International Journal of Applied Research in Mechanical Engineering

Volume 1 | Issue 1 Article 4

July 2011

Effect of Micro-Vortex Generator in Hypersonic Inlet

Vivek V. Kumar
Dept. of Aeronautical Engineering Hindustan University, vivekvkumar86@yahoo.com

Surendra Bogadi
Dept. of Aeronautical Engineering Chennai, India, surendra_bogadi@gmail.com

Recommended Citation

Kumar, Vivek V. and Bogadi, Surendra (2011) "Effect of Micro-Vortex Generator in Hypersonic Inlet," *International Journal of Applied Research in Mechanical Engineering*: Vol. 1 : Iss. 1 , Article 4.
DOI: 10.47893/IJARME.2011.1003
Available at: https://www.interscience.in/ijarme/vol1/iss1/4

Effect of Micro-Vortex Generator in Hypersonic Inlet

[1]Vivek V.Kumar, [2]Surendra Bogadi
[1,2]*ME (Aeronautical); Dept. of Aeronautical Engineering*
Hindustan University
[2]*Asst.Professor; Dept. of Aeronautical Engineering*
Chennai, India
E-mail : vivekvkumar86@yahoo.com

Abstract—In the present study computational tests were carried out to get an understanding of the flow field in a pure mixed-compression hypersonic inlet at a free stream Mach number of 7 and an altitude of 35km. Structured meshes have been used for depicting the motion of fluid inside the inlet. First, a grid has been selected after conducting a grid study. Two dimensional simulations were carried out with standard sst k-ω model using FLUENT. Computational results are compared with the available data. The results obtained from the computational tests revealed several important flow field details at hypersonic speeds. The basic shock structure inside the inlet was obtained. The boundary layer formed inner side of the engine had an adverse pressure gradient on the top ramp. Due to this the boundary layer thickens and the static pressure starts to decrease whose effect leads till the trailing edge of inlet. By providing small wedge shaped Micro-Vortex Generator (MVG) where the shock-boundary layer occurs we can smooth the boundary layer formed inside the inlet. Thus there will be more efficient compression than the actual case. The results obtained in the present series of tests, could help the hypersonic inlet design optimization at off-design condition

Keywords-hypersonic; inlet; MVG; CFD

I. INTRODUCTION

The recent advances in aerospace vehicle technology, demands the best possible propulsive balance to provide better thrust and minimum drag. Air-breathing engines are the best choice for the propulsion due to its minimum weight. The efficiency of such engines depends upon the quality of air entering for combustion. To as certain that the inlets are to be designed such that they contribute to their maximum efficiency and minimum drag. The understanding of the flow field inside such inlets at off-design conditions is necessary to predict the overall performance. At hypersonic and supersonic speeds the inlet passage flow field becomes an important performance deciding parameter.

Most of the current research on scramjet inlets around the world has been on swept sidewall compression inlet [1-4]. In this type of inlet the sidewall shocks are in different planes, reducing the total in-plane turning the flow must encounter to obtain the desired compression. In a mixed compression inlet, the inlet cowl is designed to generate an oblique shock, while the sidewalls are undeflected with respect to the flow direction. In this case the flow turning due to the forebody shock(s) and the cowl are in the same (vertical) plane. Since the boundary layer at the inlet-entrance can be quite thick with respect to the inlet height, it has been felt that further turning in the vertical direction would greatly increase the chances of separation, because of shock-boundary layer interactions [5].

By providing small wedge shaped Micro-Vortex Generator (MVG) [3] where the shock-boundary layer occurs we can smooth the boundary layer formed inside the inlet. Thus there will be more efficient compression than the actual case. In the present work, flows field analysis of a pure mixed-compression inlet (henceforth referred to as *mixed inlet*) with and without MVG has been carried out. The primary objectives were to document the flow field within these inlets and compare the performance of the two inlets for a chosen set of freestream conditions.

II. COMPUTATIONAL SETUP

The flow field was simulated using a commercially available FLUENT CFD package. The analysis was run on a laptop Windows computer powered by Intel Core i5 processor. The computer was configured with 4 GB RAM and 1 GB swap space. High Reynolds number sst k-ω model was used for turbulent flow studies.

III. MODEL DESCRIPTION

Based on the literature study carried out, a forebody-inlet configuration, a longitudinal cross-section of which is sketched in Fig.1, was identified for the study. Taking advantage of the geometrical symmetry about the vertical plane, only half the configuration was modeled. The leading edges of the forebody, cowl and sidewall surfaces were modeled as sharp, and the effect of finite bluntness was neglected.

IV. MIXED INLET

The inlet is of a rectangular cross section, which is basically a convergent duct with three zones, each of a different level of convergence. The initial zone (Z-I), had the highest convergent angle, followed by the middle zone (Z-II), which had a moderate level of convergence, and finally Z-III had a convergent angle in between. The area contraction ratio, which is defined as the ratio of the entry area to the exit area, is 2.65.

V. MVG DETAILS

Based on the literature study carried out, a model of Micro Vortex Generator (MVG), a longitudinal cross-section of which is sketched in Fig.2, was identified for the study.

The 2-D computational meshes for the mixed inlet configuration were structured meshes and had approximately 70000 cells. The meshed model of inlet with MVG had approximately 75420 cells and structure mesh chosen for the study. The difference in the mesh density between the mixed inlet and the inlet with MVG was primarily due to the necessity of having a high grid density in the area of MVG was vortices are forming in the latter case.

Appropriate boundary conditions were applied on the surfaces of the model for the flow simulation. Freestream pressure conditions were applied on the exit plane. Inviscid, laminar and turbulent flows were simulated. Inviscid flow studies were done in order to compare the results with the available data and to bring the viscous effects. The freestream Reynolds number was 1.1948 x 10^6 per meter.

VI. RESULTS AND DISCUSSIONS

Though a considerable amount of data was gathered, the results are presented mainly in terms of line plots of static pressure and contours of Mach number and static pressure. In the line plots, static pressure has been normalized with free stream value 830 Pa. These plots are presented for the three surfaces of the inlet model, i.e. ramp, upper surface and the cowl plane. The exit parameters considered are those at x= 0.142 m and not at the actual exit (x=0.14m), since flow acceleration takes place close to the exit due to the freestream pressure condition applied on the exit plane.

A. Inviscid Flow

Mixed Inlet

Fig.3 shows the general shock structure, with the two forebody shocks, the cowl shock as well as the corner shock within the inlet (originating from the cowl at x=4.668m). In inviscid flow, the forebody shocks intersected each other just upstream of the inlet, at an axial location x=3.5m. For the validation of the obtained result the inviscid flow on the upper surface is taken.

Upper Surface (Fig. 4)

On the upper surface, a small pressure rise can be seen at x=0.1m. This is due to the flow field that exists at this point, where the cowl shock and the expansion wave originating at the shoulder interact. Slightly downstream of this point, the pressure rise is due to the cowl shock impinging on the surface. Fig. 5 shows the validation of pressure plot with the available data.

Cowl (Fig. 6)

The plot shows an initial compression due to the flow passing through the cowl shock, followed by another pressure rise due to the corner shock at x=4.668m . A kink is seen in the plot at x=4.65m due to the effect of the expansion waves originating from the upper surface shoulder.

B. Laminar Flow Plots

The pressure profiles for the laminar flow for the mixed inlet showed no qualitative differences from the inviscid plots; the magnitudes of pressures vary, but the shapes of the plots remain, in general the same. The pressures in laminar flow are marginally higher, owing to the increased contraction effects due to a thin boundary layer, as well as boundary layer displacement effects resulting in slightly exaggerated wedge angles.

C. Turbulent Flow Plots

Mixed Inlet (Fig. 7)

Turbulent flow field within the inlet is much more complex than the inviscid or laminar flow fields. This is caused by the thick boundary layer and various viscous interactions that take place. The area averaged Mach number at inlet entry was found to be lower than that observed for inviscid and laminar flows. Consequently the static pressure at inlet entry was higher. This is because of the thick turbulent boundary layer, whose displacement effect increases the contraction ratio, resulting in greater compression. Moreover, the displacement of the boundary layer has the effect of increasing the wedge angle and hence the shock angles. This can be seen from the static pressure rise across the cowl shock, which occurs slightly upstream in turbulent flow as compared to inviscid/laminar.

Mixed Inlet with MVG (Fig. 8)

The location of MVG finalized based on the separation occurring inside the inlet (Fig. 8). At a distance of 0.10 meters from leading edge of forebody a separation was formed due to the shock-boundary layer interaction whose effect is leading till the trailing edge of inlet. At the trailing edge of MVG a vortex is produced which shown as a vector plot in the Fig. 10. Due to the effect of vortices produced the reflected shock coming from the cowl will get again reflected at the top surface without causing any interactions to the boundary layer at the ramp.

VII. CONCLUSIONS

Flow field analysis inside the mixed compression inlet was found to be within acceptable tolerances of the available data. There was little variation in the flow field results between inviscid and laminar flows. This was due to the low molecular viscosity at the altitude considered (μ=1.5287 x 10^{-5} kg/ms) and consequently a thin boundary layer. The line and contour plots presented in the report have given considerable amount of information on the nature of the flow field existing within the inlet. The comparison (Figs. 11 & 12) between the mixed inlet and the inlet with MVG shows the thinning (Figs. 9 & 10) of separation occurring in the case of mixed inlet with MVG. The amount of pressure is also gaining in the case of MVG and thereby proper compression for efficient combustion (Table I).

REFERENCES

[1] ***Scott D Holland, Internal Aerodynamics of a Generic Three-Dimensional Scramjet Inlet at Mach 10, NASA TP 3476, Jan 1995.***

[2] ***Schneider, A., and Koschel, W.W. " Detailed analysis of a mixed compression hypersonic intake ".International Society for Air Breathing Engines (ISABE), ISABE 99-7036, 1999.***

[3] ***Sang Lee, Large Eddy Simulation of Shock Boundary Layer Interaction Control Using Micro-Vortex Generator, PhD Thesis, university of Illinois, 2009.***

[4] ***Frank K. Lu, Adam J. Pierce and Yusi Shih, Experimental Study of Near Wake of Micro Vortex Generators in Supersonic Flow, AIAA 2010-4623, June 2010.***

[5] ***Carl A Trexler, Performance of an Inlet for an Integrated Scramjet concept, J Aircraft, Vol 11, No 9, 1974.***

[6] ***Frank K. Lu, Adam J. Pierce and Yusi Shih, Experimental and Numerical Study of Flow Topology Past Micro Vortex Generators, AIAA Paper 2010-4623, June 2010.***

[7] ***Donde Pratik Prakash, Hypersonic Intake Studies, M.Tech Thesis, Indian Institute of Technology, Bombay, July 2006.***

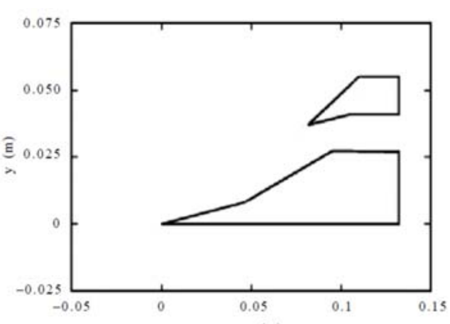
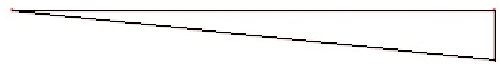


Figure 1. Dimensional sketch of mixed inlet



Length=10.315mm Height=1.57mm

Figure 2. Dimensional Sketch of MVG

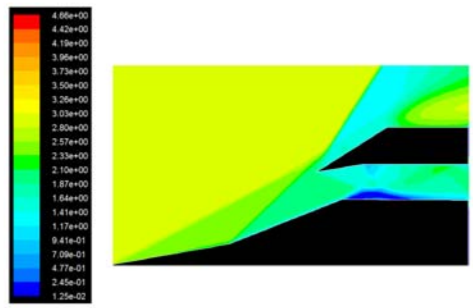


Figure 3. Mach Contour of mixed inlet (inviscid flow)

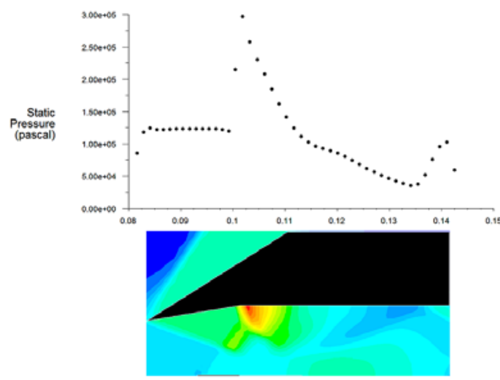


Figure 4. Mixed inlet upper surface static pressure profile (inviscid flow)

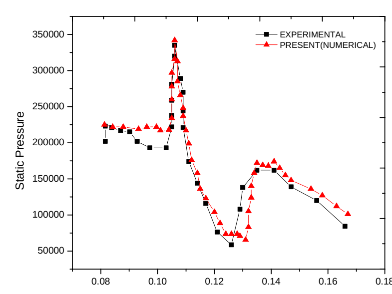


Figure 5. Validation of upper surface static pressure profile (inviscid flow)

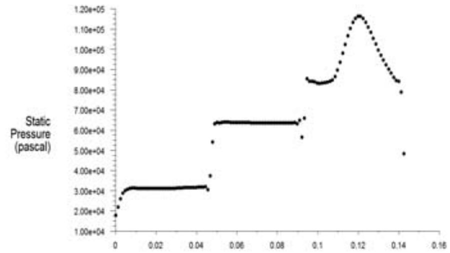


Figure 6. Mixed inlet cowl static pressure profile (inviscid flow)

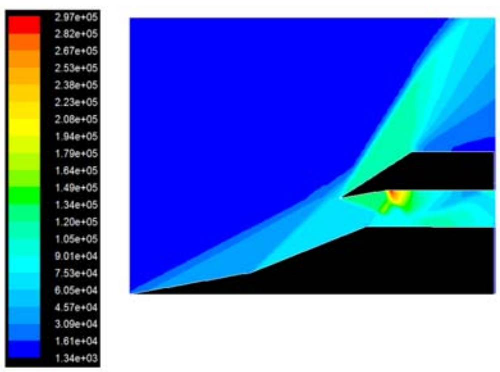


Figure 7. Contours of static pressure of mixed inlet (turbulent flow)

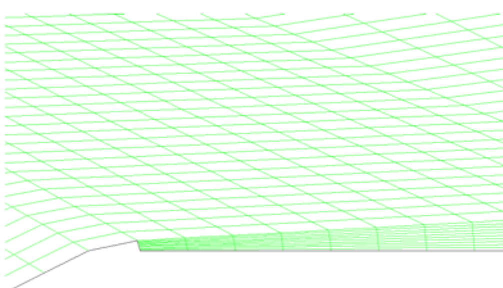


Figure 8. Location of MVG

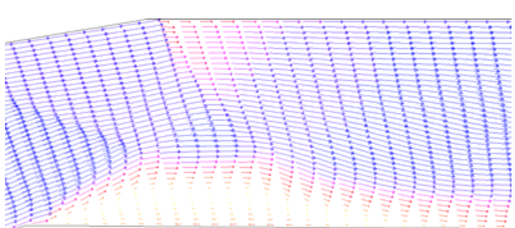


Figure 9. Detailed view of vortices in normal inlet

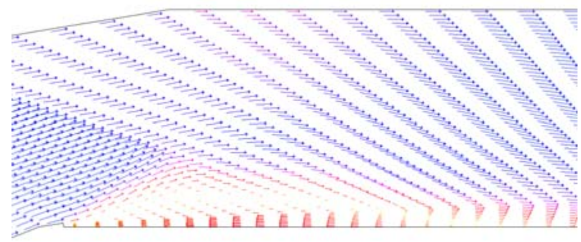


Figure 10. Detailed view of vortices in inlet with MVG

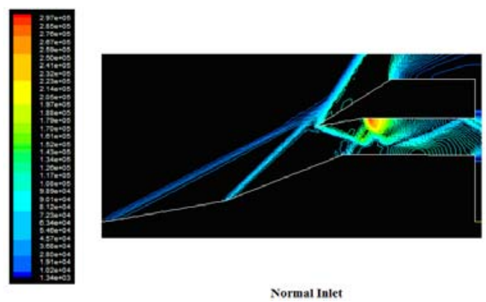


Figure 11. Static pressure contour of inlet without MVG

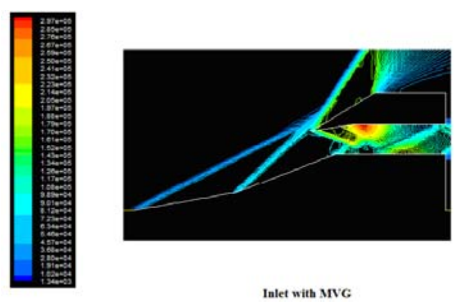


Figure 12. Static pressure contour of inlet with MVG

TABLE I. COMPARISON OF STATIC PRESSURES OF INLET WITH AND WITHOUT MVG

	Static Pressure (Pascal)	
	Entry	Exit
Without MVG	15000	45735
With MVG	15000	48963