International Journal of Scientific Research and Engineering Development-– Volume 4 Issue 3, May -June 2021

Available at ***www.ijsred.com***

RESEARCH ARTICLE OPEN ACCESS

Study Of Flow Characteristics In a Convergent And Divergent Nozzle Using Computational Fluid Dynamics

JITHENDRA SAI RAJA CHADA *SRI RAM DEEPAK AKELLA**

SATISH CHARAN JALLI*** SAI SRI HARI CHALLA****

*-**** Department of Mechanical Engineering, Pragati Engineering College, Kakinada, Andhra Pradesh, India

--************************----------------------------------

Abstract: This paper is experimental result of flow simulation in Convergent-divergent nozzle whose convergent angle is 45deg and divergent angle is 15deg and placed an pintle at the throat of the nozzle we have studied the flow using computation fluid dynamics by varying the inlet Mach parameter from 1,1.25,1.3,1.48,1.6and with an low area ratio observed the characteristics of flow in the considered nozzle and also observed the outlet Mach , max temperature , max and min pressure , and max velocity of the simulated result

Keywords: the first keyword, the second keyword, the third keyword, the forth keyword, the fifth keyword, the sixth keyword

--************************----------------------------------

INTRODUCTION:

A nozzle is the simplest structural conical tube in which hot gases flow. In the convergent and divergent nozzle, the high-temperature gases at exhaust with high velocity and pressure the nozzle converges down to the smaller area which was described as throat. The throat size has been considered based on the choke of the flow which sets the mass flow rate through the system then it expanded through the divergent section to the exit. hot Gas travels from the chamber to the converging portion and then travels through the throat and then passes through the diverging portion, and then ejected into the atmosphere. the ambient pressure is known as the backpressure. In the CD nozzle, the pressure and area decrease at the convergent portion of the nozzle as the velocity increases. coming to the divergent part velocity decreases, remaining two quantities will increase. A CD is frequently used to proselytize chemical energy into kinetic energy in a thermal chamber. The flow through the CD nozzle is isentropic. The sudden change of supersonic flow to subsonic flow occurs through a plane of discontinuity. The plane of discontinuity is called a shock wave, the flow-through shock is no longer isentropic. The great Swedish engineer Gustaf de Laval implemented his converging-diverging nozzle design for the working of his impulse turbine in the year 1888. Laval's Convergent-Divergent nozzle was first implemented in the rocket engine by Robert Goddard. The main aim of the nozzle is to rise the kinetic energy of the streaming medium, the loss of energy, and its pressure. In the case of the excessive pressure ratio of the nozzle, the fluid gains sonic velocity. The nozzle design plays a key role in giving rise to the thrust. Mass-flow rate, exhaust gas speed, and nozzle pressure determine the quantity of the trust generated.

1.Literature Review:

Mohammad Arif Hussain et al (2019)had studied the various flow the behavior of various convergent-divergent nozzles had concluded that the triangular convergent-divergent nozzles are the best(1). Muhammad Arif Budiyanto et al (2019) had studied convergent and divergent-convergent nozzle of water jet propulsion and concluded that The efficiency of the convergent nozzle is bigger than the combination nozzle (2). Raghu Ande et al (2018) had studied the effect of the divergent angel in the convergent-divergent nozzle had been concluded that the Mach number of optimum value is obtained at the divergent angle of 15°(3). Arun Kurien Reji, et al (2018) had studied the various supersonic flow in the convergent and divergent nozzle had concluded as the fluid flow inlet Mach number is low even then the existing manch number flow is supersonic speeds described that the design was well developed (4). kousik kumaar. R et al (2016) had studied the various shock ways in the supersonic convergent and divergent nozzle had been concluded that shockwave will passes outside to inside not only affected by NPR but also affected by inlet pressure it can reduce by increasing the ratio between inlet and outlet pressure(5). M.H.M. Noh, et al(2011) had studied the chocks in the convergent and divergent nozzle for the thruster application concluded that separation of flow from nozzle wall cannot be eliminated by reducing the divergence angle while maintaining the area ratio.

Also, an increase in viscous loss reduces the strength of normal shock inside the nozzle(6). G.V M Krushnarao Kotteda et al (2017) had studied the flow in a planar convergent-divergent nozzle had been concluded that the bleed is most effective in the low to moderate NPR regime effect of the throat geometry, including the discontinuity in slope, on the flow is investigated. Four different nozzle geometries with the same area ratio and length are chosen(7). Bogdan-Alexandru Belega et al (2015)had studied the flow in the convergent and divergent nozzle there design model created based on an existing parameter is in accord with the scope(8). Andreas K. Flock et al (2019) had studied the design of convergent and divergent nozzle with a constant radius of the center body had been come to conclude that Displacing the center body in angular and translational direction led to higher pressure and lower Mach number on that side where the center body was moved to. An angular displacement of 0.5◦ and a translational displacement of 4 mm was approximately equal in their impact(9). E. M. S. Ekanayake, et al (2009)had studied the simulation of a two dimensional rectangular supersonic convergent-divergent nozzle had concluded that the nozzle: pressure ratio increases the flow separation zone increases in size and moves from one wall to the opposite and back before ultimately exiting the nozzle. This transition from one wall to the other is due to the build-up of turbulent kinetic energy on the opposite wall(10). Ambareen Khan, et al (2018)had studied the flow in convergent and divergent nozzle based on the pressure control using micro-jets had been concluded that As pressure decreases velocity will increase and our results proved that the velocity is high at the exit and variation of pressure inlet to the outlet is observed by considering Mach number(11). Madhu B P, Syed Sameer et al (2017)had studied the flow in convergent-divergent nozzles and contour nozzles had concluded that the conical nozzle gives the best output that contour(12). nozzle Ric Gamble et al (2004) had nozzle section and design criteria had concluded that the designed for the appropriate air vehicle mission. As required aircraft speeds increase, so do functional requirements for the nozzle, resulting in an increasingly complex nozzle(13). H Pujowidodo, et al (2018) had studied the Converging-Diverging Nozzle for Improving the Impulse Momentum of Cross-Flow Turbine in a Bio-Micro Power Plant had been concluded that momentum flux for rotating power turbine using integration thermodynamics and CFD models has shown that the smallest pressure ratio will increase the supersonic flow pass through the CD-nozzle(14). jithendra sai raja Chada et al (2020) had studied the conceptual design and shape optimization of the pintle nozzle of a rocket had concluded that pintle has better performance among the considered models. The performance may be improved by varying the shape of the pintle. It can also be done by varying the surface modifications to the nozzle and pintle(15).

2.Methodology:

The working of the nozzle is to drive the gases faster produced by the propellant to higher velocity to obtain higher thrust. the percent of thrust is correlated to the rate of flow of mass through the engine, the exit velocity of the rate of flow through the engine the value of three of these variables are all determined by the structure of the rocket nozzle.For a direct functioning rocket propulsion system moving through a corresponding atmosphere the mount of total thrust and specific impulse represented by:

$$F = \dot{m} g_e + (p_e - p_o).A_e \rightarrow (1)$$

I. $$I_{sp} = \frac{F}{\dot{m} g_0} \rightarrow (2)$$

Area of throat	$A_t = \dfrac{\dot{m}}{p_c\sqrt{\gamma\left(\dfrac{2}{\gamma+1}\right)^{\gamma+1/\gamma-1}\dfrac{M}{R_u T_c}}} \rightarrow (3)$
Exist area	$A_e = \pi r_e^2 \rightarrow (4)$
The convergence area in the nozzle is given	$A_c = 3A_t \rightarrow (5)$
Area ratio is given as	$\varepsilon = \dfrac{A_e}{A_t} \rightarrow (6)$
The radius of the throat is given by	$r_t = \sqrt{\dfrac{A_t}{\pi}} \rightarrow (7)$
The length of the diverging nozzle is	$l_{dn} = \sqrt{\dfrac{A_c}{\pi}}\dfrac{1}{\tan\theta} \rightarrow (8)$
Length of the converging nozzle	$l_{cn} = \sqrt{\dfrac{A_c}{\pi}}\dfrac{1}{\tan\beta} \rightarrow (9)$

Calculated values for nozzle dimensions and geometry

The design values of the convergent-divergent nozzle structure are obtained through the following equations which are used commonly in every spacecraft during designed nowadays For altitude (100 km or higher) area expansion in the nozzle, given by equation '6'

Let us consider the are of the nozzle throat as:

$$A_t = 0.070685 m^2 \rightarrow (10)$$

Now by considering the value of $\varepsilon = 14.0415$ by using the formula we will get $A_e = 0.58223 m^2 \rightarrow (11)$

From equation '4'

That gives $r_e = 0.4305 m \rightarrow (12)$

And by the formula

From the equation '5'

$$A_c = 0.212055 m^2 \rightarrow (13)$$

The radius of the throat

From equation '7'

$$r_t = 0.1521 m \rightarrow (14)$$

Convergent radius:

$$r_c = \sqrt{\frac{A_c}{\pi}} \rightarrow (15)$$

$$r_c = 0.2598060 m \rightarrow (16)$$

By considering the angle of the convergence angle as $\beta = 45^{\circ} \rightarrow (17)$ and the divergence angle as $\theta = 15^{\circ} \rightarrow (18)$

And also considered the radius of curvature at the throat is 0.228m

The radius of the throat	**0.1521 m**
Area at throat	0.070685 m^2
The radius of convergent radius	0.2598060 m
Area at convergent	0.212055 m^2
Radius of exist	0.4305 m
Area of exist	0.58223 m^2
Convergence angle	45^{o}
Divergent angle	15^{o}
The radius of the curvature at the throat	0.228 m

Fromm equation '10', '11', '12', '13', '14', '16', '17', '18'

Table 2:- total values for design

The nozzle model for flow is simulated in Solid-works 2020. Solid-works by the governing under the Navier stokes equations to simulate the taken geometry. In the stream of engineering study, fluid dynamics plays a topmost role, similarly, in fluid dynamics, Bernoulli's theorem is a high obvious concept. This theorem is enunciated by Swiss mathematician Daniel Bernoulli. He described that if the flow of the fluid is level i.e., not encounter any change in gravitational potential energy then a decrease in the pressure of the fluid is harmonized with the velocity rise of the fluid. He described this theorem as during the change, the entire mechanical energy of the streaming fluid having the energy-related to fluid pressure, the potential energy of elevation of gravitation, and fluid kinetic energy been unchanged. Energy conservation is the basic idea to rise in Bernoulli's theorem. If the fluid is streaming between a horizontal tube of unlike a cross-sectional area, the fluid exerts a minimum pressure at the smallest area and vice versa.

$$p_1 + \frac{1}{2}\rho v_1^2 + \rho g y_1 = p_2 + \frac{1}{2}\rho v_2^2 + \rho g y_2 \rightarrow (19)$$

Where is described as the pressure

The ρ is described density of the fluid

The gravitation is denoted with the g

Y is described as the height from the plane

Bernoulli's equations have the drawback at a steady flow, incompressible flow, frictionless flow, and flow along streamline. As the common practical problems can be analyzed by using extended Bernoulli's Equation. As follow

$$p_1 + \frac{1}{2}\rho v_1^2 + \rho g y_1 = p_2 + \frac{1}{2}\rho v_2^2 + \rho g y_2 + H_L \rightarrow (20)$$

where H_L is described friction on wall's head loss

The Navier-Stokes equation has named after the great scientists Claudelouis Navier and George Gabriel Stokes. this equation came into performance by the principal newton's second law of motion. the scientific and the engineering stream Navier-Stokes equation plays a peculiar and functional role. The condition between all of the pressure, temperature, density, the velocity of a streaming fluid can be observed. Navier-Stokes equations are the fusion and the add-on of Euler's equations which dels with the viscosity of the stream fluid. Continuity, three time-dependent, three time-independent equations form the Navier-Stokes equation.

$$\rho \frac{\partial v}{\partial t} = -\nabla p + \rho g + \mu \nabla^2 v \rightarrow (20)$$

These equations are well combined sometimes with additional equations called energy equations to solve critical applications.'This equation is governing equation for all fluid flow

In a nozzle, the gases pass through high pressure and low velocity. The C-D nozzles in the rocket are used to boost up the exhaust gases to obtain high thrust. The model calculations are done and the design is produced in solid works 2020

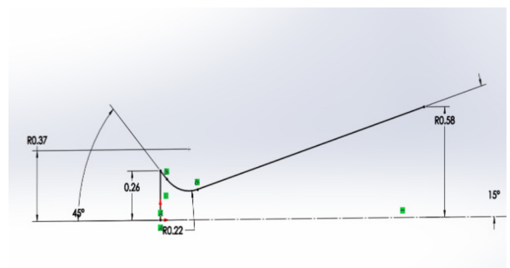

Figure 1: 2d sketch of the nozzle

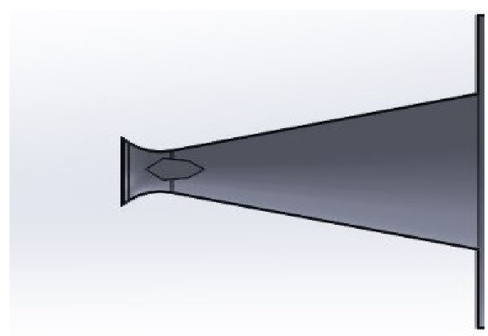

Figure 2: the cad part of the nozzle

After creating the cad part the flow simulation is done with help of the flow simulation workbench in solid works 2020 the figure 1 represents the 2 d sketch of the nozzle which is to revolve and produced as solid. a pintle is placed in the thought portion of the nozzle for obtaining best performance of the nozzle as the boundary condition the fluid which is air is entering into the nozzle at a Mach number of

1,1.25,1.3,1.48,1.6 varied respectively and observed the flow variation in the C-D nozzle the fine meshing is done and the flow boundary condition are been given such as the rocket is performing at ground level so the ambient pressure is been considered as 1 atm pressure, all the wall of nozzle are been considered as real wall the fluid or the gs is considered as air due to similar to it required properties all of this consideration are needed to analysis the flow simulation.

4.RESULTS AND DISCUSSION:

A 3d cad model is prepared and analyzed with the help of the flow simulation Solidworks workbench by varying the inlet mach from 1,1.25,1.3,1.48,1.6 and the thermodynamic parameters are been noted in the bellow table 3

Inlet Mach no.	Outlet mach no	max velocity	Max pressure	min pressure	max temperature
1	2.254	703.994m/s	316562.108 Pa	25559.718 Pa	424.842 K
1.25	2.665	779.993m/s	412485.647 Pa	16666.676 Pa	473.183 K
1.3	2.849	782.049 m/s	433580.705 Pa	17378.319 Pa	483.801 K
1.48	3.064	904.044 m/s	515498.749 Pa	16595.182 Pa	534.372 K
1.6	4.025	1413.468 m/s	575379.041 Pa	8679.348 Pa	565.650 K

Table 3: this table is value of outlet mach ,max velocity ,max pressure ,min pressure and max temperature with respective to inlet mach

The graph between the Mach numbers and other parameters.

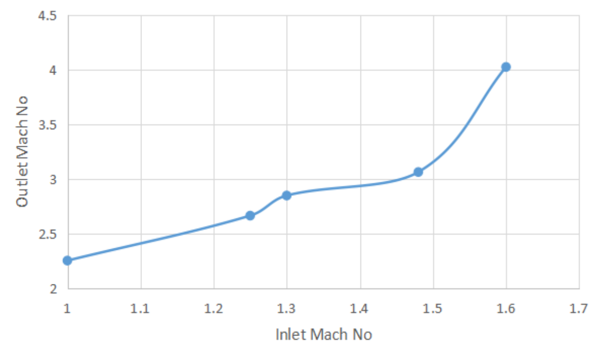

Figure 3 inlet Mach vs outlet MACH

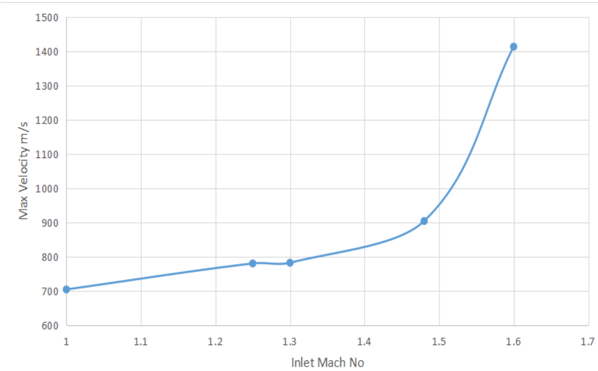

Figure 4 inlet Mach vs max velocity

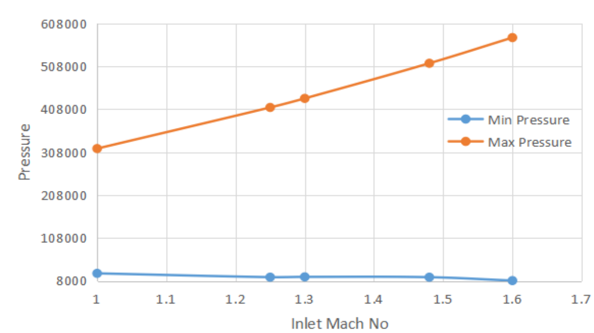

Figure 5 inlet Mach vs min and max pressure

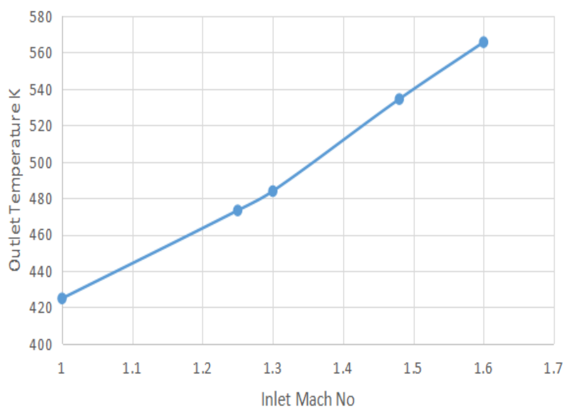

Figure 6 inlet Mach vs outlet temperature

In the figure 3 is the graph between the outlet Mach and the inlet Mach by observing that the inlet Mach number from 1.5 to 1.6 there was rapid a change of Mach 3 to 4 resepectively

In the figure 4 is the graph between the inlet mach and the max velocity by observing that the inlet Mach number from 1.5 to 1.6 there was rapid a change of max velocity of 900m/sec to 1400m/sec resepectively

In the figure 5 is the graph between the inlet mach and the max & min pressure by observing that the inlet Mach number the value of min pressure decreases but the change in the max pressure is optimally linear.

In the figure 6 is the graph between the inlet mach and the temperature by observing that the inlet Mach number from 1.3 to 1.5 there was a rapid change of temperature of 480k to 540k resepectively.

5.Conclusion:

The work aimed to analyze the flow behavior along in a convergent and divergent nozzle using Computational Fluid Dynamics. The Nozzle have shown better performance with respect to the inlet mach flow. As the inlet mach number increases the outlet mach number, Maximum velocity will be increased, Max pressure are increasing and minimum pressure is decreasing with in the considered boundary conditions.

6.REFERENCES:

[1] Mohammad Arif Hussain, M.Satya Prasad, Dr. S.Nagakalyan(2019) DESIGN AND FLOW ANALYSIS OF VARIOUS CONVERGENT - DIVERGENT NOZZLES BY USING COMPUTATIONAL FLUID DYNAMICS (CFD)-International Journal of Innovative Research in Advanced Engineering (IJIRAE) ISSN: 2349-2163 Issue 05, Volume 6 (page number 372 to 379) doi://10.26562/IJIRAE.2019.MYAE10087.

[2] Muhammad Arif Budiyanto, Jansen Novri, Muhammad Idrus Alhamid, Ardiyansyah Analysis of Convergent and Divergent-Convergent Nozzle of Waterjet Propulsion by CFD Simulation-AIP Conference Proceedings 2062, 020066 (2019) https://doi.org/10.1063/1.5086613

[3] Raghu Ande, Venkata N. Kumar Yerraboina Numerical Investigation on Effect of Divergent Angle in Convergent-Divergent Rocket Engine Nozzle-CHEMICAL ENGINEERING TRANSACTIONS VOL. 66, 2018 from(page number 787 to 792) DOI: 10.3303/CET1866132

[4] Arun Kurien Reji, G. Kumaresan, Archit S Menon, Jithesh Parappadi, Harikrishna A. P, Ajaykrishnan Mukundan International Journal of Pure and Applied Mathematics Volume 119 No. 12 2018, 2135-2142 (page number 2135 to 2142)

[5] kousik kumaar. R, kesavan.M DESIGN AND CFD ANALYSIS OF SHOCK WAVES OVER SUPERSONIC CD NOZZLE international journal of latest trends in engineering and technology (IJLTET) ISSN:2278-621X (page number 502 to510)

[6] M.H.M. NOH, A.H.A. HAMID, R. ATAN, AND H. RASHID NUMERICAL INVESTIGATION OF CHOCKED CONVERGING-DIVERGING NOZZLES FOR THRUSTER APPLICATIONIIUM Engineering Journal, Special Issue, Mechanical Engineering, 2011(page number 10to 18)

[7] V M Krushnarao Kotteda· Sanjay Mittal Flow in a planar convergent-divergent nozzle Article in Shock Waves · May 2017 DOI: 10.1007/s00193-016-0694-4

[8] Bogdan-Alexandru Belega, Trung Duc Nguyen ANALYSIS OF FLOW IN CONVERGENT-DIVERGENT

ROCKET ENGINE NOZZLE USING COMPUTATIONAL FLUID DYNAMICS -INTERNATIONAL CONFERENCE of SCIENTIFIC PAPER AFASES 2015 Brasov, 28-30 May 2015

[9] Andreas K. Flock· Ali Gülhan Design ofconverging-diverging nozzles with constant-radius center body CEAS Space Journal (2020) 12:191–201 (page number 191 to 201) https://doi.org/10.1007/s12567-019-00286-4

[10] E. M. S. Ekanayake, J. A. Gear, Y. Ding Flow simulation of a two dimensional rectangular supersonic convergent-divergent nozzle ANZIAM J. 51 (EMAC2009) (page number C377–C392), 2010

[11] Ambareen Khan, Abdul Aabid, S. A. Khan CFD analysis of convergent-divergent nozzle flow and base pressure control using micro-JETS International Journal of Engineering & Technology, 7 (3.29) (2018)(page number 232-235)

[12] Madhu B.P, Syed Sameer, Kalyana Kumar M, Mahendra Mani G CFD ANALYSIS OF CONVERGENT-DIVERGENT AND CONTOUR NOZZLE International Journal of Mechanical Engineering and Technology (IJMET) Volume 8, Issue 8, August 2017, pp. 670–677, Article ID: IJMET_08_08_073

[13] Gamble, Dwain Terrell, P.E. Rich DeFrancesco NOZZLE SELECTION AND DESIGN CRITERIA AIAA-2004-3923

[14] H Pujowidodo, A I Siswantara2, G G R Gunadi, A Daryus The Study of Converging-Diverging Nozzle for Improving the Impulse Momentum of Cross-Flow Turbine in a Bio-Micro Power Plant ICB2018 IOP Conf. Series: Earth and Environmental Science 209 (2018) 012054 IOP Publishing doi:10.1088/1755-1315/209/1/012054

[15] JITHENDRA SAI RAJA CHADA, A. PHANI BHASKAR, PRIYA CHANDINI RAGU STUDY ON CONCEPTUAL DESIGN AND SHAPE OPTIMIZATION OF PINTLE NOZZLE OF A ROCKET i-manager's Journal on Mechanical Engineering, Vol. 10 No. 4 ,August - October 2020 (page number 14 to 21)

Biographical notes

Mr. Jithendra Sai Raja Chada is an under graduate student pursuing his Bachelor's Degree (BTech) in Mechanical Engineering at Pragati Engineering College. He has published 14 papers in referred international journals. His current research interests include CFD, Industrial Design, Nano Materials, Manufacturing Processes, Heat Transfer, Additive Manufacturing, Advanced Machining, Energy Recovery Systems.

Mr. Sri Ram Deepak Akella is an under graduate student pursuing his Bachelor's Degree (BTech) in Mechanical Engineering at Pragati Engineering College.His current research interests include CFD, Industrial Design, Heat Transfer.

Mr. Satish Charan Jalli is an under graduate student pursuing his Bachelor's Degree (BTech) in Mechanical Engineering at Pragati Engineering College. He has published 7 papers in referred international journals. His current research interests include CFD, Industrial Design, Heat Transfer.

Mr. Sai Sri Hari Challa is an under graduate student pursuing his Bachelor's Degree (BTech) in Mechanical Engineering at Pragati Engineering College. He has published 7 papers in referred international journals. His current research interests include CFD, Industrial Design, Heat Transfer.