Thrust Augmentation by Analysis of Flow in Supersonic Nozzle using CFD

Girisha K L[#1], Chandru B T[#2], Dr. Matrthi B H[#3],Ravindra A R[#4]
Mechanical Engineering Department, EWIT, Bengaluru-91
KARNATAKA, INDIA
#1 P G STUDENT,girish.kl0308@gmail.com, #2 ASSOCIATE PROFESSOR,#3 PROFESSOR & HOD.
#4P G STUDENT,ravindra.0905@gmail.com

***Abstract*— Rocket propulsion depends on thrust, thrust interns depends on newton's III law. In order to obtain thrust power in rocket, convergent divergent nozzle has been used. In order to obtain maximum velocities in convergent –divergent nozzle expansion takes place where Mach No. in the convergent nozzle has been subsonic and sonic and in divergent part it has been supersonic.**

In order to obtain maximum thrust geometry parameters(D/Do*) will be varied and analysis has been carried out. After optimizing, the required geometry further analysis has been carried out at different altitude. It has been shown that, a saturation pressure has been reached at a particular altitude, which is the main aim of this project.

Again optimize geometry has been taken for analysis, where throat radius has been varied and it has shown that considerable amount of thrust has been increased.

Modelling and Meshing has been done through ICME CFD, analysis carried out through CFX and result obtained through CFD POST.

***Keywords*— Thrust, Altitude, Pressure, supersonic, sonic, throat.**

I. 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.

The nozzle aim 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 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 C-D nozzle.

C nozzles speed subsonic fluids. if the nozzle pressure ratio is more sufficient the flow will extend sonic velocity towards the narrowest point (i.e. the nozzle throat). In this condition, the nozzle is said to occur choked. Convergent divergent nozzles can therefore accelerate fluids that possess choked in the convergent section as for as supersonic speeds. This C-D process is greater efficient than allowing a convergent nozzle to expand supersonically out ward.

The divergent section also certain that the direction from the escaping gases is backwards and sideways trying not contribute to thrust.

Propulsion system is a system fundamental obeys Newton's laws that force is proportional as for as rate of change of momentum, and that action and equal. Thrust chamber is one example system that works using these laws besides some time ago systems that is turbojet and ramjet. cavity is by expelling stored subject, called the propellant. This thrust can range from mega-Newton to mille-Newton. Thrust chamber system can be main spacecraft propulsion that is missile launcher, assist-take-off engines for airplanes and even ejection of crew escape capsules.

OBJECTIVES

- The objectives is to increase thrust
- Analysis of flow through C-D Nozzle for different altitudes
- Obtained maximum uniformly of flow by achieving linear thrust
- To achieve linear velocity which will lead to maximum thrust
- To prove CFD is an effective tool for analysis

GOVERNING EQUATIONS

The equations are based on the conservation of mass, momentum and energy. The conservation equations are related to the rate of change in the amount of that property within an arbitrary control volume to the rate of transport across the control volume surface and the rate of the production within that volume.

The examples for Nervier-Stokes equations for compressible flows of governing equations. The following examples include heat source (q*) and body forces (b).

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{V}\right) = 0$$

Linear Momentum Conservation:

$$\frac{\partial(\rho\vec{V})}{\partial t} + \nabla\cdot\left(\rho\vec{V}\vec{V} - \underline{\underline{T}}\right) = \rho\vec{b}$$

Energy Conservation in total energy form:

$$\frac{\partial(e_o\rho)}{\partial t} + \nabla\cdot(e_o\rho\vec{V} - \underline{\underline{T}}\cdot\vec{V} + \dot{\vec{q}}_c + \dot{\vec{q}}_r) = \rho(\vec{V}\cdot\vec{b} + \dot{\vec{q}})$$

Where: ρ is the fluid density, vector V is the flow velocity, T is the stress tensor, b is the body forces,e_0 is the internal energy, q_c is the conduction heat transfer flux and q_r is the radiation heat transfer flux and q is the heat source.

BOUNDARY CONDITION

Inlet Location
- Total pressure=44 bar.
- Total temperature=3400k
- Medium intensity=5%

Outlet Location
- Boundary details
- Relative pressure= Atmospheric pressure.

Wall
- Option: no slip wall

Smooth wall

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

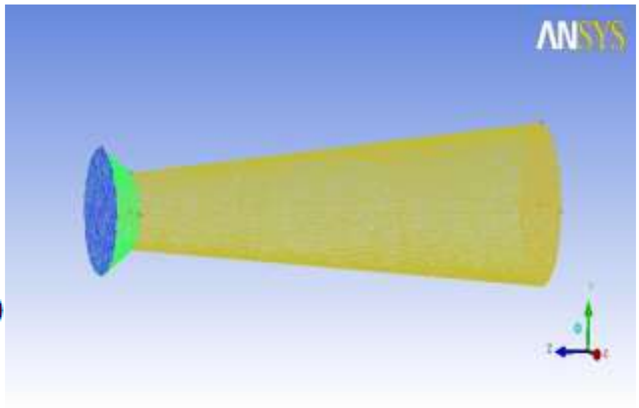


Fig. Meshing geometry

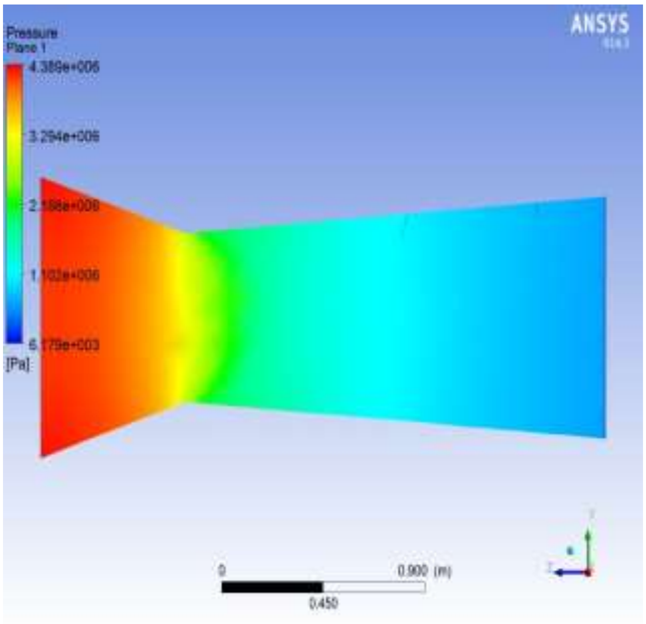


Fig. Pressure Plane Geometry

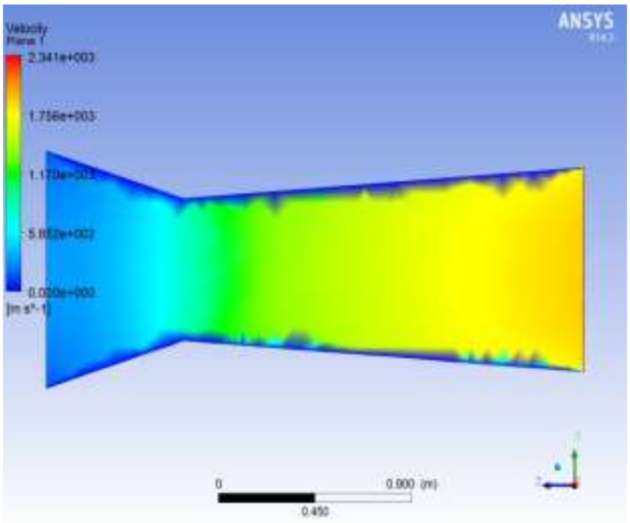


Fig. Velocity Plane Geometry

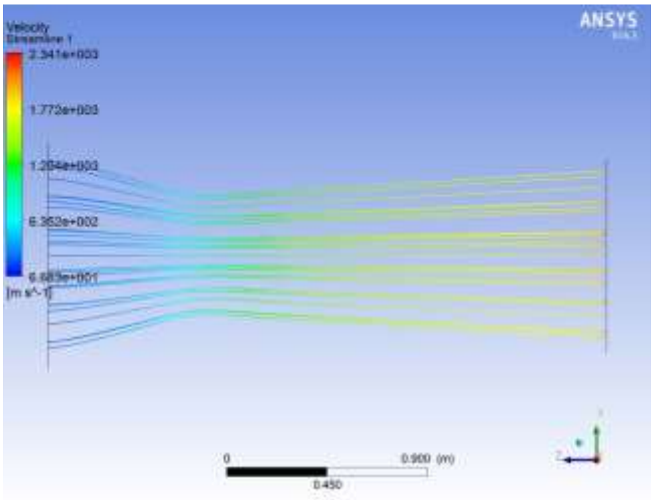


Fig. Velocity Streamline Geometry

Divergence angle	D*/D$_0$	VELOCITY AT OUTLET(m/s)	MACH NO	THRUST (KN)
4	0.3	2600	2.25	2147.60
	0.4	2430	2.1	2007.18
	0.5	2000	1.9	1652.00
	0.6	1900	1.65	1569.40
7	0.3	2550	2.24	2106.30
	0.4	2500	2.1	2065.00
	0.5	2100	1.8	1734.60
	0.6	1800	1.7	1486.80
10	**0.3**	**2650**	**2.26**	**2188.90**
	0.4	2350	2	1941.1
	0.5	2050	1.75	1693.30
13	0.3	2500	2.2	2065.00
	0.4	2300	2	1899.80
	0.5	1900	1.7	1569.40

Thrust= $\dot{m} \times V$
=826kg/s × 2600 m/s
=2147.6 N

Theoretical calculation of Mach number

$$M = \frac{V}{C} = \frac{V}{\sqrt{\gamma RT}},$$

$$C=\sqrt{\gamma RT} = \sqrt{0.4 \times 0.2817 \times T \times 1000}$$

$P^*=P_0(\frac{2}{\gamma+1})^{(\gamma/\gamma-1)}=44(\frac{2}{1.4+1})^{(1.4/1.4-1)}$

$P^*=41.27bar$,

$T^*/T_0=(\frac{p^*}{po})^{(\gamma-1/\gamma)}T^*=4300(\frac{41.27}{44})^{(1.4-1/1.4)}$

$T^*=3338.34K$,

$$C=\sqrt{1.4 \times 0.287 \times 3338.34 \times 1000}$$

$C=1158.617$ m/s,

M $=\frac{V}{C}$V from fig, 5.1(g) Mach number plane geometry model-1

$M=\frac{V\ from fig}{C\ from calculation}$

M=1151/1158.164 =0.9938 $\cong$1

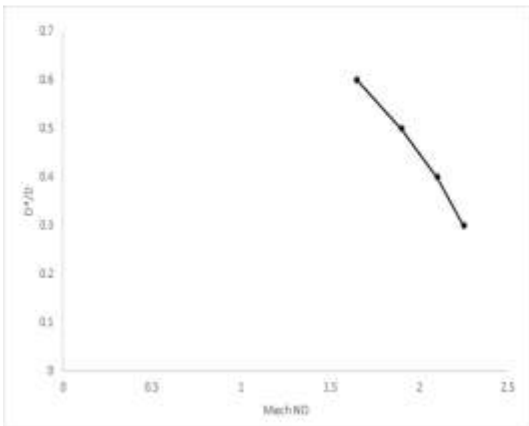
M=1

Mach number from fig, is 1 clearly above calculation is validated by CFD

Comparison of Mach number for theoretical and simulation

	Theoretical	Simulation	Difference (%)
Mach number	0.9938	1	0.0062

Graph:



D*/Do v/s Mach number

Tabulation for mach number for different 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**

CONCLUSIONS

Initially, simulations were run. The steps followed to find the solutions were as follows:

1. Generation of nozzles shapes using an ICEM-CFD.
2. Generation of the computational mesh using a grid software package ANSYS ICEM-CFD.
3. Simulations of the flow-fields and thermal analysis using ICEM CFD
4. After the above process, thrust has been calculated.
5. It has been observed that Mach number and velocity at the outlet is more for divergence angle 10^0-ε model with $D^*/D=0.3$
6. 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.
7. 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.
8. 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.
9. 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.
10. At throat radius has been provided in order to achieve maximum discharge so that exact sonic condition has to be achieved in order to obtain maximum velocity and hence maximum thrust.
11. Finally based on higher Mach number and velocity at the outlet, the divergence angle 10^0-ε with $D^*/D=0.3$ and throat radius of 0.6 model has been optimized.

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.

