World Academy of Science, Engineering and Technology
International Journal of Aerospace and Mechanical Engineering
Vol:2, No:4, 2008

Performance Improvement of a Supersonic External Compression Inlet by Heat Source Addition

Mohammad Reza Soltani, Mohammad Farahani, and Javad Sepahi Younsi

***Abstract*—**Heat source addition to the axisymmetric supersonic inlet may improve the performance parameters, which will increase the inlet efficiency. In this investigation the heat has been added to the flow field at some distance ahead of an axisymmetric inlet by adding an imaginary thermal source upstream of cowl lip. The effect of heat addition on the drag coefficient, mass flow rate and the overall efficiency of the inlet have been investigated. The results show that heat addition causes flow separation, hence to prevent this phenomena, roughness has been added on the spike surface. However, heat addition reduces the drag coefficient and the inlet mass flow rate considerably. Furthermore, the effects of position, size, and shape on the inlet performance were studied. It is found that the thermal source deflects the flow streamlines. By improper location of the thermal source, the optimum condition has been obtained. For the optimum condition, the drag coefficient is considerably reduced and the inlet mass flow rate and its efficiency have been increased slightly. The optimum shape of the heat source is obtained too.

***Keywords*—**Drag coefficient, heat source, performance parameters, supersonic inlet.

I. INTRODUCTION

FOR the airplanes and missiles that incorporate air-breathing engines, an inlet is needed to provide required air by the engine from the free-stream conditions. Inlets are one of the most important parts of these vehicles since it has the greatest contribution in producing the thrust with minimum losses, Fig. 1, [1]. Operation of the combustion is ensured by the incoming air with sufficient mass flow rate, total pressure, and Mach number. Therefore inlets are designed so that air is brought into engine with small disturbances and acceptable condition for combustion with minimum drag and total pressure loss [2]. High speed flow is always accompanied by the shock waves that reduce the overall efficiency considerably. Today large numbers of fighters have external compression inlets. These inlets have subcritical, critical and supercritical operating conditions, Fig. 2. The best performance occurs in the critical condition where the normal shock is tangent to the inlet lip. In this condition the oblique shock that forms at the spike nose passes at the inlet lip, thus preventing the spilled air and consequently additive drag vanishes. However, since the critical condition is almost always unstable due to the free stream disturbances and changes in the flight Mach number; most inlets operate in subcritical condition. Consequently the inlet goes through subcritical or supercritical operation and its parameters vary. To prevent these undesired phenomena most inlets are design for the subcritical operation, hence it is allowed that some air deflects to the outer cowl surface. As seen from Fig. 2, the subcritical operation has undesirable effects on the performance.

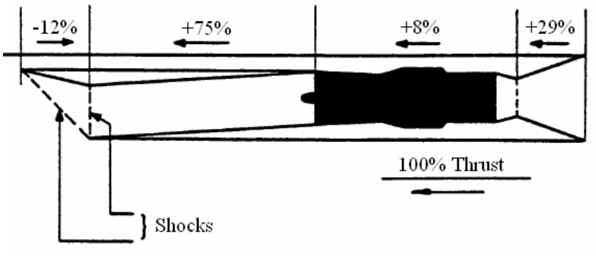


Fig. 1 Example of thrust distribution over a supersonic engine installation at M∞=2.2, [1]

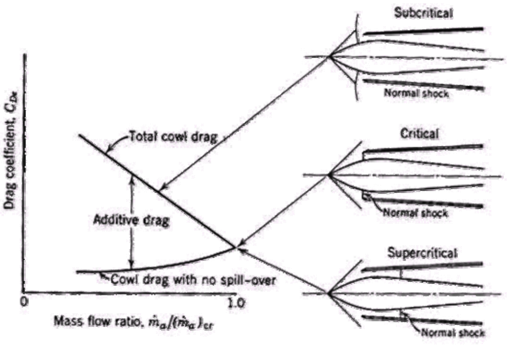


Fig. 2 Inlet performance in subcritical, critical and supercritical operation, [2]

Manuscript received April 30, 2008. This work was supported by the Engineering Research Institute, Tehran, Iran.

Mohammad Reza Soltani is with the Department of Aerospace Engineering, Sharif University of Technology, Tehran, Iran (e-mail: m.soltani@sharif.edu).

Mohammad Farahani is with the Department of Aerospace Engineering, Sharif University of Technology and Engineering Research Institute, Tehran, Iran (e-mail: farahani@ae.sharif.edu).

Javad Sepahi Younsi is with the Department of Aerospace Engineering, Sharif University of Technology, Tehran, Iran (e-mail: j_sepahi@ae.sharif.edu).

To reduce these, using an inlet with variable geometry is desirable. Using aerospikes and recently MRD (multi-row-disk) devices can reduce drag [3].However the design becomes too complex in addition to the extra cost and weight that must be considered. Heat Addition into the flow field at a proper place and a proper rate can have positive effects in the inlet performance. Macheret *et al.* studied the effects of applying a heat source on a two dimensional inlet operating in the hypersonic regime [4]. Further they attempted to control the upstream flow of an inlet by making a plasma environment. They have shown that heat addition to a supersonic flow over a cone can increase the lift force and decrease the drag force [5-7]. Kremeyer investigated an energy deposition method and apparatus over the past several years [8]. His numerical simulations indicate wave drag reduction up to 96% through the use of such an apparatus. Also the propulsive gain of Kremeyer investigation was consistently positive, meaning that the energy saved due to drag reduction was consistently greater than the amount of energy invested (e.g. deposited ahead of the vehicle). The highest ratio of energy-saved/energy invested was ~6500%.

In this study the heat has been added to the flow field in two ways; i) the front portion of the spike is heated, ii) An imaginary heat source has been added at off-body point upward the cowl lip. These methods are applied to a supersonic inlet by using numerical simulation. This inlet under study is an axisymmetric one with external compression which has been tested at a free stream Mach number of 2. All tests were conducted at a trisonic wind tunnel in Iran. Surface pressure distribution on spike and cowl, boundary layer data, and the inlet performance parameters have been obtained from the wind tunnel tests. The experimental results have been used to validate the numerical predictions.

The flow field around the inlet without heat addition has been solved by a commercial CFD code. The results of the numerical solution were compared with the experimental data. Numerical simulation parameters such as boundary conditions or grid distribution have been modified until the comparisons were acceptable. After confirming the results of numerical solution, heat has been added to the flow and its effect on the drag coefficient, mass flow rate and the efficiency of inlet (ratio of total pressure in the inlet end to the free stream total pressure) has been investigated. Note that all other conditions were similar to those of the wind tunnel ones.

Warming up of the front portion of the spike is easier to apply, in comparison with the adding imaginary heat source, it reduces drag coefficient, but it also reduces considerably inlet mass flow rate too.

In the second method, effects of position, size, and shape of the heat source on the inlet performance have been studied and the optimum conditions were obtained. The best location of the thermal source has been obtained which is in the upstream location but not very far from the oblique shock forming at the nose of the spike. Results show that the thermal source deflects the flow streamlines passed the inlet and improves the inlet performance parameters. For the optimum condition, the drag coefficient is reduced considerably and the inlet mass flow rate and its efficiency have been increased. In addition, elongation of the bow shocks which forms in front of the thermal source had no contact with the spike surface, so there is no separation due to the shock and boundary layer interaction. A bow shock forms in front of the heat source. By controlling the position of this shock the spilled air has been reduced while the inlet efficiency has been increased.

The heat source causes the streamlines that were supposed to spill over the inlet to deflect through it. The inlet efficiency can be increased by increasing the number of oblique shocks that forms at the inlet entrance. A common way for increasing the number of these shock waves is to change the spike surface inclination. However, this method causes the flow to separate behind the shocks; hence the shocks should not be allowed to have any contact with the surface. The shock in front of the heat source that is located away from the spike wall can do this. It decelerates the flow without flow separation, since there is no shock-boundary layer interaction.

In this work the mechanisms of heat source addition or warming up the spike are not investigated, but some methods such as supplying RF energy to a volume that has been pre-ionized by a laser and using hot-air jets, can be used [4].

II. COMPUTATIONAL CONSIDERATIONS

The flow field has been solved by a commercially numerical flow simulation tool. The base problem (without heat) was simulated and the results compared with experimental data of the same model. These experiments were conducted in Iran at a free stream Mach number of 2, total pressure of 0.85 bar and total temperature of 298 K.

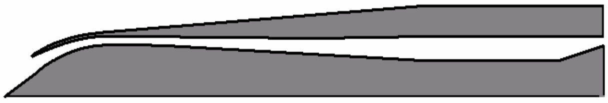


Fig. 3 Schematic of the inlet

Fig. 3 shows the geometry of the inlet used in this investigation. It is an axisymmetric inlet and was designed for a Mach number of 2. At its design conditions, the inlet operates at subcritical condition that has some spillage. Numerical solver uses a hybrid grid with triangular and quadrilateral cells, Fig. 4. This grid has been generated so that it can properly simulate the regions near the boundary layer and the regions with high gradient in the flow properties or in geometry. The flow is compressible, viscous, turbulent, and axisymmetric. The outlet boundary condition of the inlet has a significant effect on the accuracy of the data. In this boundary condition the static outlet pressure is set to be the ambient pressure. With these considerations the differences between the numerical results and the experimental data were minimized as illustrated in Fig. 5.

III. EXPERIMENTAL RESULTS

An axisymmetric inlet was tested in a supersonic speed.

The model had a fixed geometry with an L/d of 4.8. The tests were conducted in a 60×60 cm^2 cross section supersonic wind tunnel [9]. Data have been obtained at a free stream Mach number of 2 for various mass flow rates. Both static and total pressures over the cowl and the spike surfaces were measured. Locations of the static pressure orifices and the boundary layer rakes are shown in Fig. 5-a. In addition, the flow was visualized using a Schlieren technique. Fig. 5 compares the experimental data with the corresponding numerical predictions. Fig. 5-a shows the static pressure distribution over the spike surface. Comparison of the cowl static pressure are presented in Fig. 5-b. Total pressure distribution at the end of the inlet diffuser which was measured by an inner rake are shown in the Fig. 5-c. In this figure the ratios of the total pressure to free stream static pressure at a position of x/d=3.4 are shown. Boundary layer profile at a position of x/d=4 on the cowl surface is compared in Fig. 5-d. Total pressure around the body as well as the static pressure on the cowl surface were measured by a boundary layer rake. The flow velocity has been calculated from these data assuming constant total temperature. As seen in Fig. 5, a-d, the numerical predictions compare well with the experimental measurements.

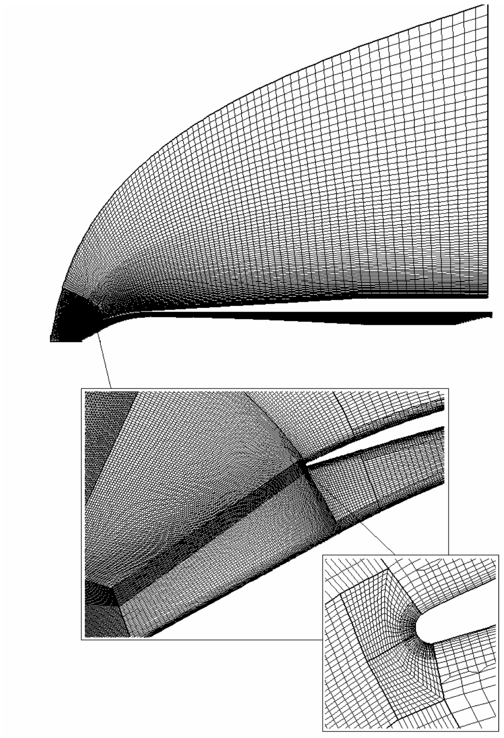
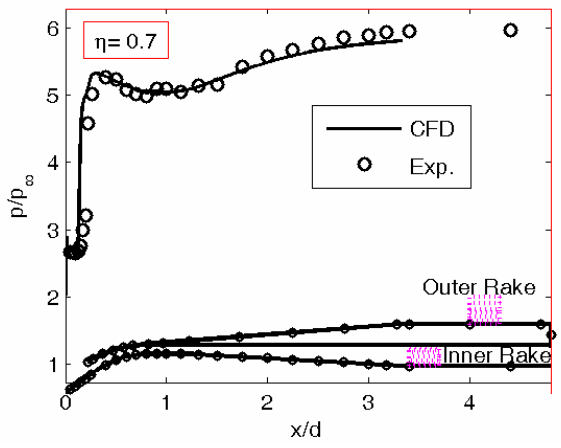
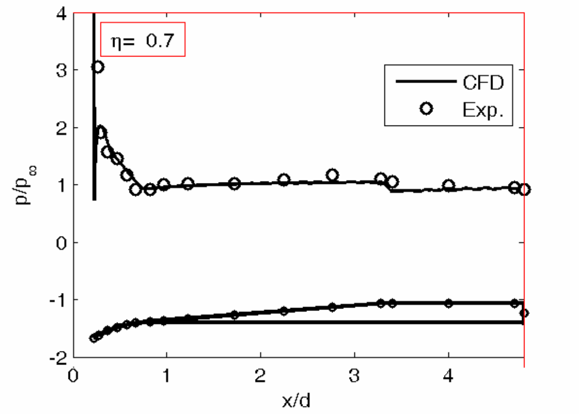


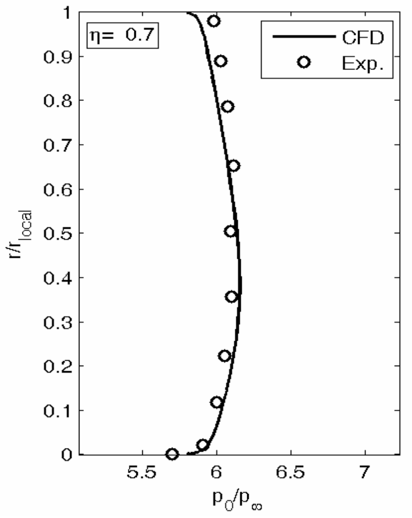
Fig. 4 Grid used for computational procedure



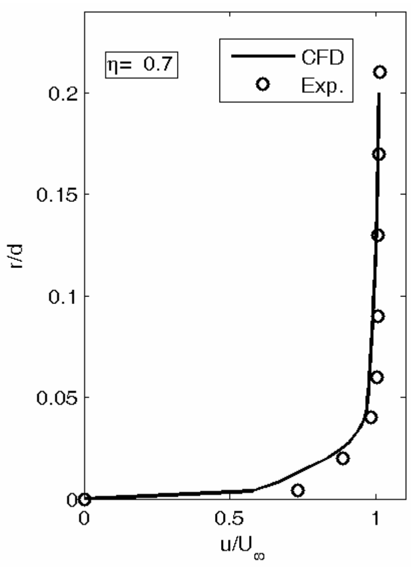
5-a) Static pressure on the spike



5-b) Static pressure on the cowl



5-c) Total pressure distribution in the inlet diffuser at position of x/d = 3.4



5-d) Boundary layer profile over the cowl surface at position of x/d = 4

Fig. 5 Comparison of the numerical predictions with present experimental results at Mach number of 2

IV. Numerical Studies

After relying on the numerical simulation, the effects of heat addition on the inlet performance were investigated. When the spike is heated the flow separation was anticipated, to prevent this, roughness was added to the spike surface was roughed. The effects of the heat source position, shape, and size on the flow field have been studied too. All of these cases are at the same free stream conditions for the case where there was no heat source (the base case) in the flow field. The results of the heat source cases have been compared with the base one. The heat source parameters that mentioned above, was changed until the optimum conditions for each of the variables were obtained. The effects of the source parameters on the inlet performance i.e. drag coefficient, mass flow rate and its efficiency were investigated and were compared with the similar inlet without the heat source.

V. Results and Discussion

Heat addition by inserting a thermal source is easier to apply in comparison with the second method and it reduces the drag too, but it has uninterested effect on the inlet mass flow rate and considerably reduces it. In addition, a small disturbance can separate the flow on the spike wall. Therefore the first manner is not suitable.

In all cases with the heat source, the drag coefficient reduced considerably, but at some positions the existence of the source reduced the mass flow rate and the efficiency when compared with the base data. Thus an optimal position was found where these undesired effects were minimized either. When the source was moved toward the upstream of the spike oblique shock the drag was further reduced and an improvement in the mass flow rate and the inlet efficiency was obtained, Fig's 6-8.

Fig. 6 shows variations of the ratio of the drag coefficient with the longitudinal location of the thermal source. Note that the plus sign has been assigned to the downstream direction and x/d=0 is the location of the spike. It seems that by moving the source from the upward position toward the downward location, the inlet drag force at first decreases slightly followed by a sharp increase as the source gets close to the cowl lip. There is an optimum location for the thermal source where the drag has its minimum value at that location. From this figure it is clearly noticed that there exists an optimum location for the thermal source where the drag has the minimum value at that location. This optimum position of the source from this figure is found to be at about x/d= -0.03, i.e. the source must be located in front of the spike nose. Variations of the ratio of the inlet efficiency or pressure recovery with the source position are shown in Fig. 7. As the source moves downstream, x/d= -0.1 to x/d=0.08 the efficiency decrees until reaches its minimum value at x/d=0.08, by moving the source further toward cowl lip, it is seen that the efficiency increases, Fig. 7. The main reason for the behaviour of the efficiency, as shown in Fig.7 treatment is related to the inlet shock structure. When the source is located upstream of the spike nose, the corresponding weak oblique shock decelerates the flow slightly, thus it weakens the nose shock and these two shocks together cause a reduction in the total pressure loss in comparison with a single shock for the base case. Movement of the thermal source toward the cowl lip has similar effects on the ratio of the inlet mass flow rate as seen from Fig. 8; i.e. it decreases the mass flow rate from x/d= -0.1 to x/d=0.08 followed by an increase. The deflection of the flow stream lines due to the existence of the thermal source changes the mass flow rate. Fig's 6-8 show the effects of the heat source on the inlet performance. In each figure the performance parameters were identified with those of before applying the heat source. These results clearly show an improvement in the inlet performance where a thermal source is added at a position of x/d<-0.03.

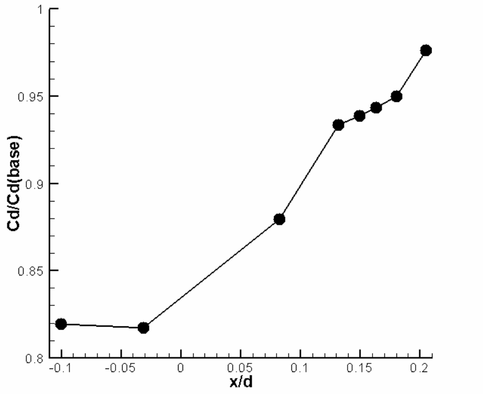


Fig. 6 Effect of source position on the inlet drag coefficient

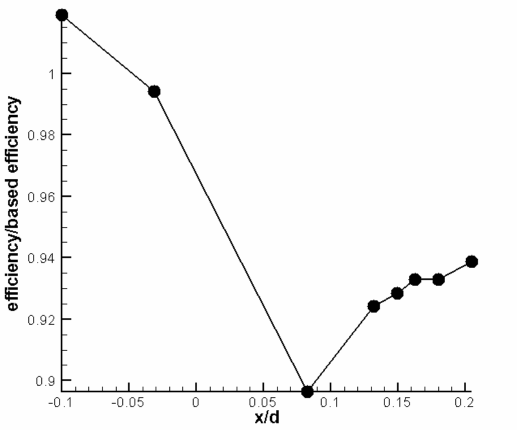


Fig. 7 Effect of source position on the inlet efficiency

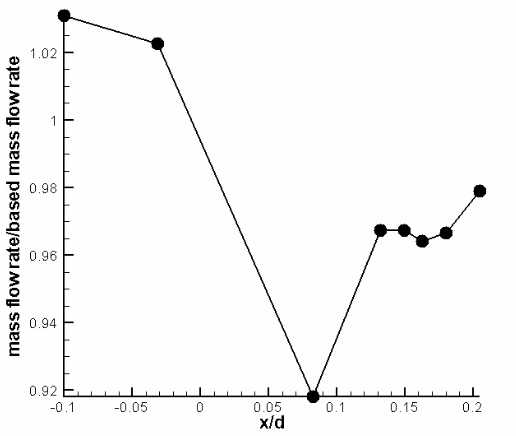


Fig. 8 Effect of source position on the inlet mass flow rate

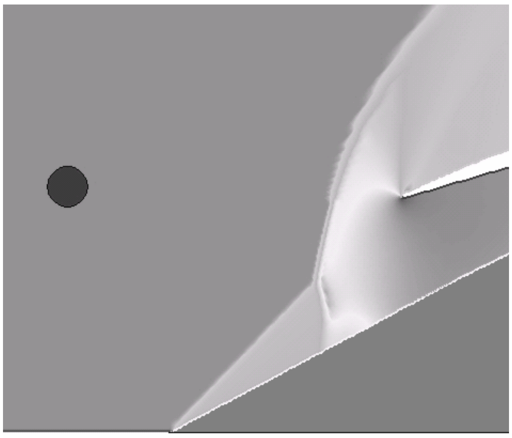


Fig. 9 Static pressure contours in the front portion of the inlet for the base case (without heat source)

Source position at x/d= -0.1 together with the static pressure contours without the source are presented in Fig. 9. This figure shows the shock structures for the base case, also the shape and position of the heat source at the most upstream location i.e. x/d= -0.1 are specified. In this picture the source is not apply yet.

Thus from Fig's 6-8 it is obvious that if the source is moved upward at x/d= -0.1,there will be a 22% reduction in the drag coefficient, about 2% increase in the efficiency and finally the mass flow rate would increase about 3% in comparison with the flow field without source. However, this is not the optimum position because for this situation, x/d= -0.1, the distance between the source shock and the inlet nose shock is too large. Therefore, when the flow elements pass from the first shock, they may accelerate before reaching the inlet, terminally with a strong normal shock above the spike wall. The formation of the strong normal shock causes separation as seen from Fig. 10. However, the experimental results for this intake at the same condition do not show any sign of flow separation at x/d= -0.03, Fig. 11. Nevertheless, for the aforementioned conditions where the heat source was applied the drag coefficient reduced by about 22.4%, along with a 0.6 percents reduction in the efficiency, and an increase of about 2.3% in the mass flow rate, in comparison with the flow field without the thermal source. Therefore based on the above findings one can conclude that the best location of the heat source is at x/d≈ -0.03.

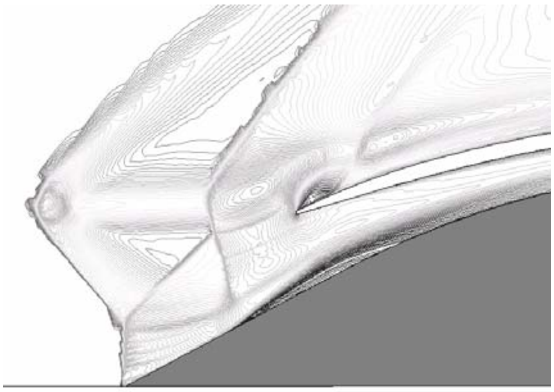


Fig. 10 Static pressure contours for the source which has been located at x/d= -0.1

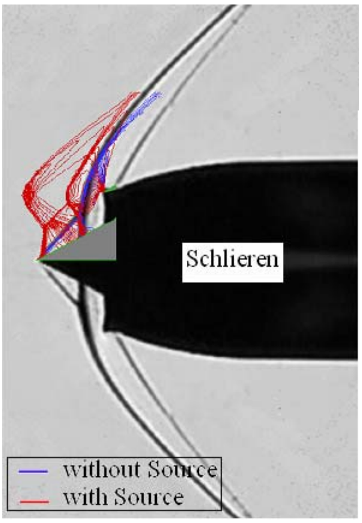


Fig. 11 Static pressure contours for the case with a source which has been located at x/d= -0.03

Fig. 11 shows the static pressure contours in front of the inlet with and without the heat source together with the shock structures that was visualized from the Schlieren technique. As seen from this figure, the numerical code has properly predicted the shock characteristics. The numerical solutions are for the case when a heat source has been placed at a location of x/d= -0.03. When the heat source is in its best position, x/d= -0.03, the corresponding reduction in the drag coefficient is due to a decrease in the spillage mass flow rate. In this situation the oblique shock that is generated in front of the heat source deflects the streamlines and guides them toward the inlet entrance, Fig. 12. Therefore, the streamlines at their new arrangement reduce the inlet spillage and the corresponding additive drag.

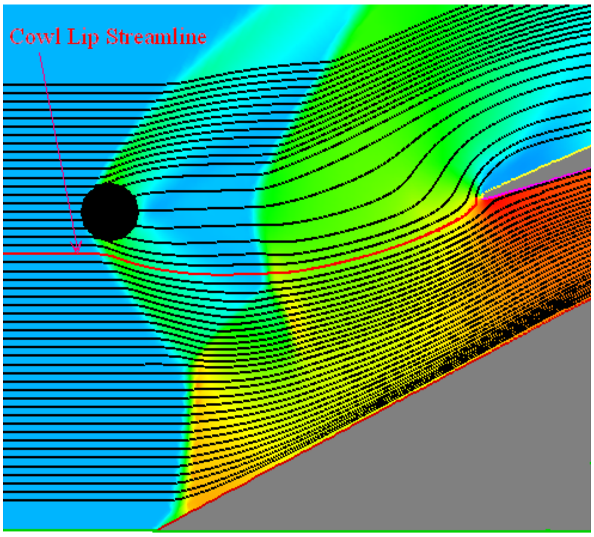


Fig. 12 Stream lines and static pressure contours for the case with a source location at x/d= -0.03

The reduction of the spilled air decreases the additive drag and consequently the total drag of the inlet reduces. It further increases in the mass flow rate too. In addition, due to the generation of a weak shock in front of the heat source the flow passes through an extra shock and the heat source shock together with the nose shock are weaker than the single nose shock that appears for the base case (without the heat source). This phenomenon decreases the total pressure losses and consequently the inlet efficiency increases. Generation of the shock in front of the inlet entrance deflects the streamlines. Now consider a streamline that passes from the cowl lip (as specified in Fig. 12), integration of the pressure distribution along this streamline (relative to a far field streamline) indicates a force which is added to the inlet drag. This force is called pre-entry, spillage or additive drag. Additive drag is a major portion of the total drag of a supersonic inlet. The heat source can have a reverse effect on those streamlines that pass through the inlet entrance and converge them together, in contrast to the inlet shocks that diverge them, Fig. 13. With this description, the heat source has a considerable effect on the additive drag of the inlet. The stream lines that pass through the top of the source are deflected upward, thus the flow rotates before it reaches to the cowl lip as seen in Fig. 13. Therefore, the shock which has formed at vicinity of the cowl lip (as is seen in Fig. 11) is weaker and the pressure drag in this region decreases.

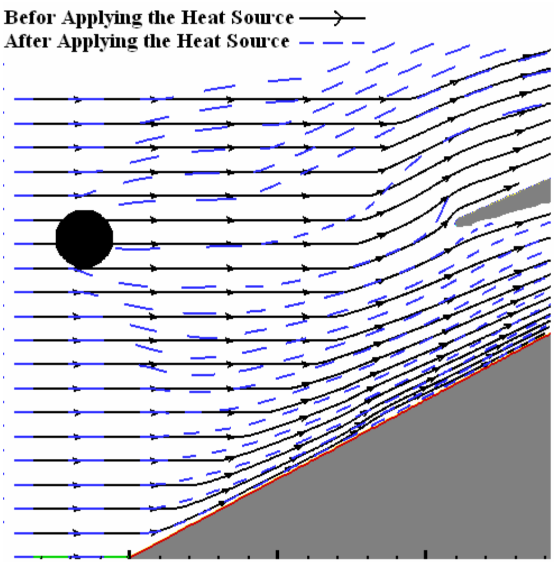


Fig. 13 Effects of the heat source which located at x/d= -0.03 on the stream lines in front portion of the inlet

The total drag of the inlets consists of friction and pressure drag. Fig's 14-15 compare the boundary layer profiles for the base case and the case with the heat source. Fig. 14 shows the boundary layer profile on the surface of the cowl at distance of x/d=4.8 from the spike nose. It is clearly seen that the heat source varied the boundary layer profile significantly and in its inner layer the velocity is larger than the region of outer layer. The heat source has increased the momentum in the boundary layer. The boundary layer profiles over the spike and the inner surface of the cowl for both cases are shown in Fig. 15. Fig's 14 and 15 show that the heat source has slight effect on the friction drag since the slope of boundary layer profile as seen from these figures has not changed considerably. Therefore, it is concluded that skin friction coefficient (C_f) and consequently the friction drag are not affected significantly. To study the effects of the thermal source on the variations of pressure drag, surface pressure distributions over the spike and the cowl are presented in Fig's 16 and 17. It is observed from these figures that the pressure in the front portion of the cowl is slightly reduced. Thus the pressure drag at this section of the inlet has been reduced. In addition, the pressure distribution on the spike surface has been altered in such away that in a small region in front of the spike, the pressure has increased and in its remaining parts the pressure distributions with and without the heat source are the same. Corresponding to the Fig. 11 the heat source affects the shock pattern. It was mentioned before that the inlet wave drag is reduced. Therefore one can conclude that, the heat source reduces the cowl pressure drag considerably because at zero angle of attack the contribution of the front portion of the cowl in the pressure drag is great and the end part of the cowl that has a uniform cross section does not have significant contribution on the pressure drag. Therefore, based on our primary investigation of the effect of thermal source on the inlet drag it can be concluded that heat addition decreases the pressure and the additive drag more than the skin friction drag.

After finding the best location of the source with an annulus shape and fixed heat flux, the optimum heat source size at this location was found too. Study shows that the best shape of the heat source had rectangular cross section that is stretched in the flow direction.

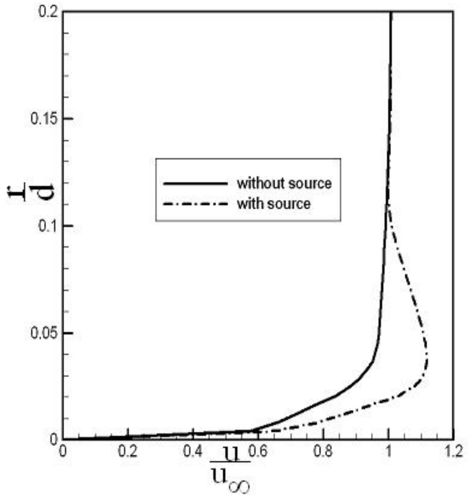


Fig. 14 Effects of the thermal source on the cowl boundary layer profile, x/d=4

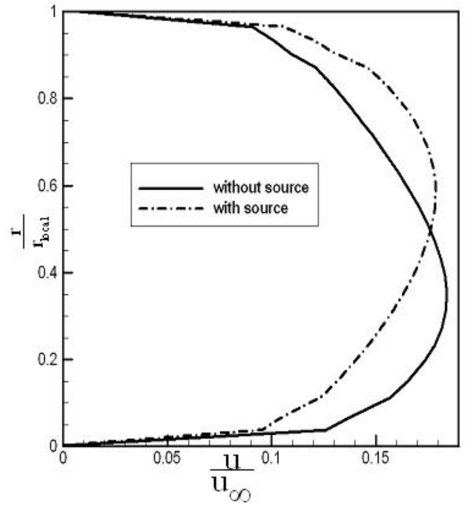


Fig. 15 Effects of the heat source on the spike boundary layer profile, x/d=3.4

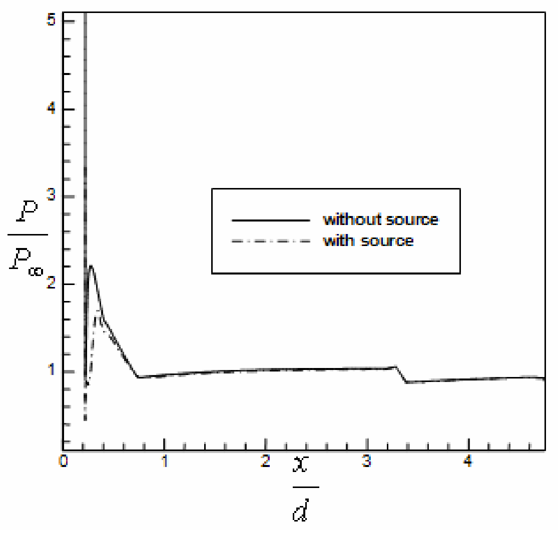


Fig. 16 Pressure distribution over the cowl

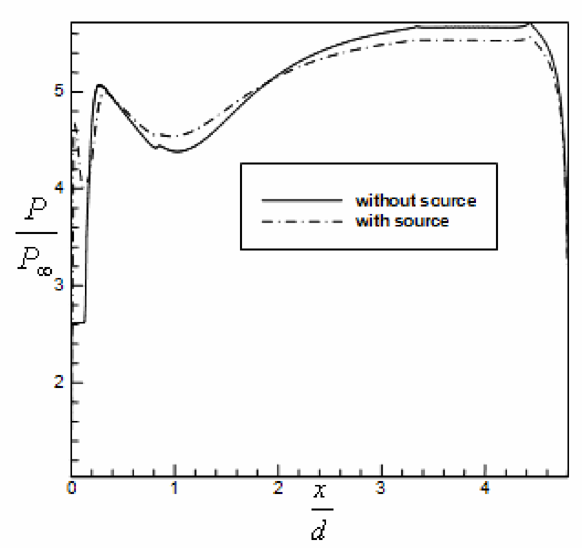


Fig. 17 Pressure distribution over the spike

The performance parameters of the inlet in presence the thermal source were investigated in off design conditions too. A summary of this investigation is presented in table 1. Results show that the source in its optimum location has considerably improved the inlet drag at Mach numbers of greater or smaller than its design Mach number.

TABLE I
EFFECT OF THE HEAT SOURCE ON THE VARIOUS PERFORMANCE PARAMETERS AT DIFFERENT M∞

Parameter	M=1.8	M=2	M=2.2
C_d	-20.6%	-24%	-22.1%
efficiency	+2.2%	+1.6%	-1.6%
$\dot{m}$	-2.9%	-4.8%	-7.8%

VI. CONCLUSION

In this study an axisymmetric supersonic inlet was simulated numerically. The numerical results were compared with the experimental data to ensure the computational data. After these comparisons the effect of heat addition to the flow was investigated in two methods. In the first method the front portion of the spike was heated and the result indicated a reduction in the drag coefficient. However, this method of heat addition reduced the inlet mass flow rate considerably. Therefore, it was concluded that the method was not suitable.

In the second method, a heat source was placed at a distance away from the body. At the first phase of this investigation, effects of the heat source position on the flow field investigated. After finding the best location of the source, its optimum size and shape were obtained using try and error method. For each case effects of adding the heat source on the drag coefficient, mass flow rate and the efficiency of the inlet were studied. Investigations of the heat source position showed that applying the source always reduced the drag coefficient; however at some locations, it may reduce the inlet efficiency and its mass flow rate. With moving the source position to front of the spike nose its best location was found to be upstream but not very far from the nose oblique shock. In this situation, the drag coefficient reduced considerably and the inlet efficiency as well as the inlet mass flow rate increased negligibly. Applying an annulus heat source in the optimum position reduced the drag coefficient by about 22%.

After finding the best location of the thermal source, studies were focused on specifying the best size and shape of the heat source. These investigations showed that the best shape of the heat source was a rectangular cross section that stretched in the free stream flow direction.

ACKNOWLEDGMENT

The financial support of the engineering research institute is greatly appreciated. Also, the authors express their sincere thanks to the personal of the QRC for their valuable help in conducting the experiments.

REFERENCES

[1] J. Roskam, *Airplane Design*, Part 6. Ottawa: Roskam Aviation and engineering Corporation, 1987, pp. 159-164.

[2] E. L. Goldsmith, J. Seddon, *Practical Intake Aerodynamic Design*. London: Blackwell Scientific Publications, 1993.

[3] H. Kobayash *et al*., “Study on variable-shape supersonic inlets and missiles with MRD device,” ScienceDirect, 2006, pp. 1-11.

[4] S. O. Macheret, M. N. Shneider, R. B. Miles, “Scramjet inlet control by off-body energy addition: a virtual cowl,” 41st AIAA Aerospace Sciences Meeting and Exhibit, Reno, Jan. 2003, pp. 1-19.

[5] I. G. Girgis, M. N. Shneider, S. O. Macheret, G. L. Brown, R. B. Miles, “ Steering moments creation in supersonic flow by off-axis plasma heat addition,” Journal of Spacecraft and Rockets, vol. 43, no. 3, May-June 2006, pp. 607-613.

[6] B. McAndrew, J. Kline, R. B. Miles, “Aerodynamic control of a symmetric cone in compressible flow using microwave driven plasma discharges,” 41st AIAA Aerospace Sciences Meeting and Exhibit, Reno, Jan. 2003, pp. 1-8.

[7] R. B. Miles *et al*., “Plasma control of shock waves in aerodynamics and sonic boom mitigation,” 32nd AIAA Plasmadynamics and Lasers Conf. and 4th Weakly Ionized Gases Workshop Conf. & Exhibit, Anaheim, CA, June 2001, pp. 1-13.

[8] K. Kremeyer, “Lines of pulsed energy for supersonic/hypersonic drag reduction: generation and implementation,” AIAA-2004-0984, 2004, pp. 1-11.

[9] M. R. Soltani, M. Taiebi-Rahni, M. Farahani, M. R. Heidari, “Flow measurements around a long axisymmetric body with varying cross section,” AIAA-05-50, 2005.