ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 4 – No.2, April 2017

FLOW ANALYSIS IN A CONVERGENT-DIVERGENT NOZZLE USING CFD

Rahul Rai
Student in Department of Aeronautical Engineering, Punjab Technical University, India
rai.rr898@gmail.com

Danish Khan
Student in Department of Aeronautical Engineering, Punjab Technical University, India
danishkhan351996@gmail.com

Vikas Kumar Chauhan
Student in Department of Aeronautical Engineering, Punjab Technical University, India
Chauhan.vk018@gmail.com

ABSTRACT
CFD is a branch of Fluid Mechanics which rely on numerical methods and algorithms to solve and analyze problem that involves fluid flow. CFD analysis has been used to analyze flow pattern of supersonic rocket nozzle at various degree of divergent angle, mach numbers etc. This paper aims to study the behaviour of flow in convergent divergent nozzle by analyzing various parameters like static pressure, temperature and velocity in Mach No using computational fluid dynamics software (C.F.D).These results were further plotted comparing them with analytical values. In this section we prove the continuity Equation in the C-D Nozzle with necessary data.

Keywords
Static Pressure, Continuity Equation, Velocity Plot, Mach number, CFD, Nozzle

1. INTRODUCTION
Nozzle is used to convert the chemical-thermal energy generated in the combustion chamber into kinetic energy. The nozzle converts the low velocity, high pressure, high temperature gas in the combustion chamber into high velocity gas of lower pressure and temperature. The one-dimensional inviscid isentropic flow in a convergent-divergent nozzle is a classical text book problem which can be further solved by using Cascade Theory and 2-d Flow analysis theories. Multi dimensionality and viscous effects like wall boundary layer and flow separation drastically alter the flow in a CD nozzle. The over-expanded flow regime in CD nozzles of different shapes and sizes has been a subject matter of numerous investigations because of their wide range of applications.

Computational Fluid Dynamics (CFD) is an engineering tool that assists experimentation. Its scope is not limited to fluid dynamics; CFD could be applied to any process which involves transport phenomena with it. To solve an engineering problem we can make use of various methods like the analytical method, experimental methods using prototypes. The introduction of Computational Fluid Dynamics has overcome this difficulty as well as revolutionized the field of engineering.

CONVERGENT-DIVERGENT nozzle is designed for attaining speeds that are greater than speed of sound. the design of this nozzle came from the area-velocity relation $(dA/dV)=-(A/V)(1-M^2)$ M is the Mach number (which means ratio of local speed of flow to the local speed of sound) A is area and V is velocity.

The following information can be derived from the area-velocity relation:

1. for incompressible flow limit, i.e. for M tends to zero, AV = constant. This is the famous volume conservation equation or continuity equation for incompressible flow.

2. For $M < 1$, a decrease in area results in increase of velocity and vice versa. Therefore, the velocity increases in a convergent duct and decreases in a Divergent duct. This result for compressible subsonic flows is the same as that for incompressible flow.

3. For $M > 1$, an increase in area results in increases of velocity and vice versa, i.e. the velocity increases in a divergent duct and decreases in a convergent duct. This is directly opposite to the behaviour of subsonic flow in divergent and convergent ducts.

4. For $M = 1$, $dA/A = 0$, which implies that the location where the Mach number is unity, the area of the passage is either minimum or maximum. We can easily show that the minimum in area is the only physically realistic solution.

Regime	Mach
Subsonic	<1.0
Transonic	0.8-1.2
Sonic	1,0
Super-Sonic	1.0-5.0
Hyper-Sonic	5.0-10
High-Hypersonic	>10

Table.1

From table.1 at transonic speeds, the flow field around the object includes both sub- and supersonic parts. The transonic period begins when first zones of $M>1$ flow appear around the object. Supersonic flow can decelerate back to subsonic only in a normal shock; this typically happens before the trailing edge.

As **M<1** dA, dp have the same sign. Thus, as A increases, p increases.

dA , du have opposite signs. Thus as A increases, u decreases.

Diverging duct in subsonic flow: pressure increases, speed decreases.

ISSN (O): 2393-8609

International Journal of Aerospace and Mechanical Engineering

Volume 4 – No.2, April 2017

Converging duct in subsonic flow: pressure decreases, speed increases.

As **M>1** dA, dp have opposite signs. Thus as A increases, p decreases.

dA, du have the same sign. Thus as A increases, u increases.

Diverging duct in supersonic flow: pressure decreases, speed increases.

As M=1 dA/dx is 0. Thus we have either a maximum or minimum of area.

The maximum area case is not of much interest, since there is no way to reach Mach 1 at this point, with flow from either direction. So the case of interest is where the area becomes a minimum: a "throat".

2. BASIC RELATIONS

The Continuity equation is

$A_x V_x \rho_x = \rho_t . A_t V_t$

The steady flow energy equation is as follows

$$\frac{Q-W}{m} - \left(h + \frac{v^2}{2} + gz\right)t - \left(h + \frac{v^2}{2} + z\right)x$$

The following equations have been derived using continuity and steady flow energy equation. Where,

P- Pressure (Pa)

T- Temperature (K)

V - Velocity (m/s)

g- Gravitational Acceleration (m/s^2)

z- Height (m)

A- Area (m^2)

C_p - Specific heat at constant pressure (J/Kg K)

C_v – Specific heat at Constant Volume (J/Kg K)

h- Enthalpy (J)

3. STANDARD DIMENSIONS

The Dimensions of the C-d nozzle for the analysis are given below:

Total length of nozzle= 125mm

Inlet diameter= 50mm

Throat diameter= 20mm

Outlet diameter = 70mm

4. BOUNDARY CONDITIONS

Inlet pressure =49300 Pascal

Inlet Velocity = 440 m/s

Inlet Temperature = 299.9k

5. COMPUTER SIMULATION

CFD is an engineering tool that assists Experimentation. The following steps were performed in CFD of nozzle:

5.1 Modeling: The 2-Dimensional modeling of the nozzle was done using Ansys Modeling software

5.2 Meshing: After modeling of the nozzle, its meshing was done using ANSYS ICEM CFD software. The mesh as created using no method and the number of Nodes 5719 and number of elements are 5544.

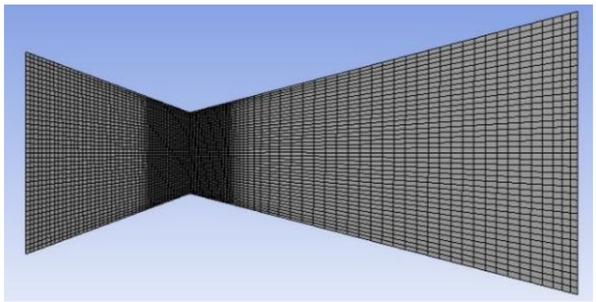


Fig 1: Meshing of Nozzle

5.3 Pre-Processing: Pre-processing of the nozzle was done in ANSYS FLUENT. 2-D and double precision settings were used while reading the mesh. The mesh was scaled since all Dimensions were initially specified in mm. The mesh was checked in fluent and no critical errors were reported.

5.4 Solution: The Solution Was Converged After 500 Iterations. And The Order of Scaled Residuals Was below Solution controls.

Counter number:5

Solution initialization

Compute from: inlet

Velocity Contours are calculated in Mach Numbers and pressure in Terms of Static Pressure.

5.5 Post Processing: It is a process which gives the results data as output. You can get output in the form of counters display curves, deform shapes and in form of graphs.

6. RESULT AND DISCUSSIONS

Static Pressure Contours: The pressure is maximum at the inlet and goes on decreasing till the outlet. There is sudden decrease in pressure due to shock wave just after the throat section.

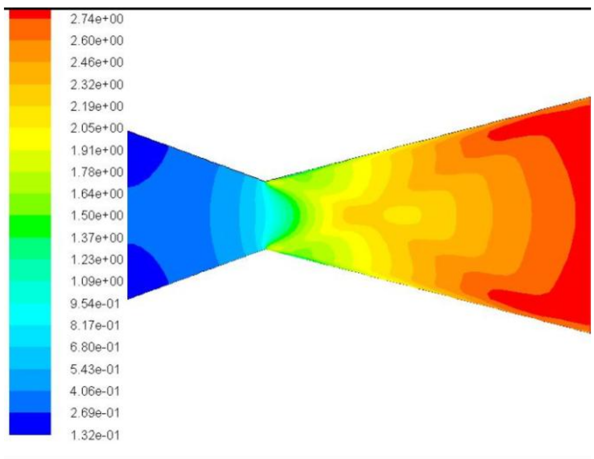


Fig. 2: Pressure Contours

Velocity Contours: The velocity is minimum at the inlet and goes on increasing till the nozzle exit. The velocity magnitude is Mach 1 at the throat section of the nozzle. This condition is known as choked flow condition.

ISSN (O): 2393-8609

International Journal of Aerospace and Mechanical Engineering

Volume 4 – No.2, April 2017

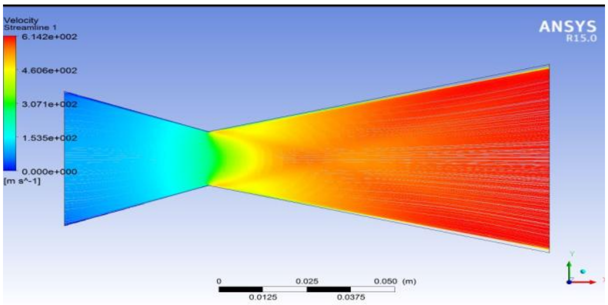
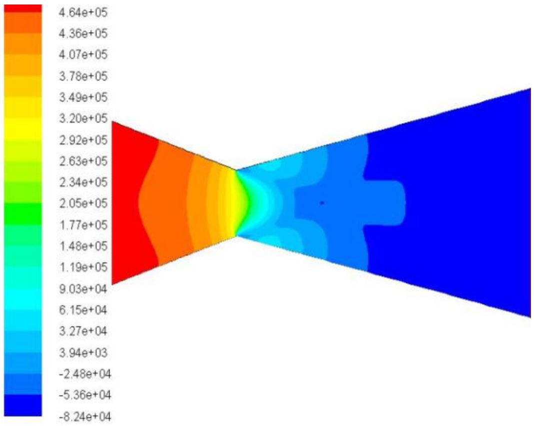


Fig. 3: Velocity Contours

7. CONCLUSION

The results obtained by Computational Fluid Dynamics (CFD) are almost identical to those obtained theoretically. And the Results of the Continuity equation and the plots and contour of Velocity and Static Pressure in C-D Nozzle are given by Following diagram



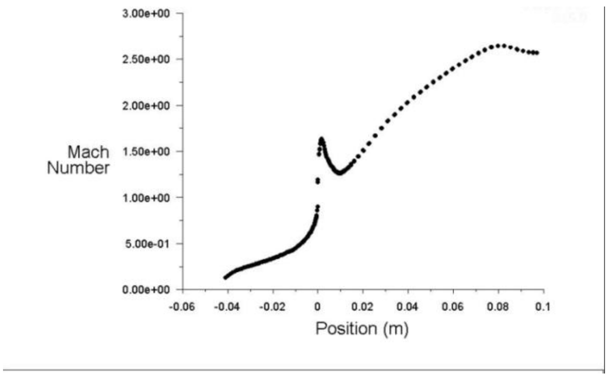


Fig. 4: Mach Variation along the wall position

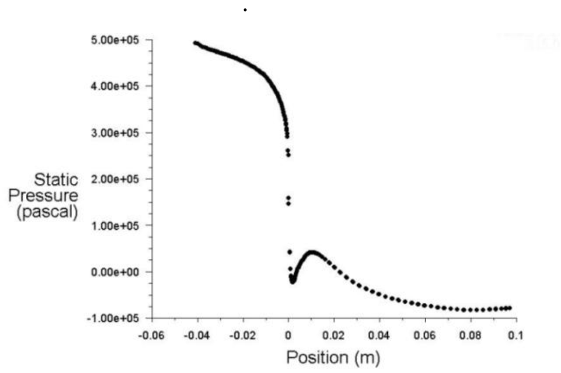


Fig. 5: Plot of variation of static pressure along walls of nozzle

8. REFERENCES

[1] B.V.V. NAGA SUDHAKAR, B PURNA CHANDRA SEKHAR, P NARENDRA MOHAN, MD TOUSEEF AHMAD "Modeling and simulation of Convergent-Divergent Nozzle Using Computational Fluid Dynamics" Published at www.irjet.net Volume: 03 Issue: 08 | Aug-2016

[2] Gutti Rajeswara Rao, U.S. Ramakanth, A. Lakshman "Flow Analysis in a Convergent-Divergent Nozzle Using CFD" International Journal of Research in Mechanical Engineering Volume 1, Issue 2, October-December, 2013, pp.136-144, © IASTER 2013 www.iaster.com, ISSN Online: 2347-5188 Print: 2347-8772

[3] Charles G Martin- Ansys Workbench-Fluent "Convergent-Divergent Nozzle" video at http://m.youtube.com/watch?v=P_jdywkbRQ4