RESEARCHERID
THOMSON REUTERS
[ICESTM-2018]
ISSN 2348 – 8034
Impact Factor- 5.070
GLOBAL JOURNAL OF ENGINEERING SCIENCE AND RESEARCHES

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

M. Anand[1] & V.Gowtham Reddy[2]

[1&2]Department of Mechanical Engineering, Vikas College of Engineering and Technology, Nunna, Vijayawada, Andhra Pradesh, India

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 should be available at every section of the nozzle so as to design the nozzle shape, insulation and cooling arrangements. This paper aims at providing theoretical formulae to calculate the above. 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

A de Laval nozzle was invented by Gustaf de Laval, a Swedish inventor. It is a converging-diverging type of nozzle, generally employed to provide supersonic jet velocity at the exit of the nozzle. In this paper, analysis of de Laval nozzle is carried out theoretically by formulating required nozzle equations and the results have been validated by computer simulation using the CFD software ANSYS FLUENT. Firstly, velocity, temperature and pressure have been calculated theoretically at different cross-sections of the nozzle using the formulated equations. Secondly, the theoretical results are verified with the help of computer simulation approach.

II. THEORETICAL FORMULATION OF NOZZLE

The equations used below are for one dimensional nozzle flow. It corresponds to the idealization and simplification of full two or three dimensional flow equations and real aero-thermochemical behaviour.
The suffixes ch, th, x denote chamber, throat, and cross section at a particular length from the inlet of the nozzle respectively. Nomenclature of symbols used is as follows:
P – Pressure (Pa)
T – Temperature (K) V – Velocity (m/s)
g – Gravitational acceleration (m/s^2) z – Height (m)
A – Area (m^2)
Cp – Specific heat at constant pressure (J/kg K) Cv – Specific heat at constant volume (J/kg K) γ – Adiabatic index (Cp/Cv)
h – Enthalpy (J)
R – Specific gas constant (J/kg K)
ρ – Density (kg/m^3)
$\dot{Q}$ – Heat input to the system (J) $\dot{W}$ – Work done by the system (J)
m– Mass flow rate (kg/s)

ISSN 2348 – 8034
Impact Factor- 5.070

Consider a gas stored at temperatures T_C and pressure P_C in the chamber. The gas is completely expanded in a convergent-divergent nozzle. The gas is assumed to be ideal, the process is adiabatic and the expansion is isentropic. The chamber conditions are assumed to be constant thus providing a steady mass flow rate.

The continuity equation is

$$\rho_x \cdot A_x \cdot V_x = \rho_{th} \cdot A_{th} \cdot V_{th}$$

The steady flow energy equation is as follows:

$$\left(h + \frac{V^2}{2} + gz\right)_{th} - \left(h + \frac{V^2}{2} + gz\right)_x \qquad \frac{\dot{Q} - \dot{W}}{\dot{m}} =$$

The following equations have been derived using Continuity equation and Steady flow energy equation:

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

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

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	(A_x/A_{th})	Velocity (m/s)	Temp. (K)	Pressure (bar)
Convergent	1.500	169.72	3240.08	87.33
Convergent	1.2	482.91	3194.12	79.89
Throat	1	1030.46	2972.98	50.96
Divergent	1.200	1335.28	2760.50	32.08
Divergent	4	2051.23	2137.56	10.98
Outlet	7.142	2387.52	1724.90	1.68

III. COMPUTER SIMULATION OF NOZZLE

CFD is an engineering tool that assists experimentation.
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 are presented in the table given below.

ISSN 2348 – 8034
Impact Factor- 5.070

Table II nozzle dimensions

Parameter	Dimension
Total Nozzle Length (mm)	484
Inlet Diameter (mm)	166.6
Throat Diameter (mm)	34.5
Outlet Diameter (mm)	183.0
Chamber Length (mm)	99.93
Convergent Angle (deg)	32
Divergent Angle (deg)	11.31
Throat Radius of Curvature (mm)	70
Convergent Radius of Curvature (mm)	40

B. Meshing

After modelling of the nozzle, its meshing was done using ANSYS ICEM CFD software. The mesh was 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.

Patch dependent method was used for meshing. Mesh quality was above 0.3 after smoothening was applied

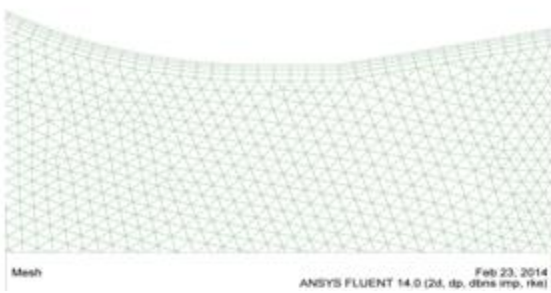

Fig. 2 Mesh (Close up view)

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.

Table III problem setup

General	Solver type: Density-based 2D Space: Axi-symmetric [3]
Models	Energy equation : On Viscous model : Standard k-ε model, realizable, enhanced wall treatment.
Materials	Density: ideal gas Cp = 1880J/kg K γ = 1.19 Viscosity = 8.983x10^{-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)

ISSN 2348 – 8034
Impact Factor- 5.070

Table IV solution

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

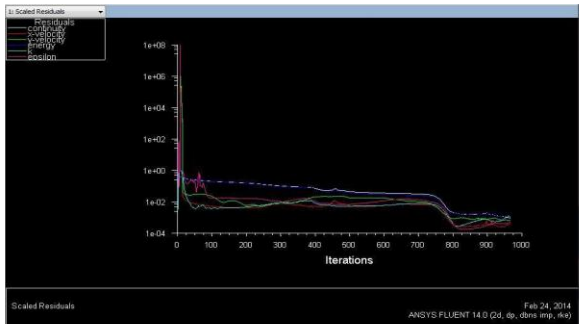

Fig. 3 Scaled Residuals

The criterion for convergence of the scaled residuals was set according to the table below:

Table V convergence criteron

Residual	Absolute criteria
continuity	0.001
x-velocity	0.001
y-velocity	0.001
energy	0.001
k	0.001
epsilon	0.001

The solution was converged after 977 iterations. And-3the order of scaled residuals was below 10e

Table VI plot setup

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

IV. RESULTS AND DISCUSSION

The axis was mirrored. 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.

ISSN 2348 – 8034
Impact Factor- 5.070

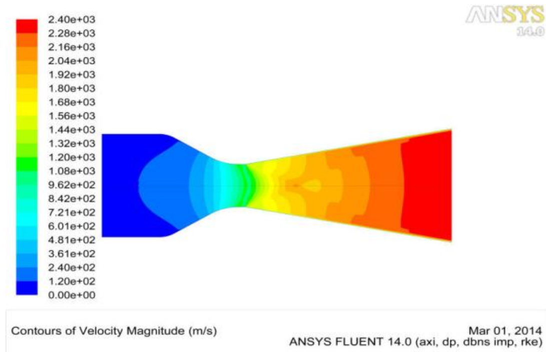

Fig. 4 Contours of Velocity Magnitude (m/s)

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.

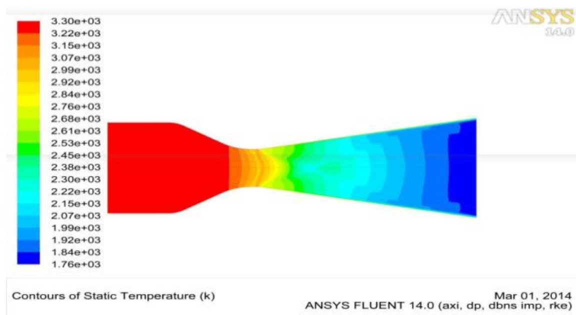

Fig. 5 Contours of Static Temperature (K)

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 .
4)

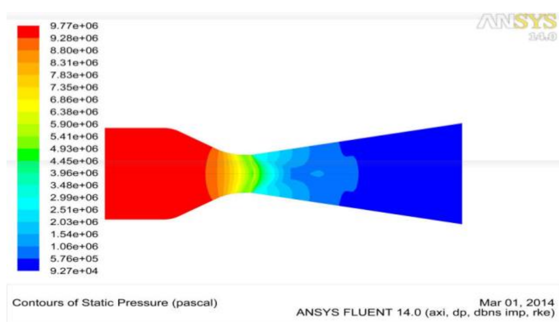

Fig. 6 Contours of Static Pressure (pascal)

V. 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.

Table VII velocity comparison

Section	(A_x/A_{th})	Velocity (m/s)	
		Theoretical	**CFD**
Convergent	1.500	169.72	163.20
Convergent	1.2	482.91	469.32
Throat	1	1030.46	1080.38
Divergent	1.200	1335.28	1321.57
Divergent	4	2051.23	2070.39
Outlet	7.142	2387.52	2400.32

Table VIII temperature comparison

Section	(A_x/A_{th})	Temperature (K)	
		Theoretical	**CFD**
Convergent	1.500	3240.08	3248.90
Convergent	1.2	3194.12	3220.79
Throat	1	2972.98	2921.81
Divergent	1.200	2760.50	2751.30
Divergent	4	2137.56	2070.83
Outlet	7.142	1724.90	1760.89

Table IX Pressur omparison

Section	(A_x/A_{th})	Pressure (bar)	
		Theoretical	**CFD**
Convergent	1.500	87.33	92.86
Convergent	1.2	79.89	82.15
Throat	1	50.96	49.53
Divergent	1.200	32.08	34.86
Divergent	4	10.98	7.14
Outlet	7.142	1.68	0.927

(1) CFD considers the factors like boundary layer effects, shock waves, radial velocity component and so on, which leads to some variance from theoretical results.
(2) The variation in the results of theoretical calculations and CFD are quite insignificant.
(3) It thus establishes the fact that one-dimensional simplified nozzle analysis is sufficient to predict the nozzle performance.

VI. ACKNOWLEDGEMENT

Sincerely grateful to The Chairman of VCTN, Sri N.Narsi Reddy for presenting them with a rare opportunity to undertake such a highly coveted project.
The valuable guidance and suggestions received from V.Gowtham Reddy.

The appreciate suggestions from M.V.U.Bhaskara Rao, Head of Mechanical Engineering Department, VCTN.

ISSN 2348 – 8034
Impact Factor- 5.070

REFERENCES

[1] *George P. Sutton and Oscar Biblarz, "Rocket Propulsion Elements", A Wiley- Interscience Publication, Seventh Edition, 2001, (pp 1-99).*
[2] *K.Ramamurthi, "Rocket Propulsion", Macmillan publishers India, 2012 edition, (pp 54-89).*
[3] *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).*
[4] *Yunus A. Çengel and John M. Cimbala, "Fluid Mechanics", Tata McGraw-Hill New York, Second edition, (pp 853- 910).*
[5] *Biju Kuttan P and M Sajesh, "Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics", The International Journal Of Engineering And Science (Ijes), Volume 2, No 2, 2013, pp 196-207.*