ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET) Vol. 3, Issue 1, January 2016

Design and Analysis of a Rocket C-D Nozzle

Kundan Kumar.K[1], Malathi.K[2], Elayakumar S[3#]

[1,2,3]*Department of Aeronautical Engineering, Dhanalakshmi Srinivasan college of Engineering and Technology.*
Chennai. tamilnadu
[1]*kabirkundan19@gmail.com*
[2]malathisaylu@gmail.com
[3]elayakumaryp@gmail.com

Abstract — de Laval nozzles are mechanical devices which are used to convert the thermal and pressure energy into useful kinetic energy. The values of temperature, pressure and velocity is needed at every section of the nozzle so as to design the shape, insulation and cooling arrangements for the nozzles. The aim of this project is to calculate the above parameters in theoretical formula and visualizing in the CFD analysis. The validation of these formulae is carried out using the Computational Fluid Dynamics (CFD) software ANSYS Fluent.

Keywords — De Laval nozzle, Theoretical equations, Computational Fluid Dynamics, ANSYS Fluent.

I. INTRODUCTION:

De laval nozzle was invented by Gustaf de Laval, a swedish inventor. This type of nozzles are most commonly used in supersonic vehicles and rockets. The type of nozzle is convergent-divergent nozzle, which is employed to provide supersonic velocity flows at the nozzle exit. In this journal, the theoretical analysis of de Laval nozzle is carried out by formulating required nozzle equations and verifying the results by using the CFD software(ANSYS FLUENT). First we will theoretically calculate the velocity, temperature and pressure at different cross sections of the nozzle using the formulae and then the results are verified by using simulation software.This provides a large number of advantages as we can able to visualize the flow in the nozzles.

Fig 1 .C-D NOZZLE

II. THEORETICALFORMULATIONOF NOZZLE:

The equations used here are for the findings of one dimensional nozzle flow. It helps in the idealization of full two or three dimensional flow equations and real aero thermochemical behaviour.

The Nozzle equation is derived using the continuity equation and steday state energy equations.

Nomenclature of symbols used are as follows:

T - Temperature (K)

P - Pressure (Pa)

Cp - Specific heat at constant pressure (J/kg K)

Cv - Specific heat at constant volume (J/kg K)

V - Velocity (m/s)

g - Gravitational acceleration (m/s²)

Z - Height (m)

A - Area (m²)

R - Specific gas constant (J/kg K)

h - Enthalpy (J)

γ - Adiabatic index

ρ - Density (kg/m³)

Q - Heat input to the system (J)

W- Work done by the system (J)

$\dot{m}$ - Mass flow rate (kg/s)

Consider a gas stored at a pressure Pc and at temperature T in the chamber.The gas completely expanded in a convergent-divergent nozzle.The gas is ideal,the process is adiabatic and expansion is isentropic.The condition is

All Rights Reserved © 2016 IJARTET

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET) Vol. 3, Issue 1, January 2016

assumed to be constant thus providing a steady mass flow rate.

The continuity equation is

$\rho_1 A_1 v_1 = \rho_2 A_2 v_2$

The steady flowEnergy Energy equation is as follows:

$$(h_1 - h_2) = (V_2^2 - V_1^2)/2$$

The following equations is derived using Continuity equation and Steady flow Energy equation:

$$\frac{A_x}{A_{th}} = \left(\frac{T_{th}}{T_x}\right)^{\frac{1}{\gamma-1}} \times \frac{\sqrt{\gamma R T_{th}}}{V_x} \quad (1)$$

$$C_p \times T_{th} + \frac{V_{th}^2}{2} = C_p \times T_x + \frac{V_x^2}{2} \quad (2)$$

Solving equation (1) and equation (2) simultaneously, we get the values of velocity (V_x) and temperature (T_x) at the required section of the nozzle. Pressure at the section can be calculated using isentropic laws.

TABLE I
THEORETICAL RESULTS

Section of nozzle	(Ax/Ath)	Velocity(m/s)	Temp.(K)	Pressure(bar)
Convergent	1.6	179.1	3260.2	88.33
Convergent	1.4	499.91	3694.21	75.77
Throat	1	1120.64	2675.89	48.98
Divergent	1.2	1239.68	2456.07	33.80
Divergent	3	2198832	2137.65	11.50
Outlet	7.26	2378.25	1765.59	1.39

III. ANSYS SIMULATION

CFD is an simulation tool which helps in finding of the experimental results.

The following steps were performed in CFD of nozzle: Modelling, meshing, pre-processing, solution, post-processing.

A.Modelling

The 2-Dimensional modelling of the nozzle was done using CATIA-V5 and file was saved in .stp format. The dimensions of the de Laval nozzle is given below.

TABLE II
NOZZLE DIMENSION

Parameter	Dimension
Total Nozzle Length (mm)	484
Inlet Diameter (mm)	166.6
Throat Diameter (mm)	34.5
Outlet Diameter (mm)	179.0
Chamber Length (mm)	99.93

All Rights Reserved © 2016 IJARTET

Convergent Angle (deg)	35
Divergent Angle (deg)	11.25
Throat Radius of Curvature (mm)	70
Convergent Radius of Curvature (ntm)	40

C.Pre-processing

Pre-processing of the nozzle made in ANSYS FLUENT. 2-Dimensional and double precision settings were used while reading the mesh. The mesh is scaled down because all dimensions are in mm. The mesh was checked in fluent software and no errors was founded out.And then the next process is carried out which is giving Boundary Conditions.

TABLE III
PROBLEM SETUP

General	Solver type: Density-based 2D Space: Axi--symmetric
Models	Energy equation:On Viscous model : Standard k-e model, realizable, enhanced wall treatment
Materials	Density: ideal gas Cp = 1880J/kg K Adiabatic index = 1.19 Viscosity= 8.983×10^{-5}Pa. s Thermal conductivity = 0.0142 W/Mk Mean molecular mass = 27.7 g/mol
Boundary conditions	Inlet Pressure = 100bar Inlet Temperature = 3300K Outlet Temperature = 1700K (For initialization purpose only)

TABLE IV
SOLUTION

Solution controls	Courant number = 5
Solution initialization	Compute from : Inlet
Run calculation	Check case No. of iterations: 2000 Click Calculation

All Rights Reserved © 2016 IJARTET

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET) Vol. 3, Issue 1, January 2016

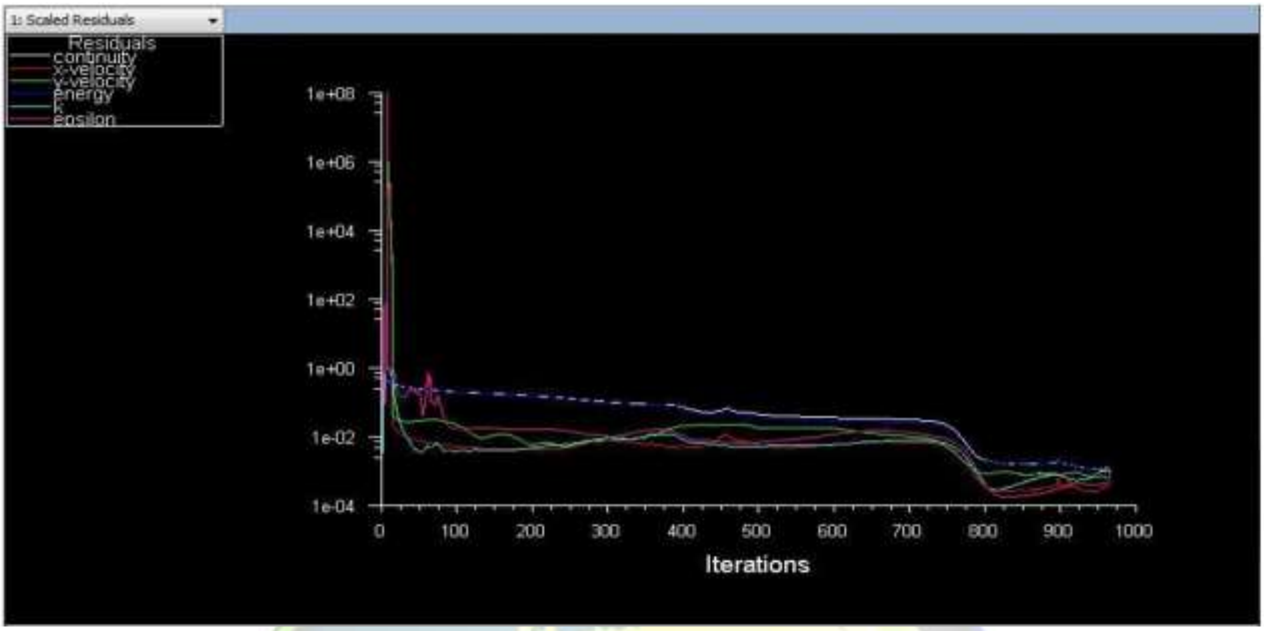

Fig. 3 Scaled Residuals

TABLE VI
PLOT SETUP

Graphics & Animation	Usingthe contour option to get the mach number contour, static pressure contour, total temperature contour, turbulent intensity contour
Plots	Using the XY plots for the mach number Vs position, static pressure Vs position plots

D.Results and Discussion

The contour plots of pressure,temperature and velocity are as follows

1) Velocity Contours:

The velocity contour is used so as to know the change in velocity from the inlet to the exit. The velocity is minimum at the inlet and goes on increasing till the nozzle exit. The maximum velocity at the throat is Mach 1. This is known as choked flow. The velocity at the nozzle exit is 2498.56m/s which is around Mach 3.05.

All Rights Reserved © 2016 IJARTET

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET) Vol. 3, Issue 1, January 2016

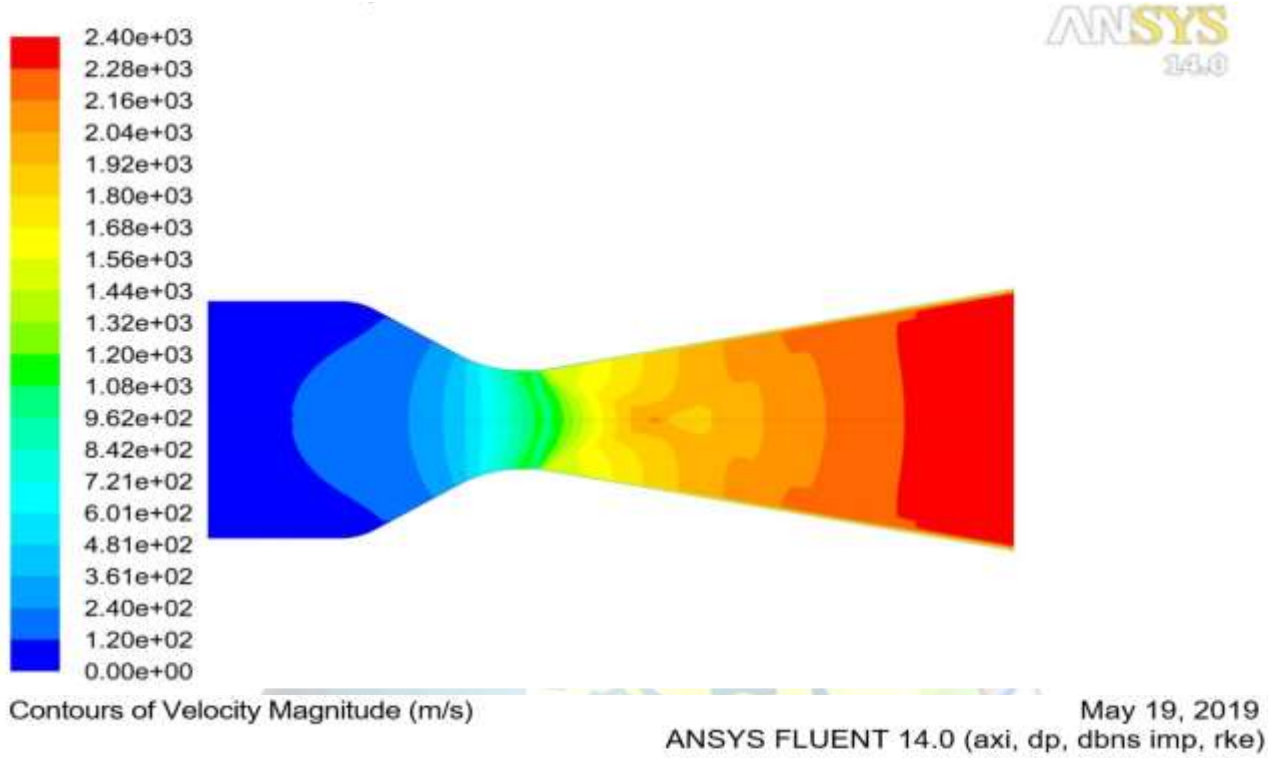

Fig. 4 Contours of Velocity Magnitude (m/s)

2) Temperature Contours: The temperature is very important factor which should be considered for the findings of cooling techniques, or else structural deformation occurs at the nozzle.The temperature is maximum at the inlet and minimum at the exit. The maximum temperature at the exit is 1801.8K.

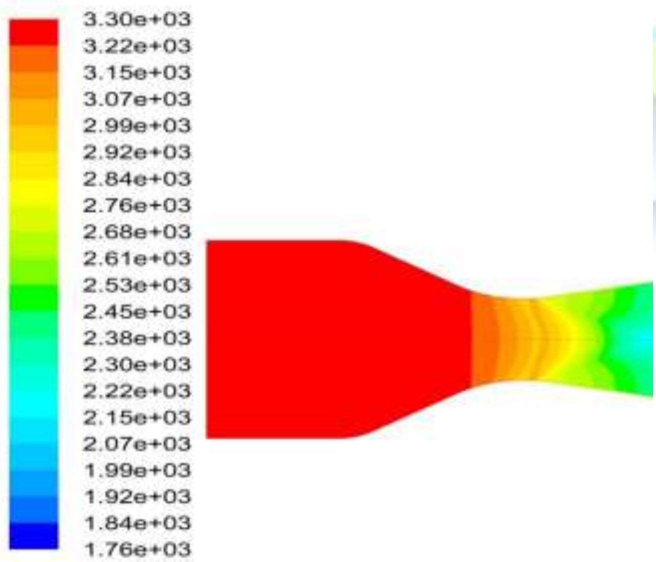

Fig. 5 Contours of Static Temperature (K)

CONCLUSION:

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

All Rights Reserved © 2016 IJARTET

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET) Vol. 3, Issue 1, January 2016

TABLEVII
VELOCITY COMPARISON

Section	(Ax/Ath)	Velocity (m/s)	
		Theoretical	CFD
Convergent	1.500	170.25	165.60
Convergent	1.2	482.91	469.32
Throat	1	1032.26	1079.38
Divergent	1.200	1335.28	1321.57
Divergent	4	2048.23	2086.39
Outlet	7.142	2387.52	2400.32

TABLEVIII
TEMPERATURE COMPARISON

Section	**(Ax/Ath)**	**Temperature (K)**	
		Theoretical	**CFD**
Convergent	1.500	3256.08	3248.90
Convergent	1.2	3269.12	3224.79
Throat	1	2989.98	2945.81
Divergent	1.200	2760.50	2751.30
Divergent	4	2137.56	2070.83
Outlet	7.142	1724.90	1760.89

TABLE IX
PRESSURE COMPARISON

Section	(Ax/&)	Pressure (bar)	
		Theoretical	CFD
Convergent	1.500	86.54	93.25
Convergent	1.2	81.65	84.46
Throat	1	50.96	49.53
Divergent	1.200	32.58	36.24
Divergent	4	10.98	7.14
Outlet	7.142	1.84	0.978

(1) The values of CFD and Theoratical methods are slightly different because the CFD software considers the factors like boundary layer effects, shock waves, velocity component and more, Therefore there is difference in the results.

(2) The difference in the results of theoretical calculations and CFD are quite small so they can be used for the calculation of the above parameters.

(3) From the Result it is clear that the one-dimensional nozzle analysis is enough to find out the nozzle performance therefore giving clarity for the selection of the nozzle.

REFERENCES

[1] A comprehensive project report titled “Computational analysis of a supersonic nozzle”, by TEAM18

All Rights Reserved © 2016 IJARTET

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET) Vol. 3, Issue 1, January 2016

[2] Quintao, Karla K, “Design and optimization of nozzle shapes for maximum uniformity of exit flow”[2012].*FIU electronic theses and dissertations.* Paper 779
[3] Mbuyamba, Jean-Baptiste Mulumba, “Calculation and design of supersonic nozzles for cold gas dynamic spraying using MATLAB and ANSYS FLUENT”[2013]
[4] Hagemann, Gerald, Hans Immich, Thong Nguyen and Gennady Dunmov,” Advanced Rocket nozzles”, *Journal of propulsion and power 14.5*(1998).
[5] “Water channel for supersonic flow investigation”, Chemical Engineering and Technology group, Bhabha Atomic Research centre
[6] “Design and optimization of de Laval nozzle to prevent shock induced flow separation”, Aeronautical Engineering Department, Hindustan University George P. Sutton and Oscar Biblarz, “Rocket Propulsion Elements”,
[7] A Wiley- Interscience Publication, Seventh Edition, 2001, (pp 1-99).
[8] K.Ramamurthi, “Rocket Propulsion”, Macmillan publishers India, 2012 edition, (pp 54-89).
[9] K.M.Pandey and S.K.Yadav, “CFD Analysis of a Rocket Nozzle with Two Inlets at Mach 2.1”, Journal of Environmental Research and Development, Vol 5, No 2, 2010, (pp 308-321).
[10] Yunus A. Çengel and John M. Cimbala, “Fluid Mechanics”, Tata McGraw-Hill New York, Second edition, (pp 853- 910).

All Rights Reserved © 2016 IJARTET