International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 05 | May - 2022 Impact Factor: 7.185 ISSN: 2582-3930

Analytical investigation over the Four Inlet C & D Nozzle Using Ansys

SRI RAM DEEPAK AKELLA*[1] **GOLLAPALLI ARAVIND[2]**

[1]Undergraduvate research scholar Department of Mechanical Engineering, Pragati Engineering College, Kakinada, Andhra Pradesh, India
[21]Undergraduvate Student Department of Mechanical Engineering, Pragati Engineering College, Kakinada, Andhra Pradesh, India

---***---

Abstract – The engine's interpretative thrust production, such as nozzles, has been restructured for higher staging. Modern expansions of combustion systems, such as rocket nozzles, will be adjusted to meet the needs of today's applications. In this paper an analytical study of fluid flow in a convergent-divergent nozzle with 4 inlets the nozzle is design with a convergent angle of 45 and a divergent angle of 15. we have observed the CFD of the C-D nozzle by swerving the inlet Mach parameter from 2,2.5,3,3.5,4,4.5,5 This study observed the usage of the usage of the four inlets in the convergent and the divergent nozzle rather that a single inlet the study is caried out in the in the 2-D Simulation the Design of the nozzle is performed in the Solid-Works software and the simulation is performed in the Ansys fluent Software. The counter of the flow is been observed. The results concluded that in the analytic study the usage of the 4-Inlet is better than the single inlet.

***Key Words*:** Ansys, convergent-divergent nozzle, nozzle, CFD, Mach number

1.INTRODUCTION

Rocket motors generate purport by scorching liquid or solid propellants, or hybrid propellants. Rocket motor impulse divaricate from its air-breathing mainly in that rocket have autonomous propellants. Chemical rocket impulse systems are restricted into two typical types as stated in whether they burn propellants stored as solid or as a liquid. The rocket motor uses a nozzle to ripen hot exhaust to generate thrust, the amount of thrust generated by the engine lean on the mass flow rate over the engine and exit velocity of flow. Rocket motors are reaction engines, generating thrust by emitting mass reward, under Newton's third law. The nozzle passes the static high-pressure high-temperature gas toward briskly moving gas at near–ambient pressure. In the C-D nozzle, gas travels from chamber to converging portion and then navigates to the minimal cross-section and passes over the diverging portion, emitted toward the atmosphere. The flow over the CD nozzle is isentropic. A CD nozzle is frequently used to persuade chemical energy into K-E. The nozzle design plays a pivotal role in developing the thrust. Many researchers make work to stimulate supersonic flow through rocket nozzle with combustion chamber to conscious of the flow dynamics and variation of flow properties in the ignition chamber through the nozzle. Some analyzed static pressure and temperature for a rocket nozzle-duct with more than one inlet at different Mach numbers and analyzed the change in temperature in the combustion chamber rapidly and in convergent parts of the nozzle and the temperature warning in the outlet of a nozzle. JITHENDRA SAI RAJA CHADA et al (2021) had conducted a study on peculiarity in a C&D nozzle using CDF and concluded that the design is best for the low area ratio rocket design (1) K.M.Pandey et al had experimented on Rocket Nozzle with Four Inlets at Mach 2.1 and concluded under the x and y plot, its heat due to high velocity and temperature and then temperature increase briskly in the combustion chamber and convergent part of the nozzle and after that temperature waning in the outlet of the nozzle. (2) K.M. Pandey et al had organized the D Analysis of Four Jet Flow at Mach 1.74 The conflict between four jets rises with the rise of L. The tougher the conflict is, the larger the Reynolds shear stress and turbulent energy are. The intervention between the four jets rises as the spacing between four nozzles decrease. (3) kousik kumaar. R et al (2016) had studied the different shock ways in convergent and divergent nozzles had been pronounced that shockwave travels outside to inside not only posh by NPR but also precious by inlet pressure it can shrink by increasing the fraction between the inlet and outlet pressure (4). Arun Kurien Reji, et al (2018) had studied the different supersonic streams in the C-D nozzle had accomplished that the fluid flow inlet Mach number is less even at the leaving Mach number stream are supersonic speeds termed that the proposal was well developed (5). H Pujowidodo et al (2018) had deliberate the C-D Nozzle for refining the Impulse Momentum of Cross-Flow Turbine in a Bio-Micro Power Plant decided that momentum flux for rotating power turbine utilizing integration thermodynamics and CFD, replicas have shown that the slightest pressure ratio will intensification the supersonic stream spread through the CD-nozzle(6) nozzle Ric Gamble et al (2004) had the nozzle section and design rule had achieved that the effect for the apt air vehicle mission. As mandatory aircraft fleetness surges, so do functional fulfillments for the nozzle, succeeding in a progressively intricate nozzle (7). E. M. S. Ekanayake, et al (2009) had premeditated the simulation of a two-dimensional rectangular supersonic convergent-divergent nozzles had established that the nozzle: pressure ratio growth the flow leave-taking zone increases in size and moves from one wall to the contrast and back before eventually exiting the nozzle. This alteration from one wall to the supplementary is due to the build-up of turbulent K-E on the contrasting wall (8). Mohammad Arif Hussain et al (2019) had studied the various flow behavior of various convergent-divergent nozzles had clinched that the triangular convergent-divergent nozzles are unsurpassed (9). Bogdan-Alexandru Belega et al (2015) had studied the flow in the convergent and divergent nozzle their design model shaped based on a prevailing parameter is in consensus with the choice (10). Raghu Ande et al (2018) had premeditated the upshot of the divergent angel in the convergent-divergent nozzle had been clinched that the Mach number of finest assessments is developed at the divergent angle of 15° (11). Muhammad Arif Budiyanto et al (2019) had studied convergent and the divergent-convergent nozzle of water jet propulsion and concluded that the efficiency of the convergent nozzle is advanced than the combination nozzle (12). Andreas K. Flock et al (2019) had deliberate the design of convergent and divergent nozzle with a constant radius of the center, the body had been come to resolve that Displacing the center body in angular and translational direction led to maximum pressure and lower Mach number on that side where the center body was moved to. The angular displacement of 0.5◦ and a translational displacement of 4 mm were roughly equal in their impact (13). Ambareen Khan, et al (2018) had deliberate the flow in convergent and divergent nozzle based on the pressure control using micro-jets had been observed that as pressure reduces velocity will upturn and our results ascertained that the velocity is maximum at the exit and discrepancy of pressure inlet to the outlet is observed by considering Mach number (14). M.H.M. Noh, et al (2011) had studied the wedges in the convergent and divergent nozzle for the thruster presentation concluded that departure of the stream by the nozzle a wall cannot be jettisoned by reducing the divergence angle while maintaining the area ratio. As well as the surge in viscous loss reduces a forte of normal shock inside the nozzle (15). G.V M Krushnarao Kotteda et al (2017) had deliberate the flow in a planar convergent-divergent nozzle had been concluded that the bleed is most effective in the low to reasonable NPR regime validate the throat geometry, together with the discontinuity in slope, on the flow is probed. Four different nozzle geometries with an equal area ratio and length are chosen (16). Madhu B P, Syed Sameer, et al (2017) had scrutinized the flow in C-D nozzles and contour, nozzles had clinched that the conical nozzle gives the best output that contour (17).

2. METHODOLOGY

the principal underneath the Navier stokes equalities to put on the taken geometry. In the flow of engineering training, fluid dynamics theatres an uppermost protagonist, similarly, in fluid dynamics, Bernoulli's theorem is an in height palpable impression. This theorem is uttered by Swiss mathematician Daniel Bernoulli. He termed that if the stream of the fluid is level i.e., not happenstance any change in gravitational potential energy at that point a reduction in the pressure of the fluid is harmonized with the velocity rise of the fluid. He proposed this theorem as during the alteration, the all-inclusive mechanical energy of the streaming fluid having the energy related to fluid pressure, the potential energy of promotion of gravitation, and fluid K-E (KINETIC-ENERGY) been unchanged. Energy preservation is the elementary inkling to the upswing in Bernoulli's theorem. If the fluid is streaming between a horizontal tube of disparate cross-sectional area, the fluid employs a minimum pressure at the slightest area and vice versa.

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

Where is pronounced as the pressure

The is described density of the fluid

The gravitation is signified with the g

Y is termed as the height from the plane

Bernoulli's equations have the shortcoming at a steady stream, incompressible stream, frictionless stream, and flow along streamline. As the collective practical problems can be analyzed by using protracted Bernoulli's Equation. As follow

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

where $H_L =$ is termed as friction on wall's head loss

The Navier-Stokes equation has baptized after the great scientists Claudelouis Navier and George Gabriel Stokes. this equation came into the routine by the foremost newton's second law of motion. the scientific and the engineering flow Navier-Stokes equation plays a weird and handy role. The circumstance among all of the pressure, temperature, density, the velocity of a streaming fluid can be discerned. Navier-Stokes equations are the blending and the supplementary of Euler's equations which dels with the viscosity of the stream fluid. permanence, three time-dependent, three time-independent equations form the Navier-Stokes equation.

$$\rho \frac{\partial v}{\partial t} = -\nabla p + \rho g + \mu \nabla^2 \quad \rightarrow (3)$$

These equations are well combined

Collective every so often with additional equations called energy equations to unravel judicious appliance. 'This equation is governing equation for all fluid stream

3. DETAILED PROCEDURE:

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

$$F = \dot{m} g v_e + (p_e + p_o).A_e \quad \rightarrow (4)$$

$$I_{sp} = \frac{F}{\dot{m} g_0} \quad \rightarrow (5)$$

The nozzle design parameter as follow

Table 1: design parameters of rocket nozzle

The nozzle is designed in solid works and converted into planer surface and saved in igs format and exported to Ansys software

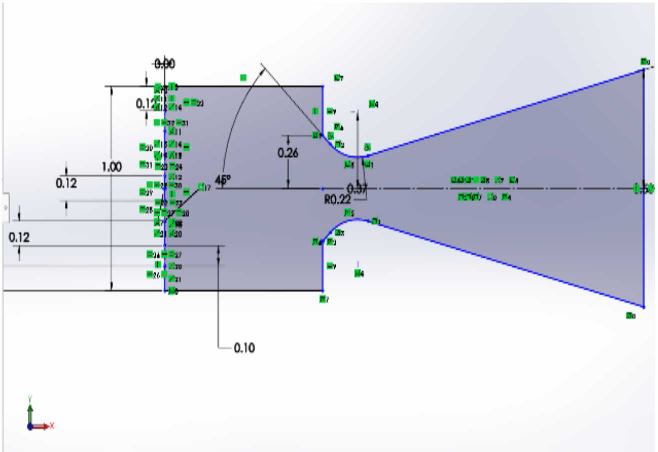

Fig1: the sketch of the rocket nozzle

The surface design:

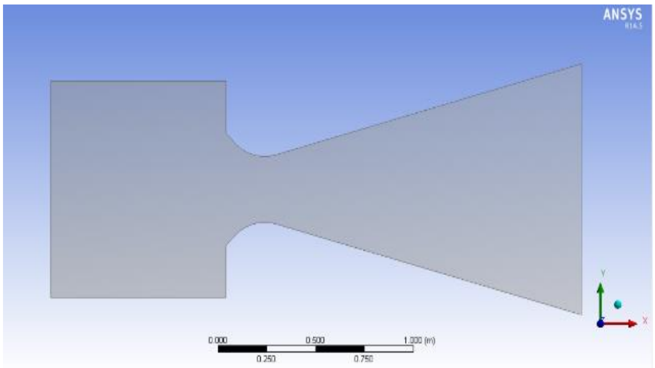

Fig 2: planner surface of rocket design

The inlet is place as follow with a length of 0.1 and internal distance between the inlet are 0.12 as show in figure

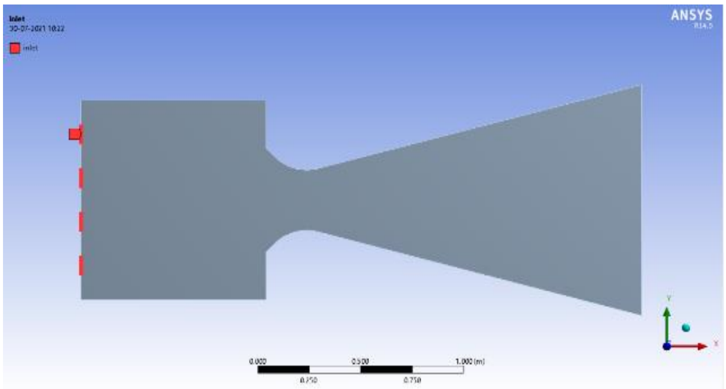

Fig3: the nozzle inlet placement

The meshing of the nozzle is as follow

Use advanced size function: proximity and curvature

Min size: 4.3219e-004M

Max size: 5e-02M

Minimum edge length: 1e-003m

With a refinement on the face 3

The outlet edge sizing in 125 divisions

The inlet edge sizing in 30 divisions

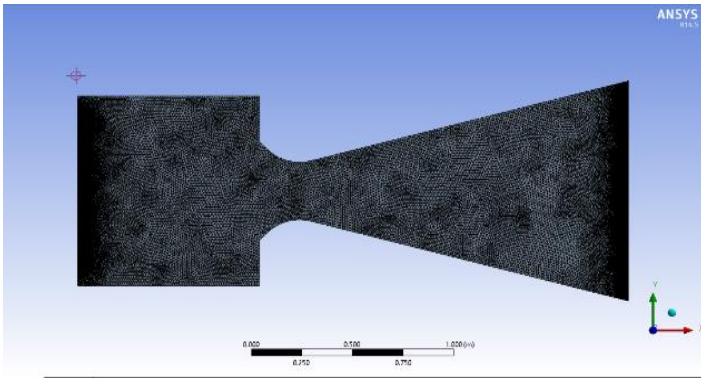

Fig4: meshing of the nozzle

The simulation is set in k-epsilon model with standard wall functions the fluid is air and density are incompressible-ideal-gas and viscosity is Sutherland the operation conduction is normal ATP and temperature 300k whose gravity is acting in positive x direction the inlet velocity is varied from 2,2.5,3,3.5,4,4.5,5 Mach with a temperature of 1200k and outlet is considered pressure out at 1bar pressure with temperature of 300k simulation with 10,000 iterations

4. RESULT AND DISCUSSION

The whole simulation done on Ansys fluent software 14.5 version in 2d model and simulated with different inlet Mach as 2,2.5,3,3.5,4,4.5,5 and also by considering the outlet as static pressure and results as follow

	velocity		outlet	pressure	
Mach no.	min	max	mach	min	max
2	0	1040.421	3.033297	-147398	17256.24
2.5	0	1288.186	3.755644	-207442	32454.32
3	0	1538.008	4.483988	-309829	38188.64
3.5	0	1777.105	5.181064	-417161	44360.36
4	0	2039.44	5.945889	-551707	62099.73
4.5	0	2315.798	6.751598	-695409	78172.18
5	0	2542.986	7.413953	-859483	84047.42

Table 2: this table with values of the inlet Mach number with outlet Mach; max and min static pressure; max and min velocity

The radius of the throat	**0.1521 m**
Area at throat	0.070685 m²
The radius of convergent radius	0.2598060 m
Area at convergent	0.212055 m²
Radius of exist	0.4305 m
Area of exist	0.58223 m²
Convergence angle	45°
Divergent angle	15°
The radius of the curvature at the throat	0.228 m

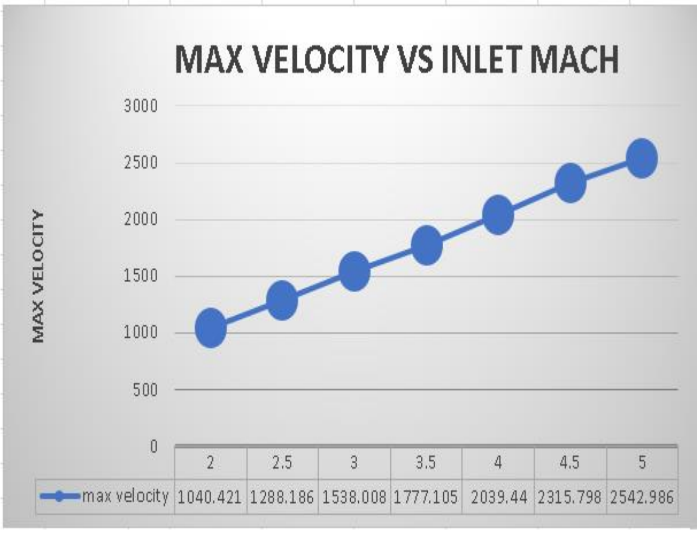

Fig5: max velocity vs inlet Mach

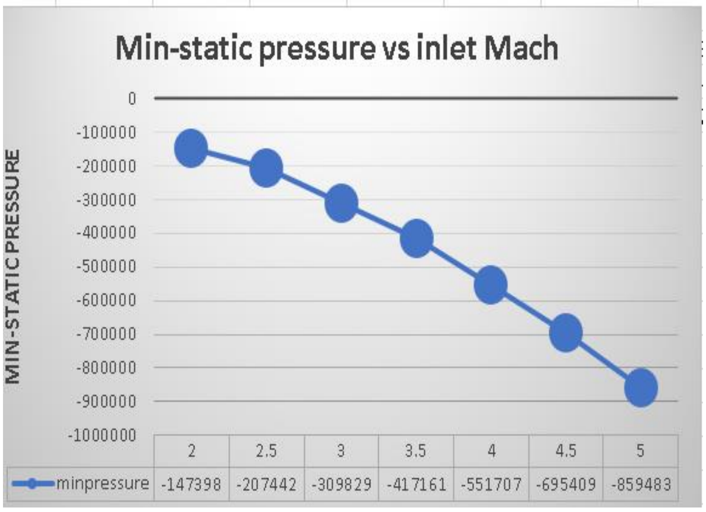

Fig6: min static pressure vs inlet Mach

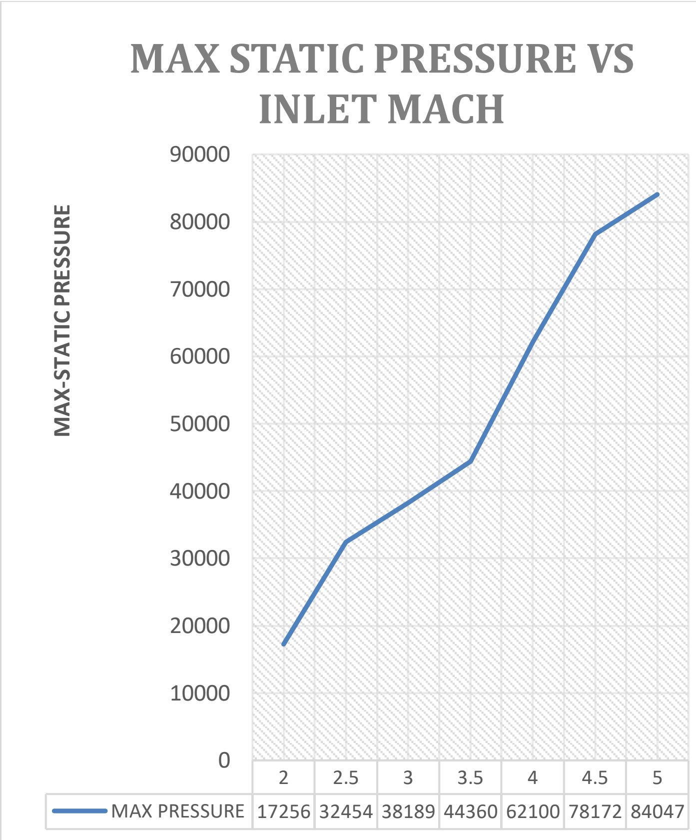

Fig 7: max static pressure vs inlet Mach

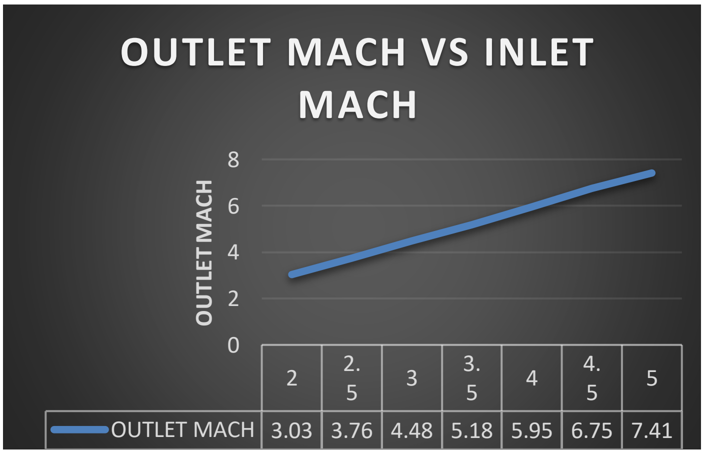

Fig8: Outlet Mach vs Inlet Mach

Hear Fig 5 represent the velocity and inlet Mach the velocity is linear and increase with respective of the inlet Mach

And Fig6 is the plot between inlet Mach and min pressure Fig7 is the plot between inlet Mach and max pressure here are some pressure counters of the simulation for Mach 4.5 and 5

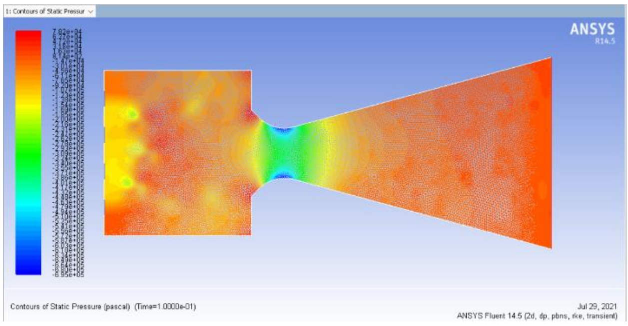

Fig12: static pressure counter of the inlet Mach 4.5

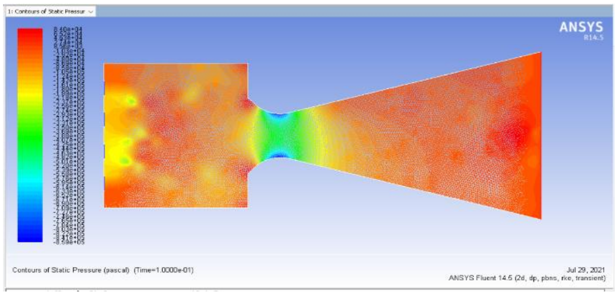

Fig13: static pressure counter of the inlet Mach 5.0

Hear The fig 8 represent outlet Mach vs inlet Mach and observed that the out Mach will be increased more than the initial one

Form observing the fig 9 to fig 11 we can see that the velocity is maximum at the throat of the nozzle and gradual again reduce and went off

4. CONCLUSIONS

This analytical study concluded that the usage of the four inlet is very much use full that using an single inlet since this study is purely analytical the Experimental validation is further to be investigated for gain the optimal result. By conclusion of this study it show that in analytical or theoretical form the Four inlet nozzle is more useful than the normal Nozzle.

ACKNOWLEDGEMENT

The heading should be treated as a 3rd level heading and should not be assigned a number.

REFERENCES

[1] JITHENDRA SAI RAJA CHADA, SRI RAM DEEPAK AKELLA et al Study of Flow Characteristics in a Convergent and Divergent Nozzle Using Computational Fluid Dynamics International Journal of Scientific Research and Engineering Development-– Volume 4 Issue 3, May -June 2021

[2] K.M. Pandey, S.K. Yadav CFD Analysis of a Rocket Nozzle with Four Inlets at Mach 2.1 International Journal of Chemical Engineering and Applications, Vol. 1, No. 4, December 2010 ISSN: 2010-0221

[3] K.M. Pandey and Virendra Kumar CFD Analysis of Four Jet Flow at Mach 1.74 with Fluent Software International Journal of Chemical

Engineering and Applications, Vol. 1, No. 4, December 2010 ISSN: 2010-0221

[4] 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)

[5] 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)

[6] 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:10.1088/1755-1315/209/1/01205

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

[8] 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

[9] Muhammad 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

[10] Bogdan-Alexandru Belega, Trang 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

[11] 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

[12] 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

[13] 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

[14] 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)

[15] 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)

[16] 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

[17] Madhu B.P, Syed Sameer, Kalyan 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

BIOGRAPHIES:

MR. SRI RAM DEEPAK AKELLA. Is an undergraduate research scholar who is perusing his bachelors in the field of mechanical engineering at Pragati Engineering college his area of research is His current research interests include CFD, aircraft and rocket design, thermal engineering, heat transfers, combustion and propulsion systems, material and manufacturing technologies, nanomaterials, and robotics. He had published some paper in the international and the national journals

Mr. GOLLAPALLI ARAVIND. Is an undergraduate research scholar who is perusing his bachelors in the field of mechanical engineering at Pragati Engineering college his area of research is His current research interests include Design of Automobile, material and Manufacturing technologies, and CFD.