Modelling and Analysis of C-D Nozzle Using CFD

Ketki Shirbavikar[1], Susmita Waghmare[2], Aditya Yadav[3], Ayush Yadav[4], Yugandhar Bhangale[5], and Srushti Sale[6]

[1]*Professor, Department of Mechanical Engineering, Vishwakarma Institute of Technology, Pune, Maharashtra, India*

[2,3,4,5,6]*TY Students, Department of Mechanical Engineering, Vishwakarma Institute of Technology, Pune, Maharashtra, India*

***Abstract*— This research was conducted to introduce a more efficient and powerful nozzle for steam turbine applications. This research paper will help explain various concepts related to De Laval nozzles. In this paper, pressure fluctuations are used to describe nozzle behavior. In an era of increasing computational fluid dynamics research in industry, the implementation of more efficient and powerful nozzles has become important. Contours and graphs of pressure and velocity variations are plotted against nozzle length to better understand the concepts. Fluctuations in flow parameters such as pressure and velocity are visualized using computational fluid dynamics. CFD provided a realistic approach to testing nozzles.**

***Keywords*— CFD, De Laval, Nozzle, Pressure, Velocity**

I. INTRODUCTION

The combustion chamber's chemical heat energy must be transformed into kinetic energy by the nozzle. By utilizing the nozzle, the low-velocity, high-pressure hot gas produced in the combustion chamber can be converted to high-velocity gas at low-pressure temperature. The De Laval nozzle was developed by a Swedish engineer of French descent who aimed to create a more efficient steam engine. As it directs the jet against the wheel, the nozzle is essential in converting the thermal energy of the hot, high-pressure steam from the boiler into kinetic energy. De Laval discovered that the most efficient conversion occurs when the nozzle narrows first, increasing the jet's speed to the speed of sound before expanding again. The expansion of the jet above (but not below) the speed of sound further increases the velocity of the jet, leading to efficient conversion of thermal energy into motion. Sir Isaac Newton first introduced the theory of air resistance in 1726, which states that aerodynamic forces depend on the density and velocity of liquids and the shape and size of moving objects. Newton's theory was followed by other theoretical solutions to the problem of fluid motion. Today, steam turbines are the preferred power source for power plants and large ships, but they typically have a different design. However, for rocket engines, the De Laval nozzle is an ideal design. A potent engineering tool used to simulate and resolve issues involving transport phenomena is computational fluid dynamics (CFD). CFD can predict the outcome of a problem before experimentation, which is much faster and less expensive than experimental methods involving prototypes.

II. LITERATURE REVIEW

The nozzle, according to research, is a crucial part of a power cycle plant because it converts heat enthalpy into kinetic energy for rotary power turbines. The cross-sectional nozzle's flow properties are crucial for driving the turbine blade because they absorb the most drop enthalpy and momentum flux. CFD modelling was used to study converging-diverging steam nozzle design in order to improve the crossflow turbine model for a micropower bioenergy source. In this study, a converging-diverging nozzle was replicated in order to examine the flux of momentum. In order to establish the proper input, throat, and outflow dimensions as the computational domain, a mathematical model of compressible flow was made using EES® tools. Analytical comparisons between the various flow parameters, such as pressure, temperature, and velocity distribution, were made in order to determine the most accurate approximation of momentum flux for turbine demand. Steam fluid had a maximum speed of 1.3 Ma and a mass flow of 0.978 kg/s (3.52 tons/hour) at pressure ratios of 0.5 and 200 C. A larger cross-sectional area could be made to improve flow.

A CD-nozzle with a throat diameter of 2 mm and an outlet diameter of 2.4 mm will pass more supersonic flow at the lowest pressure ratio, according to the analysis of momentum flux for rotating power turbines using integrated thermodynamics and CFD models. When steam was subjected to a pressure ratio of 0.5 and a temperature of 200 C, it reached a top speed of 1.3 Ma and a mass flow of 0.978 kg/s (3.52 tons/hour). A larger cross-sectional area could be made to improve flow. [16]

The performance of rockets must be increased through the use of better, fully integrated propulsion systems. The supersonic convergent-divergent design method (C-D nozzle) is covered in this project. The ideal gas will flow isentropically, which is the basis for the conical and contour C-D nozzle designs. The method of characteristics and the stream function is used by computer programming to create a high-efficiency nozzle profile for isentropic, inviscid, irrotational supersonic flows of any working fluid for any user-defined exit Mach number. The designed nozzle area ratio for the chosen fluid and desired exit Mach number is used to compare the theoretical area ratios. Using a commercial Computational Fluid Dynamics (CFD) code, it is independently verified that the nozzle geometry the code generated is accurate. Using ANSYS-FLUENT, flow simulations on nozzles were carried out to confirm the isentropic flow.[17]

The steam jet refrigeration (SJR) method is widely used in the industry, where steam from a steam generator is directed through a nozzle and into a tank filled with liquid. The nozzle converts pressure energy into kinetic energy, causing the liquid to briefly evaporate before being released into the condenser. The cooled water from the condenser is then used for heat transfer to cool the product. In this study, ANSYS FLUENT software was used to simulate the vapor pressure characteristics of different nozzle designs, including convergent, convergent-parallel, and convergent-divergent types. The study's findings, which were validated by experimental data, were presented through visualizations of the pressure in the nozzles. The results showed that there were varying pressures along the different nozzle types, which were related to changes in nozzle geometry. The FLUENT analysis results were found to be consistent with the experimental findings.[18]

Steam energy can be efficiently converted into kinetic energy through the use of a nozzle that produces a high-velocity gas or steam jet. The effects of critical pressure ratio on nozzle performance are discussed in relation to various nozzle designs, including their length, throat area, and exit size. The nozzle's primary function is to generate a high-speed steam jet to drive turbines. In impulse turbines, freely rotating blades are struck by the high-velocity steam jet to produce shaft work. In reaction turbines, high-velocity steam is emitted from free-moving nozzles, and the steam's reaction force against the nozzle creates motion and work. Friction in the nozzle reduces the steam's velocity while increasing its degree of superheat or final dryness fraction. The steam that is further superheated at the end of expansion provides additional benefits. These conclusions are drawn based on a discussion of saturated flow, supersaturation effects, steam discharge velocity and mass, and various nozzle passage types. Experimental data confirm these findings. [19]

In a reheat-gas turbine, a binary-flow system is created by integrating numerous steam nozzles with the first-stage annular-gas nozzle. Originally used for internal vane cooling, the steam is then expanded and accelerated to extract work. The vanes have steam nozzles in a "fat-body" design, with cooling achieved through reverse serpentine flow and trailing-edge impingement. Internal trailing-edge steam nozzles are used to diffuse or shock waves to the boundary layer inside the trailing edge to promote heat transfer. Steam blanketing is employed to increase film cooling effectiveness and reduce nozzle-profile loss by decreasing surface viscosity and limiting the expansion of suction-side after-shock waves. To further enhance vane-film cooling efficiency, it is suggested that a new vane shape be used in combination with a gas turbine combustion system. The integration of the gas/steam nozzle in the binary-flow system enhances the nozzle efficiency and exit velocity. The primary gas stream experiences minimal suction-side losses after shockwaves, achieving a Mach 1.15 to 1.20. Profile-friction losses are reduced as the surface steam viscosity is about two percentage points lower than the main gas stream. Steam is delivered to the vane's leading edge through backwards-curved weep holes and tangential steam nozzles to cool the nose and enhance downstream vane steam blanketing efficiency. A steam-cooled combustor with skew hairpin coils encased both internally and externally can be employed to construct a segmented-annular-type combustor.[20]

Steam turbines are a widely used type of prime mover that effectively converts thermal energy from steam into mechanical energy. They are commonly used in power plants and industrial applications where process power is required. The steam turbine is an ideal heat engine and prime mover due to its high thermal efficiency and superior power-to-weight ratio. Steam turbines are particularly effective in generating energy because they produce circular motion and are responsible for over 80% of the world's energy generation. The steam turbine's rotor plays a crucial role in its efficiency in generating steam. Compared to steam engines, steam turbines exhibit significantly higher thermal efficiency. This study has provided a better understanding of how steam turbines are manufactured and the materials and processes involved. The challenges associated with steam turbines have also been explored, which should be taken into consideration by designers. These notes aim to help engineers gain a deeper understanding of steam turbine components so that they can address related issues with confidence and knowledge.[21]

Computational fluid dynamics (CFD) is a branch of fluid mechanics that utilizes numerical techniques and algorithms to solve and analyze fluid flow problems. CFD analysis is employed to comprehend the flow patterns of supersonic rocket nozzles at various divergence angles and Mach values. This study examines the behavior of flow in a convergent-divergent nozzle by employing computational fluid dynamics software to evaluate variables. The obtained results are compared to analytical values by creating plots. The comparison reveals that the results obtained from theoretical analysis and CFD are quite similar. [22]

C-D Nozzle

De Laval nozzles are commonly utilized to achieve supersonic velocities and are among the most popular nozzle types. They are tube-shaped and narrow in the middle, forming an asymmetric hourglass shape. The cross-sectional area of the converging nozzle decreases towards the throat, where the fluid reaches its maximum velocity. However, converging nozzles can only accelerate fluids up to the speed of sound [Ma=1]. As a result, a branch is added to the nozzle to accelerate the fluid to supersonic speed [Ma > 1]. These types of nozzles are called convergent-divergent nozzles or De Laval nozzles.

III. MODELLING

A. Design of Nozzle

The De Laval nozzle was modelled using Ansys design modeler. The nozzle was outlined using the lines from the points option, and a 2D surface was created using the surface from edges command. The following are the dimensions of the nozzle.

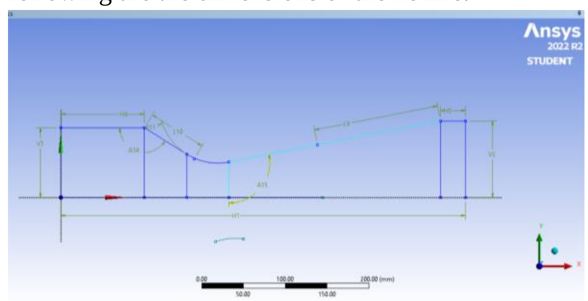


Fig. No. 1: C-D Nozzle Dimension

Table No. 1: Standard Dimensions

Parameter	Dimensions
Total Nozzle Length (mm)	484
Inlet Diameter (mm)	166
Throat Diameter (mm)	34
Outlet Diameter (mm)	183
Chamber Length (mm)	99
Convergent Angle (deg)	32
Divergent Angle (deg)	11
Throat Radius Curvature (mm)	70

B. Meshing

The De Laval nozzle model with the given dimensions meshed using Ansys Design Modeler. Meshing involves transforming a model from an infinite number of particles to a finite number of particles. In mesh mode, the mesh details were set to Physics Preference: CFD, with meshing relevance set to 100. Proximity and curvature options were selected to account for the curved nature of the nozzle. High smoothing was applied, and the number of cells on the gap was set to 50. The fine mesh resulted in 12,427 elements and 12,760 modes. The mapped surface mesh was utilized for the two surfaces, with the center of relevance finely set, and square elements used.

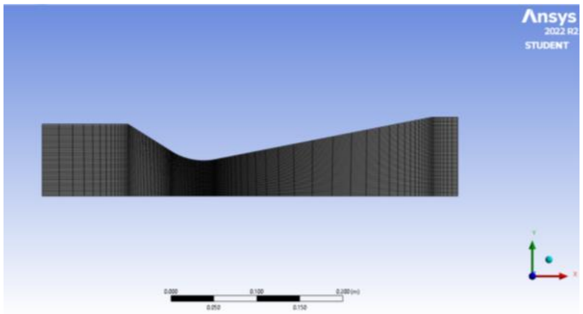


Fig. No. 2: Meshed Model of a Nozzle

C. Pre-processing

Nozzles were pretreated with ANSYS FLUENT. 2D and double precision settings were used when reading mesh. The mesh was scaled because originally all dimensions were specified in mm. The mesh was checked fluently and reported no significant errors.

Table No. 2: Pre-processing Details

General	Solver Type: Density Based 2D Space: Planar Time: Steady
Materials	Density: Ideal Gas Cp: 1.006.43 J/kg K Mean molecular mass: 28.966 g/mole

D. Cases

$$\boldsymbol{Pressure\ Ratio} = \frac{\boldsymbol{Outlet\ Pressure}}{\boldsymbol{Inlet\ Pressure}}$$

Table No. 3: Cases of Different Pressure Ratio

Case No.	Pressure Ratio
Case I	0.8
Case II	0.5
Case III	0.1
Case IV	0.01

IV. RESULTS AND DISCUSSIONS

Table. No. 4: Pre-Processing Details of Given Cases

Cases	Boundary Condition
Case I	Inlet Pressure = 100000 Pa Inlet Temperature = 300 K Outlet Pressure = 80000 Pa Outlet Temperature = 300 K
Case II	Inlet Pressure = 100000 Pa Inlet Temperature = 300 K Outlet Pressure = 50000 Pa Outlet Temperature = 300 K
Case III	Inlet Pressure = 100000 Pa Inlet Temperature = 300 K Outlet Pressure = 10000 Pa Outlet Temperature = 300 K
Case IV	Inlet Pressure = 100000 Pa Inlet Temperature = 300 K Outlet Pressure = 10000 Pa Outlet Temperature = 300 K

A. *Case I: Pressure Ratio = 0.8*

Velocity Variation

The velocity initially increases towards the throat, reaches a maximum at the narrowest point, and then decreases towards the exit. The lowest velocity occurs at the entrance section with a value of 80.7 m/s, while the maximum velocity of 471.608 m/s is obtained at the throat.

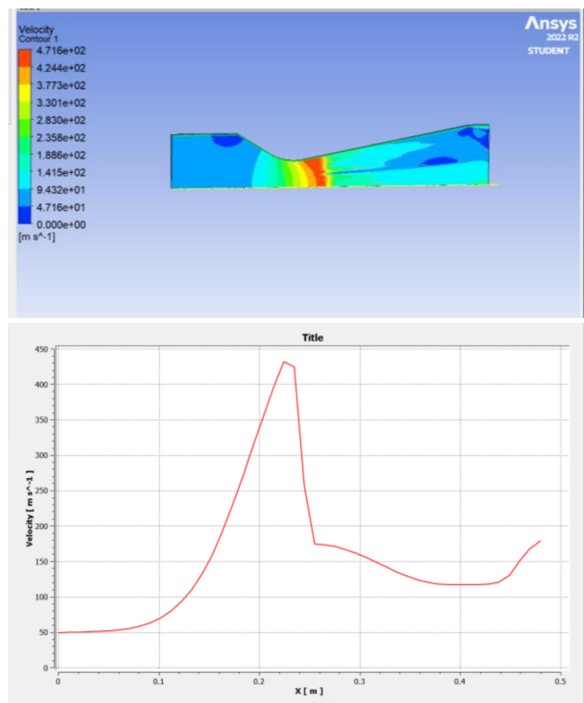


Fig. No. 3: Velocity Contour and Graph of Case I

Pressure Variation

The pressure is highest at the inlet section and lowest at the throat. The maximum pressure value obtained is 101009 Pa, while the minimum pressure value is 20297.1 Pa.

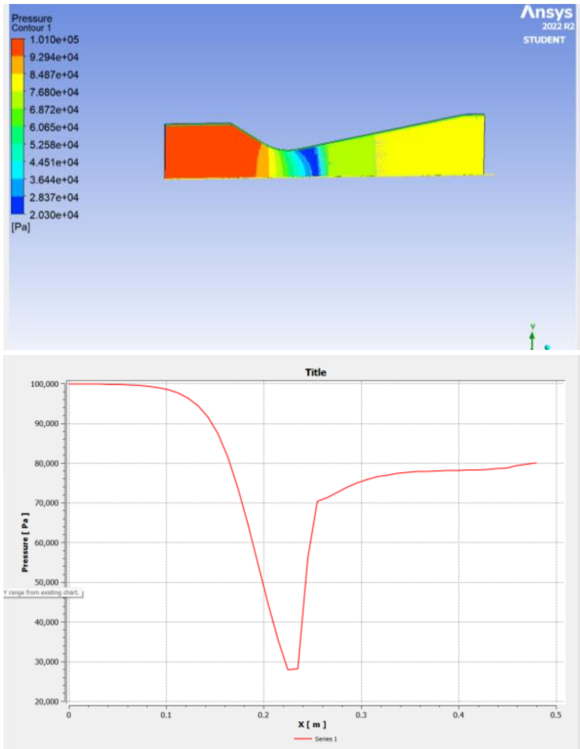
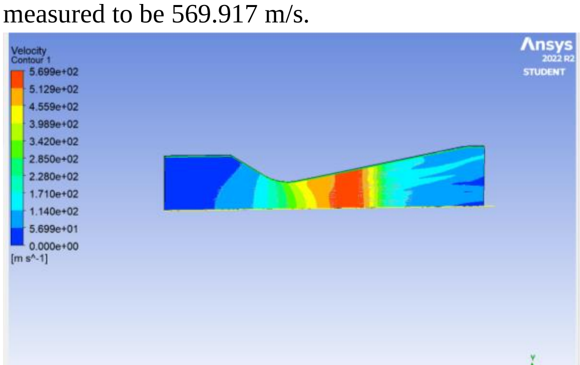


Fig. No. 4: Pressure Contour and Graph of Case I

B. Case II: Pressure Ratio = 0.5

Velocity Variation

The velocity of the fluid passing through the De Laval nozzle follows a pattern of initially increasing towards the neck, reaching a maximum at the throat, and then decreasing towards the exit section. The minimum velocity of the fluid is observed at the entrance section and is measured to be 65.1 m/s. The maximum velocity of the fluid is observed at the throat and is measured to be 569.917 m/s.



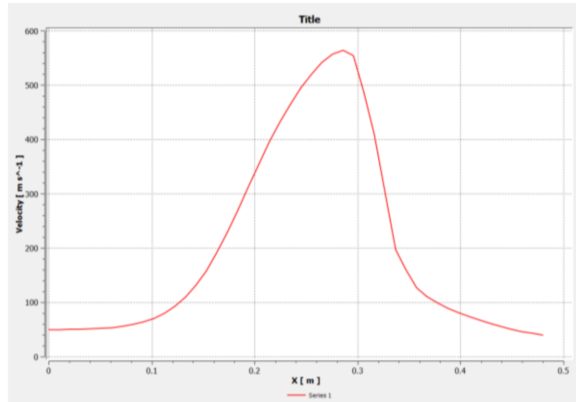


Fig. No. 5: Velocity Contour and Graph of Case II

Pressure Variation

The pressure distribution shows that the maximum pressure occurs at the inlet section and the minimum pressure occurs at the throat. The maximum pressure value is found to be 101009 Pa while the minimum pressure value is 6771.91 Pa.

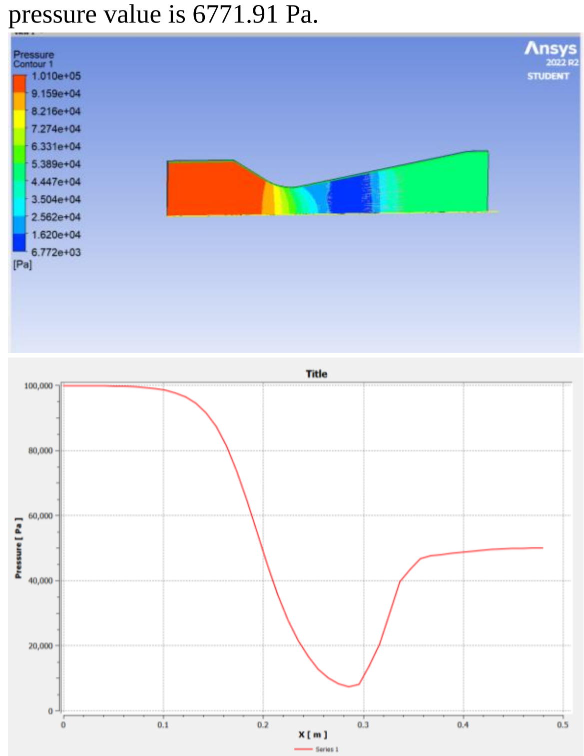


Fig. No. 6: Pressure Contour and Graph of Case II

C. Case III: Pressure Ratio = 0.1

Velocity Variation

The velocity of the fluid first increases towards the throat and then decreases. The minimum velocity occurs at the entrance section, the maximum at the throat, and decreases again at the exit section. The minimum speed is 75.7 m/s, and the maximum speed is 643.42 m/s.

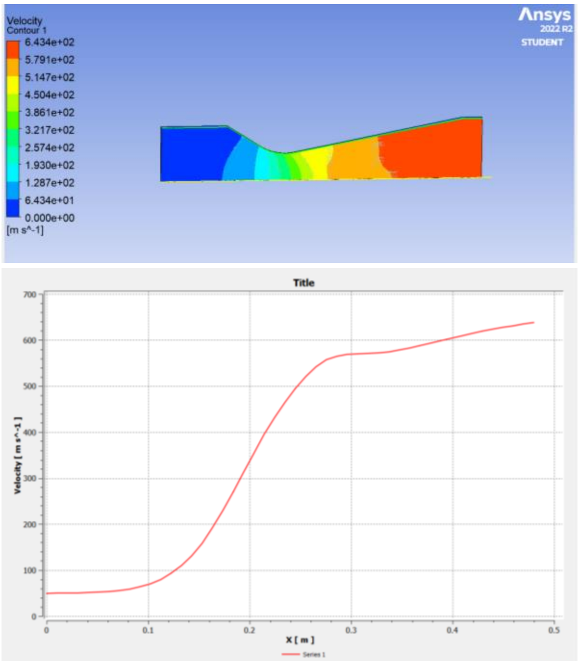


Fig. No. 7: Velocity Contour and Graph of Case III

Pressure Variation

The pressure profile displays a maximum value at the inlet section and a minimum value at the throat. The maximum pressure is 101012 Pa, while the minimum pressure is 1753.73 Pa.

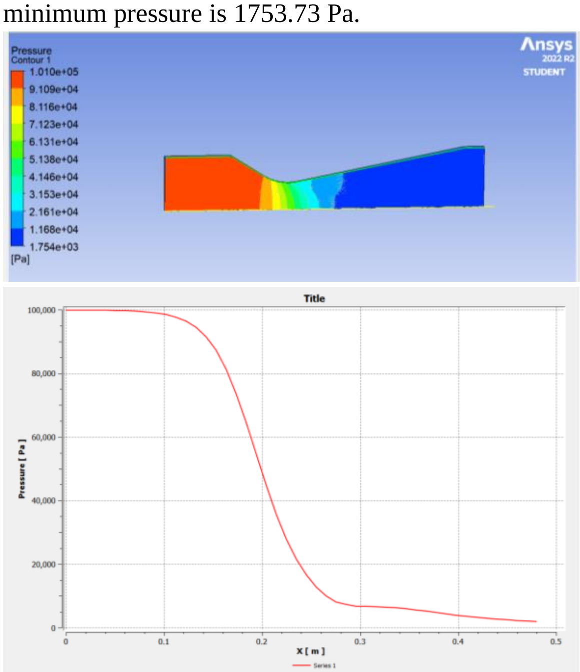


Fig. No. 8: Pressure Contour and Graph of Case III

D. Case IV: Pressure Ratio = 0.01

Velocity Variation

The velocity of fluid first increases towards the narrowest point of the nozzle, which is the throat, and then decreases as it moves towards the wider exit section. The minimum velocity is observed at the entrance section, while the maximum velocity occurs at the throat. In the present case, the minimum velocity is found to be 75.7 m/s, while the maximum velocity is found to be 643.42 m/s.

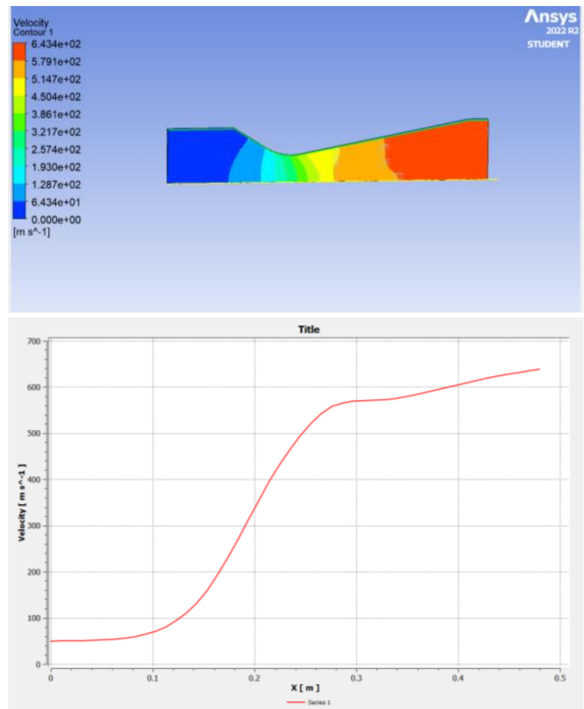
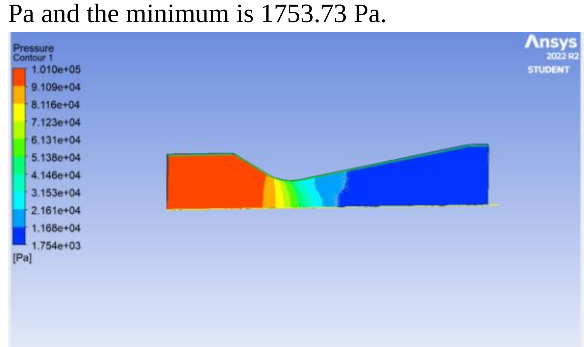


Fig. No. 9: Velocity Contour and Graph of Case IV

Pressure Variation

Pressure is maximum at the inlet section and minimum at the throat. The maximum pressure value is 101012 Pa and the minimum is 1753.73 Pa.



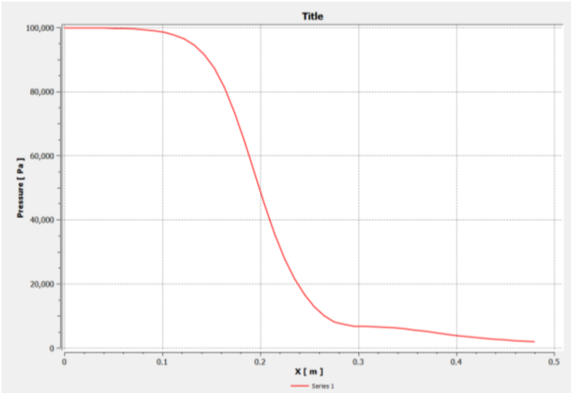


Fig. No. 10: Pressure Contour and Graph of Case IV

VII. CONCLUSION

This study provides an overview of the fundamental principles of the convergent-divergent nozzle. By using computational fluid dynamics, we have determined the location of the normal shock wave in the diverging section of the C-D nozzle. Our analysis has shown that the efficiency of the nozzle is directly proportional to the pressure ratio; the lower the pressure ratio, the higher the efficiency of the nozzle. Additionally, we have observed that there is no significant change in velocity between the pressure ratio of 0.1 and 0.01, indicating that the efficiency change is constant beyond a certain pressure ratio.

REFERENCES

[1] Nikhil Deshpande and Suyash Vidwans, “Theoretical and CFD Analysis of De Laval Nozzle”, vol.2, No 4, April 2014.

[2] Venkatesh V and C Jayapal Reddy, “Modelling and simulation of Supersonic nozzle using computational fluid dynamics,”vol. 2,2015, No 6.

[3] Mr Lijo P Varghese, Mr Rajiv Saxena And Dr R.R. Lal, Analysis of the Effect of Nozzle Hole Diameter on CI Engine Performance Using Karanja Oil-Diesel Blends. International Journal of Mechanical Engineering and Technology (IJMET), 4(4), 2013.

[4] Yunus Cengel and John Cimbala,” Fluid Mechanics” 3rd edition,

[5] Nagesha S, Vimala Narayanan, S. Ganesan and K. S.Shashishekar, CFD Analysis in an Ejector of Gas Turbine Engine Test BED. International Journal of Mechanical Engineering and Technology (IJMET),5(9), 2014.

[6] Malay S Patel, Sulochan D Mane and Manikant Raman, Concepts and CFD analysis of De-Laval Nozzle. International Journal of Mechanical Engineering and Technology (IJMET), 7(5), 2016.

[7] Prosun Roy, Abhijit Mondal, Biswanath Barai, CFD Analysis of Rocket Engine Nozzle, International Journal of Advanced Engineering Research and Science (IJAERS) Vol-3, Issue-1, ISSN: 2349-6495 (2016)

[8] B.Krishna Prafulla, V. Chitti Babu and P. Govinda Rao, CFD Analysis of ConvergentDivergent Supersonic Nozzle, International Journal of Computational Engineering Research, Vol-03, Issue-5 (2013)

[9] Samynathan. R, Durairaj.S, Manikandan.M, Balaji. E, Design and CFD analysis of flow through C-D nozzle and Aerospike nozzle at different operating conditions, International Journal of Innovative Research in Technology, Science & Engineering (IJIRTSE), ISSN: 2395-5619, Volume – 2, Issue – 6 (2016)

[10] C.Satheesh, A.Arulmurugu, Design and analysis of c-d nozzle increase the efficiency using CFD, International Journal of Modern Trends in Engineering and Research, e-ISSN: 2349-9745, p-ISSN: 2393-8161 (2015)

[11] Pandey K.M.; Singh A.P.; CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software, International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, ISSN: 2010-0221 (2010)

[12] P. Padmanathan, Dr S. Vaidyanathan, Computational Analysis of Shockwave in Convergent Divergent Nozzle, International Journal of Engineering Research and Applications (IJERA), ISSN: 2248-9622, Vol. 2, Issue 2, pp.1597-1605 (2012)

[13] Natta, Pardhasaradhi.; Kumar, V. Ranjith.; Rao, Dr Y.V. Hanumantha.; Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (CFD), International Journal of Engineering Research and Applications (IJERA),

ISSN: 2248-9622, Vol. 2, Issue 5, pp.1226-1235 (2012)

[14] Pardha samadhi 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.

[15] 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.

[16] Pujowidodo, H., Siswantara, A. I., Gunadi, G. G., & Daryus, A. (2018). The study of converging-diverging nozzle for improving the impulse-momentum of the cross-flow turbine in a bio-micro power plant. IOP Conference Series: Earth and Environmental Science, 209, 012054.

[17] Ms. K. Sree Lakshmi, & Mr. K. Venkatesh. (2016). Modelling and Simulation of Supersonic Nozzle Using Computational Fluid Dynamics. International Journal and Magazine of Engineering Technology, Management and Research, 3 (2016, 9, September).

[18] Firman, & Anshar, M. (2018). Study on steam pressure characteristics in various types of nozzles. Journal of Physics: Conference Series, 979, 012084.

[19] Kingsley E. Madu, & Emmanuel I. Nwankwo. (2018). Effects of Friction on critical pressure ratio of a nozzle.

[20] Rice, I. G. (1984). The integrated gas/steam nozzle with steam cooling: Part II — design considerations. Volume 4: Heat Transfer; Electric Power.

[21] A. Sudheer Reddy, MD. Imran Ahmed, T. Sharath Kumar, A. Vamshi Krishna Reddy, & V. V Prathibha Bharathi. (2014). Analysis Of Steam Turbines. International Refereed Journal of Engineering and Science (IRJES), 3(2).

[22] B.V.V. NAGA SUDHAKAR, B PURNA CHANDRA SEKHAR, P NARENDRA MOHAN, & MD TOUSEEF AHMAD. (2016). Modelling and simulation of Convergent-Divergent Nozzle Using Computational Fluid Dynamics. International Research Journal of Engineering and Technology (IRJET), 03(08).