Modeling and simulation of Convergent-Divergent Nozzle Using Computational Fluid Dynamics

Article · July 2016

1 author:

PONUGUPATI NARENDRA MOHAN
Acharya Nagarjuna University

Modeling and simulation of Convergent-Divergent Nozzle Using Computational Fluid Dynamics

B.V.V. NAGA SUDHAKAR[1], B PURNA CHANDRA SEKHAR[2], P NARENDRA MOHAN[3], MD TOUSEEF AHMAD[4]

[1]*Department of Mechanical Engineering, University College of Engineering and Technology, Acharya Nagarjuna University, A.P, INDIA.*

[2,3,4] *Asst. Professor, Department of Mechanical Engineering, University College of Engineering- and Technology, Acharya Nagarjuna University, A.P, INDIA.*

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 conducted to analyze flow pattern of supersonic rocket nozzle at various degree of divergent angle, mach numbers etc. This paper aims to study the behavior of flow in convergent divergent nozzle by analyzing various parameters like pressure, temperature and velocity using computational fluid dynamics software(C.F.D).These results were further plotted comparing them with analytical values.

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. Swedish engineer of French descent who, in trying to develop a more efficient steam engine, designed a turbine that was turned by jets of steam. The critical component – the one in which heat energy of the hot high-pressure steam from the boiler was converted into kinetic energy – was the nozzle from which the jet blew onto the wheel. De Laval found that the most efficient conversion occurred when the nozzle first narrowed, increasing the speed of the jet to the speed of sound, and then expanded again. Above the speed of sound (but not below it) this expansion caused a further increase in the speed of the jet and led to a very efficient conversion of heat energy to motion. The theory of air resistance was first proposed by Sir Isaac Newton in 1726. According to him, an aerodynamic force depends on the density and velocity of the fluid, and the shape and the size of the displacing object. Newton‟ s theory was soon followed by other theoretical solution of fluid motion problems. All these were restricted to flow under idealized conditions, i.e. air was assumed to posses constant density and to move in response to pressure and inertia. Nowadays steam turbines are the preferred power source of electric power stations and large ships, although they usually have a different design-to make best use of the fast steam jet, de Laval‟s turbine had to run at an impractically high speed. But for rockets the de Laval nozzle was just what was needed.

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 analytical method is very complicated and difficult. The experimental methods are very costly. If any errors in the design were detected during the prototype testing, another prototype is to be made clarifying all the errors and again tested. This is a time-consuming as well as a cost-consuming process. The introduction of Computational Fluid Dynamics has overcome this difficulty as well as revolutionized the field of engineering. In CFD a problem is simulated in software and the transport equations associated with the problem is mathematically solved with computer assistance. Thus we would be able to predict the results of a problem before experimentation.

2. BASIC ISOMETRIC RELATIONS

The properties i.e. pressure, temperature and velocity at throat are find out by the following relations

$$\frac{p_t}{p_0} = \left[\frac{2}{(\gamma+1)}\right]^{\frac{\gamma}{\gamma-1}}$$

$$\frac{T_t}{T_0} = \left[\frac{2}{(\gamma+1)}\right]$$

$$v_t = \sqrt{\frac{2\gamma}{\gamma+1} RT_0} = \sqrt{\gamma RT_t}$$

Velocity and temperature values at different cross sections are by the following formulae:

The continuity equation is

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

The steady flow energy equation is as follows

$$\frac{Q-W}{m} = (h + \frac{V^2}{2} + gz)_t - (h + \frac{V^2}{2} + gz)_x$$

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)

γ - Adiabatic Index(C_p/C_v)

h – Enthalpy(J)

R – Specific gas constant(J/Kg K)

ρ – Density(Kg/m^3)

Q – Heat input to the system(J)

W – Work done by the system(J)

m^0 – Mass flow rate(Kg/s)

3. 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

Name	Tungsten
Symbol	W
Atomic Number	74
Atomic Mass	183-84 amu
Melting Point	3410.0°C (3683.15 K,6710.0° F)
Boiling Point	5660.0° C (59933.15K,10220.0°F)
Number of protons/Electrons	74
Number of protons	110
Classification	Transition Metal
Crystal Structure	Cubic
Density	19.3g/cm^3
Color	silver

4. COMPUTER SIMULATION

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

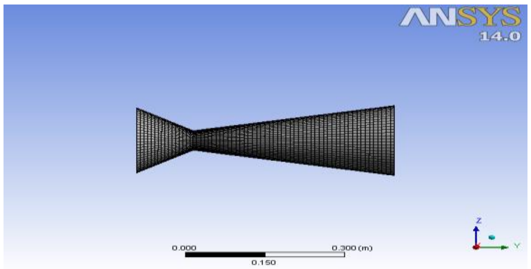
a. Modeling
b. Meshing
c. Pre-Processing
d. Solution
e. Post-Processing

a. 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 2.

b. Meshing

After modeling of the nozzle, its meshing was done using ANSYS ICEM CFD software. The mesh as created of trigonal elements with element size1mm 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 finer

.



c. 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.

d. Solution

The Solution Was Converged After 977 Iterations. And The Order Of Scaled Residuals Was Below

Solution controls Coutrant number : 5

Solution initialization Compute from : inlet

Run calculation number of iterations 2000

Residuals	Absolute Criteria
Continuity	0.001
X-Velocity	0.001
Y-Velocity	0.001
Energy	0.001
K	0.001
Epsilon	0.001

Table 1 Criteria of Convergence

e. Post Processing

Graphics & Animations	Contour option is used to plot velocity magnitude, Pressure, temperature, intensity
Plots	Use XY plots to get static pressure vs position Velocity vs velocity magnitude

Table 2 dimensions of the de Laval nozzle

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

5. Results and discussions

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.

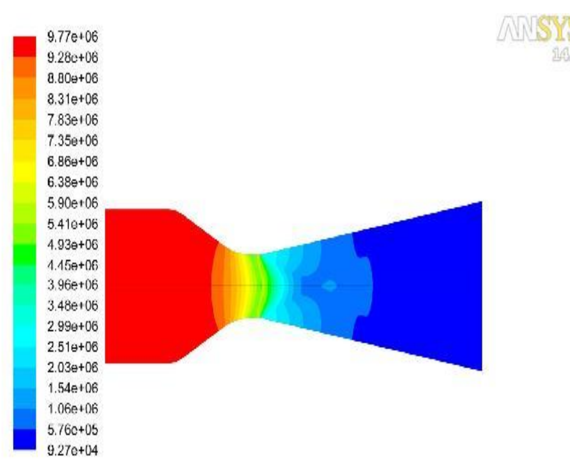


Fig: Pressure contours

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.

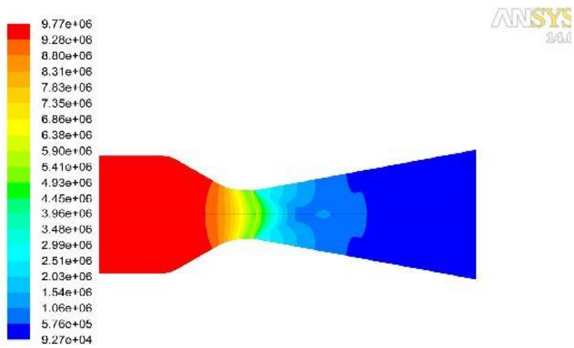


Fig: Temperature contours

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. The velocity at the nozzle exit is 2400.32 m/sec, which is around Mach 3.03.

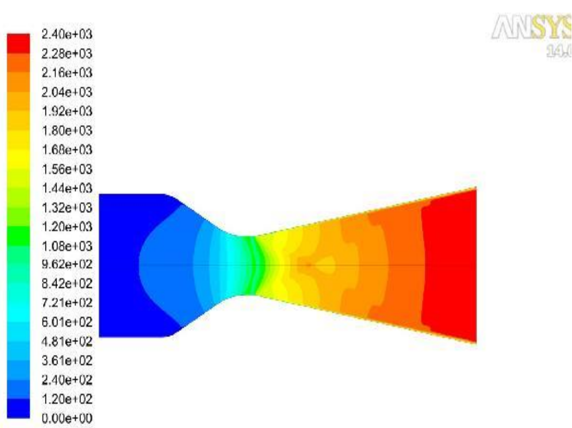


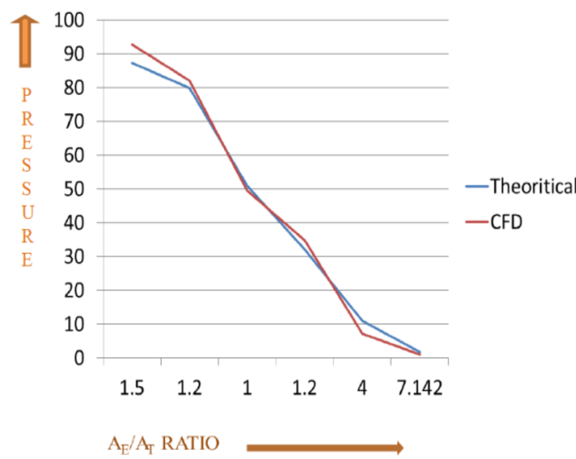
Fig: velocity contour

6. 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.

Comparison of pressure(bar)

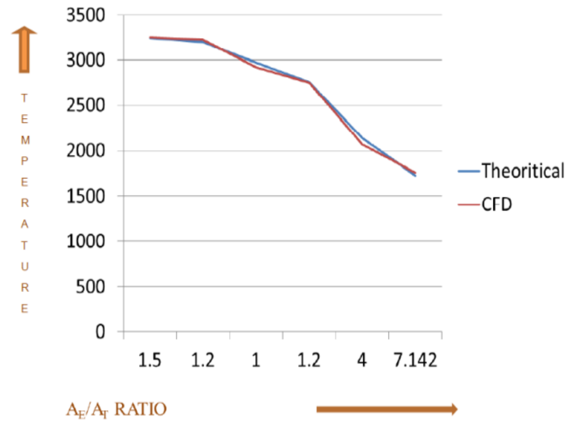
Section	A_E/A_T	Theoretical	CFD
Convergent	1.50	87.33	92.86
Convergent	1.20	79.89	82.15
Throat	1.0	50.96	49.53
Divergent	1.20	32.08	34.86
Divergent	4.0	10.98	7.14
Outlet	7.142	1.68	0.927



Graph between Area Ratio &Pressure

Comparison of Temperature(K)

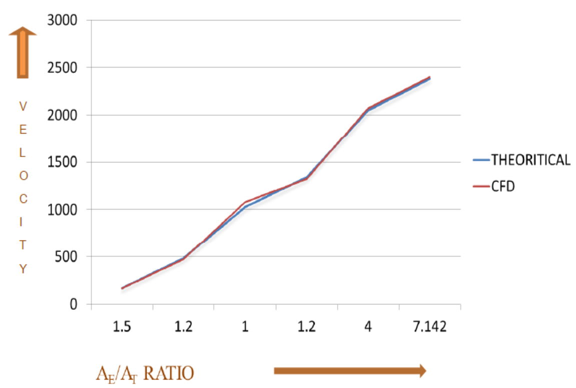
Section	A_E/A_T	Theoretical	CFD
convergent	1.50	3240.08	3248.90
convergent	1.20	3194.12	3220.79
throat	1.0	2972.98	2921.81
divergent	1.20	2760.50	2751.30
divergent	4.0	2137.56	2070.83
outlet	7.142	1724.90	1760.89



Graph between Area Ratio & Temperature

Section	A_E/A_T	Theoretical	CFD
convergent	1.50	169.71	163.2
convergent	1.20	482.91	469.32
throat	1.0	1030.46	1080.38
divergent	1.20	1335.28	1321.57
divergent	4.0	2051.23	2070.39
outlet	7.142	2387.52	2400.32

Comparison of velocity(m/s)



Graph between area Ratio & Velocity

References

[1]. Pardha saradhi 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

[2]. "Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (CFD)" . 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"

[3]. 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.

[4]. George P. Sutton and Oscar Biblarz, "Rocket Propulsion.

[5]. [K.Ramamurthi, "Rocket Propulsion", Macmillan publishers India, 2012 edition, (pp 54-89). Elements", A Wiley- Interscience Publication, Seventh Edition, 2001, (pp 1-99).