International Journal of Emerging Technologies in Engineering Research (IJETER)
Volume 1, Issue 2, July (2015) www.ijeter.everscience.org

Intensification of Trust by Analysis of flow Through Supersonic Nozzle using CFD Simulation

Anjappa S.B
Global Academy Of Technology Bangalore, India.

Dr. M S Rajagopal
Professor & HOD Mechanical Engineering Department GAT Bangalore, India.

H V Harish
Asst professor Aeronautical Engineering Department NMIT Bangalore, India.

Abstract – The optimization design of a convergent divergent nozzle has been conducted to understand the phenomena of supersonic flow through it at various divergent angles and different throat to that of inlet diameter ratio .Here k-ε model has been used for flow analysis. The boundary conditions were specified according as for as the available experimental information. Mach number is found have an increasing trend with increase in divergent angle there by obtaining an optimal divergent angle. Here throat diameter to that of inlet diameter will be varied and analysis will be carried out. Based upon the Mach number we are optimizing the geometry. Further analysis has been carried at different altitude. ICEM CFD has been used for modeling and meshing and CFX has been used for solving.

Index Terms – Supersonic Nozzle, Machnumber, CFD.

1. INTRODUCTION

A Nozzle to be a device designed as for as restraint the characteristics of a fluid flow (especially as for as increase velocity) as it exits an enclosed cavity via an orifice.

A nozzle is frequently a pipe or tube from make cross sectional area and it can occur used as for as direct or modify the flow from a fluid (liquid or gas). They are frequently used as for as restraint (control) the rate from flow, speed, direction, mass, shape, and or the pressure from the stream that emerges from them.

The aim of a nozzle is to increase the kinetic energy of the flowing medium towards the expense from its pressure and internal energy. Nozzles can occur delineate while convergent (narrowing down from a wide diameter to a smaller diameter in the direction from the flow) or divergent (expanding from a smaller diameter as for as a larger one). A de Laval nozzle is a convergent section followed from a divergent section and to be often called a convergent-divergent nozzle.

2. METHODOLOGY

Variable nozzle diameter concept used to calculate geometry of CD nozzle. Here D^*/D_0 (0.7, 0.6, 0.5, 0.4, 0.3) and for different divergence angle of 4,7,10 and 13 degree carried.

From the above analysis whichever gives highest Mach number has been taken for analysis at different altitude (feet)

Important point to be noted that the area ratio variation has been in such a way it must satisfy choked condition (Mach number 1 at throat)

A computational fluid dynamics software package was used to simulate fluid and thermal flows through nozzles. ANSYSICEM-CFD was used for mesh generation and ANSYS CFX for flow analysis. The analysis was made turbulence models K-ε. The results have been obtained through CFD-POST.

2.1. Boundary Condition

Inlet Location

- Total pressure=44 bar.
- Total temperature=3400k
- Medium intensity=5%
- Wall=smooth

Nozzle dimensions (reference paper)

Inlet diameter (D) =1m

Exit diameter (D_0) =0.861m

Mass flow rate=826 kg/sec

Convergent length=0.64m

D*=throat diameter

2.2. GEOMETRY MODEL-4 (4^0 k-ε) D^*/D=0.3

From figure 1 it is clear that Mach number at throat is 1 which satisfies maximum discharge and based on outlet Mach number thrust is calculated. From above figure it is clear that in convergent part Mach number is subsonic (M<1), at throat it is sonic (M=1) and at divergent it is supersonic (M>1).

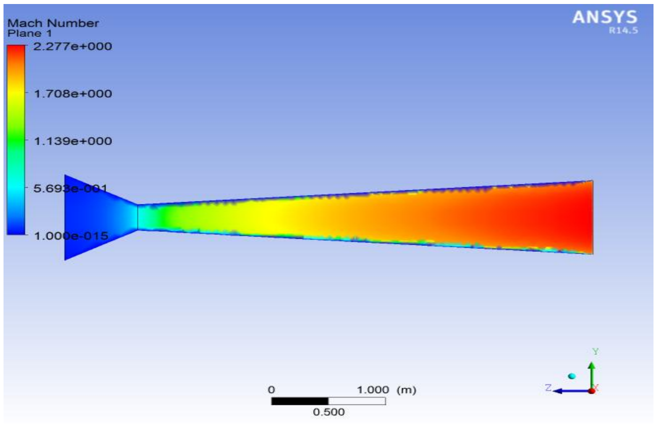

Figure 1 Mach number Plane Geometry

2.3. GEOMETRY MODEL (10^0k-ε) D*/D=0.3

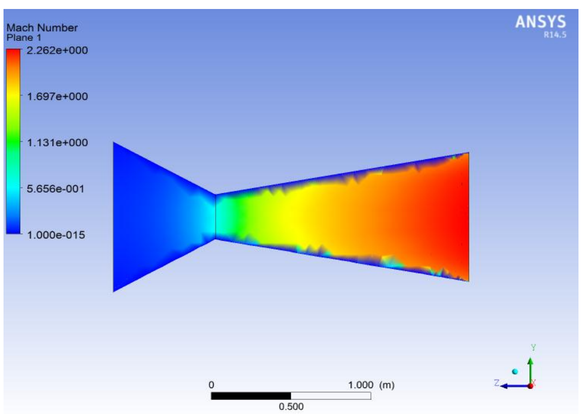

Figure 2 Mach number geometry

3. CFD RESULTS FOR VARIOUS ALTITUDES

After optimizing 10 degree d*/d_o=0.3 from above analysis it has been chosen for analysis at different altitude.

For 10000ft:

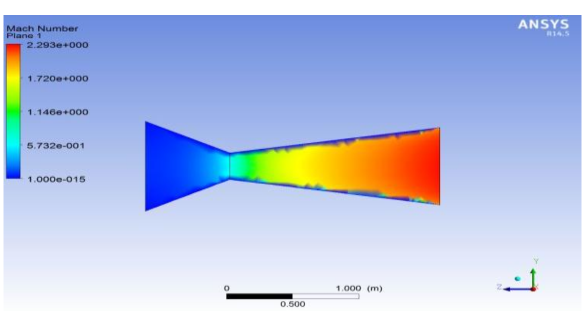

Figure 3 Mach number

For 20000ft:

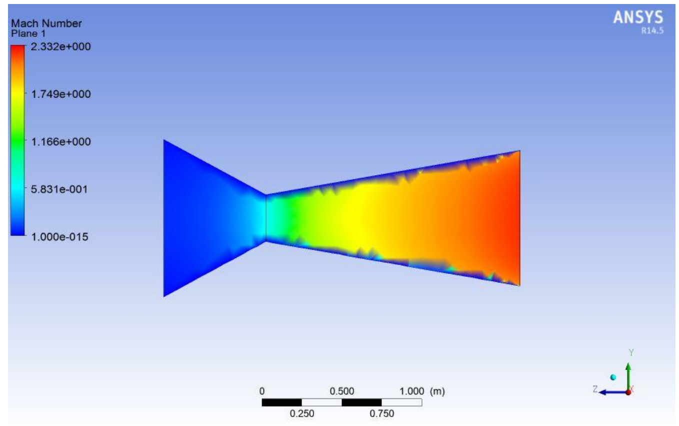

Figure 4 Match Number

For 40000ft:

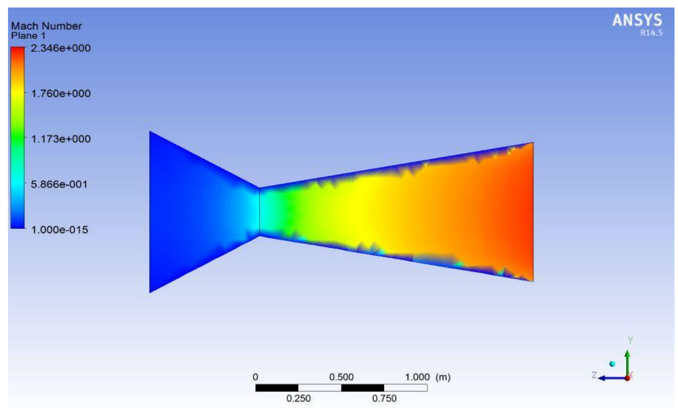

Figure 5 Mach number for 40000ft

For 50000ft, 55000ft, 60000ft, 65000ft:

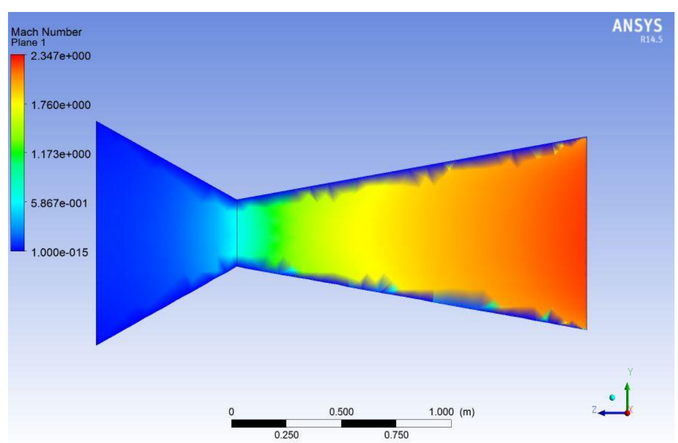

Figure 6 Match Number For 50000ft, 55000ft, 60000ft, 65000ft:

3.1. Tabulation for Various Altitudes:

ALTITUDE (ft.)	MACH No	VELOCITY (m/s)
10000	2.293	2681
20000	2.332	2726
25000	2.340	2735
30000	2.344	2739
35000	2.345	2741
40000	2.346	2742
50000	2.347	2743
55000	2.347	2743
60000	2.347	2743
65000	2.347	2743

3.2. GRAPH

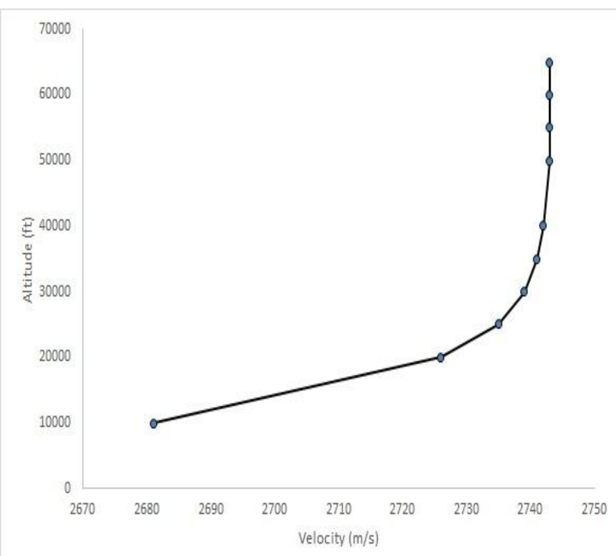

Figure 7 Altitude vs Velocity

4. CONCLUSIONS

The steps followed to find the solutions were as follows:

1. Generation of 14 nozzles shapes using an ICEM-CFD.
2. Generation of the computational mesh using a grid generator software package ANSYS ICEM-CFD.
3. Simulations of the flow-fields and thermal analysis were run in the CFD Software package ANSYS CFX for the 14 shapes, using k-ε turbulence models with a total of 14 simulations.
4. The standard deviations of the pressure, Mach number and velocity results at the exit of the nozzles were calculated.
5. After the above process, thrust has been calculated.
6. It has been observed that Mach number and velocity at the outlet is more for divergence angle 10^0-ε model with D^*/D=0.3
7. As the divergence angle is increased with increase in the ratio of D^*/D, the divergent length will decrease and Mach number at exit will be decreased.
8. From the simulation it has been observed that divergence angle 4° with D^*/D=0.7, divergence angle 7° with D^*/D=0.7, divergence angle 10° with D^*/D=0.6 & 0.7, and divergence angle 13° with D^*/D=0.6 & 0.7, the divergent length is less than convergent length so that expansion process is terminated due to back flow. Hence analysis is not carried out.
9. Finally based on higher Mach number and velocity at the outlet, the divergence angle 10^0-ε with D^*/D=0.3 model has been optimized.
10. Further analysis has been carried out at higher altitude. Where we found that saturation level has been reached where Mach number and velocity was constant between 55000ft to 65000ft.

REFERENCES

[1] K.M.Pandeyworked on the topic of "Wall Static Pressure Variation in Sudden Expansion in Flow through De-Laval Nozzles at Mach 1.74 and 2.23: A Fuzzy Logic Approach"

[2] K.M.Pandeyworked on the topic of "Wall Static Pressure Variation in Sudden Expansion in Cylindrical Ductswith Supersonic Flow: A Fuzzy Logic Approach"

[3] K.M Pandey, Singh, A.P.; CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software, International Journal of Chemical Engineering and Applications.

[4] K.M.Pandey worked on the topic of "Studies onPressure Loss in Sudden Expansion in Flow through Nozzles: A Fuzzy Logic Approach" (IJERA), Vol. 2, Issue 5, September- October 2012.

[5] K. M. Pandey and E.Rathakrishnan worked on thetopic of "Influence of Cavities on Flow Development in Sudden Expansion "Vol 5, No 2, 2010.