ANALYSIS OF FLOW IN A DE LAVAL NOZZLE USING COMPUTATIONAL FLUID DYNAMICS

CH. Sriivasa Chakravarthy[1], R.Jyothu Naik[2]

Mechanical Engineering,Narasarao Peta Engineering College, Narasaraopet, A.P., India[1]

Assistant Professor, Mechanical Engineering, Narasaraopet Engineering College, A.P, India [2]

Chakri.ch81@gmail.com[1]

Jyothu.naik@gmail.com[2]

Abstract

De Laval nozzles are mechanical devices which are used to convert the thermal and pressure energy into useful kinetic energy. Nozzle is a device designed to control the rate of flow, speed, direction, mass, shape and the pressure of the stream that exhaust from them. Convergent divergent nozzle is the most commonly used nozzle since in using it the propellant can be heated in combustion chamber. The general range of exhaust velocity is 2 to 4.5 kilometer per second. In this project the designing and analysis of CD nozzle geometry is done in the CFD (Computational Fluid Dynamics software). Firstly the design of nozzle is made in CATIA V-5 software and then the nozzle geometry is further analyzed in ANSYS FLUENT software in order to analyze the flow inside the CD nozzle and to get the view of the behavior of fluid inside the convergent-divergent section of nozzle. And compare these results with manual calculation obtained by standard nozzle equations.

1. INTRODUCTION

A Rocket engine is a jet engine that uses specific propellant mass for forming high speed propulsive exhaust jet. Rocket engines are reaction engines and obtain thrust in accordance with Newton's third law of motion. Most rocket engines are internal combustion engines. Rocket engines are in a group that have maximum exhaust velocities, are the lightest, and are the least energy efficient of all types of jet engines. The rockets are powered by exothermic chemical reactions of the rocket propellants used.

Hot exhaust gases expand in the diverging section of the nozzle as the Mach number increases from one. The pressure of these gases decreases as energy is used to accelerate the gas. The nozzle is usually designed in such a way that the exit area is large enough to accommodate the requisite mass flow. It is under this condition that the thrust reaches maximum and the nozzle is said to be adapted, also called optimum or correct expansion. Rocket engines produce thrust by creating a high-speed fluid exhaust. This fluid is generally always a gas which is created by high pressure (10-200 bar) combustion of solid or liquid propellants, consisting of fuel and oxidizer components, within a combustion chamber.

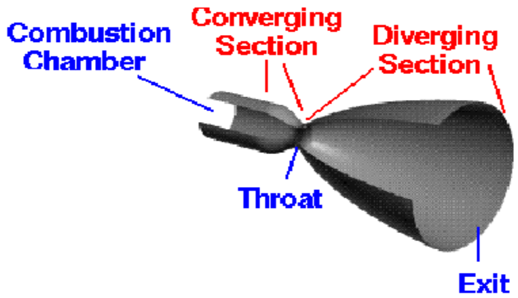


Fig: Parts of rocket nozzle

2. LITERATURE REVIEW

K. M. PANDEY, S. K. YADAV [1] "**Cfd Analysis Of A Rocket Nozzle With Two Inlets At Mach 2.1**". Ms. B. Krishna Prafulla, Dr. V. Chitti Babu and Sri P. Govinda Rao [2] "**CFD Analysis of Convergent Divergent Supersonic Nozzle".** Biju Kuttan P, M Sajesh [3] **"Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics". Pardhasaradhi Natta**, V. **Ranjith Kumar [4] Dr. Y. V. Hanumantha Rao "Flow Analysis of**

Rocket Nozzle Using Computational Fluid Dynamics (CFD)". K.M. PANDEY , Member IACSIT and A.P. Singh [5] "CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software".

3. MATHEMATICAL FORMULATIONS

Newton's Third law of motion: It states that for every action, there is always an equal and opposite reaction; or the mutual actions of two bodies upon each other are always equal.

Standard Dimensions

- To draw the nozzle in ANSYS FLUENT, the standard dimensions of nozzle are taken from International Journal of Mechanical and Production Engineering.
- Total length of nozzle= 484 mm
- Inlet diameter= 166 mm
- Throat diameter= 35mm
- Outlet diameter = 183 mm
- Convergent angle=32 degrees
- Divergent angle =11 degrees

Boundary Conditions

- Inlet pressure =100 bar
- Inlet temperature= 3300k

Nozzle Properties

Name: **Tungsten**
Symbol: **W**
Atomic Number: **74**
Atomic Mass: **183.84 amu**
Melting Point: **3410.0 °C (3683.15 K, 6170.0 °F)**
Boiling Point: **5660.0 °C (5933.15 K, 10220.0 °F)**
Number of Protons/Electrons: **74**
Number of Neutrons: **110**
Classification: **Transition Metal**
Crystal Structure: **Cubic**
Density @ 293 K: **19.3 g/cm^3**
Color: **Silver**

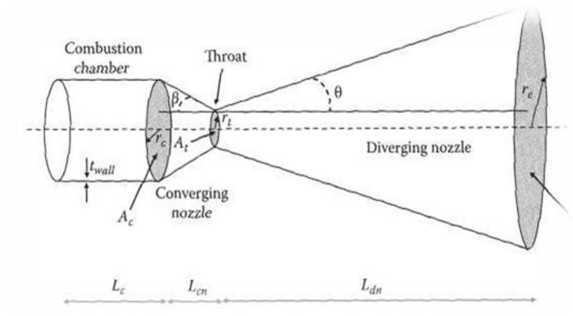


Fig: Line diagram of De Laval nozzle

Table-1: Theoretical Results Obtained By Using Basic Relations with Different Mach Numbers

section	A_E/A_T	Velocity (m/sec)	Pressure (bar)	Temperature (k)
convergent	1.5	169.71	87.33	3240.08
convergent	1.2	482.91	79.89	3194.12
throat	1	1030.46	50.96	2972.98
divergent	1.2	1335.28	32.08	2760.50
divergent	4	2051.23	10.98	2137.56
outlet	7.142	2387.52	1.68	1724.90

4. COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process i.e. on a computer). Computational Fluid Dynamics, or CFD, is the use of mathematical techniques to model fluid flow. You can use CFD to create a virtual prototype of your product or process in order to better understand its performance and improve its design.

Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of

- mathematical modeling (partial differential equations)
- numerical methods (discretization and solution techniques)
- software tools (solvers, pre- and post-processing utilities)

APPLICATIONS OF CFD

Applications of CFD are numerous.

i. Flow and heat transfer in industrial processes (boilers, heat exchangers, combustion equipment, pumps, blowers, piping, etc.).
ii. Aerodynamics of ground vehicles, aircraft, missiles.
iii. Film coating, thermoforming in material processing applications.
iv. Flow and heat transfer in propulsion and power generation systems.– Ventilation, heating, and cooling flows in buildings.
v. Chemical vapor deposition (CVD) for integrated circuit manufacturing.
vi. Heat transfer for electronics packaging applications.

5. COMPUTER SIMULATION

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

1. Modeling
2. Meshing
3. Pre-Processing
4. Solution
5. Post-Processing

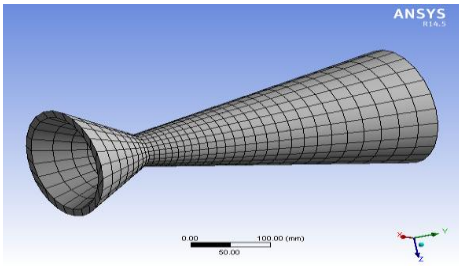
Modeling The 2-Dimensional modeling of the nozzle was done using CATIA-V5 and file was saved in .stp format. The dimensions of the de Laval nozzle are presented in the table given below.

Table 2: Nozzle dimensions

Parameter	Dimensions
Total nozzle length(mm)	484
Inlet diameter(mm)	166.6
Throat diameter(mm)	34.5
Outlet diameter(mm)	183
Chamber length(mm)	99.93
Convergent angle(deg)	32
Divergent angle(deg)	11.31
Throat radius curvature(mm)	70
Curvature(mm)	40

Meshing

After modeling of the nozzle, its meshing was done using ANSYS ICEM CFD software. The mesh as created of trigonal elements with element size 1mm near the wall of the nozzle, five prism Layers of 0.4 mm height and height ratio 1.3 were created so as to capture boundary layers finely.



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.

Solution

The Solution Was Converged After 977 Iterations. And The Order Of Scaled Residuals Was Below 10e-3.

Table-3: Solution

Solution controls	Coutrant number : 5
Solution initialization	Compute from : inlet
Run calculation	Check case Number of iterations = 2000 Click calculation

Table-4 convergence criteria

Residuals	Absolute criteria
continuity	0.001
x-velocity	0.001
y-velocity	0.001
Energy	0.001
K	0.001
Epsilon	0.001

Post Processing

Table-5: Graphics, Animation and Plots

Graphics & Animation	use contour option to get velocity magnitude, static pressure , total temperature and turbulence intensity contours
plots	use XY plots to get static pressure vs position ,velocity magnitude vs position and turbulence intensity vs position

6. RESULTS AND DISCUSSIONS

Following are the contour plots that were obtained
1)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. The velocity at the nozzle exit is 2400.32 m/sec, which is around Mach 3.03.

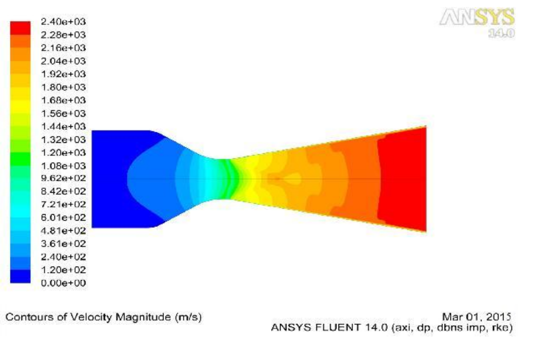


Fig.Contours of velocity

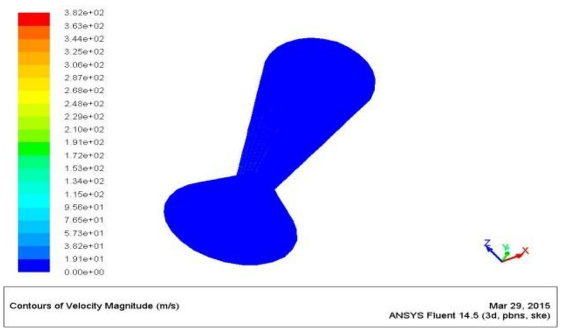


Fig.Contours of velocity magnitude

2) Temperature Contours: The temperature is maximum at the inlet and goes on decreasing till the outlet. The magnitude of temperature at the outlet is 1760.89 K.

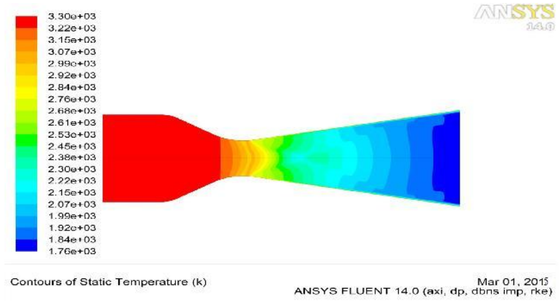


Fig.Contours of temperature

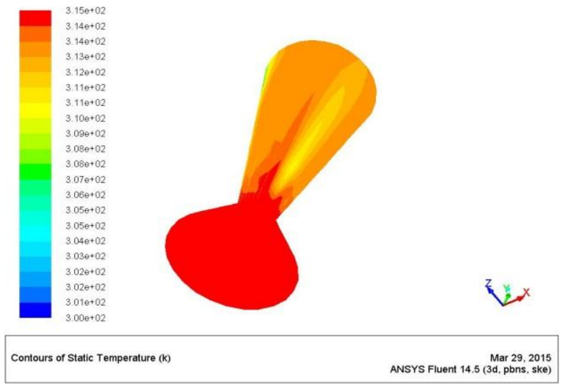


Fig.Contours of static temperature

3) Pressure Contours: The pressure is maximum at the inlet and goes on decreasing till the outlet. The static pressure at the outlet is 0.927 bar. There is sudden decrease in pressure due to shock wave just after the throat section.

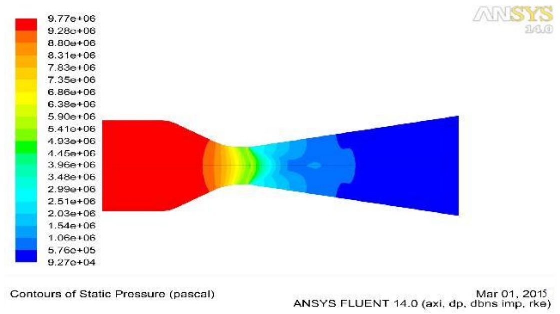


Fig.Contours of pressure

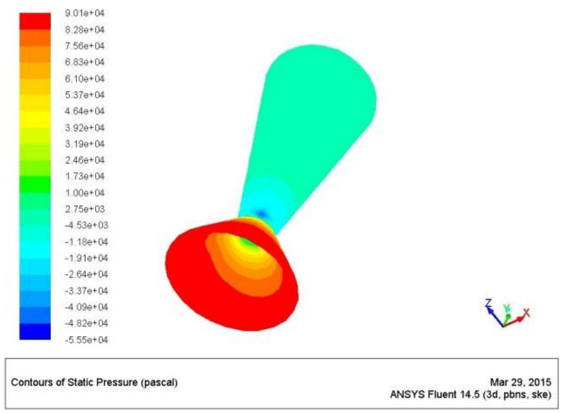


Fig.contours of static pressure

7. CONCLUSIONS

The results obtained by Computational Fluid Dynamics (CFD) are almost identical to those obtained theoretically. The tables below compare theoretical results to CFD results.

Table-6: Comparison of values in taken from theoretical and from CFD

Section	A_E/A_T	**Velocity**		**Temperature**		**Pressure**	
		Theoretical	**CFD**	**Theoretical**	**CFD**	**Theoretical**	**CFD**
convergent	1.50	169.71	163.2	3240.08	3248.90	87.33	92.86
convergent	1.20	482.91	469.32	3194.12	3220.79	79.89	82.15
throat	1.0	1030.46	1080.38	2972.98	2921.81	50.96	49.53
divergent	1.20	1335.28	1321.57	2760.50	2751.30	32.08	34.86
divergent	4.0	2051.23	2070.39	2137.56	2070.83	10.98	7.14
outlet	7.142	2387.52	2400.32	1724.90	1760.89	1.68	0.927

8. REFFERENCES

1. K. M. PANDEY , S. K. YADAV Journal of Environmental Research And Development Vol. 5 No. 2, October-December 2010 , "CFD ANALYSIS OF A ROCKET NOZZLE WITH TWO INLETS AT MACH 2.1"
2. Ms. B. Krishna Prafulla, Dr. V. Chitti Babu and Sri P. Govinda Rao International Journal of Computational Engineering Research , Volume 03 , Issue 5 "CFD Analysis of Convergent – Divergent Supersonic Nozzle"
3. Biju Kuttan P, M Sajesh ;The International Journal Of Engineering And Science (Ijes) Volume 2 ,Issue 2 , Pages: 196-207 , 2013, Issn: 2319 – 1813 Isbn: 2319 – 1805 "Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics"
4. Pardhasaradhi Natta, V. Ranjith Kumar, Dr. Y. V. Hanumantha Rao International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 Vol. 2, Issue 5, September- October 2012, pp.1226-1235 "Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (CFD)"
5. K.M. PANDEY , Member IACSIT and A.P. Singh , International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, August 2010 ISSN: 2010-0221 "CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software"
6. A.A. Khan and T.R Shem Bharkar, "Viscous Flow Analysis In A Convergent Divergent Nozzle" proceeding of the international conference on aerospace science and Technology, Bangalore, India, June 26-28,2008
7. H . K .Versteeg and W. Malala Sekhara An Introduction To Computational Fluid Dynamics, British library Cataloguing pup, 4th edition,1996
8. David C. Will Cox Turbulence Modeling for CFD second edition 1998
9. Kazuhiro Nakahahi " Navier Strokes Computations Of Two And Three Dimensional Cascade Flow Fields"