Modelling and Simulation of Supersonic Nozzle using ANSYS Workbench with New Measuring Data

Jyoti Kumari[1] C. S. Koli[2]
[1]M. Tech Scholar [2]Assistant Professor
[1,2]Department of Mechanical Engineering
[1,2]SRCEM Banmore

Abstract— This paper deals with modelling and simulation of supersonic nozzle using computational dynamics. We are considering the C-D (convergent-divergent) nozzle and analysis of supersonic nozzle by changing the parameters which are presently uses in the rocket have been conducted. Study also focused on discussing about the material using at the time of manufacturing of nozzle. First axisymmetric nozzle geometries were drawn on design modeller of ANSYS then analysis has done using fluent. The heat transfer rate of the outer wall and inner wall of the nozzle has been calculated and compared for various nozzle configurations. Static Pressure contours of various nozzle configurations have been plotted. Velocity and Mach number contours of various nozzle configurations have also been plotted. Graph of Mach number and heat transfer coefficient has also shown. After comparing the results it has observed by the contours that, nozzle with greater expansion gives high heat transfer coefficient compared to the conical nozzle. Parameters and laminar models considered in the study were found to give results which agreed with experiment and analytical data.

Key words: Quality Function Deployment; Supersonic Nozzle

I. INTRODUCTION

Basically the nozzle used to modify the flow of fluid. We considered CD types of nozzles, in which flow velocity much higher that sonic velocity and these types of nozzle are used in propelling nozzles are used in jet engines, Specially this supersonic nozzle are used in rockets (for providing sufficient trust to move upwards). They also used in supersonic gas turbine engine (for increasing the air intake when air requirement of engine is very high).

The main purpose of this article is to find out the effects and output of fluid flow while change the parameter of supersonic nozzle using ANSYS CFD.

In convergent divergent nozzle, when fluid is flowing in nozzle then continuous changes in the flowing stream occur are,

– Changes the cross sectional area
– Wall friction
– Energy friction, such as external heat exchange, etc.

Now we compare and study about the different parameter of CD nozzle. In the design of convergent-divergent nozzle the relation is obtained from the area velocity relation $(dA/dV) = -(A/V)(1-M^2)$ Where M is Mach number, A is cross-sectional area, v is velocity or speed of fluid flow.

Converging section	Diverging section
dA<0	dA>0
dV>0	dV>0
M<1	M>1

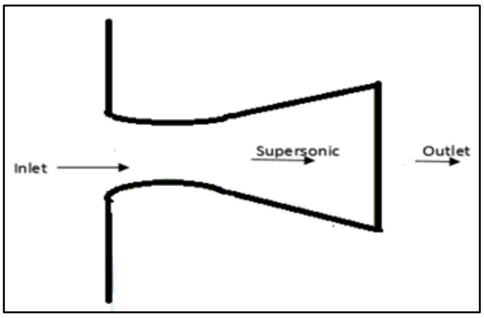

Fig. 1: Nozzle

In above figure 1 we can observe different things:

A. *In converging sections (dA<0)*

– When flow is subsonic ($M<1 \rightarrow (1-M^2)>0$); speed increases ($dv>0$); Mach number increases ($M>0$); but pressure and temperature decreases.
– When flow is supersonic ($(M>1) \rightarrow 1-M^2)>0$); speed decreases ($dv<0$); Mach number decreases ($M<0$); but pressure and temperature increases.

B. *In diverging section (dA>0)*

– When flow is subsonic ($M<1 \rightarrow (1-M^2)>0$); speed decreases ($dv<0$); Mach number decreases ($M<0$); but pressure and temperature increases.
– When flow is supersonic ($(M>1) \rightarrow 1-M^2)>0$); speed increases ($dv>0$); Mach number increases ($M>0$); but pressure and temperature decreases.

II. MODELLING

The model of supersonic nozzle is created in ANSYS WORKBENCH. In the modelling we give parameter for designing the model (nozzle), we also make the nozzle symmetric for maintaining the continuity equation as discuss above.

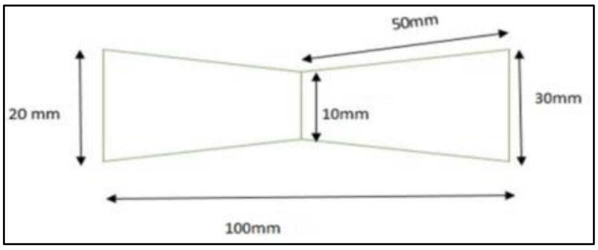

Fig. 2: Geometry of the nozzle

Table 1 Design parameter of convergent- divergent nozzle

Inlet diameter	20 mm
Outlet diameter	30 mm
Throat diameter	10 mm
Length	100 mm
Pressure	2 bar
Ideal gas viscosity	Sutherland

Table 1: Modelling

III. MESHING

After the modelling is completed the meshing is to be done. We are using a fluid fluent module for performing a meshing. Automatic method is using here for meshing and the mesh type is selected as all quad.

In study of any fluid flow we basically concern about three basic principles:

- Conservation of mass
- Conservation of momentum
- Conservation of energy

Physics preference	CFD
Solver preference	CFX
Advanced size function	Proximity and curvature
Relevance centre	Fine
Smoothing	High

Table 2: Details and sizing input at the time of mesh

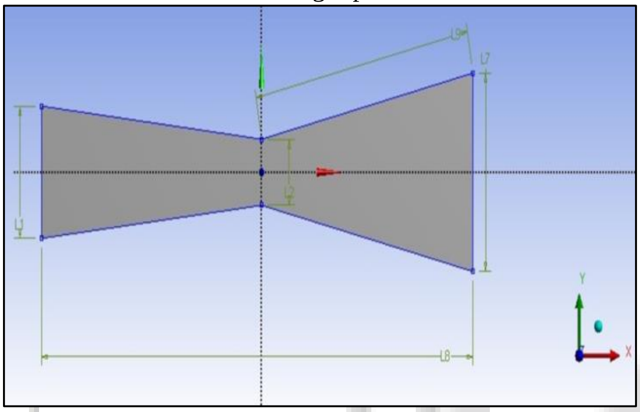

Fig. 3: Meshed view of the geometry

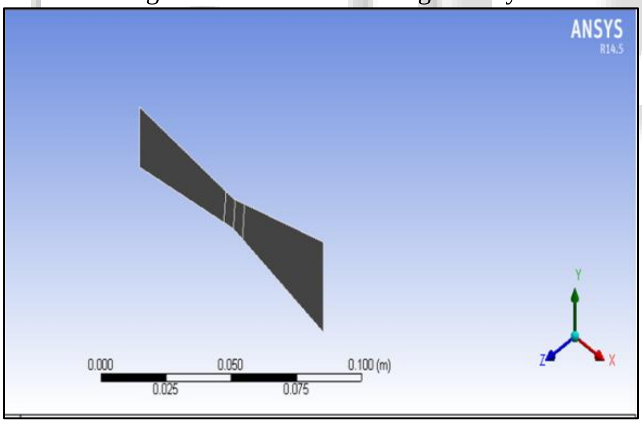

Fig. 4: Meshing generated

A. Boundary Conditions

- Inlet
- Outlet
- Walls

Boundary condition has done in ANSYS WORKBENCH. There is no possibility to given a boundary condition in FLUENT. Due to this, it is easy to analysis flow rate. The meshed file is read in Fluent to solving the problem.

IV. SOLUTION

The analysis process has to done in ANSYS FLUENT at different condition.

A. Analysis procedure:

The procedure has to follow at the time of meshing and results are validated.

Procedure	**Details**
Problem setup: General – solver	Type: Density Base, Velocity: Absolute
Models	ON: Energy based equation, Viscous : Laminar
Materials	Fluid: Air, Density: ideal gas, Viscosity: Sutherland
Boundary condition	Inlet: pressure inlet, Gauge total pressure:2e5, Outlet: pressure outlet
Reference value	Compute from: inlet, Reference Zone: Solid surface body
Monitors	Create walls- CD1, Select print to console and plot
Initialization	Standard initialization, Compute from inlet
Solution	Solution controls, Run calculation: Enter the number of iteration, Click calculation

Table 3: Problem Setup

The remaining values are:
The density of air = 1.225 kg/m³
Specific heat (Cp) = 1006.3 j/kg
Thermal conductivity = 0.0242W/m-k
Viscosity = 1.71e-05
Reference temperature = 273.11K
Effective temperature =110.56K.

V. RESULT AND DISCUSSION

The solution is run with given reference and input.

In figure 4 we analyse the effects of static pressure on the nozzle. We can easily see that the maximum pressure is exerted on side walls of the nozzle. The minimum pressure is -5.72e+05 and maximum pressure is 6.55e+04. The minimum effect is produced on the middle part or throat of the nozzle. In a minimum effect is starting from -5.72e+05 and its going to -4.76e+05.

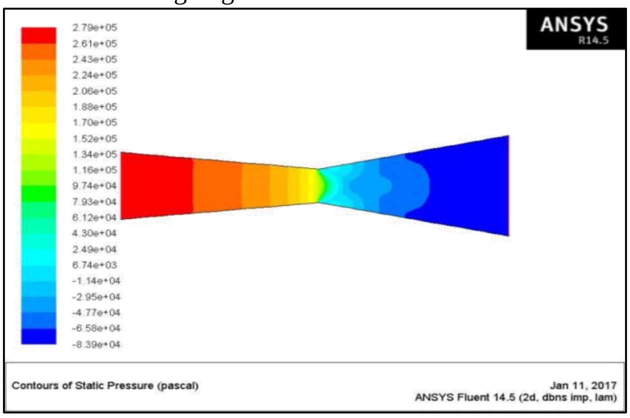

Fig. 5: Contours of Static pressure

Pressure contour: In figure8. The pressure is increases at the inlet then sudden decreases due to shock, the maximum number of pressure is -5.715e+005 and minimum pressure is 6.546e+004.

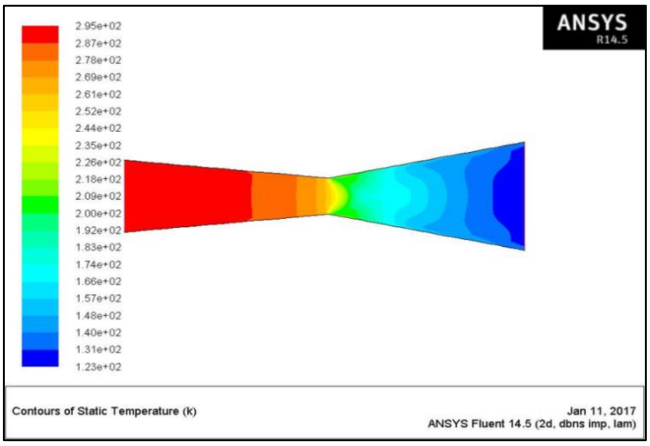

Fig. 6: Contours of static temperature

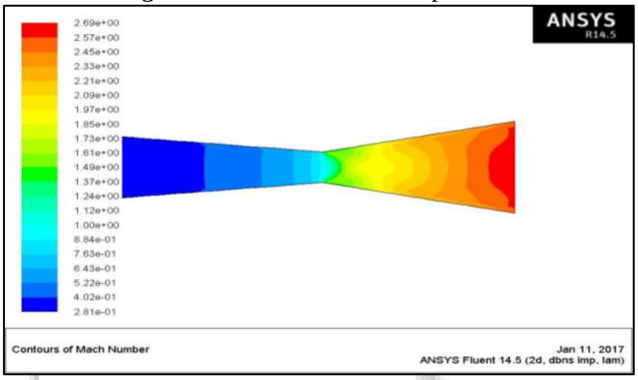

Fig. 7: Contours of Mach number

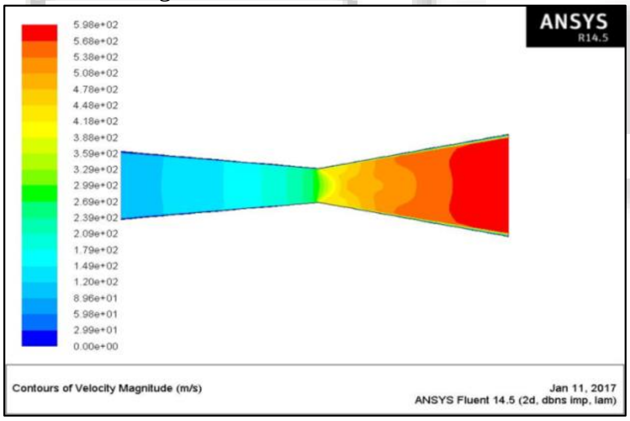

Fig. 8: Contours of velocity magnitude of nozzle geometry.

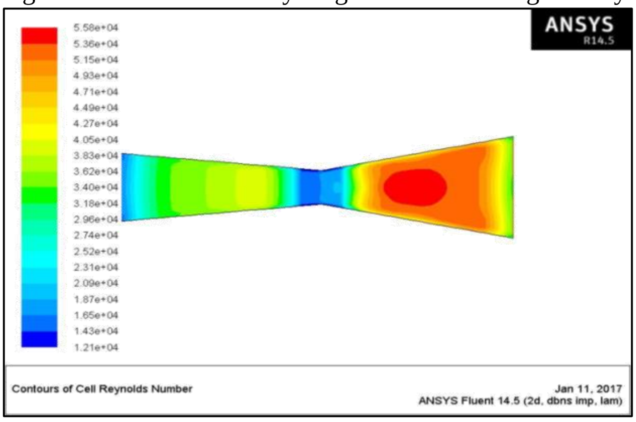

Fig. 9: Contours of Cell Reynolds number of nozzle geometry.

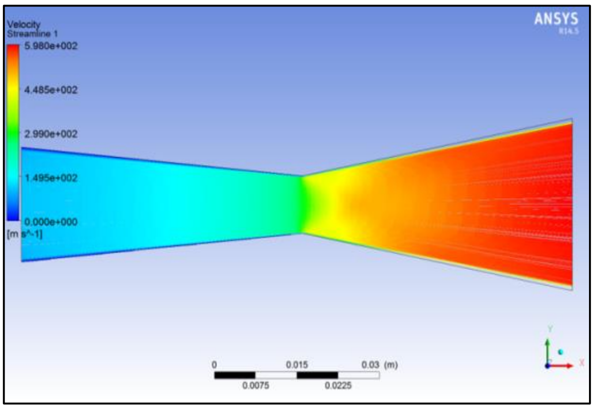

Fig. 10: Velocity stream line of nozzle geometry.

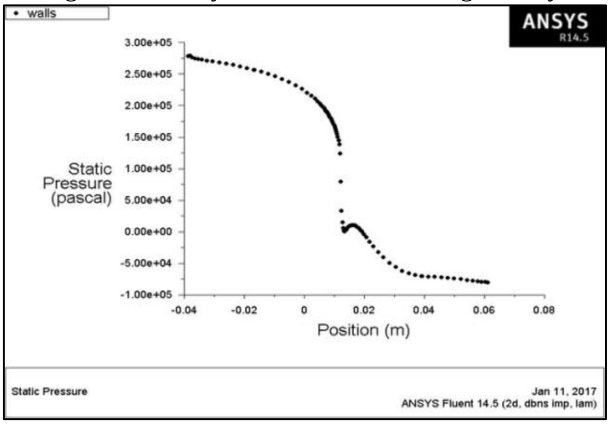

Fig. 11: Plot between static pressure to nozzle length

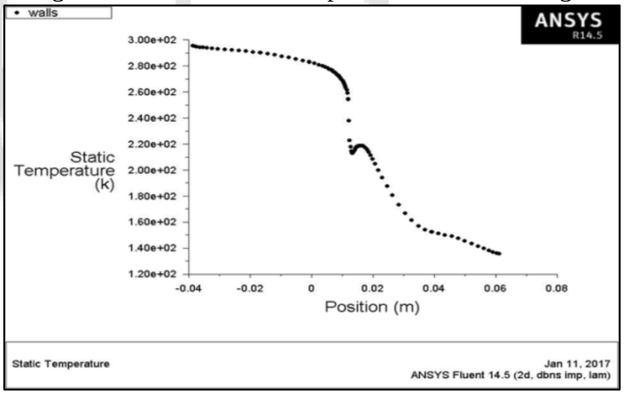

Fig. 12: Plot between static temperatures to nozzle length

VI. CONCLUSION

CFD analysis of supersonic nozzle is done using varying properties and varying boundary conditions with different equation. After successfully completing this simulation based experiment, the decisions were finally confined.

By the analysing we can conclude the Sutherland equation and K epsilon laminar model exactly simulates the flow flied of supersonic conditions "to capturing the behaviour of flow of fluid in nozzle and pressure and temperature effects on the nozzle with using of ANSYS fluent." The lesser effects of pressure of fluid flow which is striking on the walls of nozzle.

- The effect of pressure in the inlet is high with 2.79e+05, then the pressure goes decreasing to the outlet with -8.39e+04.
- The effect of static temperature at inlet is high with 2.95e+02, then the pressure goes decreasing to the outlet with 1.23e+02.

- The effect of Mach number in the inlet is low with 2.81e-01, then the Mach number goes increases to 2.69e+00 in outlet.
- The effect of velocity magnitude in the inlet is low with 1.20e+02, then the velocity increasing in the outlet with 5.98e+02.

REFERENCES

[1] Adam Neale, Dominique Derome, Bert blocken and Jan Carmelite, "CFD calculation of convective heat transfer coefficients and validation- parts2: turbulent flow" 41-kyoto, April 3rd to 5th 2006.
[2] KM Pandey, SK yadav "CFD analysis of a rocket nozzle with four inlets at Mach 2.1", international journal of chemical engineering and application (IJCEA),Vol.1, 4dec 2010.
[3] Srikrishna C. Srinivasan "CFD modelling and analyzing of an air-jet facility using ansys Fluent" CA-951932, May 2012.
[4] Pardhasaradhi Natta, V.ranjith Kumar, Dr.Y.V Hanumantha Rao, "Flow Analysis of rocket Nozzle using computational Fluid Dynamics (CFD), International journal of engineering research and application (IJERA),Vol.2, issue:5sept.-oct.2012,pp. 1226-1235.
[5] Manish Sharma, T.ratna, "Flow analysis over an F-16 aircraft using computational fluid dynamics" ISO 9001:2008,Vol 3, issue5, may 2013, pp.339.
[6] Gutti rajeswara Rao, US Ramakanth, Alakshman, "Flow analysis in a convergent divergent nozzle using CFD", international journal of engineering research and application, (IASTER), Vol.2, issue: 5, October-December, 2013, Pp164-144
[7] Hanamant, K.N Kumar, Dhanawade H, DJ sanghvi "Natural convection heat transfer flow visualization of perforated fin array using CFD simulation", international journal of research of engineering and technology (IJRET), Vol: 02, issue 12, Dec2013, pp-483-490.
[8] Biju Kuttan P, M sajesh "Optimization of divergent angle of a rocket engine nozzle using computational fluid dynamics", The international journal of engineering and science (IJES), Vol.2, issue: 2, 2013, pages-196-207.
[9] Christopher E. Cordell, Jr. and Robert D. Braun, "An analytical approach to modelling supersonic retro propulsion flow flied components", AIAA 2014-1093, 13-17Jan 2014.
[10] Nirmith ku, Dr. S srinivas, "Modeling and simulation of rocket nozzle", IJAEGT, Vol-2, issue-09, September 2014, Pp.: 988-995.

