OPTIMUM DESIGN OF ROCKET NOZZLE FOR ENHANCEMENT OF THRUST USING CFD

Allama prabhu S [1], Dr. K S Shashishekar [2] & H V Harish [3]
[1] Final year student Thermal power Engineering-M-Tech,SIT, Tumkur, India.
[2] Professor & Academic Dean Department of Mechanical Engineering, SIT, Tumkur. India.
[3]Assistant Professor, Department of Aeronautical Engineering, NMIT, Bangalore, India.
E-mail: prabhuhosahalli@yahoo.com

ABSTRACT: ***Analysis of flow through C-D nozzle has been carried out , ideal gas equation used for designing the nozzle, based on that design flow analysis carried out. The inlet boundary conditions were specified according to the available experimental information from journal. Numerical study has been conducted using two-dimensional axis-symmetric model at different divergence angle .Governing equations were solved using the finite-volume method in ANSYS CFX-13 software. The nozzle geometry has been calculated theoretically by using boundary conditions parameters. The present study is aimed at investigating supersonic flow at different divergence angle, and different area ratio keeping throat dia intact, since the variations in the parameters like the Mach number, pressure, are being analyzed and nozzle design will be optimized. Further for different altitude analysis has been carried out (5000, 10000, 20000, 25000 & 50000 feet). A Computational Fluid Dynamics (CFD) software package, ANSYSCFX-13, was used to simulate the compressible, viscous gas flow-field in seven nozzle shapes, including the heat transfer analysis. The results of turbulence models, k-ε were computed for different divergent angle, different area ratio and compared. Further based on Mach number and thrust, nozzle design has been optimized .Model and meshing done by ICEM-CFD, analysis done by CFX-13 and results obtained by CFD-POST. Geometry has been redesigned theoretically, by analyzing the flow parameters and hence optimizing the geometry of nozzle.***

Key words ***:supersonic nozzle, ICEM-CFD CFX-13, Mach number.***

I. INTRODUCTION

Aerospace propulsion system is a system in which the fundamental principle obeys Newton’s laws, namely that force is proportional to rate of change of momentum, and that action and reaction are equal and opposite. Thrust chamber is one example of a system that works using these laws besides other systems i.e. turbojet and ramjet. Thrust chamber is a system that provides thrust by expelling stored matter, called the propellant. This thrust can range from mega-Newton to milli-Newton. Thrust chamber system can be main spacecraft propulsion i.e. rocket engine, auxiliary spacecraft propulsion, satellite launcher, missile launcher, assist-take-off engines for airplanes and even ejection of crew escape capsules.

Small thrusters are usually used as secondary spacecraft propulsion system in which some of the functions are station keeping. Liquid propellant thrust chamber generally consists of injector, combustion chamber and convergent-divergent (C-D) nozzle or De-Laval nozzle. Propellant is injected through an injector and burned in combustion chamber and produces hot gases. The gases move towards the nozzle throat at a subsonic flow. This process continues until there is enough pressure builds up to force a part of them out of a nozzle throat at supersonic velocity. A carefully designed nozzle’s divergence part is vital as it will determine the overall efficiency of the thruster.

This paper is organized as, section II deals with methodology followed in this design, section III describes the results obtained, and section IV is concluded with the future scope on this work.

II. METHODOLOGY

In order to create the nozzles shapes, we have taken design variables inputs. By using ideal gas flow equation geometry of nozzle has been arrived. Nozzle geometry calculated according to the boundary condition

- Inlet pressure = 44bar(Po)
- Inlet temperature = 3400k(To)
- Mass flow rate = 826kg/s.
- P*= 0.528 Po ,
- p*= 0.528×44 = 23.232.
- T*= 3400/1.2 = 2833k

V*=(RT*/p*)=(287×2833/23.23×100000)
v*=0.3499 m^3/kg(specific volume)
Velocity at throat V*=(44.72√Cp(To-T*)
V*=1066m/s.
Area of throat A*=(mv*/V*)=(826×0.3499/1066)
A*=0.2744m2
D*=0.591mt(throat diameter).
To find out let diameter

Boundry condition for Atomspheric condition
INLET
Pressure - 44bar
Temperature - 3400k
Outlet-atmospheric
Wall-Adiabatic
At outlet pressure will vary according to altitude.

Boundry condition for out atmospheric condition is 1atm
$(To/T_1)=(Po/p_1)^{(\gamma-1/\gamma)}$
$T1=To/(Po/p1)^{(\gamma-1/\gamma)}=1163k$
$T_1=1163k$
$V1=RT_1/P_1=(287\times1163)/1.013\times105$
V1=3.2955m3/kg(sp volume)
Velocity at out let = $(44.72\sqrt{Cp(To-T1)}$
V1=2120m/s
Area at outlet=$(mv_1/V_1)=(836\times3.2955)/2120$
Area at outlet A1=1.283mt sq.
Diameter at outlet=1.278mt.

III. RESULTS

This section describes the results obtained for different models at different conditions.

a). RESULTS FROM POST CFD(model-1)4⁰ area ratio 0.46

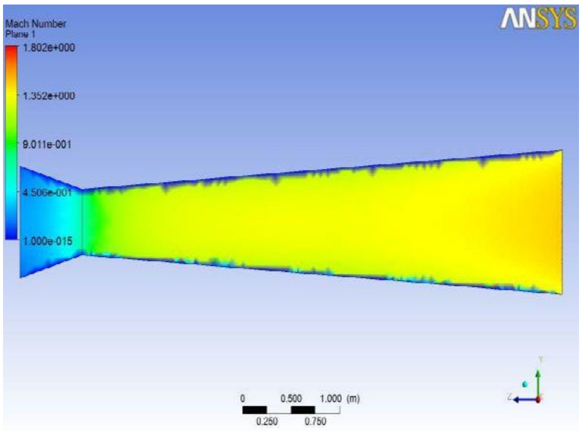

Fig. 1 : Mach number for the model 1

b).RESULTS FROM POST CFD(model-2)7⁰ area ratio 0.46

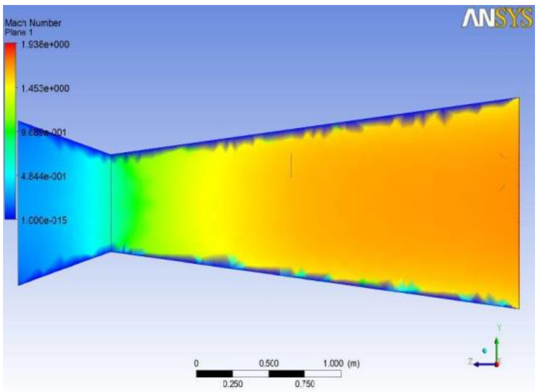

Fig. 2 : Mach number for the model 2

c). RESULTS FROM POST CFD(model-3)10⁰ area ratio 0.46

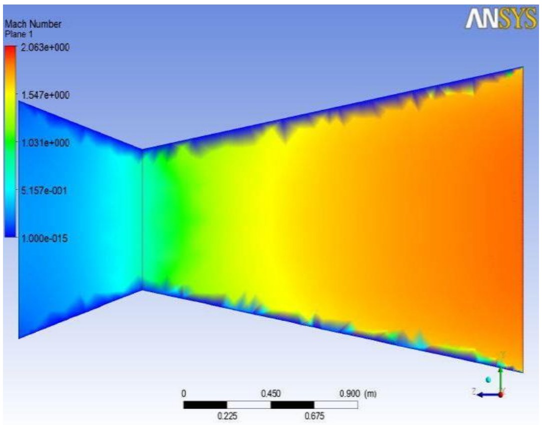

Fig. 3 : Mach number for model 3

d). RESULTS FROM POST CFD(model-4) 13⁰area ratio 0.46

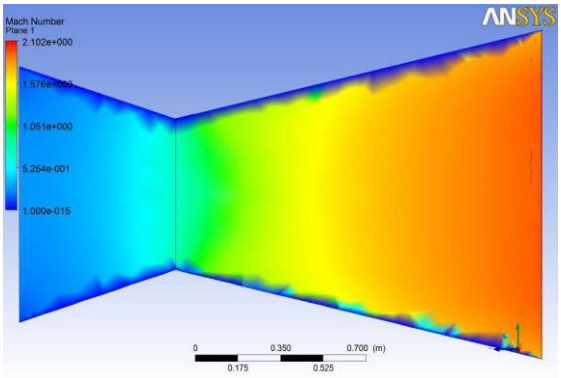

Fig. 4: Mach number for model 4

e). RESULTS FROM POST CFD(model-5)15⁰ area ratio 0.46

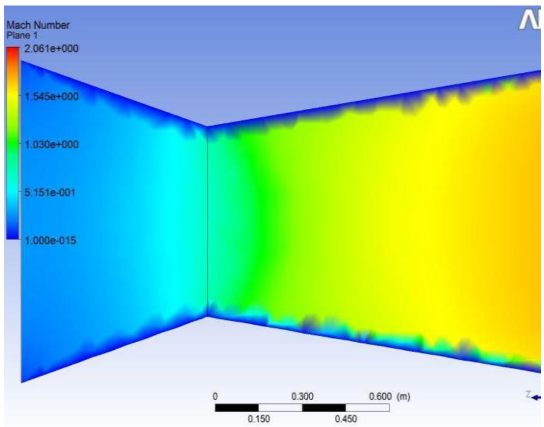

Fig 5.: Mach number for model 5

f). RESULTS FROM POST CFD(model-6)13⁰ area ratio 0.46 (HIGHER ALTITUDE-5000FEET)

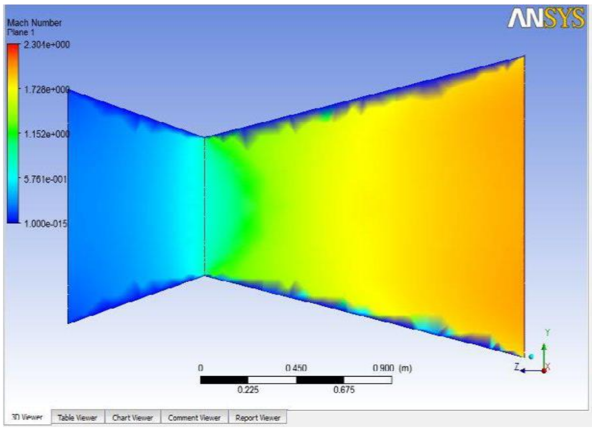

Fig. 6: Mach number for model 6

g). RESULTS FROM POST CFD(model-7) 13⁰ area ratio 0.46 (HIGHER ALTITUDE-10000FEET)

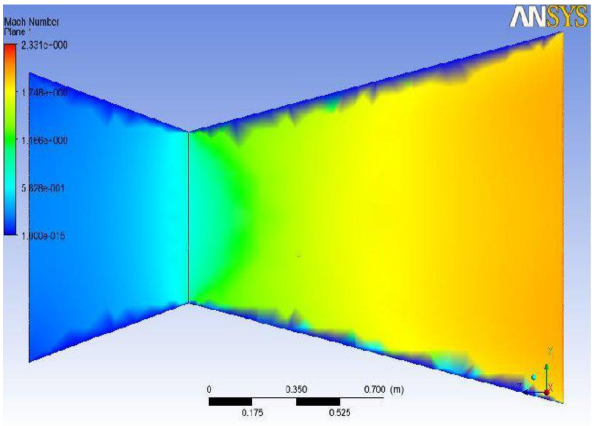

Fig. 7: Mach number for model 7

h). *RESULTS FROM POST CFD (model-8) 13⁰ area ratio 0.46(HIGHER ALTITUDE-20000FEET)*

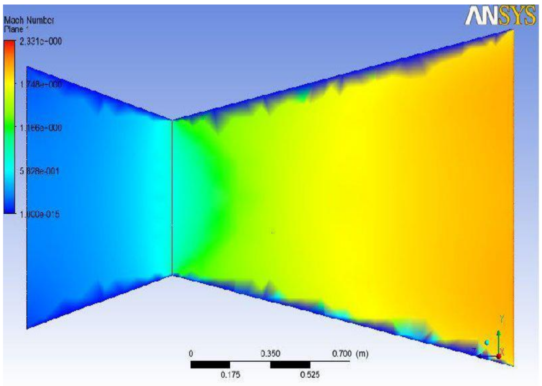

Fig. 8: Mach number for model 8

i). *RESULTS 13⁰ area ratio 0.46(HIGHER ALTITUDE-25000FEET)*

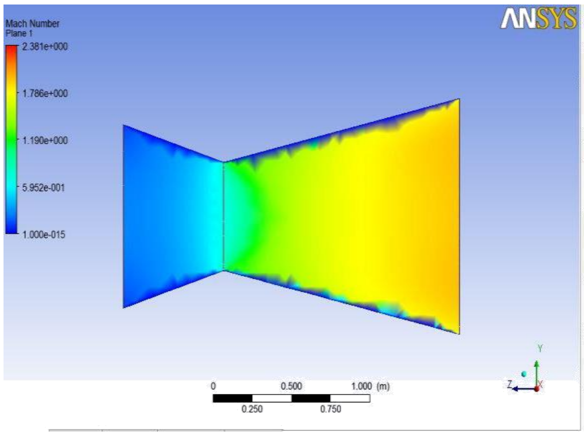

Fig. 9: Mach number for higher altitude

TABLE 1

SL NO	MODEL	ALTITUDE FEET	AREA RATIO	VELOCITY m/s	MACH NUMBER	THRUST KN
1	13⁰	5000	0.46	2300	2.1	1899
2	13⁰	10000	0.46	2350	2.2	1941
3	13⁰	20000	0.46	2400	2.28	1982
4	13⁰	25000	0.46	2550	2.32	2106

IV. Conclusion and Future Scope.

1 Simulations of the flow-fields and thermal analysis were run in the CFD software package ANSYS CFX-13 for the 19 shapes, using k-_ turbulence model with a total of 19 simulations.
2 The standard deviations of the pressure, Mach number , velocity and thrust results at the exit of the nozzles were calculated .
3 After this process, some conclusions were made. The highest mach number at exit was obtained at 13⁰ divergent angle and 0.46 area ratio (throat to outlet) which clearly shows whatever the throat and exit area designed initially according to ideal gas equation is validated .
4 After optimizing 13⁰ divergent angle and 0.46 area ratio ,considering the design, analysis of flow has been carried out at higher altitude
5 At higher altitude up to 50000 feet can be carried, above that stagnated pressure at entry is not enough to operate this is due to very less back pressure.
6 Finally based on mach number at outlet, outlet velocity, thrust and above discussion 13⁰ divergent angle and 0.46 area ratio (throat to outlet) model has been optimized.

REFERENCES

[1] D. G. Shepherd, Aerospace propulsion (American Elsevier Publishing Company, Inc., 1972).

[2] J. B. Pearson, D. B. Landrum, and C. W. ssHawk, “Parametric Study of Solar Thermal Rocket Nozzle Performance,” NASA Technical Memorandum 111354, (1994).

[3] S. P. Grisnik and T. A. Smith, “Experimental Study of Low Reynolds Number Nozzles,” NASA Technical Memorandum 89858, (1987).

