International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 07 | July - 2022 Impact Factor: 7.185 ISSN: 2582-3930

Computational Study of Hypersonic Scramjet Experimental Vehicle Intake

Ashvini P Desai[1], Keziya J R[2], Melini D[3], Nihal Moni George[4], Premkumar A[5]

[1] Assistant Professor, Department of Aeronautical Engineering, KCG College of Technology, Chennai, India

[2,3,4,5] Student, Department of Aeronautical Engineering, KCG College of Technology, Chennai, India

***Abstract* - To design an efficient intake for a scramjet engine which can be operated in hypersonic conditions between Mach 6 and 7. A computational fluid dynamics analysis is performed on the intake design to analyze its pressure losses, shocks and the possibility of flow separation. The intake of the scramjet engine is designed for critical Mach number of 6.5.**

***Keywords* –**Scramjet inlet, Ramp angle, Ramp length, Cowl, Oblique shock, ANSYS, Hypermesh.

I. Introduction

Before knowing about Scramjet, let us first discuss about the Ramjet engines. A ramjet is an air-breathing jet engine which uses the vehicle's forward motion to compress incoming air for combustion without a rotating compressor. Fuel is injected within the combustion chamber where it mixes with the hot compressed air and ignites. Around Mach 3, the ramjet engines workefficiently at supersonic speed and can operate up to speeds of Mach 6. However, efficiency drops downwhen the vehicle reaches it hypersonic speed.

Scramjet is similar to a ramjet engine within which the airflow through the engine remains supersonic, or greater than the speed of sound. Scramjet powered vehicles are used to operate at speeds up to Mach 15theoretically. If a scramjet combustor is supersonic, the inlet decelerates the air to a lower Mach number and increases the pressure for combustion, after which it is accelerated to a higher Mach number through the nozzle.

Current challenges in Scramjet

The current challenges in the scramjet engine take place in air induction, combustor, nozzle and structures and materials. Problems in these areas vary from the inlet to igniting the fuel during the supersonic flow. Current scramjet designs target the start of supersonic combustion to be between Mach 5 and 6.

II. Scramjet Inlet

Compression is performed internally by the shock waves to the engine. This inlet can be shorter than the mixed compression inlet. It maintains full capture at Mach numbers lower than the design point, but it requires a variable geometry to start.

Scramjet inlets are classified on the basis of compression used, there are three types.

1) *Internal compression*: In internal compressiontwo ramps will be directed inwards and oblique shock waves are created in these ramps and the flow is compressed internally. Normal shock is formed at the end of the ramps.
2) *External compression*: In external compression ramps and cowl system is used to compress the incoming air. The oblique shock formed due to the ramps impinge on the cowl lip and the flow is compressed. Normal shock if formed in the cowl.
3) *Mixed compression*: This system is also similar to external compression but the only difference is shock train is formed in the isolator and the normal shock is formed inside the isolator.

Starting of an inlet

Internal contraction ratio is defined as the ratio of flow area at the plane of cowl to the throat area. Whereas, Contraction ratio is defined as the ratio of flow area at the tip of inlet to the throat area. If theinverted contraction ratio is lower, then the value is calculated from the formula,

$$\frac{A_2}{A_0} = 0.05 - \frac{0.52}{M_0} + \frac{3.65}{{M_0}^2}$$

Software used:

The software's used in this project are HYPERMESH and ANSYS. HyperMesh is used in aviation and aerospace industry for the analysis of local structures and identification and removal of redundant material. It helps in meshing and in model construction. Whereas Ansys is a finite element modeling package for solving numerical problems. There are methods used in Ansys. One is the Graphical User Interface (GUI) and the other is to use command files. Simulations are performed using Workbench system.

Problem Statement

Generally, for any air breathing engine, the greater the pressure and concentration of air at the combustion chamber, the more combustibility takes place. For a Supersonic Combustion Ramjet engine, the incoming air velocity to the combustion chamber has to be low as possible but still in supersonic speeds. And hence, the aim of this project is the design a scramjet intake so that ram compression allows the air to be reduced to below Mach 2.5.

Design Methodology:

The design of the scramjet engine mainly depends on the Oblique shock waves. Scramjet flies at extremely high speeds and it uses the oblique shock waves to reduce the incoming velocity of air and thereby increasing the pressure of the air going to the combustion chamber. We have used the oblique shock wave relations to calculate the ramp angle and ramp lengths.

III. Design Procedure

Analytical calculation

This calculation is performed by oblique shock relation to find the appropriate design for analysis. By having the Mach number as a constant value and varying it for different shock angle, we get a static pressure ratio, density ratio, temperature and total pressure ratio. These values are used to design a model.

In the below tabular column, there are different values of oblique shock relation and a graph is plotted to find the ramp angles. For an ideal scramjet inlet, the oblique shock wave created on the ramps should impinge on the cowl tip. By considering this ideal condition, the length if the ramp is calculated by fixing cowl tip at certain height.

MODEL	RAMP ANGLES	RAMP LENGTH
1	6.5, 10.8, 15.1	1458.161, 588.231, 240.3
2	7.5, 9.6, 12.6	1342.489, 528.46, 274.5
3	9.5, 9.6 ,12.6	1227.424, 430.267, 236.1
4	11.1, 11.8, 15.1	1201.641, 359.903,160.4
5	11.1, 11.8, 9.3	1201.641, 387.542, 130.9
6	11.1, 12.8, 18.5	1227.009, 366.715, 126.3
7	10.5, 12.5, 8.5	1255.396, 316.217, 214.7

2D Design

All the ramp angles are calculated and 2D design is created in FUSION 360. Metric system is used to design the model and after creating the model it is exported as IGES file which is used in HYPERWORKS for meshing the model. The IGES file is used for creating the mesh. After meshing, the model is again exported to ANSYS for further analysis.

Meshing

Meshing for the ramp design is done using the mesh option in the HYPERMESH software. Biasing option is used to make the mesh more efficient such that the maximum number of elements is concentrated for a particular region, where more accuracy is needed and a smaller number of elements on the region is free stream. By using this biasing method computational time is reduced and more accurate results are obtained, since there is maximum limit for the number of nodes. For each design the nodes and element size vary.

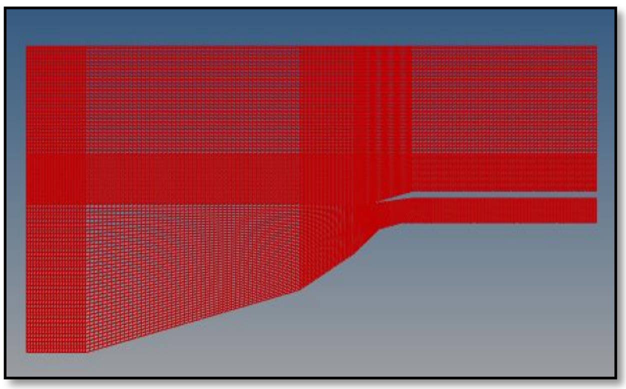


Fig: Mesh

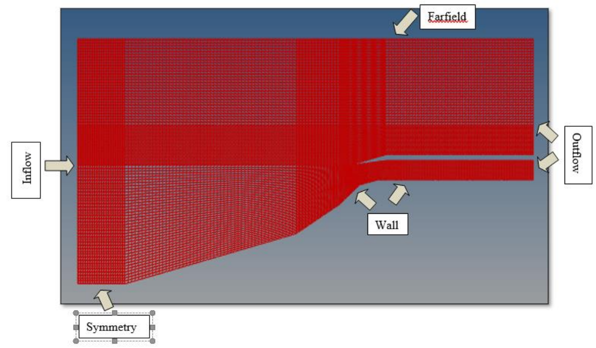
Analysis of Three Ramp Design

Setting Solver

• In Problem Setup, select General by double clicking. The only option we need to change here is the type of solver. In the Solver window, select Pressure-Based Solver.

• We will need to utilize the energy equation in order to solve this simulation. Change the type of viscosity model. Select K-ω SST model.

• In the Boundary Conditions window, select pressure far field for inlet and far field boundary. Similarly select pressure outlet for the domain and isolator outlet region.



Initial condition

• Select Solution methods, Under Spatial Discretization, ensure that the option for Flow is specified as Second Order Upwind.

• Select Solution, Initialization and select Standard Initialization. Then under Compute from, select inlet, then select Initialize.

Number of iterations

• Select Run Calculation. Change the Number of Iterations to 3000.

• Double-click Calculate to run the calculation.

Results and Discussion

Model 1:

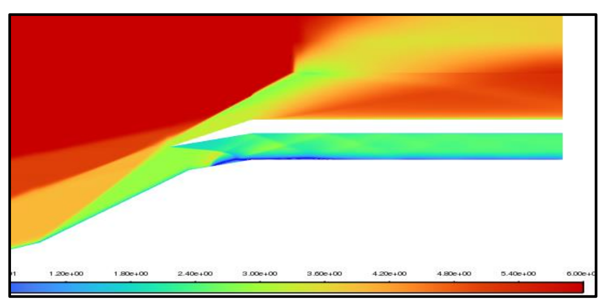


Fig : Mach contour of Model 1

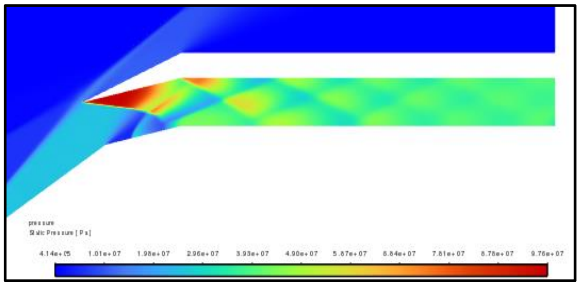


Fig : Pressure contour of Model 1

INFERENCE:

In Theoretical calculation we can see at the cowl inlet the Mach no. was found to be 3.05. In computation we found that this model has reduced to Mach 2.4 at the isolator exit, but it has high spillage drag. Also, we can see a shock train formation.

MODEL 2:

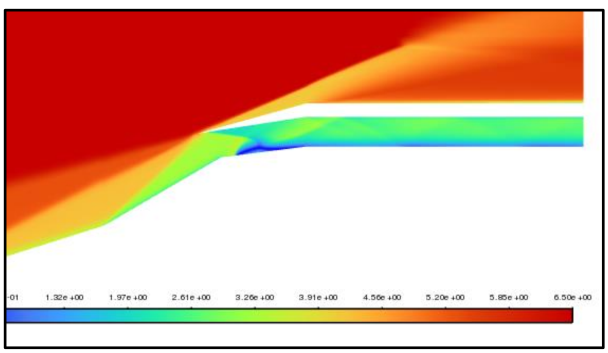


Fig :Mach contour of Model 2

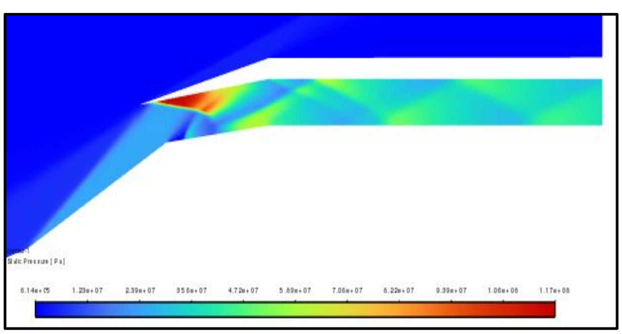


Fig : Pressure contour of Model 2

INFERENCE:

In theoretical analysis, we found that the Mach no at the end of third ramp was 3.27 and in computational analysis The Mach number is reduced to 2.6 at the isolator exit region and has not so high spillage drag and observing from the pressure contour we can see the formation of a shock train which indicates better compression strength.

MODEL 3:

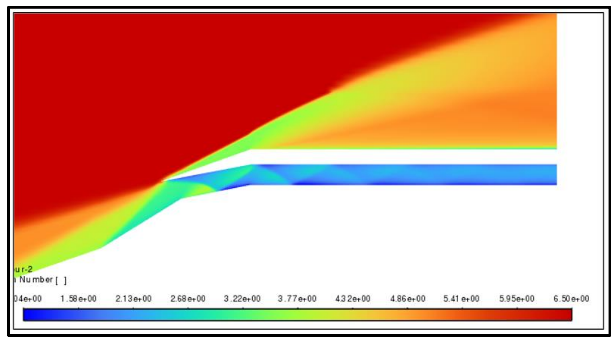


Fig : Mach contour of Model 3

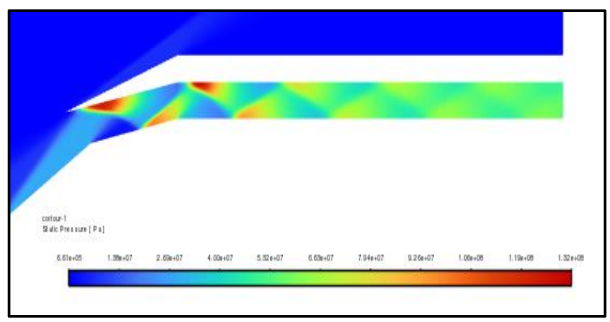


Fig : Pressure contour of Model 3

INFERENCE:

In the theoretical calculation of this model the Mach no. at the cowl inlet was found to be 3.13. In computational simulation we found that the Mach no. was decreased to around 2.24 in the isolator region. But we can observe that there is high spillage drag. The shock train formation looks healthy indicating a good compression strength.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 07 | July - 2022 Impact Factor: 7.185 ISSN: 2582-3930

MODEL 4:

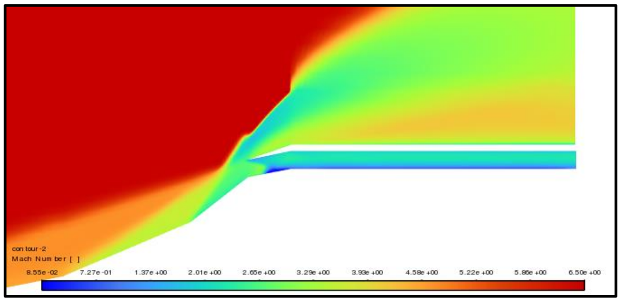


Fig : Mach contour of Model 4

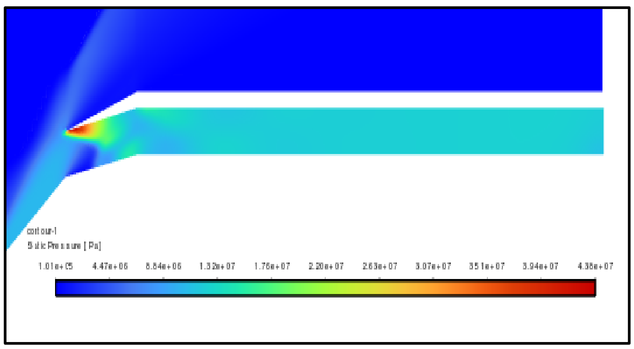


Fig : Pressure contour of Model 4

INFERENCE:

In theoretical calculation, this model showed a Mach no. of 2.718 at the end of the third ramp and during computational analysis it reduced to a Mach No. of 2.17 at the end of the isolator region. But we can see a very high spillage drag and also from the pressure contour there is a faint formation of shock train thus indicating less compression strength. From this inference this model seems to be less efficient.

MODEL 5:

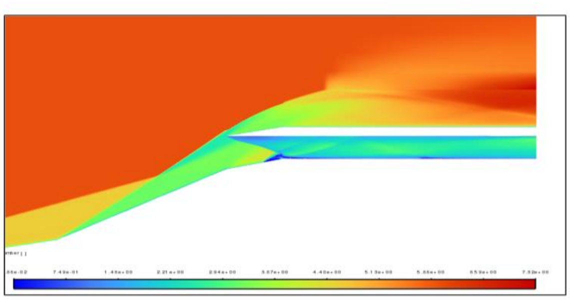


Fig : Mach contour of Model 5

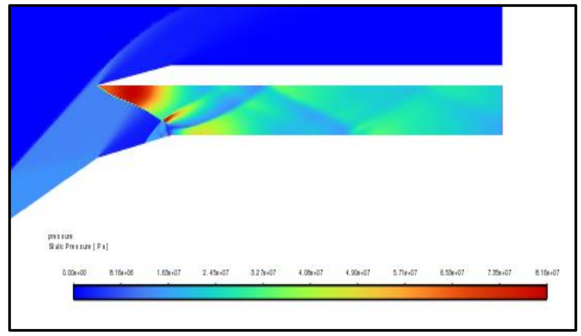


Fig : Pressure contour of Model 5

INFERENCE:

In the theoretical analysis, the Mach no. at the end of the third ramp was found to be around the range of 2.45. In computational analysis, the Mach No. has been reduced to the range of 1.89. But from the Mach contour we can see that there is considerable amount of spillage drag and also there is a formation of shock train.

MODEL 6:

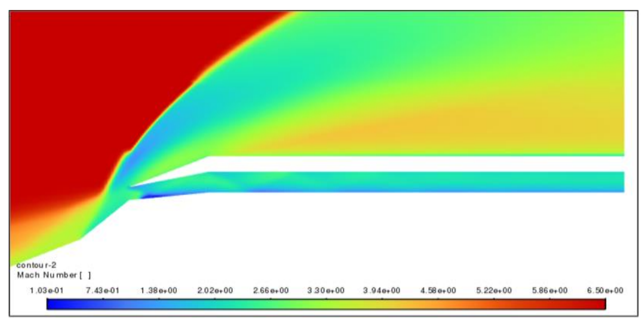


Fig: Mach contour of Model 6

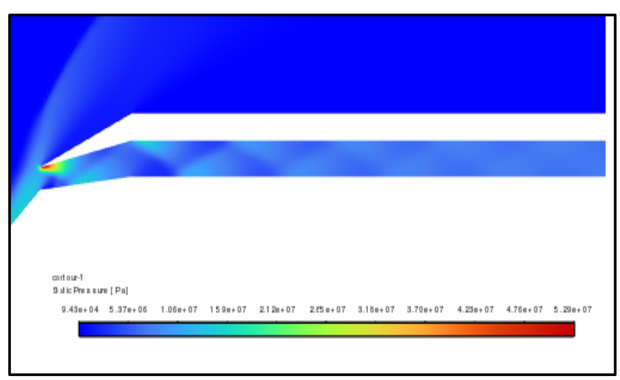


Fig : Pressure contour of Model 6

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 07 | July - 2022 Impact Factor: 7.185 ISSN: 2582-3930

INFERENCE:

Here, the Mach No. was found to be 2.44 at the end of the third ramp in theoretical calculation. In computational analysis, we found out that the Mach no. has been reduced to the range of 2.02. Also this model has a high spillage drag and the shock train formation looks weak. Hence, this model is not so efficient.

MODEL 7:

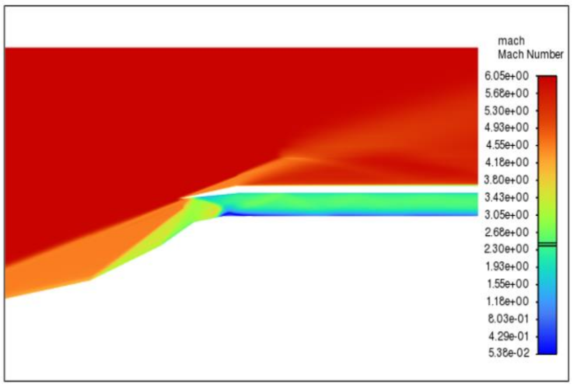


Fig : Mach contour of Model 7

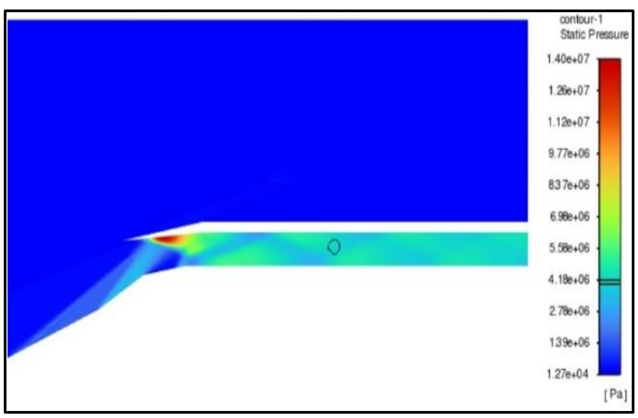


Fig : Pressure contour of Model 7

INFERENCE:

In theoretical calculation of this model, we found out that the Mach No. at the inlet cowl was around the range of 3.13. In computational analysis, the Mach no. reduced to a range of 2.4 at the isolator exit region. From the Mach contour we can also observe that there is a very less spillage drag and also there is a sufficient level of shock train formation which indicates a decent level of compression strength. Thus, this model seems to be more efficient comparatively to the other models simulated.

FINALIZED MODEL

Thus, based on the above analysis, we had understood that model 7 provided better efficiency than the rest of the models.

A point worth noting is that the spillage drag produced by this model is comparatively very small. This gives a drastic advantage as it ensures less fuel consumption due to low resistance caused by drag.

If we look at the pressure contour, we can see that the shock trail produced is less compared to many other models. Nevertheless, it produces a sufficient shock trail which means that good ram compression takes place at the inlet.

Mach No. Distribution along the entire model

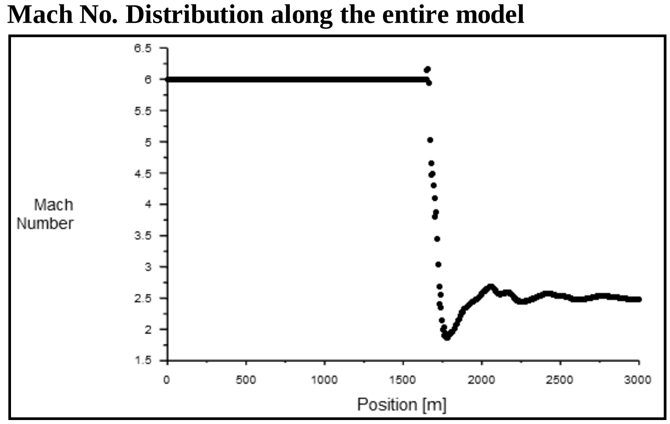


Fig: Mach No. Distribution along the entire model

Mach No. Distribution along the ramps:

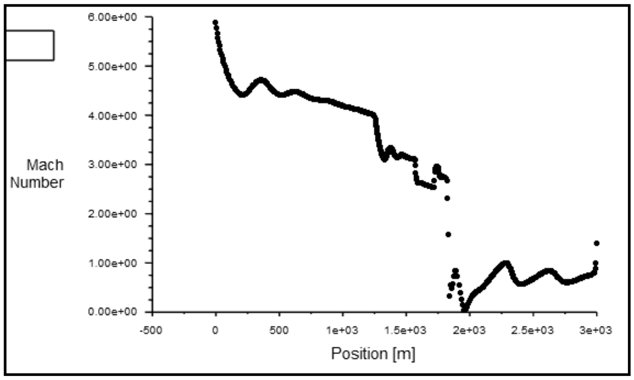


Fig: Mach No. Distribution along ramps

Static Pressure Distribution through the entire model:

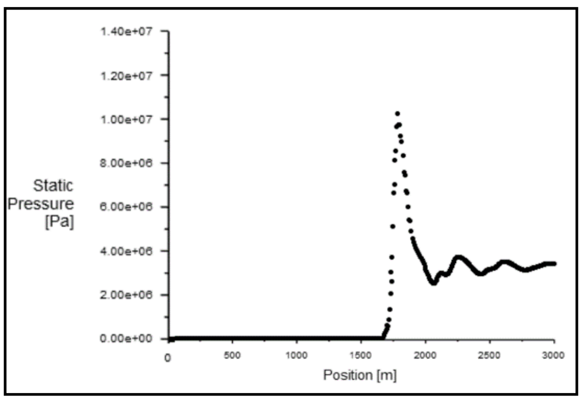


Fig: Static Pressure Distribution along the entire model

Static Pressure Distribution through the ramps:

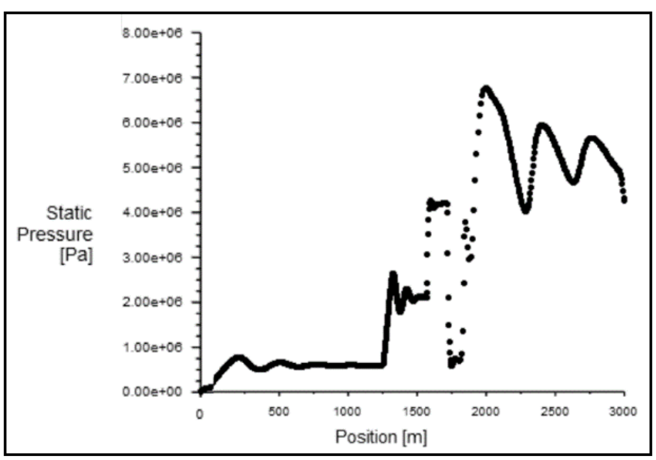


Fig: Static Pressure Distribution along ramps

Static Pressure Distribution through the cowl:

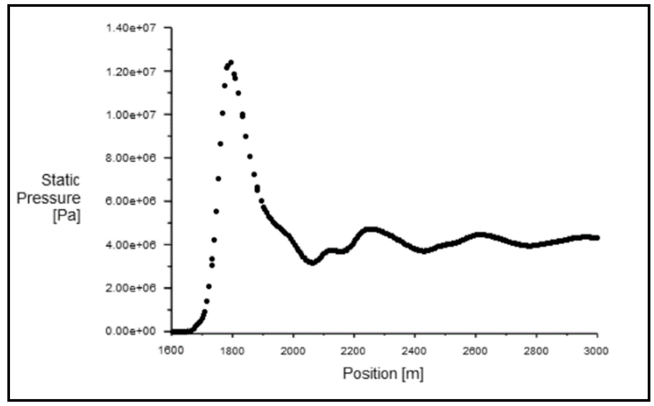


Fig : Static Pressure Distribution along cowl

The above graphs give us the Mach and Static Pressure distribution through the entire model, ramps and cowl region respectively.

CONCLUSION

Based on the analysis and graph result it is seen that Model No. 7 is performing better than the other simulated designs. In theoretical calculation of this model, we found out that the Mach No. at the inlet cowl was around the range of 3.13. In computational analysis, the Mach no. reduced to a range of 2.4 at the isolator exit region. From the Mach contour we can also observe that there is a very less spillage drag and also there is a sufficient level of shock train formation which indicates a decent level of compression strength. Thus, this model seems to be more efficient comparatively to the other models simulated. The results also indicate the advantages of using three-ramp design as it produces optimum results. Further studies can be done on scramjet intakes of even higher design Mach numbers and also by varying the ramp lengths, ICR and turning angles. Also, we're able to conclude that K- ω turbulence model specifically simulates the flow field characteristics in supersonic and hypersonic conditions, in capturing shocks at inlet and shock trains within the isolator.

FUTURE WORK

Experimental data can be used for analyzing the design in order to compare the selected design with the data obtained. Further work can be done to find how boundary layer formation in the isolator affects the total pressure ratio and strength of the shock inside the isolator.

REFERENCES:

➢ Investigation of local unstart in a hypersonic scramjet intake at a mach number of 6- Manoj K Devaraj, Prahallada jutur, Srisha M V Rao, Gopalan Jagadeesh, Ganesh T K Anavardham

➢ Mechanism study of local unstart of hypersonic inlet at high mach numberxiaoliang jiao, juntao chang, zhongqi wang, and daren yu

➢ Numerical study on the start and unstart phenomenain a scramjet inletisolator model - Jaewon lee, sang hun kangi

➢ Experimental investigation of unstart dynamics driven by subsonic spillage in a hypersonic scramjet intake at mach 6 - manoj kumar k. devaraj, prahallada jutur, srisha m. v. rao, gopalan jagadeesh, and ganesh t. k. anavardham.

➢ A Scramjet Compression System for Hypersonic Air Transportation Vehicle Combined Cycle Engines ➢ Design And Analysis Of Scramjet Inlet Atulya Sethi Amity Institute of Aerospace Engineering, Amity University, Noida,India

➢ Details of turbulence modeling in numerical simulations of scramjet intake - T. nguyen, g. schieffer, c. fischer, h. olivier, m. behr and b. reinartz

➢ A Scramjet compression system for hypersonic air transportation vehicle combined cycle engines - devendra sen , apostolos pesyridis and andrew lenton

➢ On the unsteady throttling dynamics and scaling analysis in a typical hypersonic inlet-isolator flow - K.Raja Sekar, Sengunthapuram Kandasamy Karthick, S. Jegadheeswaran, Kannan Ramaraj