Performance study of a supersonic inlet in the presence of a heat source

M.R. Soltani *, M. Farahani, J. Sepahi Younsi

Department of Aerospace Engineering, Sharif University of Technology, Tehran, P.O. Box 11155-8639, Iran

Received 20 October 2010; revised 6 February 2011; accepted 19 April 2011

KEYWORDS
Supersonic inlet; Heat source; Drag coefficient; Pressure recovery; Mass flow rate.

Abstract The flow over a supersonic inlet has been investigated experimentally and numerically at a free stream Mach number of 2 and a zero degree angle of attack. Wind tunnel tests were performed to obtain the performance parameters of the inlet and were used as a baseline and validation tools for the numerical code. A heat source was added to the flow field at a distance ahead of the inlet. The effect of heat source addition on the main performance parameters of the inlet is investigated numerically. Results show that the heat source considerably reduces drag coefficient; however, its effect on pressure recovery is not favorable. This unfavorable effect was then minimized by controlling heat source parameters, such as its location, size and shape. For the optimum condition, drag coefficient reduces considerably, inlet mass flow rate increases, but pressure recovery slightly decreases.

© 2011 Sharif University of Technology. Production and hosting by Elsevier B.V.

1. Introduction

A supersonic inlet is often considered to be one of the most important elements in the air-breathing engine of flying vehicles. The function of the inlet is to provide uniform and stable flow, at a desired velocity and pressure, to the engine face, with minimum loss in total pressure. Inlets have the greatest contribution to producing thrust with minimum losses (Figure 1, [1]). Proper operation of the combustion is ensured by incoming air with sufficient mass flow rate, total pressure and Mach number [2]. High speed flow is always accompanied by shock waves that considerably reduce overall efficiency. Today, a large number of fighters have external compression inlets, which have subcritical, critical and supercritical operating conditions (Figure 2). The best performance occurs under a critical condition where normal shock is tangent to the inlet lip. In this condition, the oblique shock that forms at the spike nose passes through the inlet lip, preventing spilled air, and consequently additive drag vanishes. However, since the critical condition, due to free stream disturbances and changes in flight Mach number, is almost always unstable, the inlet undergoes subcritical or supercritical operation and its parameters vary. To prevent these undesired phenomena, most inlets are designed for subcritical operation, i.e. it is allowed that some air deflects over the cowl surface. As seen from Figure 2, subcritical operation has unfavorable effects on inlet performance.

A variable geometry inlet can reduce the aforementioned undesirable phenomena. Aerospikes and recently developed MRD (Multi-Row-Disk) devices can reduce drag [3]. However, the design becomes too complex and expensive. Heat addition into the flow field from a proper place and at a sufficient rate can have desirable effects on inlet performance [4]. Macheret et al. [5] studied the effects of applying a heat source to a two dimensional inlet operating at hypersonic speed. Girgis et al. [6] and McAndrew et al. [7] showed that heat addition to a supersonic flow over a cone can increase lift force and decrease drag force. Miles et al. [8] outlined a series of experimental and modeling efforts that are directed toward determining the viability of using plasmas for various aerospace applications, ranging from drag reduction to power extraction. They discussed the utilization of thermal plasmas for drag reduction, vehicle steering and sonic boom mitigation. Over the past several years, Kremeyer has investigated an energy deposition method [9]. His numerical simulations indicated wave drag reduction up to 96%. In addition, the propulsive gain of the Kremeyer investigation was consistently positive,
meaning that the energy saved due to the wave drag reduction was always greater than the amount of energy invested to produce the heat source.

In the present study, a heat source was added numerically to the flow field ahead of the cowl lip. The inlet used in this study is axisymmetric with external compression. This inlet has been tested at a free stream Mach number of 2 in the wind tunnel. The test results, without the heat source, were used to validate code predictions. Surface pressure distribution over the spike and cowl, the boundary layer profile and the inlet performance parameters have been obtained from wind tunnel tests for flow without a heat source.

The flow field around the inlet has been solved by the INS (Inlet Numerical Simulation) code that has been entirely developed by the authors. The code can solve the flow field for different inlet geometries when operating at supersonic speeds. The results of the numerical solution were then compared with experimental data, and some parameters, such as boundary conditions and grid distribution were varied until acceptable accuracy was achieved. A heat source was then added to the flow, and its effect on drag coefficient, mass flow rate and the pressure recovery (ratio of total pressure at the end of the inlet to the free stream total pressure) of the inlet was numerically investigated using the INS code. Note that all other conditions, except the heat source, were similar to those of wind tunnel tests, which were done without a heat source.

Effects of the position, size and shape of the heat source on the inlet performance were studied and optimum conditions were obtained. The best location for the thermal source was found to be where the shock wave generated by the source and the spike passed through the spike nose. For the optimum condition, drag coefficient reduced considerably and inlet mass flow rate increased. However, inlet pressure recovery decreased slightly.

If a solid body was used as the source, it would create a drag force. In addition, the source must be attached to the inlet via a rod or other means. The source considered in this research is an imaginary, temperature-elevated region, not a solid body. Thus, the streamlines can pass through it. The present numerical simulation and previous work show that there exists a bow shock in front of the thermal source. This bow shock deflects streamlines through the inlet while prior to the source addition, the streamlines would have spilled over the inlet. For numerical consideration, the heat is modeled by adding a source term in the right hand side of the energy equation. As a result, although code validation has been done for flow without a heat source, with the aid of numerical considerations, it is expected that numerical results would also be valid for flow with a heat source. Furthermore, a grid resolution study has been conducted. In addition, other experimental investigations in this field confirm the benefits of heat source addition to supersonic flow.

Inlet pressure recovery can be improved by increasing the number of oblique shocks that form at the inlet entrance. A common way to increase the number of shock waves is to gradually change the surface inclination of the spike. However, this method may cause flow separation due to shock and boundary layer interaction; hence it is recommended to prevent the interaction of shocks with the surface. In the present study, it is ensured that the bow shock that forms in front of the heat source will not have any interaction with the surface. As a result, the shock will decelerate the flow without separation; a desirable phenomenon.

When an annulus heat source is placed ahead of the cowl, the incoming air is deflected through the elevated temperature and/or elevated pressure region. This streamline deflection will cause an increase in the mass flow rate entering the inlet. Note that in the present work, the mechanism of heat source addition is not investigated and will not be emphasized. But it has been proposed that application of microwave or RF energy to a pre-ionized region can be considered as a heat source [5].

2. Computational considerations

Figure 3 shows a picture of the inlet used in this investigation. It is an axisymmetric inlet designed for operating at a free stream Mach number of \( M_\infty = 2 \). Under its design conditions, the inlet operates under a subcritical condition; hence it has some air spillage.

2.1. Grid

The flow field has been solved numerically by the INS code. The grid was generated by an elliptic grid generator. The physical domain has been divided into two blocks, as shown in Figure 4. In this grid generator, the following system of elliptic Partial Differential Equations is considered:

\[
\begin{align*}
\frac{\partial \xi}{\partial x} + \frac{\partial \eta}{\partial r} &= 0 \\
\frac{\partial \chi}{\partial x} + \frac{\partial \eta}{\partial r} &= 0
\end{align*}
\]

where \( \xi \) and \( \chi \) represent the coordinates in the computational domain, and \( x \) and \( r \) represent the coordinates in the physical...
domain. To find the location of the grid points, the above system of equations was solved numerically for $x$ and $r$. The resultant grid is structured and can easily be refined and stretched in all or part of the physical domain. Figure 5 shows the grid generated by this method.

2.2. Governing equations

In the INS code, the Reynolds–averaged Navier–Stokes equations are solved numerically. Neglecting body forces, these equations, in two dimensional (planar and axisymmetric) conservative form, are:

$$\frac{\partial \vec{W}}{\partial t} + \int_A \vec{F}_c \, dA + \int_A \vec{F}_k \, dA = \int_A \vec{F}_v \, dA + \beta \int_A \vec{Q} \, dA,$$  \tag{2}

where:

$$\vec{W} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{bmatrix}, \quad \vec{F}_c = \begin{bmatrix} \rho V_u \\ \rho u V_u + n_s p \\ \rho u V_v + n_s p \\ \rho \nu H \end{bmatrix},$$

$$\vec{F}_k = \begin{bmatrix} 0 \\ n_s \tau_{xx} + n_r \tau_{xr} \\ n_s \tau_{xx} + n_r \tau_{xr} \\ n_s \tau_{xx} + n_r \tau_{xr} \end{bmatrix},$$

$$\tau_{xx} = \frac{\partial \sigma_{xx}}{\partial x}, \quad \tau_{xr} = \frac{\partial \sigma_{xr}}{\partial r}.$$

And:

$$V_n = \vec{V} \cdot \vec{n} = n_x u + n_r v,$$

$$n_x = \frac{\Delta r}{\Delta s}, \quad n_r = -\frac{\Delta x}{\Delta s},$$

$$\Delta s = \sqrt{\Delta x^2 + \Delta r^2},$$

$$\Theta_x = u \tau_{xx} + v \tau_{xr} + \frac{k}{\rho} \frac{\partial T}{\partial x},$$

$$\Theta_r = u \tau_{xx} + v \tau_{xr} + \frac{k}{\rho} \frac{\partial T}{\partial r}$$  \tag{4}

$E$ and $H$ are the total internal energy and total enthalpy of the air, respectively. If $\alpha = 0$, the equations are for two dimensional planar, and if $\alpha = 1$, they are for axisymmetric flow. $A$ is the area of the two dimensional cell, $\Delta s$ is the length of the cell face and $V_n$ is the velocity component normal to the cell face. Air is considered to be a perfect gas:

$$P = \rho RT.$$  \tag{5}

Further, $\beta = 1$ for the flow with heat source addition, and $\beta = 0$ for flow without a heat source.

Using explicit finite volume discretization, Eq. (2) becomes:

$$W_{ij}^{n+1} = W_{ij}^n - \frac{\Delta t_j}{A_{ij}} \left[ \sum_{k=1}^4 (F_{ik})_k \Delta S_{kj} \right]$$

$$- \alpha \Delta t_{ij} \nu_{ij} + \Delta t_{ij} \left[ \sum_{k=1}^4 (F_{ik})_k \Delta S_{kj} \right]$$

$$+ \alpha \Delta t_{ij} \nu_{ij} + \beta \Delta t_{ij} Q_{ij}.$$  \tag{6}

Eq. (6) must be applied to each grid cell to find flow variables at each time step. The flow is assumed to be steady; however, the time derivative term is used to march until specified convergence is achieved. To accelerate convergence, the time step in Eq. (6), $\Delta t_{ij}$, is calculated using the local time stepping method.

2.3. Other numerical considerations

Vectors $F_c$ and $F_k$ in Eq. (2) are the convective and viscous flux vectors, respectively. These fluxes must be evaluated at the cell face in Eq. (6). In the INS code, the convective fluxes are computed by the Roe upwind scheme, since this scheme has high accuracy. Viscous fluxes are calculated by a finite volume method, consistent with the overall discretization method. To increase the accuracy of space discretization, the MUSCL approach with variable extrapolation is used. Therefore, the spatial solver of the INS is a 2nd order upwind solver.

The boundary conditions used in this investigation are presented in Figure 4. The stress terms in Eqs. (3) and (4) are computed, using the following viscosity coefficient:

$$\mu = \mu_L + \mu_T,$$  \tag{7}

where $\mu_L$ and $\mu_T$ are the laminar and the turbulent viscosity coefficients, respectively. Laminar viscosity is molecular viscosity, which has been computed by the Sutherland relation in this study. The turbulent viscosity coefficient, however, is calculated by the Baldwin–Lomax turbulence model. This algebraic, two-layer, eddy viscosity model is based on the Cebeci–Smith model with some modifications to avoid locating the edge of the boundary layer. This simple and numerically efficient model can capture major turbulent effects, and has been widely used for numerical computation of the supersonic inlet flow field [10–14].
2.4. Grid resolution study and grid quality

An intensive grid resolution study was conducted to ensure that the numerical solution is independent of grid size. In this study, the physical domain was divided into two blocks (Figure 4). Thus the grid resolution in these blocks was studied separately. For each block, INS has been used by the various grid sizes to achieve the best results. It was found that in block 1, a grid of 100 × 60 points, while in block 2, a grid of 400 × 40 points were sufficient (the left number is the number of nodes in the x direction, and the right number is the number of nodes in the r direction).

All grids were generated by an elliptic grid generator that had uniformity in the grid size and small values of grid distortion. These characteristics will significantly improve the accuracy of the numerical solution. Furthermore, it is well known that successful computations of turbulent flows require careful attention to mesh generation. This is caused by the strong interaction of mean flow and turbulence. Therefore, numerical results for turbulent flow tend to be more grid dependent than those for laminar flow. Since there exists a viscous sublayer near the wall for $y^+ < 2 \sim 8$, it is recommended that the first node (or cell centroid) should be located at distance $y^+ \leq 1$ from the wall [15]. However, a higher $y^+$ could be acceptable, as long as it is certain that this value is well inside the viscous sublayer.

In this study, a sufficiently fine grid was generated near the walls by means of grid clustering functions, while ensuring that the first point near the wall was in the laminar sublayer. Comparisons of numerical and experimental boundary layer profiles (in block 1) and the total pressure profile (in block 2) have shown that the turbulence model and the grid quality near the wall are acceptable.

The base problem (without heat addition) was simulated, and the results are compared with experimental data for the same model. All experiments were conducted at a free stream Mach number of 2, zero degree angle of attack, free stream total pressure and total temperature of 0.85 bar and 298 K, respectively.

3. Experimental facilities

Some main facilities and equipment used during the experimental study, such as the model and the pressure sensors, are described in the following sections.

3.1. Wind tunnel and model

All experiments were performed in a continuous supersonic wind tunnel; $0.4 \leq M_{\infty} \leq 3$, with a rectangular test section size of $60 \times 60 \, \text{cm}^2$ [16]. The glass windows in the sidewalls of the wind tunnel allow observation of the flow pattern and the shock waves over the nose of the inlet via the Schlieren system and a high speed camera (1000 frames/s).

A picture of the model installed in the wind tunnel test section is shown in Figure 6. The inlet is an axisymmetric external compression one, which has been designed and fabricated especially for this research. The model has a fixed geometry, with an $L/d$ (length/diameter) of 4.8. The design Mach number of the model is 2, and the nose apex semi angle is $28^\circ$. The mass flow rate passing through the inlet can be varied via a plug located at the end of its diffuser.

3.2. Pressure sensors and test procedure

The cowl surface pressure distribution was measured using seventeen highly sensitive Honeywell 5 and 10 psi pressure transducers located at different positions. At a location of $x/d = 4$ ($x$ is measured from the tip of the spike) on the cowl surface, a specially designed boundary layer rake, with eleven static and total pressure probes, was installed to measure the boundary layer profile under different conditions. The spike surface pressure distribution was measured with twenty eight pressure probes. Another rake, equipped with nine total pressure sensors, was designed and installed at the end of the diffuser. This rake was used to measure the pressure recovery of the inlet. Figure 7(a) shows the locations of static pressure orifices and boundary layer rakes.

The inlet model was tested at a free stream Mach number of 2 and at zero degree angle of attack. Static and total pressure data, as well as Schlieren pictures, were obtained for all tests.

4. Comparison of experimental and numerical data

Figure 7 compares measured experimental data with corresponding numerical predictions. Figure 7(a) shows static pressure distribution over the spike surface. Similar comparisons, but for the cowl surface, are presented in Figure 7(b). Total pressure distribution at the end of the inlet diffuser, measured by an inner rake located at $x/d = 3.4$, are compared with predicted results in Figure 7(c). The boundary layer profile at a location of $x/d = 4$ on the cowl surface is compared with CFD results in Figure 7(d). Total pressure around the body, as well as static pressure on the cowl surface, were measured by a special boundary layer rake. Flow velocity has been calculated from these data, where the total temperature was assumed to be constant. As seen from Figure 7, numerical predictions compare well with experimental measurements for all cases examined in this investigation. Furthermore, Figure 8 compares the static pressure contours in front of the inlet, together with shock structures that were visualized from the Schlieren technique. From this figure, it is clearly seen that the numerical code has accurately predicted the shock characteristics.

5. Numerical studies of heat addition effect

After gaining confidence in numerical simulation and predictions, as shown in Figures 7 and 8, the effects of heat addition on inlet performance were investigated. The effects of the position, shape and size of the heat source on the flow field were studied. All these cases are under the same free stream conditions, and are compared with the case with no heat source; the base case in the flow field. The heat source parameters mentioned above were changed until the optimum conditions for
Figure 7: Comparison of the numerical predictions with the present experimental results, $M_\infty = 2, \alpha = 0$. (a) Static pressure distribution on the spike; (b) static pressure on the cowl; (c) total pressure distribution in the diffuser inlet at $x/d = 3.4$; and (d) boundary layer profile over the cowl surface at $x/d = 4$.

Each variable were obtained. The effects of various source parameters on inlet performance, i.e. drag coefficient, mass flow rate and pressure recovery, are investigated, and are further compared with the base case, i.e. no heat source. Figure 9 shows the static pressure contours in front of the inlet with the heat source, which is compared to the base case. The numerical solution is for the case when a heat source has been placed at a location of $x/d = -0.03$ in front of the spike.

6. Results and discussion

Comparisons of numerical predictions for all cases containing a heat source with the base case show that drag coefficient has been considerably reduced, but for some locations of the heat source, mass flow rate and pressure recovery were reduced. Thus, it was decided to find an optimum position, where the aforementioned effects were eliminated or minimized. When the source was moved further upstream, the drag continued to decrease and an improvement in mass flow rate and inlet pressure recovery (in comparison with other places) was achieved (Figures 10–12). Note from these figures that all data are non-dimensionalized, with respect to baseline results.

Figure 8: Comparison between experimental and numerical shock structures for the base case. (a) Static pressure distribution on the spike; (b) static pressure on the cowl; (c) total pressure distribution in the diffuser inlet at $x/d = 3.4$; and (d) boundary layer profile over the cowl surface at $x/d = 4$.

Figure 9: Numerical shock structures for the base case and for the one with a source located at $x/d = -0.03$.

Figure 10: Effect of source position on inlet drag coefficient.
the plus sign has been assigned to the downstream direction and \( x/d = 0 \) is the location of the spike nose. It seems that by moving the source from the upward position toward the face of the spike, the source gets close to the cowl lip and the inlet drag force increases. It is evident that one cannot find the best location of the heat source from this figure alone, and other performance parameters must be considered. Variations of the ratio of inlet pressure recovery with source positions are shown in Figure 11. As the source moves downstream, \( x/d = -0.1 \) to \( x/d = 0.08 \), pressure recovery decreases and reaches its minimum value at \( x/d = 0.08 \). By moving the source further toward the cowl lip, passing over the spike, it is seen (from Figure 11) that pressure recovery increases again. The main reason for this behavior is related to the inlet shock structure. In addition, adding heat to a supersonic flow decreases its total pressure [17]. Consequently, the presence of the thermal source has decreased \( P_0 \), independent of its position. The source shock interacts with the inlet nose shock and varies the entire shock arrangement in the vicinity of the inlet; both external and internal. Total pressure loss for this new flow structure with an extra shock, which is caused by the heat source, is more than that for the base case. Moving the thermal source toward the cowl lip has similar effects on the ratio of the inlet mass flow rate, as seen from Figure 12, i.e. it decreases the mass flow rate from \( x/d = -0.1 \) to \( x/d = 0.08 \), followed by an increase for greater \( x/d \)'s. The deflection of flow streamlines, due to the existence of the thermal source, alters the mass flow rate.

From Figures 10–12, it is clearly seen that the best location for the heat source is at \( x/d \approx -0.1 \), because at this location, the drag coefficient is considerably reduced, mass flow rate is slightly increased (in comparison with the base case) and the inlet pressure recovery loss has been minimized.

Figure 12 also shows that when the heat source is moved further upstream from \( x/d = -0.1 \), the mass flow rate reduces considerably, i.e. at \( x/d \approx -0.2 \) and \( 1/\nu_{base} \approx 0.95 \). To investigate the reason for this phenomenon, the flow pattern for two different heat source positions, \( x/d = -0.1 \) and \( x/d = -0.17 \), are compared (Figure 13). For both cases, it is seen that the flow separates. However, for the case where the source is located at \( x/d = -0.17 \), the separation point is in the vicinity of the inlet entrance (Figure 13(b)). While Figure 13(a) shows that this point occurs well inside the inlet. In fact, for this case, \( x/d = -0.17 \), the maximum separation height is approximately at the inlet entrance. However, for the optimum case \( x/d = -0.1 \), the maximum separation height is inside the inlet. Therefore, for \( x/d = -0.17 \), the effective cross section area of the inlet is smaller than for the case where the source is located at \( x/d = -0.1 \). Hence, the mass flow rate for this case decreases, as seen from Figure 13(b). When the thermal source moves further upstream, the spike nose is downstream of the source bow shock, so the Mach number at this location reduces and the angle of the nose oblique shock increases, as seen from Figure 14. In this figure, the inlet shock wave structures have been illustrated for various positions of heat source.

Therefore, based on the numerical study for this specific inlet, the best location of the source is predicted to be at \( x/d = -0.1 \). If the source is placed here, there will be a 22% reduction in drag coefficient. 1% decrease in pressure recovery and finally mass flow rate will increase about 3%, in comparison with base case values. It should be further clarified that this location is calculated based on the assumption that the reflected shock, due to the spike and the heat source, passes through the spike nose (see Figure 14). Consequently, to achieve the best performance for this supersonic external compression inlet, the heat source should be placed at a location where the reflected shock generated due to the source and spike passes through the spike nose. In this location, the oblique shock generated by the spike nose, which pushes streamlines away from the inlet, has been eliminated.

This shock, however, does not change the streamline direction considerably. Figure 15(a) shows streamlines that are displaced after passing through the spike oblique shock for the base case, that will cause the spillage mass flow rate. When the heat source is located at its optimum position, \( x/d = -0.1 \), the corresponding reduction in drag coefficient is due to a decrease in the spillage mass flow rate. In this situation, the oblique shock generated in front of the heat source deflects the streamlines, and this phenomenon will eliminate the displacement seen in Figure 15(a). Therefore, for the thermal heat case, the streamlines are guided toward the inlet entrance (Figure 15(b)). This new arrangement of streamlines reduces both the inlet spillage mass flow rate and the corresponding additive drag.

The reduction of spilled air over the inlet decreases the additive drag and consequently reduces the total drag of the inlet. It further increases the mass flow rate slightly, as seen from Figure 12.

As mentioned before, generation of the shock wave in front of the inlet entrance deflects the streamlines. To further elaborate on this phenomenon, consider a streamline that passes from the cowl lip (as shown in Figure 15(b)). If the pressure distribution along this streamline is investigated
(relative to another one located far from the one under consideration), a force will be obtained that is added to the inlet drag. This force is called pre-entry or additive drag [2]. This part of the drag has a major contribution to the total drag of a supersonic inlet. The heat source affects the streamlines that pass through the inlet entrance; hence they converge (a) \( x/d = -0.1 \).

(b) \( x/d = -0.17 \).

Figure 13: Effect of heat source location on Mach contours and on position of maximum separation height. (a) \( x/d = -0.1 \); and (b) \( x/d = -0.17 \).

Figure 14: Comparison of flow pattern (Mach number contours) for various heat source locations (min. and max. values kept fixed for each case, so the legend is the same for all).

Figure 15: Streamlines and dimensionless static pressure. (a) No thermal source (base case); and (b) thermal heat located at \( x/d = -0.1 \). (Figure 16) and their corresponding spillage drag is reduced significantly. With this description, the heat source has a considerable effect on the additive drag of the inlet. The streamlines that pass through the top of the source are further deflected upward when compared to a similar base case. Thus the flow may rotate before it reaches the cowl lip, as seen from Figure 16. Therefore, the shock that has formed in the vicinity of the cowl lip (as seen from Figure 15(a)) is weaker, and the pressure drag in this region decreases.

After the best location for the source with an annulus shape, as well as a fixed heat flux, was obtained, the source size and shape at this optimum location were further investigated. The numerical results show that for the best shape, the heat source should have a rectangular cross section, with its larger side located along the flow direction.

The performance parameters of this inlet, in the presence of the thermal source, were investigated under off design conditions also. A summary of this investigation is presented...
These investigations showed that the best shape for the heat source was a rectangular cross section stretched in the free stream flow direction. For off design operation cases, the heat source also had some improvements on the aforementioned parameters.

Acknowledgments

The financial support of the Engineering Research Institute is greatly appreciated. In addition, the authors express their gratitude to the personnel of the QRC for their valuable help in conducting the experiments.

References

[1] Roskam, J., *Airplane Design, Part 6*, Roskam Aviation and Engineering Corporation, Ottawa, Canada, pp. 159–164 (1987).
[2] Goldsmith, E.L. and Seddon, J., *Practical Intake Aerodynamic Design*, Blackwell Scientific Publications, London, UK, pp. 110–148 (1993).
[3] Kobayashi, H., et al., “Study on variable-shape supersonic inlets and missiles with MRD device”, *Acta Astronaut.*, ScienceDirect (2007).
[4] Soltani, M.R., et al., “Performance improvement of a supersonic external compression inlet by heat source addition”, *WASET*, 40(51), pp. 267–274 (2008).
[5] Macheret, S.O., et al., “Scramjet inlet control by off-body energy addition: a virtual cowl”, *AIAA 2003-0032*, pp. 1–18 (2003).
[6] Girgis, I.G., et al., “Steering moments creation in supersonic flow by off-axis plasma heat addition”, *J. Spacecr. Rockets*, 43(3), pp. 607–613 (2006).
[7] McAndrew, B., et al., “Aerodynamic control of a symmetric cone in compressible flow using microwave driven plasma discharges”, *AIAA 2002-0093*, pp. 1–7 (2003).
[8] Miles, R.B., et al., “Plasma control of shock waves in aerodynamics and sonic boom mitigation”, *AIAA 2001-3062*, pp. 1–12 (2001).
[9] Kremeyer, K., “Lines of pulsed energy for supersonic/hypersonic drag reduction: generation and implementation”, *AIAA-2004-0984*, pp. 1–11 (2004).
[10] Smith, C.F. and Smith, G.E., “Two stage supersonic inlet (TSSI): 10-inch model calculations”, *NASA, CR-2005–213287* (2005).
[11] Lu, P.J. and Jain, L.T., “Numerical investigation of inlet buzz flow”, *J. Propul. Power*, 14(1), pp. 90–100 (1998).
[12] Goldale, S.S. and Kumar, V.R., “Numerical computations of supersonic inlet flow”, *Internat. J. Numer. Methods Fluids*, 36, pp. 597–617 (2001).
[13] Salowski, B., et al., “Evaluation and application of the baldwin-lomax turbulence model in two-dimensional compressible boundary layers”, *NASA, TM-105810* (1992).
[14] Kumar, A., “Numerical simulation of scramjet inlet flow fields”, *NASA, Technical Paper 2517* (1986).
[15] Blazek, J., *Computational Fluid Dynamics: Principles and Applications*, Elsevier Science, London, UK, pp. 225–245 (2001).
[16] Soltani, M.R., et al., “Flow measurements around a long axisymmetric body with varying cross section”, *AIAA-05-50* (2005).
[17] Anderson, J.D., *Modern Compressible Flow with Historical Perspective*, 3rd ed., McGraw-Hill, New York, USA (2003).

Mohammad Reza Soltani has a Ph.D. in Aerodynamics from the University of Illinois at Urbana-Champaign, USA, and is now Professor in the Aerospace Engineering department of Sharif University of Technology, Tehran. His research interests include Applied Aerodynamics, Unsteady Aerodynamics Wind Tunnel Testing, Wind Tunnel Design and Data Processing.

Mohammad Farahani was born in 1978 in Arak, Iran. He obtained his B.S. and M.S. degrees in Aerospace Engineering from the Department of Aerospace Engineering at Sharif University of Technology, Tehran, Iran, where he is now a Ph.D. candidate in the same subject. He works in the field of Experimental Aerodynamics.

Javad Sepahi Younsi was born in 1985 in Younsi, Gonabad, Iran. He obtained his B.S. and M.S. degrees in Aerospace Engineering from the Department of Aerospace Engineering at Sharif University of Technology, Tehran, Iran, where he is now a Ph.D. candidate in the same subject. He works in the field of Numerical and Experimental Aerodynamics.

Table 1: Effect of heat source on the various performance parameters at different $M_{\infty}$

| Parameter | $M_{\infty} = 1.8$ (%) | $M_{\infty} = 2$ (%) | $M_{\infty} = 2.2$ (%) |
|-----------|------------------------|---------------------|------------------------|
| $C_d$     | −16                    | −22                 | −24                    |
| $\bar{m}$ | −0.9                   | +3.1                | −3.9                   |
| Pressure recovery | +0.7                 | −1                  | −4.8                   |

Figure 16: Effects of heat source located at $x/d = −0.1$ on streamlines.