r}\frac{\partial (r\tau_{rr})}{\partial t}+\frac{\partial \tau_{rz}}{\partial z}-\frac{\tau_{\theta\theta}}{r}\right]$$

z-momentum

$$\rho\left\{\frac{\partial v_z}{\partial t}+v_r\frac{\partial v_r}{\partial r}+v_z\frac{\partial v_r}{\partial z}\right\}=-\frac{\partial p}{\partial r}+\left[\frac{1}{r}\frac{\partial (r\tau_{rz})}{\partial r}+\frac{\partial \tau_{zz}}{\partial z}\right]$$

Energy

$$\rho\left\{\frac{\partial e}{\partial t}+v_r\frac{\partial}{\partial r}\left(e+\frac{p}{\rho}\right)+v_z\frac{\partial\left(e+\frac{p}{\rho}\right)}{\partial z}\right\}$$

$$=-\frac{1}{r}\frac{\partial (rq_r)}{\partial r}-\frac{\partial q_z}{\partial z}+\mu\Phi$$

where the last term in the energy equation denotes viscous dissipation.

2.1 Equation of State

$$p=\rho RT$$

The shear stresses in the momentum equations are defined as

$$\tau_{rr}=-\mu\left[2\frac{\partial v_r}{\partial r}-\frac{2}{3}(\nabla . v)\right]$$

$$\tau_{\theta\theta}=-\mu\left[2\left(\frac{1}{r}\frac{\partial v_\theta}{\partial \theta}+\frac{v_r}{r}\right)-\frac{2}{3}(\nabla . v)\right]$$

$$\tau_{zz}=-\mu\left[2\frac{\partial v_z}{\partial z}-\frac{2}{3}(\nabla . v)\right]$$

$$\tau_{zr}=\tau_{rz}=-\mu\left[\frac{\partial v_z}{\partial r}+\frac{\partial v_r}{\partial z}\right]$$

where

$$(\nabla.v) = \frac{1}{r}\frac{\partial}{\partial r}\left(rv_r + \frac{1}{r}\frac{\partial v_\theta}{\partial \theta} + \frac{\partial v_z}{\partial z}\right)$$

and

$$\mu = \mu_l + \mu_t$$

Here, μ_l is the laminar viscosity. The turbulent viscosity μ_t may be related to the kinetic energy of turbulence (k) and its dissipation rate (e) by dimensional analysis and is given by

$$\mu_t = C\mu\rho k^2/\varepsilon$$

where $C\mu = 0.09$.

3. BOUNDARY CONDITIONS

The computational domain is as shown in the Fig. 2. The boundary conditions employed in the simulation of air intake flow are:

(a) Inflow

Free-stream conditions

- Total pressure, total temperature, and z–velocity component (v_z) are specified.
- $v_r = 0$.

(b) Symmetry

The radial velocity and the radial derivatives of all-flow variables are set to zero on the axis of symmetry.

(c) Wall (no heat addition)

No-slip condition for all velocity components is imposed.

$v_r = 0$, $v_z = 0$ and the wall heat flux $q_r = 0$

(d) Wall (with heat addition)

No-slip condition for all velocity components is imposed.

$v_r = 0$, $v_z = 0$, heat flux q_r specified.

(e) Outflow

- For subsonic flow, backpressure is specified at flow exit.
- With supersonic flow at the nozzle exit, the flow variables are determined from the interior domain by extrapolation.

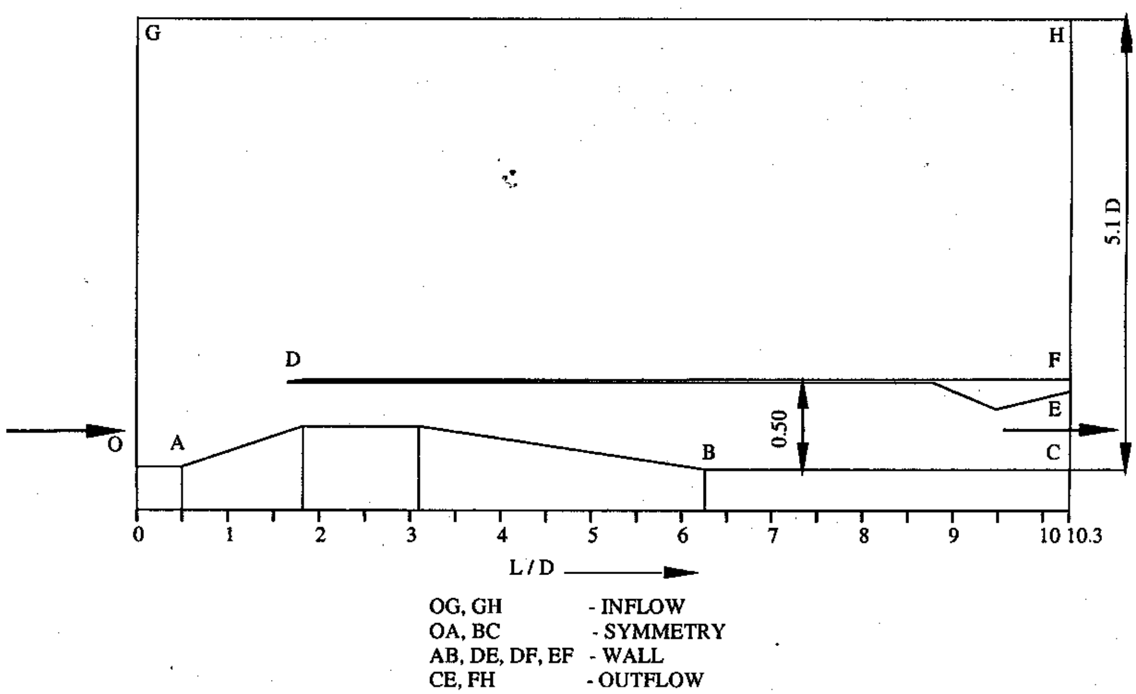


Figure 2. Computational domain

The critical and delicate issue in the numerical analysis of supersonic air intake is the handling of outflow boundary condition at the exit of the duct. Since the flow at the exit section is subsonic, it calls for some flow quantity to be specified. But the pressure rise achieved due to compression has to be estimated for predicting the air intake performance, and subsequently matching the air intake with the compressor.

A controllable supersonic convergent-divergent nozzle has been introduced before the exit plane, which makes the flow supersonic at the exit. The resulting supersonic flow has been handled with zero-gradient condition. The advantages of using a supersonic nozzle at the exit are:

(a) There is no need to specify any flow property at the exit.

(b) The flow properties at the head-end of the nozzle correspond to the air intake. Conditions at the end of the intake automatically emerge from the solution and these need not to be specified externally.

(c) Mass flow rate can be varied by adjusting the throat size.

(d) This technique simulates the ramjet engine completely, since it models all the constituents, viz., air intake, combustor, and nozzle.

4. SOLUTION METHODOLOGY

The above governing differential equations and boundary conditions have been solved using the commercial flow simulation software FLUENT. Finer grids were used in the steep-gradient zones, such as boundary layers near the walls and also at the locations of normal and oblique shocks.

The procedure employs a finite volume technique using primitive variable formulation. The conservation equations for mass, momentum, energy, species, and turbulence quantities are solved using the finite volume technique. The computational domain is divided into discrete control volumes using a general curvilinear grid. Non-staggered grid storage scheme is employed for all the nodal variables. The governing equations on the individual control volumes are integrated wrt space and time to obtain the algebraic equations for nodal unknowns, such as velocities and pressure. The equations thus obtained are solved to obtain the flow field. For turbulent flows, the kinetic energy equations are solved additionally for obtaining the distribution of effective viscosity and Reynolds stresses.

5. VALIDATION OF NUMERICAL PREDICTIONS WITH EXPERIMENTAL DATA

The experimental data generated by Nagarathinam[5] are used for validation studies. The experimental data are available for the cases where the model has been tested with both closed tunnel as well as free-jet conditions. The parameters monitored include cowl inner surface, static pressure, and total pressure at the exit.

The comparison of the predicted values for pressure recovery, diffuser exit Mach number, and percentage distortion with the experimental data are given in the Figs 3, 4 and 5, respectively. The flow distortion at the diffuser exit is given by

$$\text{Percentage flow distortion} = \left[\frac{(p_{oe}/p_{o\infty})_{max} - (p_{oe}/p_{o\infty})_{min}}{(p_{oe}/p_{o\infty})_{average}} \times 100\right]$$

The predictions match reasonably well, both qualitatively and quantitatively with the experimental data.

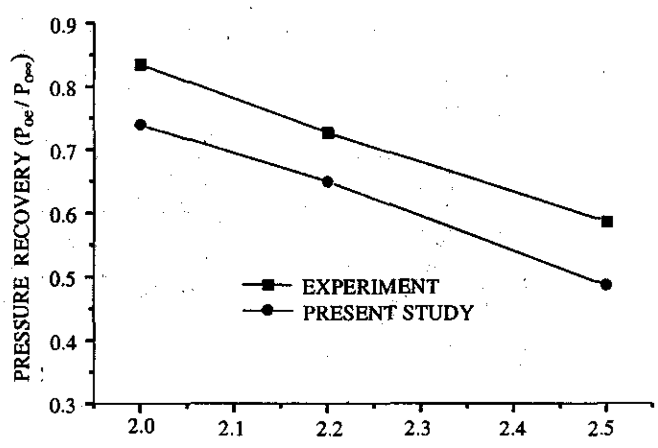


Figure 3. Variation of pressure recovery with inlet Mach number

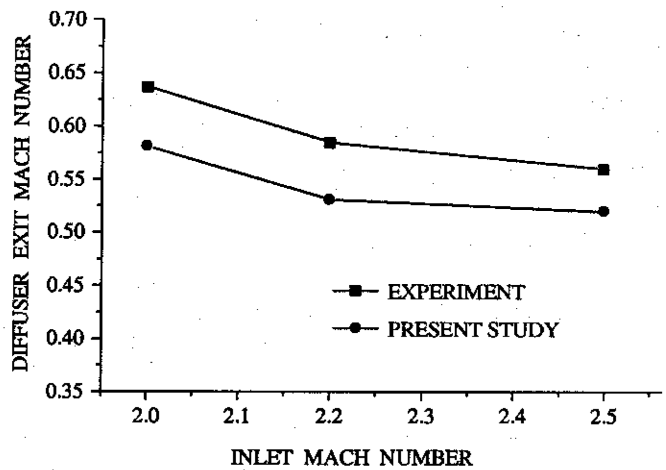


Figure 4. Variation of diffuser exit Mach number with inlet Mach number.

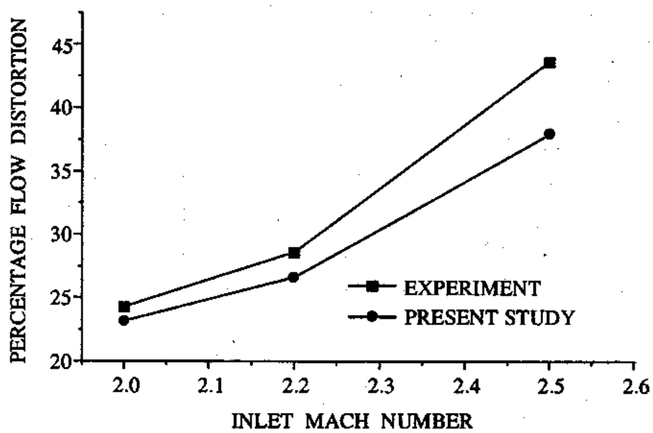


Figure 5. Flow distortion at diffuser exit for different inlet Mach number.

6. ANALYSIS OF GRID SENSITIVITY

The numerical simulation of the air intake of a ramjet engine is quite complicated because of the mixed supersonic and subsonic flow. The flow field comprises shocks emanating from the centre body and cowl, reflected shocks, and shock-boundary layer interactions. Fine meshes were employed at the expected shock locations to accurately capture the oblique shocks and their reflections as well as the terminal normal shock. The location of the grid point nearest to the wall was 0.0245 mm and the smallest grid size in the axial direction was 0.2 mm. The complexities involved in the prediction of flow properties for ramjet air intake call for a detailed analysis of grid sensitivity.

A typical grid used for the simulation is shown in the Fig. 6. Higher mesh density has been employed where the flow gradients are high, such as the shock locations as well as the boundary layer regions closer to the walls. The effects of the grids employed on static pressure distribution of cowl inner wall are shown in the Fig. 7. Three different grids with 100 × 100 nodes, 150 × 150 nodes, and 200 × 200 nodes in the axial and radial directions have been considered. From the figures, it can be inferred that the pressure recovery predicted is independent of the grid employed. However, the initial shock strength and reflected shock locations are mildly grid-sensitive.

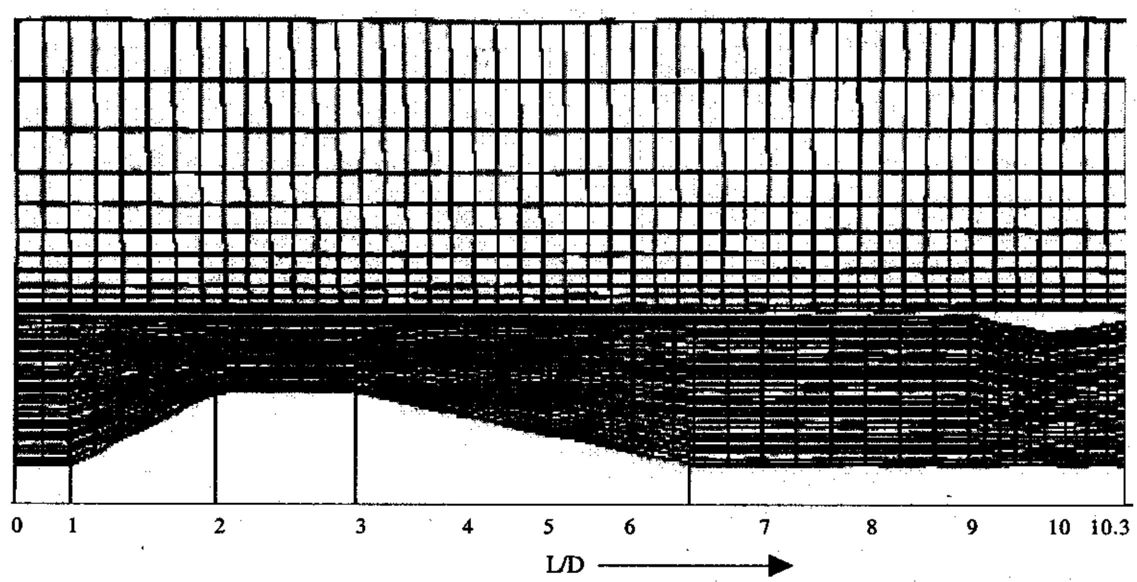


Figure 6. Typical grid employed for simulation with supersonic C-D nozzle at the exit

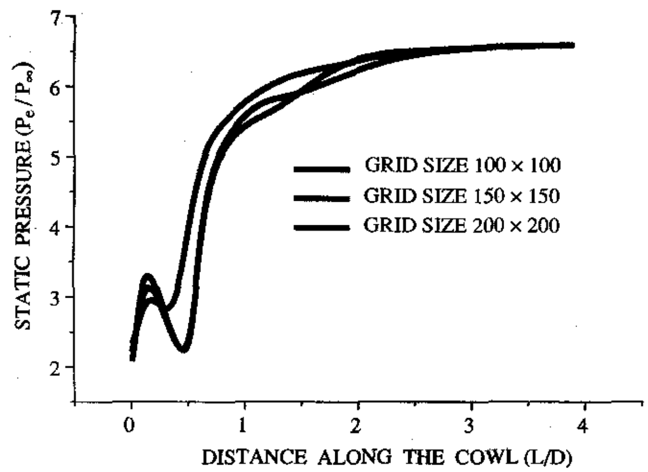


Figure 7. Static pressure distribution along the cowl inner surface.

7. RESULTS & DISCUSSIONS

Axisymmetric and two-dimensional supersonic mixed compression air intake of realistic configurations have been studied. In a ramjet engine, the incoming supersonic flow is decelerated to subsonic flow through a series of shocks. The subsonic flow is further decelerated to a low subsonic Mach number ($M < 0.3$), which is suitable for the combustion chamber operation. As a result of this diffusion process, the static pressure of the incoming airflow reaches a value 6–8 times of the free-stream condition.

7.1 Axisymmetric Mixed Compression Air Intake Performance

A comparison with the experimental data for two different inlet Mach number values (Figs 8 and 9) shows fair agreement for the cowl inner wall static pressure variation and good agreement for the pressure recovery. The differences observed in the figures between the numerical simulation and the experimental one could be attributed to the fact that it is not possible to exactly match the boundary conditions between the theoretical and the experimental investigations.

The Mach contours for the various modes of an operation of the ramjet engine are shown in Figs 10-12. It is evident that the leading oblique shock just sits at the cowl lip for the critical condition and it enters within the duct for supercritical condition;

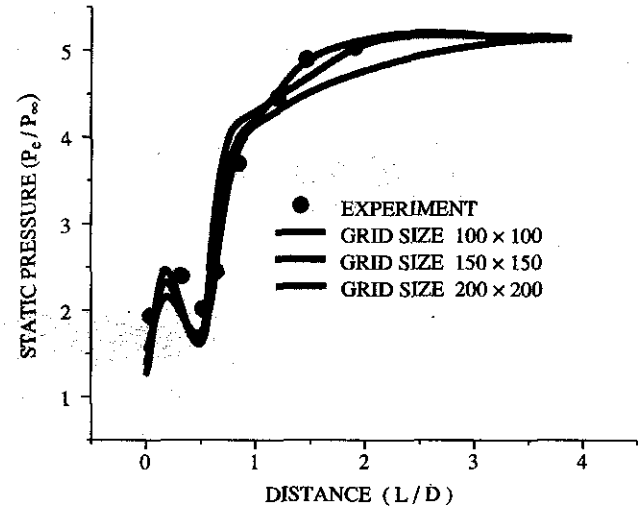


Figure 8. Cowl inner wall static pressure distribution for free-stream Mach number, M = 2.2.

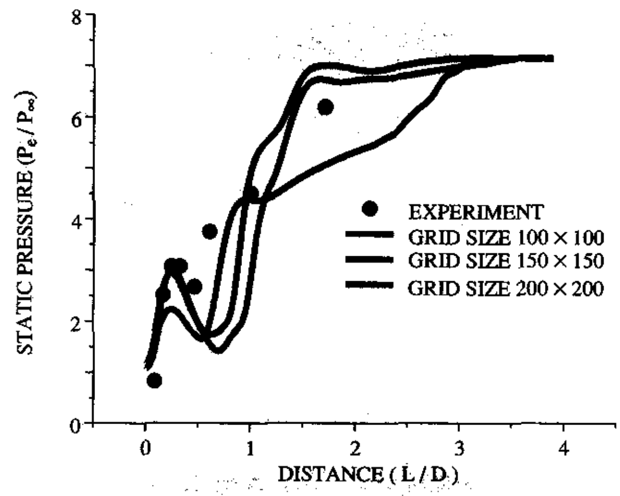


Figure 9. Cowl inner wall static pressure distribution for free-stream Mach number, M = 2.5.

on the other hand, for subcritical operation, the shock opens out, resulting in spillage of the approach mass flow. These well-known characteristics of a typical ramjet engine are predicted very well by the simulations. In the supercritical operation, maximum mass flow rate is achieved but the pressure recovery is low; on the other hand, in sub-critical operation, pressure recovery is good but the mass flow rate is less due to spillage. Further, the intake is unstable at subcritical mode of an operation. In this condition the terminal normal shock is located outside the cowl, which causes subsonic air spillage over the cowl. At critical operation, both pressure recovery and mass flow rate are closer to the required operating conditions.

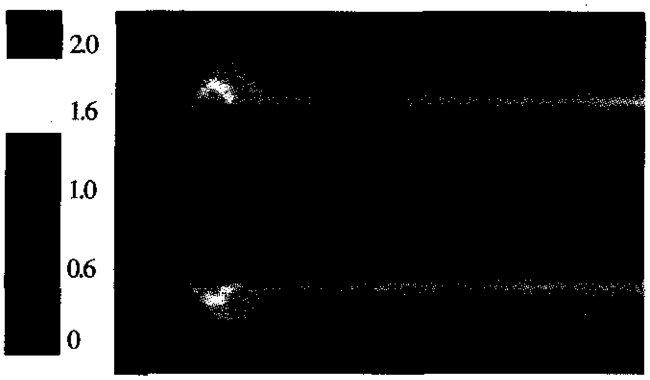


Figure 10. Various modes of operation of supersonic air intake: Mach contours for subcritical operation.

Figure 11. Various modes of operation of supersonic air intake: Mach contours for critical operation.

Figure 12. Various modes of operation of supersonic air intake: Mach contours for supercritical operation.

The static pressure distribution along the cowl inner surface and the centre body outer surface for different inlet Mach numbers are given in Figs 13 and 14, respectively. The inward movement of initial oblique shock and its reflections and the terminal normal shock, with an increase in Mach number is evident from these figures. The shock strengths (and hence static pressure recovery) also increase with Mach number. The variation of total pressure recovery with normalised mass flow rate is shown in Fig. 15 for different operating modes of a ramjet air intake. The predictions show that for supercritical condition mass flow rate is maximum with low-pressure recovery, while for subcritical operation there is a high pressure recovery with low mass flow rate which is in accordance with the well-known theory of supersonic air intake flows.

7.2 Two-dimensional Supersonic Mixed Compression Air Intake Performance

The simulation studies were carried out for the two-dimensional mixed compression air intake geometry which was experimentally investigated by Anderson and Wong[1]. Few other researchers[7,11] have also investigated this air intake configuration.

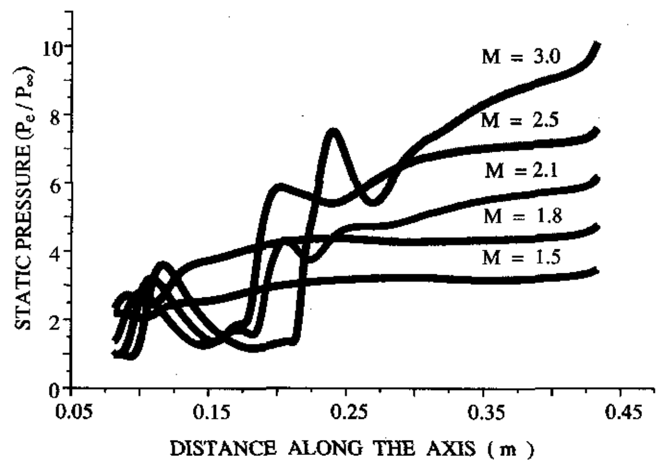


Figure 13. Static pressure distribution along the cowl inner surface.

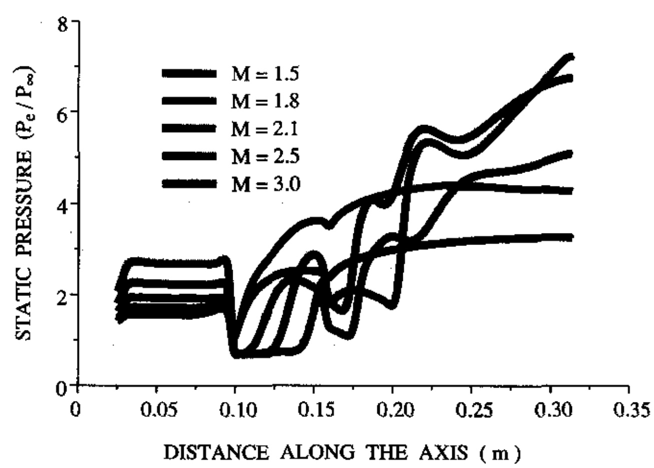


Figure 14. Static pressure distribution along the centre body

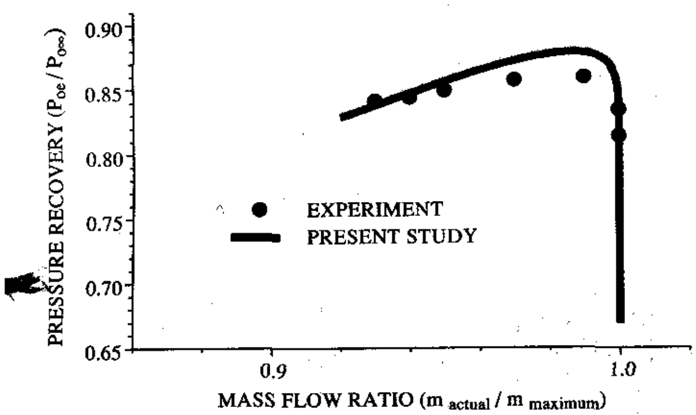


Figure 15. Air intake characteristics curve

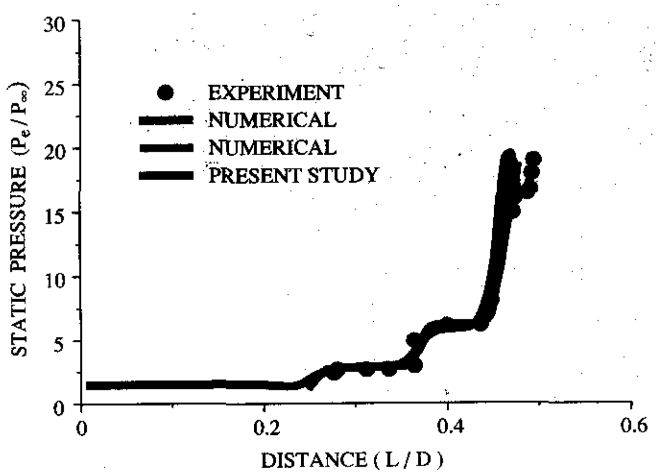


Figure 16. Static pressure distribution along cowl inner surface

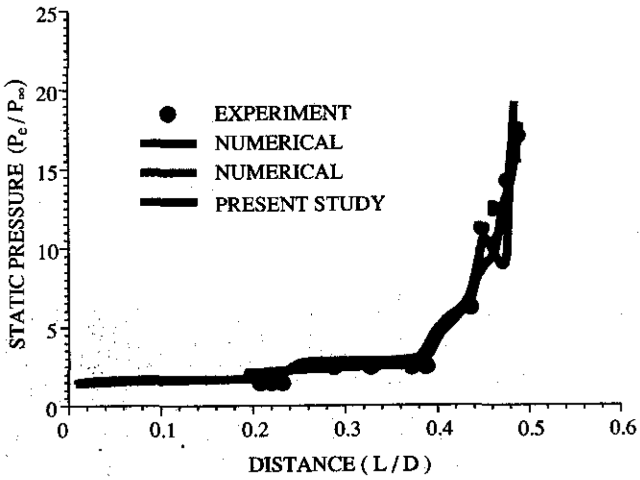


Figure 17. Static pressure distribution along the ramp outer surface.

The static pressure variations along the cowl inner surface and the ramp outer surface are shown in Figs 16 and 17. The predictions show reasonable agreement with the various results obtained by other researchers. However, it may be noted that the numerical prediction by the authors indicate a stronger shock and subsequent re-expansion (resulting in a fall of static pressure near L/D = 0.5), which are absent in the present prediction as well as in the experimental data[1]. The Mach contours for this configuration are shown in the Fig. 18.

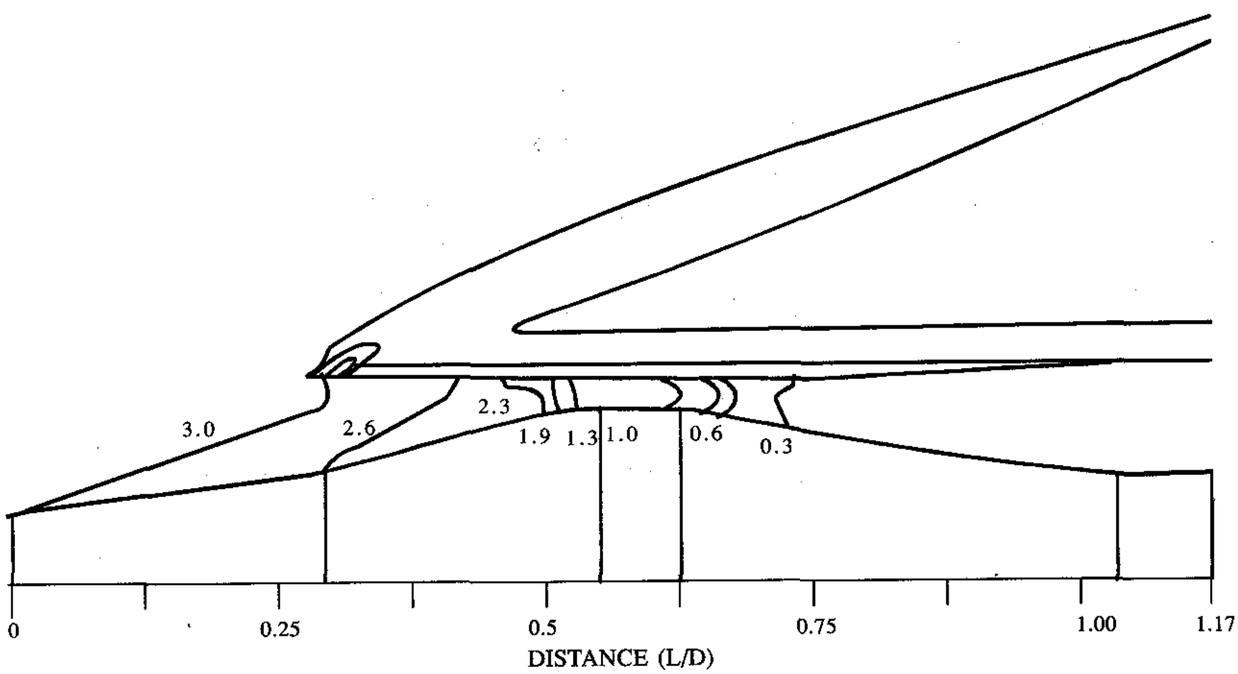


Figure 18. Mach contours for two-dimensional supersonic mixed compression air intake

7.3 Complete Ramjet Engine Model

7.3.1 *Cold Flow Studies*

For simulating the entire ramjet engine, a cylindrical combustion chamber with a convergent divergent nozzle is attached to the exit section of the air intake. The grid employed for simulation studies is shown in the Fig. 19. This arrangement is useful in analysing the ramjet engine in totality. The static pressure and Mach contours for cold flow studies for an inlet Mach number of 2.5 are shown in the Figs 20 and 21. With the enlarged combustion chamber, the terminal normal shock moves further downstream inside the air duct. The

Figure 19. Grid employed for flow simulation with heat addition in the combustion chamber

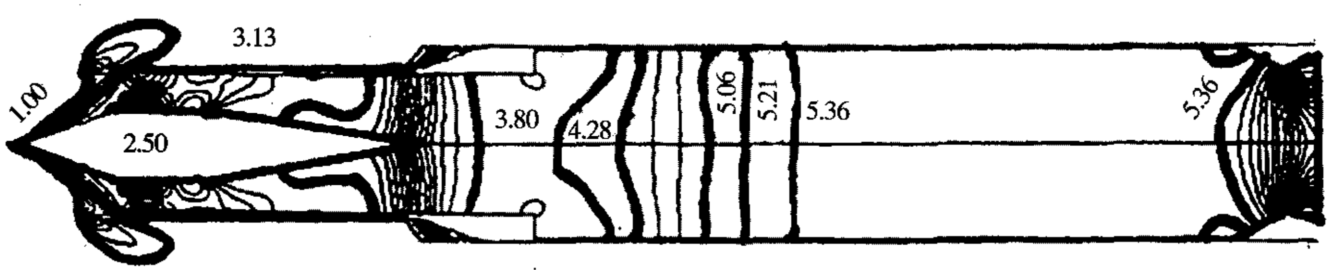


Figure 20. Static pressure (X×10^5 Pa) contours–cold flow free-stream Mach number: 2.5

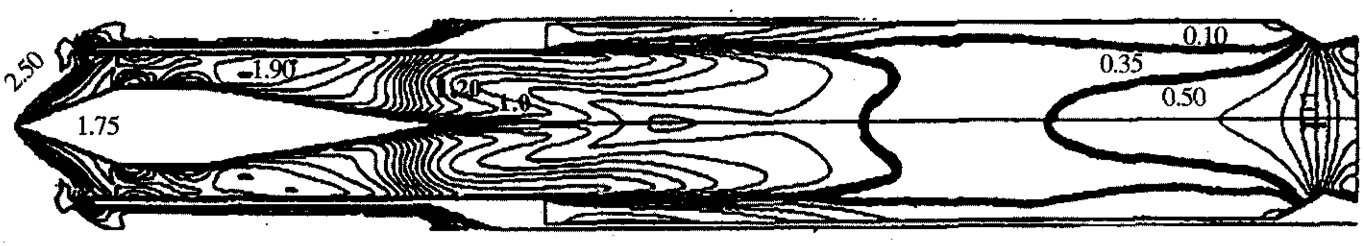


Figure 21. Mach contours–cold flow free-stream Mach number: 2.5

stream function contours are shown in the Fig. 22. A recirculation zone is observed where there is sudden enlargement in the combustion chamber area.

7.3.2 *Flow with Heat Addition in Combustion Chamber*

The heat generated by a typical ramjet engine works out to be 10.5 MW for the following operating conditions:

- Mass flow rate of 7.5 kg/s
- Temperature of 600 K at the exit plane of air intake
- Temperature of 2000 K inside the combustion chamber

In the present study, cold flow and flow with heat addition of 10 MW and 12 MW have been investigated. The thermal output of a typical ramjet engine was distributed over the combustion chamber wall for the simulation of hot test of a ramjet engine. With heat addition, the terminal normal shock moved upstream and located itself at the narrowest cross section of the air intake. The contours of static pressure and Mach number for this condition are shown in the Figs 23 and 24. The Rayleigh flow in the constant area combustion chamber also has been predicted well and the pressure drop across the combustor matched well with the theoretical one-dimensional analysis. The static pressure contours in the combustion chamber portion are shown in the Fig. 25.

Figure 22. Stream function (kg/s) contours–cold flow free-stream Mach number: 2.5

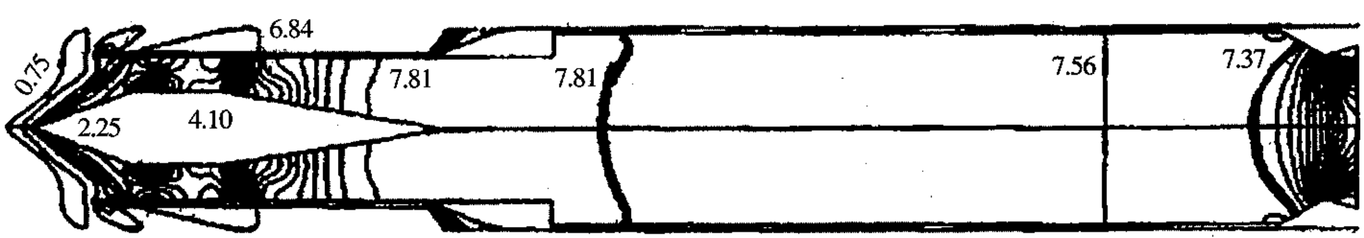


Figure 23. Static pressure (X×10^5 Pa) contours with heat addition free-stream Mach number: 2.5

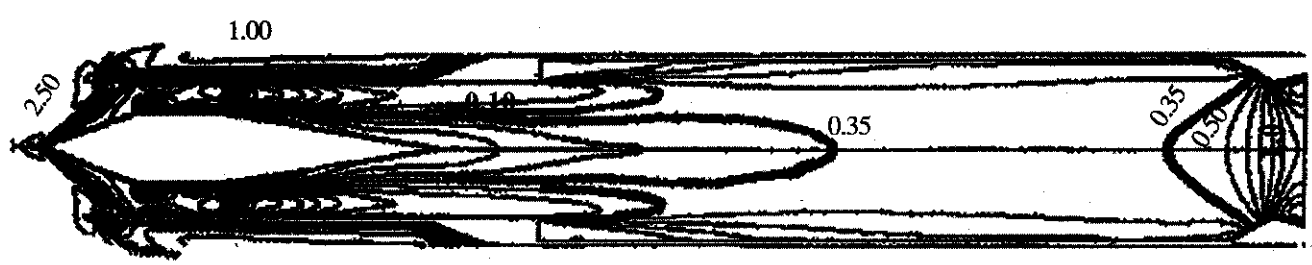


Figure 24. Mach contours with heat addition free-stream Mach number: 2.5

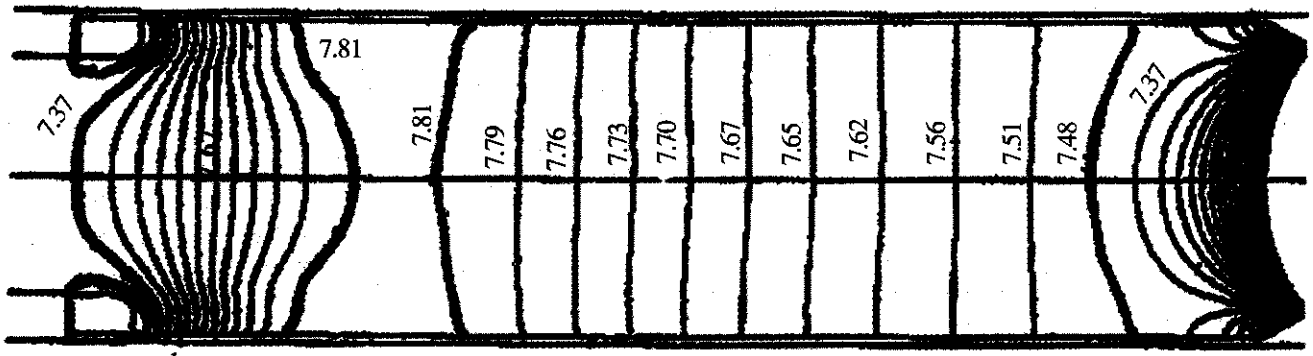


Figure 25. Stream pressure (X×10[5] Pa) contours in the combustor (with heat addition) free-stream Mach number: 2.5

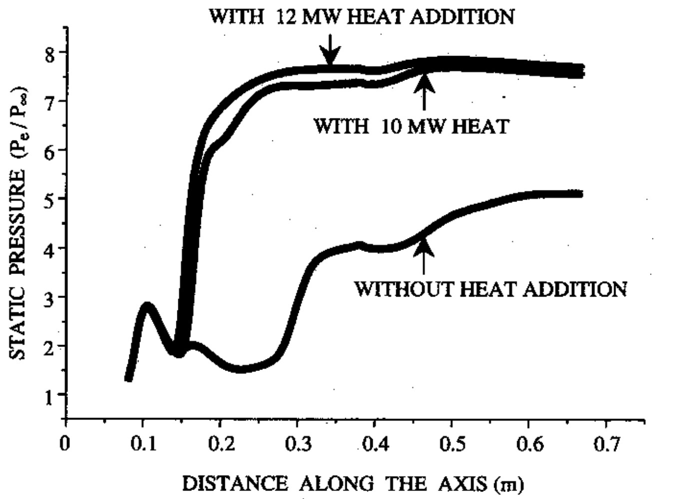


Figure 26. Effect of heat addition on the static pressure distribution along the cowl inner surface.

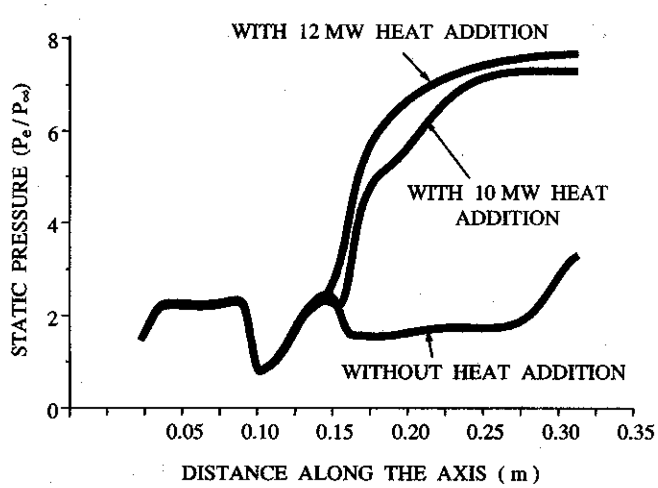


Figure 27. Effect of heat addition on the static pressure distribution along the centre body.

The static pressure distribution along the cowl inner surface and the centre body outer surface, with and without thermal input in the combustion chamber inner wall, for an inlet Mach number 2.5 are compared in the Figs 26 and 27. It is evident that heat addition significantly increases the pressure recovery for the given flow geometry.

8. CONCLUSIONS

The model developed to predict the flow in air intake is able to capture qualitatively and quantitatively the flow features, such as the shock pattern and the pressure recovery. The predictions have been compared and validated with experimental data available in the literature. The flow model developed can be effectively used to predict the flow environment in generic intake. The effects of flight Mach number on air intake performance have been studied in detail. By integrating an enlarged cylindrical chamber and a convergent divergent nozzle, the complete ramjet engine operation can be simulated.

ACKNOWLEDGEMENTS

The authors are thankful to Shri Prahlada, Director, Defence Research & Development Laboratory (DRDL), Hyderabad, for his constant encouragement to complete this study successfully. The guidance and help given by Dr M. Nagarathinam, Head, CFD Group and Dr S. Sundarrajan, Head, Programme Team, PJ-10 of DRDL, Hyderabad, at various stages are greatly acknowledged.

REFERENCES

1. Anderson, W.E & Wong. Experimental investigation of a large scale two-dimensional mixed compression inlet system–performance at design conditions, M = 3.0. NASA, USA, May 1970 Report No NASA-TM-X- 2016.

2. Knight, D.D. Numerical simulation of high speed inlets using the Navier-Stokes equations. *AIAA Journal*, 1977, 1583-589.

3. Yang, B.T. & Yu, M.H. The flow field in a suddenly enlarged combustion chamber. *AIAA Journal*, 1983, **21**(1), 92-97.

4. Noel, A.; Talcott (Jr) & Ajay Kumar. Two-dimensional viscous simulation of inlet/ diffuser flows with terminal shocks. *Journal of Propulsion*, 1985, **1**(2), 109-17.

5. Nagarathinam, M. Design and development of an axisymmetric air intake for ramjet propulsion system. Indian Institute of Technology Madras, Chennai, 1985 MS Thesis.

6. Hunter, Louis, G.; Tripp, John M. & Howlett, Douglas G. Supersonic inlet study using the Navier-Stokes equations. *Journal Propulsion*, 1986, **2**, 181-87.

7. Liang, S.M. & Chan, J.J. Numerical investigation of supersonic mixed compression inlet using an implicit upwind scheme. *Journal Propulsion*. 1992, **8**(1), 158-67.

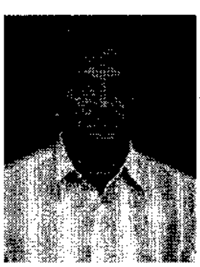
8. Freskos, G. & Penanhoat, O. Numerical simulation of the flow field around supersonic air intake. *Transactions ASME*, 1994, **116**, 116-23.

9. Wang, Shuo & Xu, Jianzhong. Investigation on the entrance flow of the supersonic inlet. *In* ISABE 97-7028, 1977.

10. Montazel, X.; Kergaravat, Y. & Blaize, M. Navier–Stokes simulation methodology for supersonic missile inlets. *In* ISABE 97–7029, 1997.

11. Mittal, Sanjay.; Jain, Vaibhav & Sullerey, R.K. Numerical investigation of supersonic mixed compression inlet using Euler equations. *In* Fifth National Conference on Air Breathing and Aerospace Propulsion (NCABE–2000). pp. 561-71.

Contributors



Mr G Raja Singh Thangadurai obtained his ME (Aerospace Engg) from the Indian Institute of Science, Bangalore, in 1991. He worked as Assistant Engineer at the Hindustan Aeronautics limited (HAL), Bangalore, from 1988 to 1989. He joined DRDO at the Defence Research & Development Laboratory (DRDL), Hyderabad, as Scientist B in 1991. Presently, he is working there as Scientist D. He has published six research papers in various journals. His research areas include: Rocket and ramjet propulsion, numerical simulation of internal flows and supersonic air intake. He is a life member of The Aeronautical Society of India.