VOL. 11, NO. 21, NOVEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

EFFECT OF CIRCULAR CAVITY ON THE PERFORMANCE OF SUPERSONIC AIR INTAKE

N. K. Gahlot, Jitesh, Vikramjeet and Vansh
Department of Mechanical Engineering, SRM University, Delhi NCR Campus, Ghaziabad, India
E-Mail: neerajkumargahlot@gmail.com

ABSTRACT

In the present investigation, computational studies were made to obtain the influence of circular cavity on the performance of a mixed compression supersonic air-intake designed for Mach 2.2 with and without back pressure. Numerical simulation has been conducted with RANS solver by using k-w turbulence model. Starting characteristic of the intake is achieved by incorporation of a cavity of various radius on the second ramp at various locations. 12.2 % of increment in TPR is obtained with circular cavity of radius 1 mm at a location of X/L = 0.1468 on account of some increment in the FD with the cavity. Results obtained could be useful for further studies to improve the intake performance.

Keywords: intake, back pressure, pressure recovery, flow distortion, starting characteristic, performance.

Nomenclature

P	=	Static pressure
P_{inf}	=	Free Stream pressure
M	=	Mach number
L	=	length of the intake
H	=	Intake height at exit from the ramp surface
R	=	Radius of the Cavity
P_o	=	Stagnation pressure
P_{oe}	=	Stagnation pressure at the exit of intake
P_{oinf}	=	Free stream stagnation pressure
X	=	Location of a point along the length of intake
Y	=	Location of a point along the height of intake
TPR	=	Total Pressure Recovery
FD	=	Flow distortion
P_b	=	Back pressure ratio

1. INTRODUCTION

The role of feeding air to the air-breathing propulsion systems is done by specialized aerodynamic ducts called as air-intakes. Air-intakes take air from atmosphere and provide it to the propulsion systems to generate thrust and to the conditioning systems. In high speed flights air-intakes act as a form of a compressor. Air-intakes take air at a higher free stream Mach number and a lower pressure and convert it to a lower Mach number and a higher pressure before feeding it to the engine. The primary goal in the design and development of an air-intake can be divided into two objectives, namely geometry and internal aerodynamics. The geometry of an air-intake must be such that it provides efficient compression generates low drag. The internal flow should be such that it provides nearly uniform flow for entering into the combustor and provides these characteristics over a wide range of operating conditions. The design of the supersonic intake is to achieve the internal aerodynamic performance necessary to meet the mission requirements. These requirements include high total pressure recovery for maximum engine thrust, low distortion for satisfactory compressor operation, tolerance to transient changes in free stream Mach number, angle of incidence, and engine corrected flow demand for safety.

Over the past half-century, a lot of research has been done on various factors, which influence the air-intake flow field. The performance of supersonic air-intake depends on several factors which involves the proper diffusion inside the duct, shock-boundary later interactions, flow separator, distortion etc. the control of flow separation inside the intake duct is a major issue which could even lead to intake un-starts. "Figure-1," shows the complete flow physics inside the mixed compression air intake. Several techniques to control these flow separations are reported in available literatures.

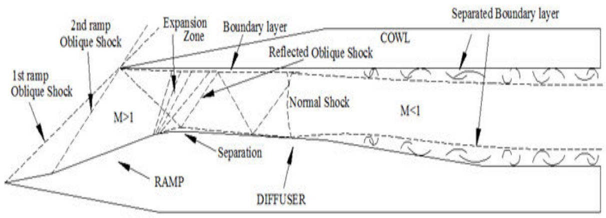


Figure-1. A schematic of flow pattern inside the mixed compression air intake [4].

Neale and Lamb [1] have experimentally studied a two dimensional mixed compression intake at a free stream Mach 2.2 had a maximum pressure recovery of 87 per cent with adoption of 2.8 % bleed the intake. The tests have emphasized the importance of the bleed slot design. Controlling the bleed at increasing rates had created clean normal shock at the entrance of the subsonic diffuser, and provided the efficient subsonic diffusion. Kim [2] has investigated, numerically, the external-compression inlet with a three-dimensional bump at Mach 2 to scrutinize the geometrical effects of the bump in controlling the interaction of a shock wave with a boundary layer and it showed that a bump-type inlet can provide an improvement in the total pressure recovery downstream of the shock wave/boundary layer interaction over a conventional ramp-type inlet. Starting characteristics of a

ARPN Journal of Engineering and Applied Sciences

2D hypersonic intake with side fencing was presented by Saha and Chakraborty [3] and it was observed that wall boundary condition for temperature has a pronounced effect in determining the starting Mach number. Das and Prasad [4] have studied on the starting characteristics of a supersonic air- intake with cowl deflection. The results have indicated that a gain in performance with cowl deflection angle is comparable to the performance with 2.8 per cent bleed, hence cowl deflection could be also thought of as an alternative to the bleeding. Two dimensional studies of a supersonic air-intake with different cowl deflections have done by Das and Prasad [5] with increase in cowl deflection angle, the separation zone seems to reduce. For free exit flow, increase in cowl deflection angle increased the overall performance. Vasana M. Don *et al.* [6] have carried out experimental and computational investigation of the flow filed behaviour of open and close cavity in a supersonic flow. With incorporation of cavity near the shock wave, a large vortex near the rear wall of cavity is shown. This could help in the reduction the separation. An improvement in the pressure recovery is reported by R. K. Jaiman *et al.* [7]. In his investigation, Computations were performed to investigate the flow fields of normal shock wave boundary layer interaction with meso-flaps. The numbers of flaps and their locations were also found to affect the stagnation pressure recovery.

Based on the above literature survey a Passive Cavity can play a major role in reducing the shock wave boundary layer interaction. The principle of the passive cavity consists of establishing a natural circulation between the downstream high-pressure side of a shock and its upstream low pressure. This circulation spreads the shock system while reducing boundary layer thickness. It is achieved by placing a cavity underneath the shock foot region "Figure-2," shows the concept of flow control by passive cavity.

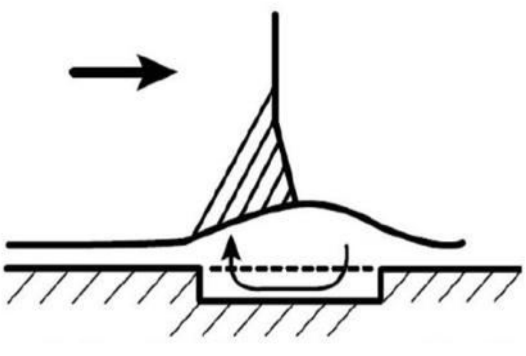


Figure-2. Flow control by passive cavity.

This study is aimed towards understanding the nature of flow in and around an intake with passive cavity. Both qualitative and quantitative tests have been performed numerically to achieve this aim by using the commercially available software FLUENT.

2. INTAKE MODEL DETAIL

To obtain the effect of passive cavity, the basic intake geometry used in [4] has been adopted for the present study. "Figure-3," shows the basic geometrical details of the intake. A passive cavity is made of different radius (R = 0.5, 1, 1.5 & 2 mm) on the second ramp of the supersonic air intake at various locations (X/L = 0.1468, 01856 & 0.2223) respectively.

Studies were carried out on rectangular mixed compression air intake designed for a free stream mach number of 2.2.

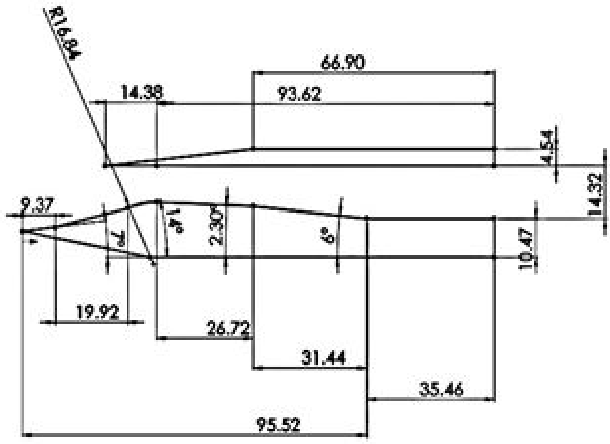


Figure-3. Geometrical details (all the dimensions are in mm)

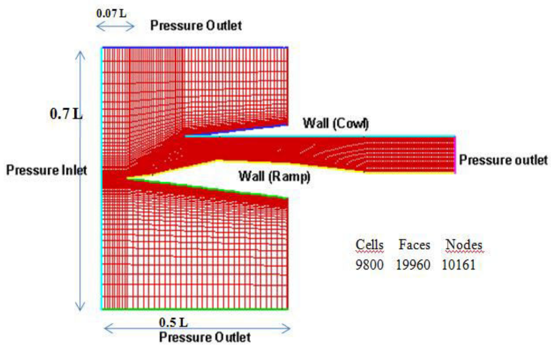
3. COMPUTATIONAL METHODOLOGY

Computations are carried out to understand the flow filed in and around the intake using the commercially available software CFD software FLUENT. The solution method in fluent can be broadly divided into three parts namely: pre-processing, solver and post processing.

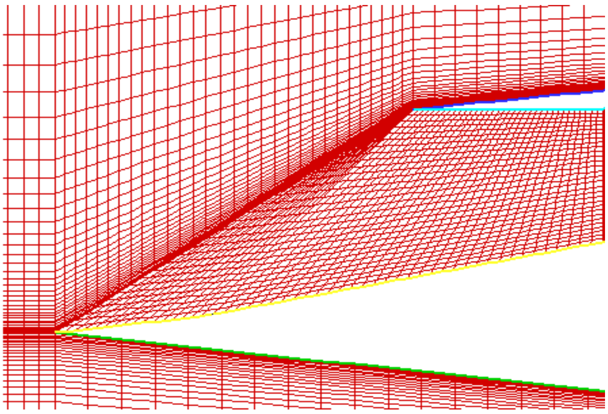
Pre-processing of the problem was done in GAMBIT. In the present study two dimensional grids were generated with and without cavity for all the cases to capture the flow physics. Structured grids with conformal mapping using quadrilateral cells with first cell distance of 0.15 mm near the wall were made very fine at the surfaces and started to coarsen as it moved away from the body. A rectangular domain has been selected. "Figure-4," shows all the details of the grid and the boundary conditions for clean model along with the computed domain. Same grid generation method and cell distance is opted for the cavity model.

ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com



(a) Computational domain with boundary conditions.



(b) Close-up view of grid near the surface.

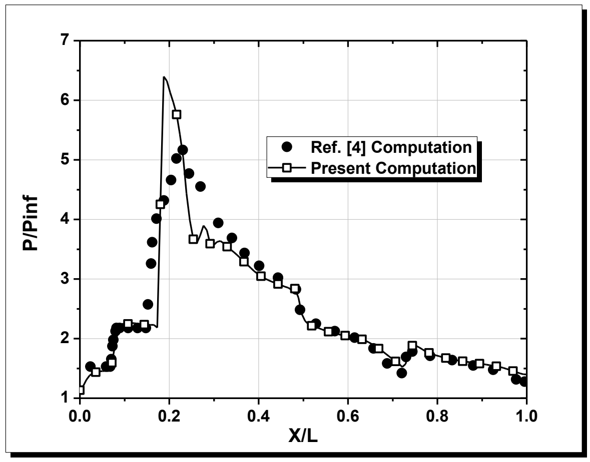
Figure-4. Surface Grid for the intake (clean model).

Once the problem is meshed and the boundary conditions are specified the meshed geometry is then imported as a 'mesh file' into FLUENT. Computations were carried out with double precision steady coupled, explicit solver scheme to solve the compressible Reynolds Averaged Navier Stokes equation. The viscous model chosen for the problem was the standard k-Omega model with turbulent intensity and viscosity ratio as inputs. Standard wall functions were used for the near wall treatment of the flow. For the present simulations, ideal conditions were used. Boundary conditions at inlet were specified by providing the stagnation and static pressures corresponding to a supersonic flow of Mach 2.2.

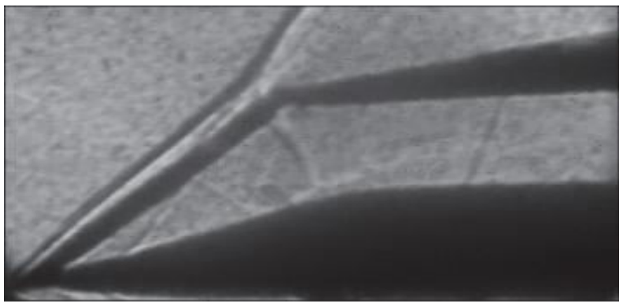
A free stream turbulent intensity of 0•5% was specified at the inlet. For supersonic outflow, all the variables were extrapolated from the interior cells to the boundary.

4. VALIDATION TEST

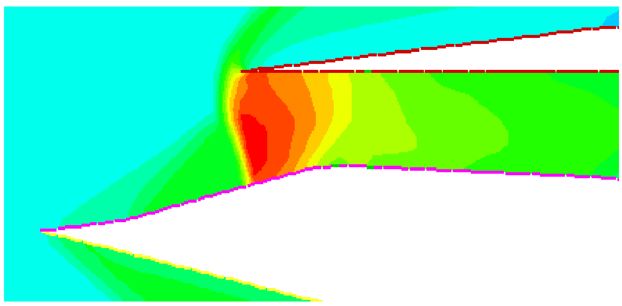
A validation test is performed to validate the adopted boundary and operating condition's to check the accuracy of the present solution. Intake geometry in present computation is similar to the earlier discussion [4]. Computational and the experimental data is taken from the ref [4] and validated with the present computation at zero degree deflection of the cowl for clean model. "Figure-5", clearly indicates a good comparison of the ramp pressure distribution at Mach 2.2,which is also can be consider as a grid comparison test. Schileren image [4] and the present density contour indicate a normal shock at the throat of the intake, which leads the air intake to unstart.



(a) Ramp pressure distribution for clean model at zero degree cowl deflection.



(b) Schileren at zero degree [4].



(c) Density contour of present computation.

Figure-5. Validation test for the present geometry (Clean model).

So now our objective is to start the air intake by adopting the cavity on the second ramp of the supersonic air intake by using the validated boundary and operating conditions.

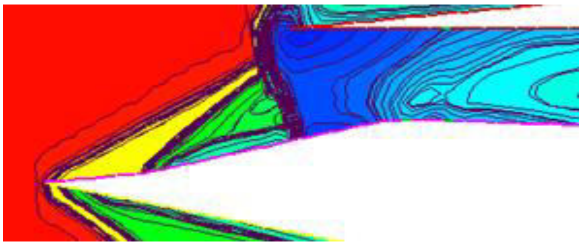
VOL. 11, NO. 21, NOVEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

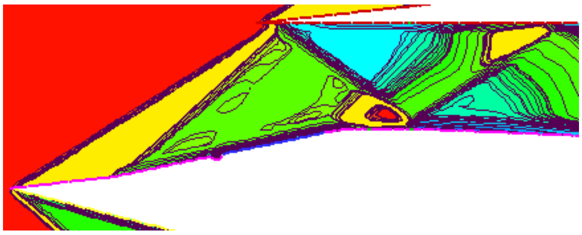
5. RESULT AND DISCUSSIONS

Computations were performed to get an understanding of the flow field around a rectangular mixed compression supersonic air intake with and without passive cavity of circular shape of four different radius (R = 0.5 mm, 1 mm, 1.5mm and 2 mm) at various locations (X/L = 0.1468, 0.1856 and 0.2223) at Mach 2.2 with and without back pressure. Result obtained is presented here.

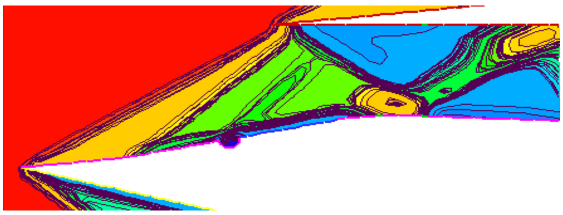
Figure-6 shows the comparison of Mach contour of clean model and intake with cavity of various radius at a location of X/L = 0.1456 for free flow at design Mach. It clearly shows that a normal shock is appearing near the throat, which is the indication of unstart phenomenon. Flow spillages can also be seen at the cowl lip. Large amount of separation can be observed after the normal shock.



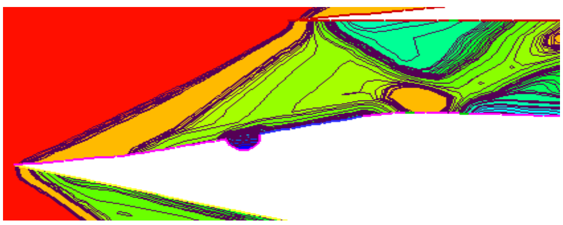
(a) Clean model.



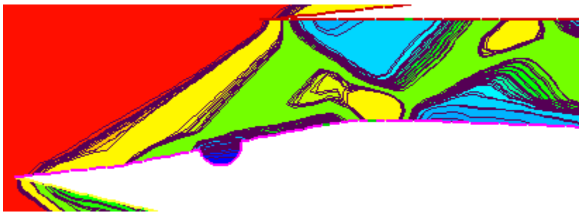
(b) Model with cavity of radius 0.5 mm.



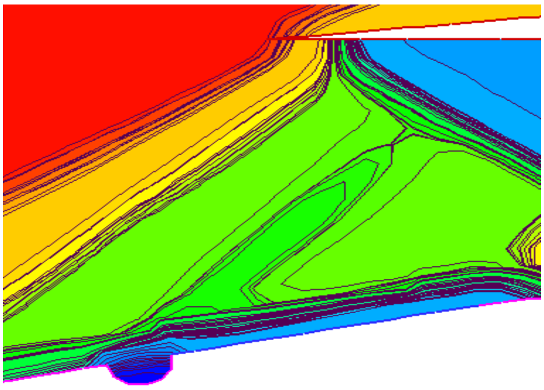
(c) Model with cavity of radius 1 mm.



(d) Model with cavity of radius 1.5 mm.



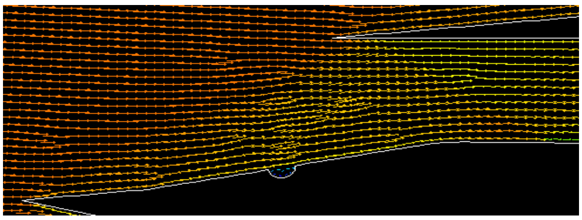
(e) Model with cavity of radius 2 mm.



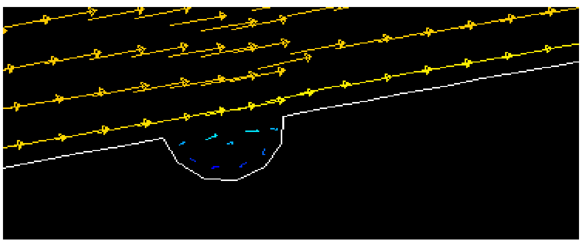
(f) Close-up view of the contour near the ramp surface with cavity Radius of 1 mm.

Fig**ure-6.** Mach contour with and without cavity (X/L= 0.1468) of intake at Mach 2.2**.**

On comparison with cavity model, starting characteristic of the intake is observed, i.e. because of the reduction in shock wave boundary layer separation at the throat. Although the intake is start in all the cases, but still a variation in separation can be seen for different radius of the cavity near the throat. "Figure-7", shows the velocity vector with the cavity model of radius 1 mm. A circulation zone is appear inside the cavity and which helps in improving the flow around the intake.



(a) Velocity vector of ramp surface.



(b) Close-up view of velocity vector.

Figure-7. Velocity vector showing the recirculation zone in the cavity.

VOL. 11, NO. 21, NOVEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

"Figure-8", indicate the density contour for the intake with and without cavity and improvement in the flow is observed. A series of oblique shock is observed and which tells about the series of compression and the smoothness of the flow.


(a) Clean model.


(b) Model with cavity of radius 1 mm at X/L = 0.1468.

Figure-8. Density contour at mach 2.2.

"Figure-9", Shows the total pressure distribution at the exit of intake with and without cavity at various location with 1 mm radius of the cavity for free flow at Mach 2.2. It clearly shows the improvement in the total pressure at the exit of intake with cavity model; however the total pressure distribution for the entire cavity model seems to be similar.

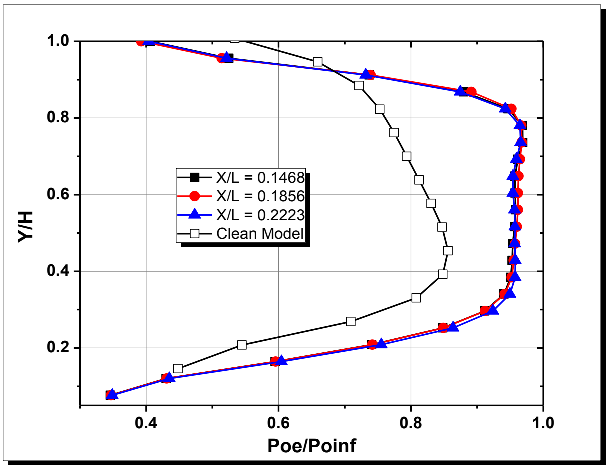


Figure-9. Total pressure distribution at the exit of the intake.

"Figure-10", indicates the ramp surface pressure distribution with and without cavity model, it shows the jump in the pressure for the clean model on the second ramp, i.e. because of the normal shock but for the case of cavity model, a drop in the pressure is observed and that's what is required for the characteristic of a started air intake.

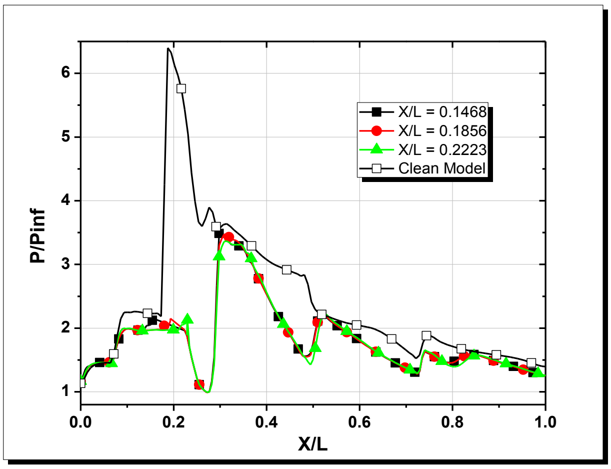


Figure-10. Comparison of ramp surface static-pressure distribution.

"Figure-11", shows the effect of the radius and position of cavity on the TPR of supersonic air intake. TPR is one of the important parameter to measure the performance of the intake. The maximum value of the pressure recovery is achieved at X/L = 0.1468 with radius of 1 mm. TPR is calculated by using equation (1).

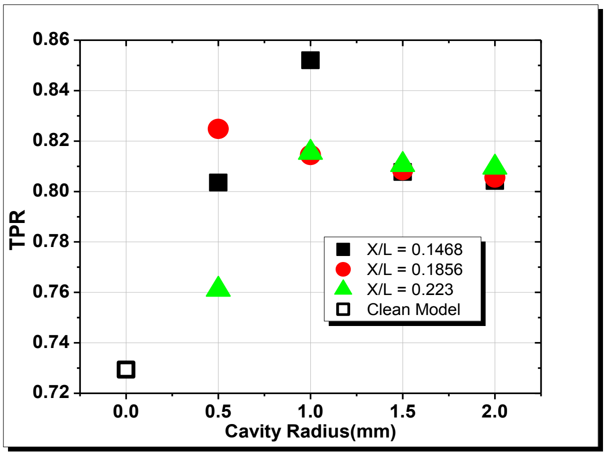


Figure-11. Effect of cavity radius on TPR.

$$TPR = \frac{1}{n}\sum_{j=1}^{n}\left(\frac{P0e}{P0j}\right)j \qquad (1)$$

Back Pressure Computations

"Figures 12 & 13", shows the ramp pressure distribution and the density contour of the supersonic air intake with cavity model at various operating conditions. As the operating conditions changes, the movement of the normal shock wave towards the throat of the intake can be seen. Similarly from the ramp pressure distribution at the location of the normal shock, a rise in the pressure distribution is observed. Separation is also observed just after the normal shock towards the ramp of the supersonic air intake.

VOL. 11, NO. 21, NOVEMBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

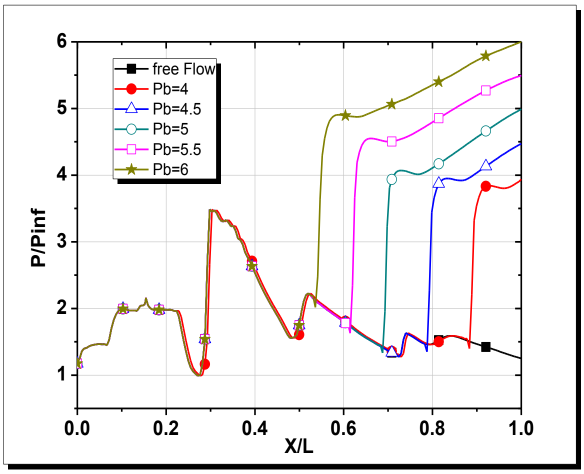
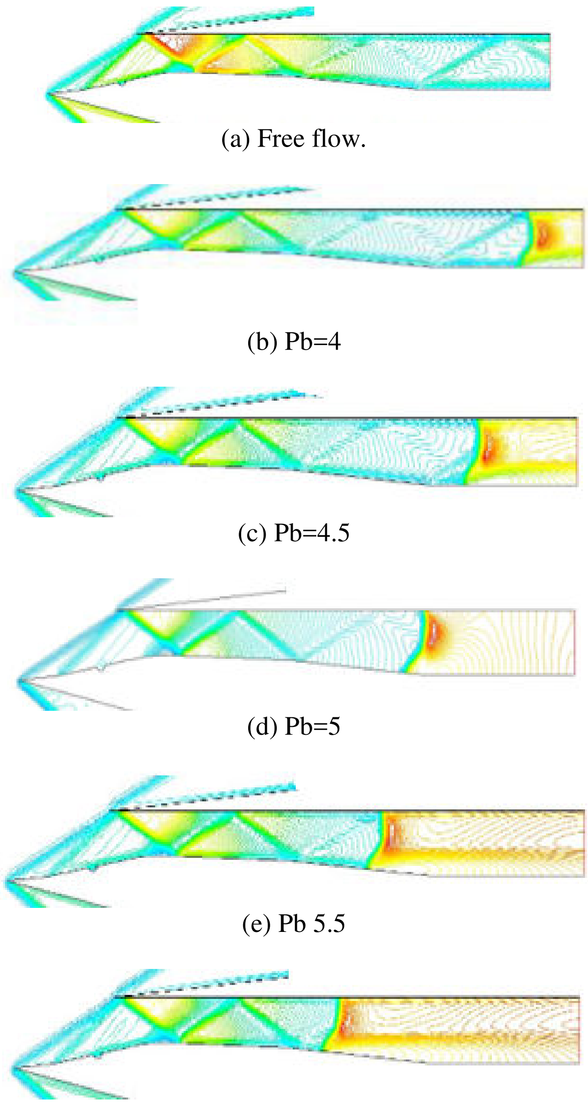


Figure-12. Ramp surface pressure distribution at various operating conditions with cavity model of radius 1mm at a location of X/L = 0.1468.



(a) Free flow.

(b) Pb=4

(c) Pb=4.5

(d) Pb=5

(e) Pb 5.5

(f) Pb=6

Figure-13. Density contour of the intake with cavity of radius 1mm at various operating conditions.

"Table-1", Shows the effect of the cavity on the Flow Distortion (FD) of the supersonic air intake. It is also an important parameter for the performance of the intake. But by using the cavity there is an increment in the flow distortion percentage, which will affect the performance as well. "Figure-14", shows the comparison of the total pressure distribution at the exit of intake with and without cavity at various operating conditions. Significant improvement in the total pressure with cavity model is observed for various back pressure. Hence the use of cavity can improved the performance of the supersonic air intake Flow distortion at the intake exit is given by

$$\text{Flow Distortion \%} = \frac{\left(\frac{P0e}{P0inf}\right)Max - \left(\frac{P0e}{P0inf}\right)min}{\left(\frac{P0e}{P0inf}\right)avg} X100$$

Table-1. Effect of Cavity on FD %.

Type of geometry	Back Pr ratio	F. D %
Clean model	4	49.96
	4.5	45.53
	5	24.44
Model with cavity (R= 1 mm)	4	70.82
	4.5	58.95
	5	43.6

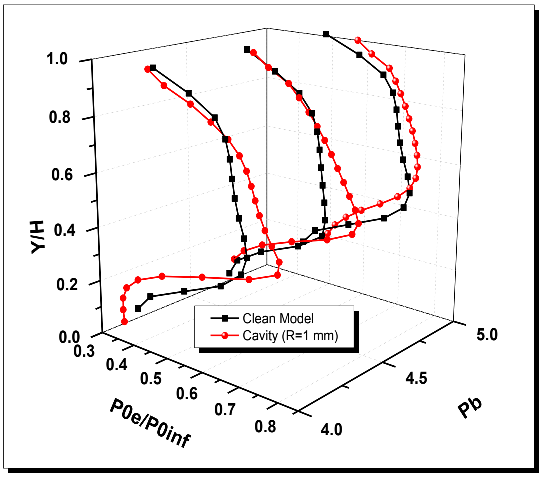


Figure-14. Comparison of total pressure distribution at the exit of the intake with and without cavity.

6. CONCLUSIONS

This study demonstrates the validation of the present computation (clean model) with the reference [4] computation as well as with experiment. It shows that intake will be in unstart conditions with zero degree cowl deflection of the intake. The main cause for the unstart condition is the normal shock wave boundary layer interaction (SWBLI) near the throat of the intake. To eliminate this SWBLI, a cavity of various radius at various location is tested in further computations. Results indicate the reduction in SWBLI near the throat and the normal shock disappears. Hence compression take place inside the intake with series of oblique shock appeared.

The effect of cavity radius and position on the performance of the supersonic air intake is also demonstrated in this study. On comparison of the clean model with cavity model, improvement in the TPR is found. All the tested cases (cavity with each and every radius and position) show the significant improvement in the flow physics inside the duct. Supersonic air intake with cavity of radius 1 mm at a location of X/L = 0.1468 shows the highest improvement in the TPR, however cavity of radius 0.5 mm at a location of X/L = 0.223 shows the minimum improvement. Increment in the total pressure at the exit of air intake with cavity of radius 1 mm at a location of X/L = 0.1468 at various operating conditions is also reported. However there is some increment in the FD is found. Hence by using the cavity of various radius at various location, overall improvement in the performance of the intake is achieved.

ACKNOWLEDGEMENT

Thanks to the SRM University for providing us their kind support during this research work. Thanks to the BIT, Mesra Ranchi for providing their labs to perform the test.

REFERENCES

[1] Neale M. C. and Lam, P. S. 1965. Tests with a Variable Ramp Intake Having Combined External/Internal Compression and a Design Mach number of 2.2. Aeronautical Research Council, C.P. No. 805.

[2] Kim S.D. 2009. Aerodynamic Design of a Supersonic Inlet with a Parametric Bump. Journal of Aircraft. 46(1): 198- 202.

[3] Saha S. and Chakraborty D. 2011. Hypersonic Intake Starting Characteristic-a CFD Validation Study. Symposium on Applied Aerodynamics and Design of Aerospace Vehicle (SAROD 2011) November 16- 18, Bangalore, India.

[4] Das S. and Prasad J. K. 2010. Unstart Suppression and Performance Analysis of Supersonic Air-intake Adopting Bleed and Cowl Bending. Journal of Institution of Engineers (India). 91: 27-35.

[5] Das S. and Prasad J. K. 2010. Starting Characteristics of a Rectangular Supersonic Air-Intake with Cowl Deflection. The Aeronautical Journal. 114: 177- 189.

[6] Vasana M. Don *et al.* 2013. Computational and experimental investigation of supersonic flow and their control. International Journal of Mathematical, Computational, Physical, Electrical and Computer Engineering. 7(1).

[7] R. K. Jaiman *et al.* 2004. Simulations of Normal SWBLI Control Using Mesoflap. Journal of Propulsion and Power. 20(2).

[8] Das S. and Prasad J. K. 2009. Cowl Deflection Angle in a Supersonic Air Intake. Defence Science Journal. 59(2): 99-105.

[9] Babinsky H. 2011. Shock Boundary Layer Interaction Flow Control with Micro Vortex Generators. Air Force Research Laboratory- Air Force Office of Scientific Research- UK- TR- 2011- 0014.

[10] Hirt S. M. and Anderson B. H. 2009. Experimental Investigation of the Application of Microramp Flow Control to an Oblique Shock Interaction. NASA/TM-2009- 215630, AIAA- 919.

[11] Lee S., Loth E., Georgiadis N. J. and De J. R. B. 2009. Effect of Mach number on Flow Past Micro-Ramps. 39th AIAA Fluid Dynamics Conference, San Antonio, Texas, AIAA 2009- 4181.

[12] Lee S., Goettke M. K., Loth E., Tinapple J. and Benek J. 2010. Microramps Upstream of an Oblique-Shock/Boundary-Layer Interaction. AIAA Journal. 48(1): 104-118.

[13] Allan B. G., Sheng Y. C. and Lin J. C. 2002. Numerical Simulations of Vortex Generator Vanes and Jets on a Flat Plate. AIAA 02-3160, 1st AIAA Flow Control Conference, St. Louis, MO.