e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

PERFORMANCE EVALUATION OF NOZZLES THROUGH CFD ANALYSIS FOR DIFFERENT MATERIALS

Mamleshwaram Chandra*1, Virendra Nayak*2

*1M.Tech Research Scholar, Dept. Of Mech. Engg., MATS School Of Engineering & IT, MATS University, Raipur, C.G., India.

*2Associate Professor, Dept. Of Mech. Engg., MATS School Of Engineering & IT, MATS University, Raipur, C.G., India.

ABSTRACT

There is a literature Gap regarding optimization of nozzle such as Design optimization of nozzles for different materials has not been seen frequently. Also Several studies have been studied but rarely the effect of geometry & flow parameters used simultaneously has been observed and Researches related to drag enhancement have also been seen very rarely. So this paper focuses on Drag enhancement through various geometry & flow parameters, material optimization for nozzle and Flow optimization through the nozzle.

Keywords: Nozzles, Ansys, CFD, Fluids.

I. INTRODUCTION

The flow of fluids pertains to the motion of a fluid caused by different imbalanced forces. Fluid flow is a fundamental aspect of fluid mechanics that specifically examines the dynamic behavior of the fluid. The fluid's mobility persists until various imbalanced forces act upon it.For instance, while pouring water from a bottle into a cup, the water flows at a higher velocity over rim of mug and a low velocity in the bottom part of the mug. In this scenario, the unbalanced force acting is gravity, which causes the water to flow until the mug is tilted and the water is contained inside it.

Fluid flow can exhibit characteristics of steadiness, unsteadiness, viscosity, or non-viscosity. In the scenario of a continuous flow of fluid along a certain path, velocity of fluid remains constant at every location. However, in the context of unstable fluid flow, the fluid's velocity varies between any two sites. Viscosity refers to the measurement of fluid thickness, and there is a wide range of viscous fluids, including oil, shampoo, and others . In uniform flow, the velocity remains constant in terms of both magnitude and direction between any two places. However, in non-uniform flow, the velocity varies from point to point at any given moment. In laminar flow, also known as streamline flow, fluid predominantly moves in parallel layers without any disturbance or interruption between them. Turbulent flow exhibits chaotic behavior characterized by rapid fluctuations in pressure and flow velocity. In rotational flow, the particles in the fluid are undergoing rotation around their individual axes, whereas in irrotational flow, the particles do not rotate about their own axes.

Fluid and flow are distinct concepts, each possessing its own precise meanings. Fluids, such as liquids and gases, are substances that lack a definite structure and readily undergo changes in response to external pressure. Flow, however, pertains to the act of moving. The phrase fluid flow encompasses the motion of both liquids and gases when they are mixed. Design process of designing vehicles, like aircraft, ships & automobiles has been changed by advancements in computational technology, software, and hardware . Various commercial software programs are utilized in the design and analysis procedures. These tools not only save the time and expenses associated with creating new designs, but also enable the examination of systems where controlled experiments are challenging or unfeasible.

In the field of fluid dynamics, several commercial Computational Fluid Dynamics (CFD) software programs are accessible for simulating the movement of fluids in or around objects. CFD has seen continuous advancements in recent decades, resulting in commercial and research codes that offer increasingly reliable and precise outcomes. When wind tunnel test data is combined with computational fluid dynamics (CFD), it may be utilized in the design process to actively influence changes in the shape, rather than only serving as a tool for validation of design. CFD is now an essential component of the engineering design and analysis process for several firms. It allows them to forecast the performance of new designs or processes prior to their actual production or implementation. An essential need for any computational fluid dynamics (CFD) instrument employed in thermal applications is the capability to accurately model flow phenomena occurring in nozzles and turbines. Computing is challenged by complex factors like pressure gradients, stream wise vortices, shocks, eddy placement, velocity distribution, stream line curvature etc.

Precise tools capable of recognizing slight variations between alternate designs are necessary to target the modest margins of improvement in nozzle and turbine design. Computational fluid dynamics (CFD) surpasses the accuracy of custom modeling tools that utilize reduced numerical methods and assumptions. Computational Fluid Dynamics (CFD) possesses various intrinsic benefits, including its ability to give prompt and cost-efficient solutions in comparison to experimental techniques. Additionally, CFD provides better accuracy when compared to empirical approaches commonly employed in the design process. Precise modeling of fluid dynamics within the nozzle is crucial for forecasting velocity and pressure patterns . The present study attempts to analyze the flow through the nozzle and estimate the best performance parameters. The resolution of flow via the nozzle exclusively pertains to a single gas phase.

1.1 Nozzles

A nozzle is a particular device employed to control the direction or characteristics of a fluid flow, specifically to augment its velocity, when it is expelled from or enters a restricted chamber or pipe. A nozzle is a conduit that may control the flow of a fluid, such as a liquid or gas, by adjusting its cross-sectional area. Nozzles are frequently used to control the speed, direction, amount, shape, pressure, and/or rate of flow of the stream that is released from them [9]. Inside a nozzle, the speed of the fluid increases while its pressure decreases.

A nozzle is a straightforward apparatus, consisting of a specifically designed conduit through which high-temperature gases pass. Nevertheless, comprehending the mathematical principles governing the functioning of the nozzle requires meticulous consideration. Nozzles exhibit a diverse range of forms and sizes, contingent upon the specific objectives of the aircraft's mission. Conventional turbojets and turboprops typically utilize a fixed geometry convergent nozzle, as seen in figure 1. Turbofan engines commonly utilize a co-annular nozzle, as seen in the top left corner. The primary flow emerges from the central nozzle, while the secondary flow emerges from the surrounding annular nozzle. The combination of the two flows results in an increase in thrust, and these nozzles also tend to produce less noise compared to convergent nozzles .

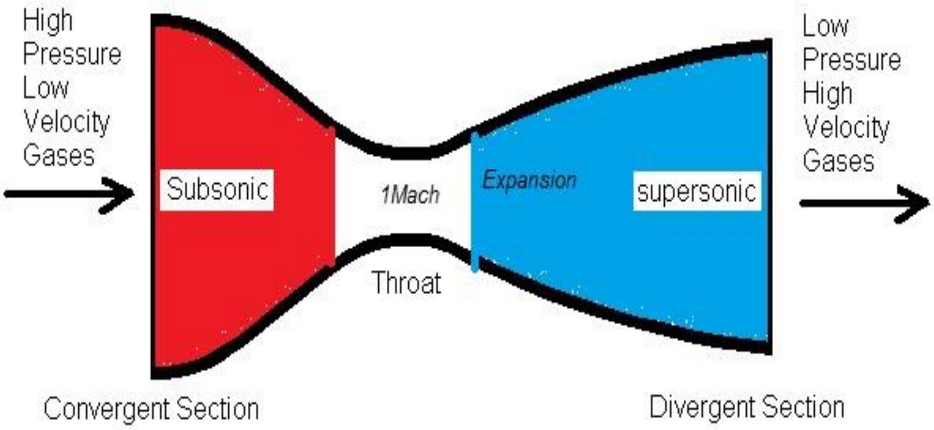

Figure 1: Common type of Convergent-Divergent Nozzle

Afterburning turbojets and turbofans need the use of a variable shape convergent-divergent (CD) nozzle, as seen in figure 1. The flow in this nozzle undergoes convergence at the minimum region or throat, followed by expansion across the divergent portion towards the outlet on the right . The flow is characterized as subsonic before reaching the narrowest part of the passage, known as the throat, but becomes supersonic after passing through the throat.

II. LITERATURE SURVEY

Nikhil D. Deshpande et al. (2014)- "Theoretical & CFD Analysis Of De Laval Nozzle"

De Laval nozzles are mechanical apparatuses that transform heat and pressure energy into practical kinetic energy. In order to construct the geometry of the nozzle, as well as the insulation and cooling arrangements, it is necessary to have access to the temperature, pressure, and velocity data at every section of the nozzle. This work seeks to present theoretical equations for calculating the aforementioned. The verification of these equations is conducted with the Computational Fluid Dynamics (CFD) program ANSYS Fluent.

Khizar Ahmed Pathan et al. (2017)- "CFD Analysis of Effect of Flow and Geometry Parameters on Thrust Force Created by Flow from Nozzle"

This research introduces a CFD analysis that examines how the shape and flow characteristics of convergent diverging nozzles affect the thrust force generated by the flow into an abruptly expanded circular duct with a greater cross-sectional area. The study is specifically investigating the resultant thrust force. The nozzles are specifically engineered to accommodate Mach values of 1.1, 1.4, 1.8, and 2.0. The CFD study is conducted by systematically altering the area ratios and nozzle pressure ratios over all Mach values. The analysis considers area ratios of 1, 2, 4, 6, 8, 10, and 12. The analysis considers nozzle pressure ratios of 2, 4, 6, 8, 10, and 12. Tables are utilized to compare the effects of various combinations of Mach number, area ratios, and nozzle pressure ratios. Based on the observations, it can be inferred that the thrust force generated by the flow through the nozzle is significantly affected by the Mach number, area ratio, and nozzle pressure ratio.

Sher Afghan Khan et al. (2021)- "CFD Analysis of CD Nozzle and Effect of Nozzle Pressure Ratio on Pressure and Velocity for Suddenly Expanded Flows"

A nozzle is a mechanical apparatus that utilizes the energy of pressure and the enthalpy of fluid to enhance the velocity of outflow and regulate the direction of fluid flow. In order to determine the shock pattern of the nozzle duct, the flow inside the nozzle must be at a supersonic speed, with a Mach number exceeding one. Empirically, the shock pattern is acquired for a nozzle with a Mach number of 2 and a nozzle pressure ratio (NPR) that is equal to or less than 7. At Mach number M = 2, the required NPR (Nozzle Pressure Ratio) for proper expansion is 7.82. Flow from the nozzles is under-expanded when the NPR exceeds 7.82. If the nozzle's NPR is less than 7.72, the flow from the nozzle is considered to be over-expanded. This work employed the computational fluid mechanics (CFD) approach to model the flow of the nozzle, relying on experimental observation. The ANSYS Fluent software was used to model a transient compressible flow of air via a supersonic nozzle in two dimensions. The simulation data were analyzed using a density-based implicit solution for a time-dependent flow. The results demonstrate that the CFD approach accurately models fluid flows and the generation of shock waves in a duct, providing valuable insights for fluid dynamics study.

Malay S Patel et al. (2016)- "Concepts and CFD Analysis of De-Laval Nozzle"

A nozzle is a device specifically engineered to regulate the properties of a fluid. Its primary function is to enhance the speed of fluid. A typical De-Laval nozzle consists of three sections: a converging component, a throat, and a diverging part. This study seeks to elucidate the majority of the ideas pertaining to the De Laval nozzle. This article discusses the operational principle of a nozzle. Theoretical study of flow is conducted at several locations within the nozzle. Computational Fluid Dynamics is employed to display the changes in flow parameters such as Pressure, Temperature, Velocity, and Density. CFD is also utilized to simulate shockwaves.

Sher Afghan Khan et al. (2018)- "CFD Analysis of CD Nozzle and Effect of Nozzle Pressure Ratio on Pressure and Velocity for Suddenly Expanded Flows"

A numerical analysis was conducted to investigate the efficacy of micro-jets in regulating the base pressure in a two-dimensional planar duct with a rapid expansion. Two micro-jets, each with a diameter of 1 mm, were positioned at 90° intervals along a pitch circle distance that was 1.5 times the diameter of the nozzle exit at the base region. These micro-jets were used as active controls. The Mach numbers measured at the entrance of the abruptly enlarged duct were calibrated to be 1.87.

The length-to-diameter ratio (L/D) of the rapidly enlarged duct was 10. Nozzles yielding the calibrated Mach numbers were operated with nozzle pressure ratio (NPR) 3, 5, 7, 9 and 11. The current study clearly shows that, for a given Mach number, the influence of NPR will lead to the greatest rise or decrease in pressure and velocity. The geometry of the convergent-divergent nozzle has been simulated and analyzed using turbulence models. The K-ε standard wall function turbulence model used in the code was verified independently using commercial computational fluid dynamics.

Prapti Joshi et al. (2020)- "Critical Designing and Flow Analysis of Various Nozzles using CFD Analysis"

The nozzle component of a rocket has undergone continuous development and study to enhance its efficiency and performance. A nozzle is a three-dimensional visualization of a two-dimensional pipe with a variable cross-sectional area. It is used to guide and increase the speed of the flow of gases generated by the combustion chamber. This research involves analysing the design of many nozzles and comparing them to current ones. This is important because in the coming years, numerous organizations are going to conduct research on other planets to study their environment and hunt for a more suitable habitat. Operating nozzles in an environment that differs from Earth's would provide challenges. Several pivotal planets have atmospheres composed of various gases such as hydrogen, helium, methane, and carbon dioxide, necessitating the preservation of nozzle efficiency. The current study includes many characteristics, including Mach velocity, temperature, and pressure. The nozzle effectiveness at various temperature conditions is analyzed by a thorough examination of nozzles using CFD simulations in ANSYS Fluent and designing in Catia V5.

Mustafa Atmaca et al. (2021)- "CFD analysis of jet flows ejected from different nozzles"

Nozzles are extensively employed to regulate the velocity, direction, mass, shape, pressure, and flow rate of a stream in many technical applications. This work introduces a computational fluid dynamic (CFD) model that predicts the performance of a jet. The model utilizes 3D models, parametric analysis, realizable k-epsilon turbulence models, and experimental measurements. Three distinct types of jet flows are discharged from three different slot nozzles: a round-shaped nozzle, a rectangular-shaped nozzle, and a 2D-contoured nozzle. This numerical investigation predicts the velocities of unconfined jets at various axial distances from the nozzle exit within the range of $0.2 \leq z/B \leq 12$, assuming a constant center velocity at the nozzle exit. The computational fluid dynamics (CFD) simulation results are compared to the experimental findings documented in the literature. These results align with the findings of previous investigations.

Uttam Kumar et al. (2018)- "CFD analysis and parameter optimization of Divergent Convergent Nozzle"

The present research focuses on doing a computational fluid dynamic analysis of a two-dimensional convergent-divergent nozzle using Ansys software. The CVM (control volume method) is utilized to solve the governing equation of a fluid flow problem, which is developed based on the specified boundary condition. The primary objective of the present study is to ascertain the most appropriate or optimal design of the convergent-divergent angle in a DC Nozzle. The nozzle parameter is determined based on the geometry of the DC nozzle. The configurations were created by varying the angle from 15 to 40 degrees in increments of 5 degrees for the convergent angle, and from 12.5 degrees to 20 degrees in increments of 2.5 degrees for the divergent angle. The study was conducted using the Fluent Workbench feature of the ANSYS program. The nozzle's input data is determined by the temperature of the exhaust gas and the pressure at the intake. The output data is acquired by Fluent in the form of a temperature plot, pressure distribution, and computed values for velocity gradient and Mach number for each combination.

III. METHODOLOGY

As noted in the previous section, in order to complete the task of current research study, a convergent-divergent nozzle is to be analyzed so that it can be optimized for its design, material & flow parameters that lead to drag enhancement through the nozzle. Flow chart of the methodology to be followed is presented in the figure 2.

Figure 2 represents several steps to be followed for the CFD simulation & analysis of the current problem. Steps are presented here for reference-

- **Concept Generation-** First step in the process is to generate the relevant concept regarding the nozzles & the same has been in the first chapter of the dissertation.
- **Literature Review-** Second step in the process of achieving the objectives for the current research study is to review the available literature so that the objectives for the current study are ascertained. Rigorous literature review has been conducted in the second chapter of the dissertation.
- **CAD Data Preparation.** After deciding upon the objectives, next step is now to develop the CAD 3D model of the nozzle having desired dimensions. For the current research study different type of models are developed. For optimizing the flow pattern across the nozzle various nozzle angles have been set &

accordingly the nozzles have been developed. Similarly different cross-sections have been chosen & accordingly the nozzles have been developed. Model development has been accomplished in Solidworks-2020 that is worldwide acclaimed to be the best solid modelling software.

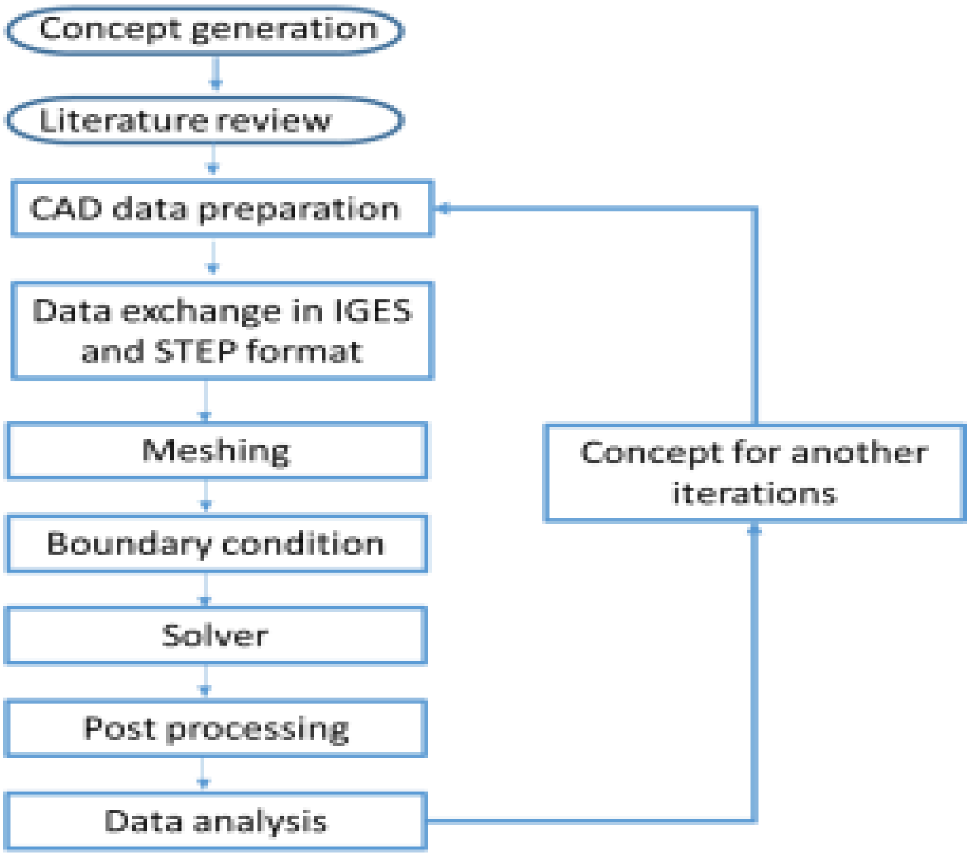

Figure 2: Flowchart of the Methodology to be Followed for the Current Research Study

- **Data Exchange-** 3D models developed in Solidworks are now imported to ANSYS CFD module for analysis conduction & for the purpose all the solid models are converted in to. iges or. step format. These format files are now employed in ANSYS for further analysis process.
- **Meshing-** Once the 3D model of the nozzle is imported into the ANSYS, it is meshed according to the conditions and its shape & size in ANSYS CFD module. Most of the time general settings are chosen when meshing any object in ANSYS but one can change several options provided in ANSYS along with shape and size of the elements generated through meshing.
- **Boundary Conditions-** In CFD module of ANSYS, next step is to provide various boundary conditions that include Nozzle dimensions, Inlet velocity, inlet pressure, Different shapes of Nozzles (Rectangular, Square & Circular), Different Nozzle Angles, Different Area ratios, Flow Parameters & Pressure Ratio etc.
- **Problem Solving-** once the boundary conditions are set, CFD solver of the ANSYS is made to solve the problem. In solving the problem 300 iterations have been chosen so that the solution converges & proper results are obtained.
- **Post Processing-** After the problem is solved, the post processing process begins where several results, contours, streamlines & solutions patterns are derived.
- **Data Analysis-** In this last phase of the methodology, data obtained is analysed for numerous permutations & combinations & the best feasible solution for the analysis of nozzle is obtained.

IRJMETS
e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

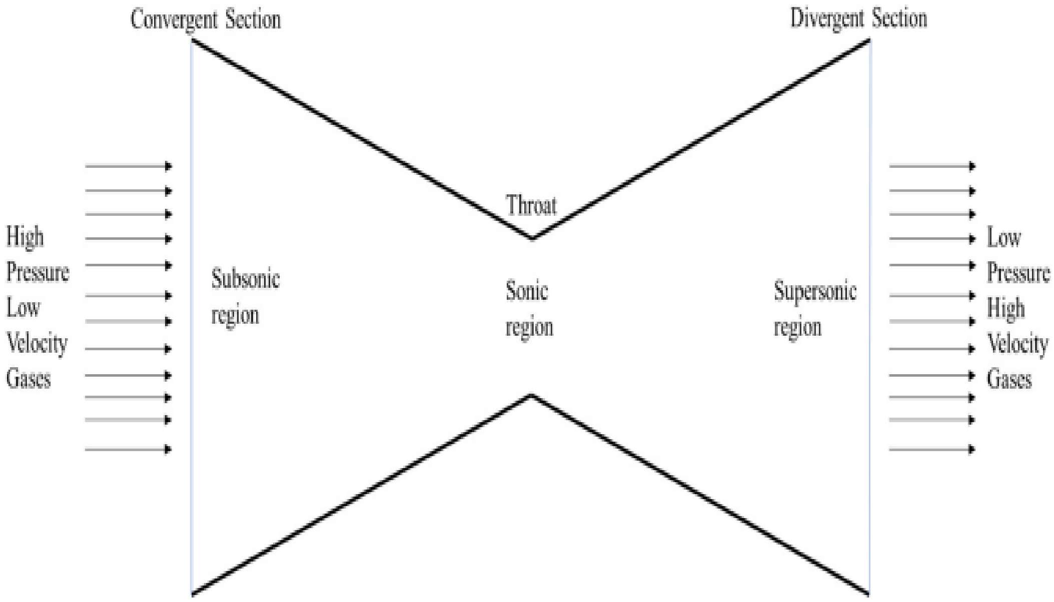

Figure 3: A Common Setup of Convergent-Divergent Nozzle

3.1 Problem Statement

Current problem analyses a convergent-divergent nozzle for different flow parameters, shapes & materials so that the flow across the nozzle is optimized. Dimensions of the nozzle to be employed for the current research study are presented in the following table.

Table 1: Dimensions of Nozzle

Parameter	Dimension
Total Nozzle Length (mm)	484
Inlet Diameter (mm)	166.6
Throat Diameter (mm)	34.5
Outlet Diameter (mm)	183
Chamber Length (mm)	99.93

3.2 Process Parameters

The convergent-divergent nozzle is to be analyzed for its performance on the basis of several performance parameters. Parameters that have been considered in the current analysis are presented here for the reference-

- Input Parameters
  - Nozzle dimensions
  - Inlet pressure
  - Flow Parameters
  - Pressure Ratio
- Output Parameters
  - pressure distribution
  - Velocity Distribution
  - Wall Shear
  - Flow characteristics
  - Drag

3.3 Formulae to be Used in Calculating Nozzle Parameters

- **Continuity Equation**

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

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

- **Steady Flow Energy Equation**

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

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

By solving equations (1) and (2) concurrently, we obtain the precise values of velocity (Vx) and temperature (Tx) at the specified region of the nozzle. The pressure at the section may be determined by applying isentropic principles. The subscripts "x" and "th" represent specific sections and throats, respectively.

The 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)

Q– Heat input to the system (J)

W– Work done by the system (J)

m– Mass flow rate (kg/s)

IV. RESULTS AND DISCUSSION

In the previous chapter, the data related with CFD analysis through ANSYS simulation software package was completed, the data was extracted & presented in tabular form. Current chapter deals about analyzing & interpreting the data. Nozzle has been analyzed for its performance on the basis of Pressure, Velocity, Temperature, Mach Number, Wall Shear, Coefficient of Drag & Drag Force. For analyzing three materials including Aluminum, Titanium & Steel have been selected. Following sections discuss numerous results obtained through analysis along with their comparison against each other

4.1 Pressure Distribution

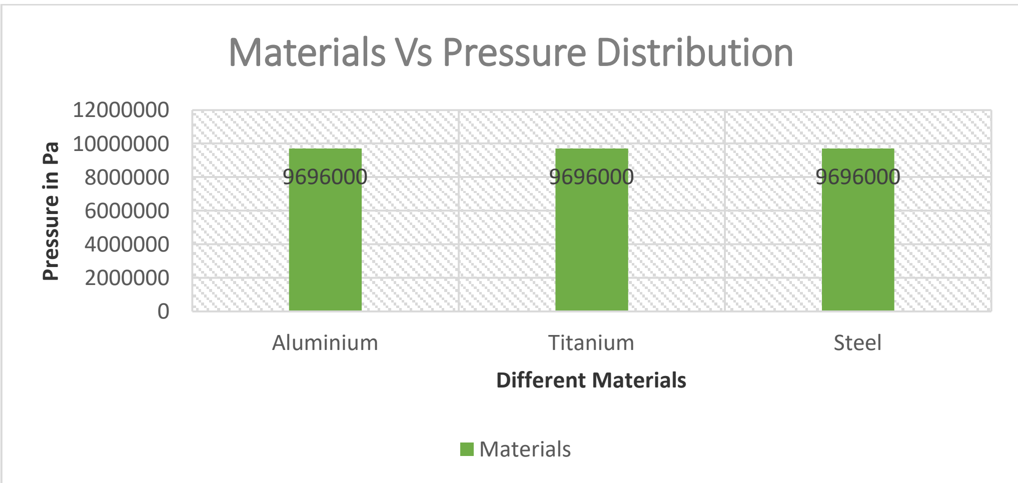

Figure 4: Pressure Distribution

Above Fig. 4 Represents the distribution of Pressure for all three materials including Aluminum, Titanium & Steel after applying the initial boundary conditions. Results of the pressure distribution across materials reveals that maximum pressure inside the nozzle remains same for all the three materials. This shows that after applying initial pressure in the problem, Flow of working fluid across the nozzle remains unaltered for all the materials. This concludes that pressure of the working fluid is independent of the material that is being used.

4.2 Velocity Distribution

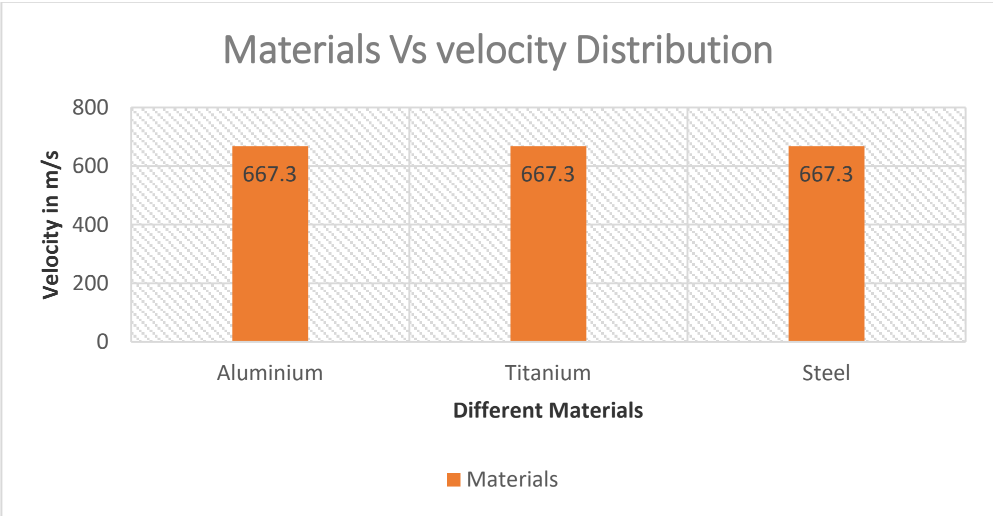

Figure 5: Velocity Distribution

Above Fig. 5 Represents the distribution of velocity for all three materials including Aluminum, Titanium & Steel after applying the initial boundary conditions. Results of the velocity distribution across materials reveals that maximum velocity inside the nozzle remains same for all the three materials. This shows that after applying initial pressure in the problem, Flow of working fluid across the nozzle remains unaltered for all the materials. This concludes that velocity of the working fluid is independent of the material that is being used.

4.3 Temperature Distribution

Below Fig. 6 Represents the distribution of temperature for all three materials including Aluminum, Titanium & Steel after applying the initial boundary conditions. Results of the temperature distribution across materials reveals that maximum temperature inside the nozzle remains nearly same for all the three materials. This shows that after applying initial pressure in the problem, temperature of working fluid across the nozzle remains nearly unaltered for all the materials. This concludes that temperature of the working fluid remains same for all materials for this input set.

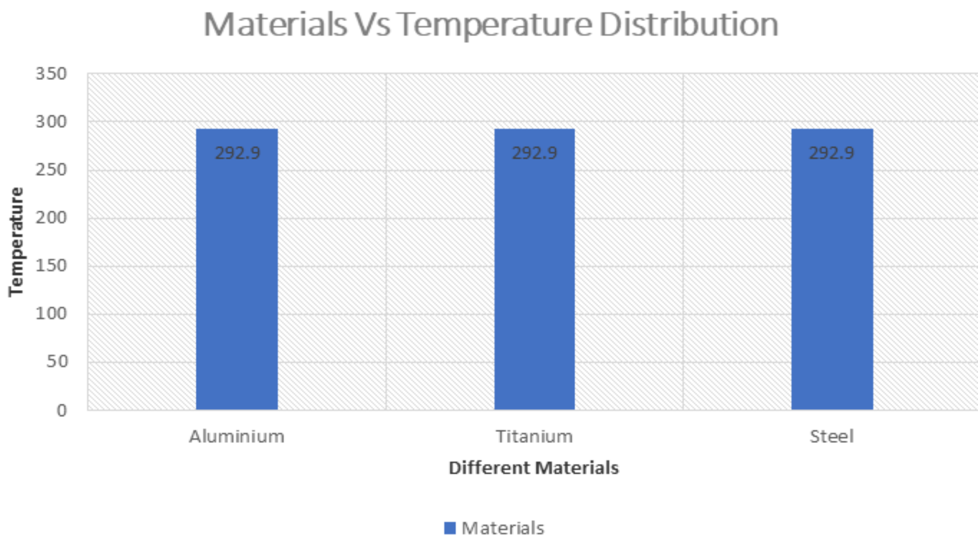

Figure 6: Temperature Distribution

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

4.4 Mach Number Distribution

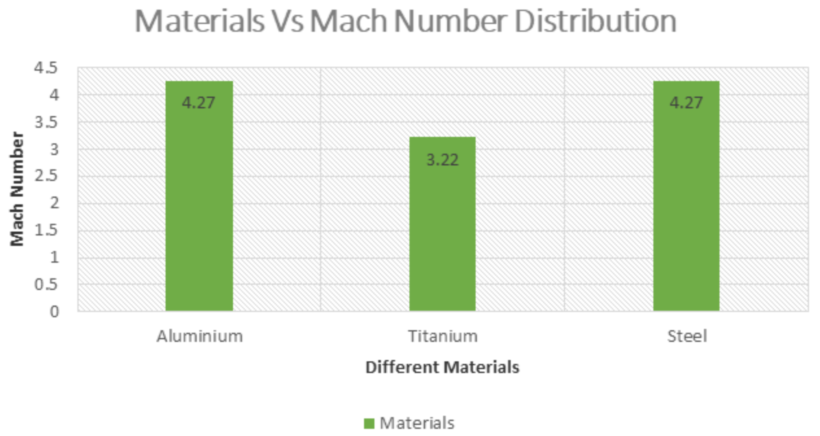

Figure 7: Mach number Distribution

Above Fig. 7 Represents the distribution of Mach Number for all three materials including Aluminum, Titanium & Steel after applying the initial boundary conditions. Results of the Mach Number distribution across materials reveals that Mach number is equal for Aluminum & Steel but reduces for steel material. This concludes that for the current application, Aluminum & Steel materials can be selected.

4.5 Wall Shear Distribution

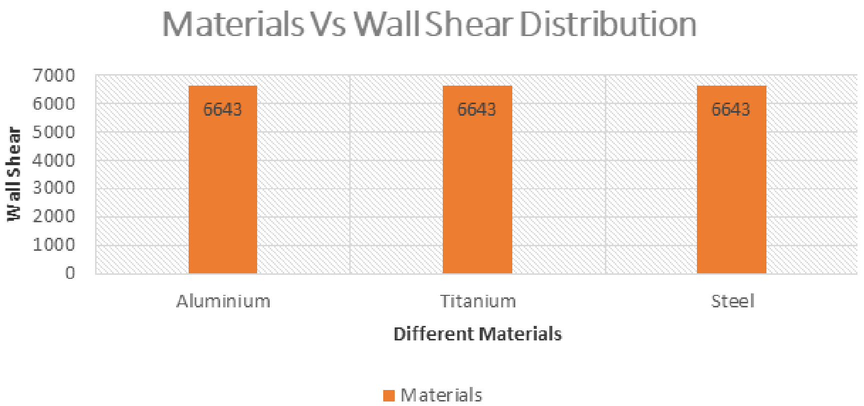

Figure 8: Wall Shear Distribution

Above Fig. 8 Represents the distribution of wall shear for all three materials including Aluminum, Titanium & Steel after applying the initial boundary conditions. Results of the wall shear distribution across materials reveals that wall shear is equal for Aluminium, Titanium & Steel all three materials. This concludes that wall shear of the working fluid is independent of the material that is being used.

4.6 Coefficient of Drag Distribution

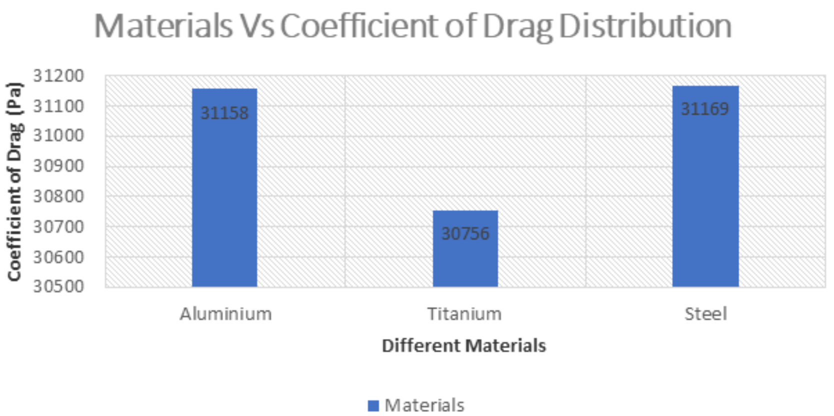

Figure 9: Coefficient of Drag Distribution

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:06/Issue:04/April-2024 Impact Factor- 7.868 www.irjmets.com

Above Fig. 9 Represents the distribution of Coefficient of Drag for all three materials including Aluminium, Titanium & Steel after applying the initial boundary conditions. Results of the Coefficient of Drag distribution across materials reveals that Coefficient of Drag is maximum for Steel which lowers for Aluminium then remains lowest for Titanium. This concludes that Coefficient of Drag of the working fluid is dependent of the material that is being used & Steel proves to be the best material out of the lot.

4.7 Drag Force Distribution

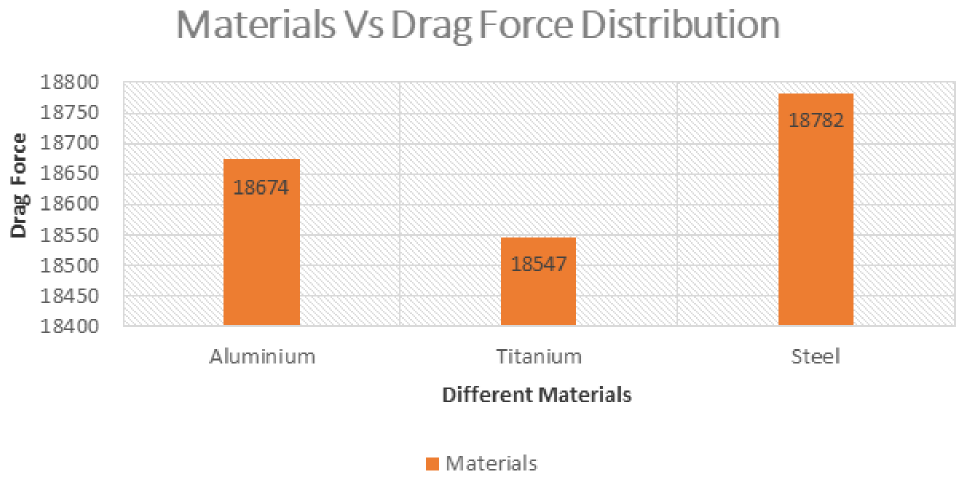

Figure 10: Drag Force Distribution

Above Fig. 10 Represents the distribution of Drag Force for all three materials including Aluminium, Titanium & Steel after applying the initial boundary conditions. Results of the Drag Force distribution across materials reveals that Drag Force is maximum for Steel which lowers for Aluminium then remains lowest for Titanium. This concludes that Drag Force of the working fluid is dependent of the material that is being used & Steel proves to be the best material out of the lot.

V. CONCLUSION

Process parameter wise conclusion for the given set of inputs is as follows-

- Pressure distribution across the materials reveals no change in pressure value for all the materials which remains same at 96.96 bar (9.696×10^6). This shows that material doesn't affect the pressure inside the nozzle for given input parameters.
- Velocity distribution across the materials reveals no change in velocity for all the materials which remains same at 667.3 m/s. This shows that material doesn't affect the velocity inside the nozzle for given input parameters.
- Temperature distribution across the materials reveals no change in temperature value for all the materials which remains same at 292.9 K. This shows that material doesn't affect the temperature inside the nozzle for given input parameters.
- Mach Number distribution across the materials reveals that for Aluminium & Steel remains same as 4.27 and for Titanium it remains to be equal to be 3.22. Hence For the current application Aluminium & Steel materials can be used.
- Wall Shear Distribution across the materials reveals no change in Wall Shear value for all the materials which remains same at 6643 Pa. This shows that material doesn't affect the Wall Shear inside the nozzle for given input parameters.
- Coefficient of Drag distribution for all the materials reveals Coefficient of Drag is highest for Steel material at 31169 which reduces for Aluminium at 31158 & remains lowest for 30756. It reveals that Steel is best suited material for the current application.
- Coefficient of Drag distribution for all the materials reveals Coefficient of Drag is highest for Steel material at 31169 which reduces for Aluminium at 31158 & remains lowest for 30756. It reveals that Steel is best suited material for the current application.

Overall, it can be concluded that Aluminium as well as Steel materials are better suitable for the current application.

5.1 Future Scope

No analysis is complete in itself and there always remains some scope of modification & improvement for further researches. Some of the points that may be useful area for futuristic research as far as nozzles are concerned, are mentioned here for reference-

- Current analysis has been conducted keeping the input pressure in mind only. Future researches may be conducted for several input parameter sets so that material selection can be generalized.
- This analysis has been conducted using k-ε, standard viscosity model using enhanced wall treatment option. Futuristic researches can be conducted using other type of wall functions, other viscosity models & inviscid physical model so as to observe the results & compare them with results of current research.
- Future researchers must look for correcting the random errors occurred during the CFD analysis of current problem for example calculations of Mach Number & Velocity.

VI. REFERENCES

[1] Satyanarayana, Ch. Varun, S.S. Naidu (2013). "CFD Analysis of Convergent-Divergent Nozzle", Acta Technica Corviniensis – Bulletin of Engineering, vol.-4, issue 3, pp. 139-144.

[2] Nikhil D. Deshpande, Suyash S. Vidwans, Pratik R. Mahale, Rutuja S. Joshi, K. R. Jagtap. "Theoretical & CFD Analysis Of De Laval Nozzle", Proceedings of 4 th IRF International Conference, Pune, 16th March-2014, ISBN: 978-93-82702-66-5, pp. 61-64

[3] Khizar Ahmed Pathan, S.A. Khan, P. S. Dabeer (2017). "CFD Analysis of Effect of Flow and Geometry Parameters on Thrust Force Created by Flow from Nozzle", 2nd International Conference for Convergence in Technology (I2CT) Siddhant College of Engineering, Pune, India. Apr 7-9, 2017

[4] Sher Afghan Khan, Omar Mohamed Ibrahim, Abdul Aabid (2021). CFD analysis of compressible flows in a convergent-divergent nozzle. Materials Today: Proceedings https://doi.org/10. 1016/j. matpr. 2021. 03. 074

[5] Malay S Patel, Sulochan D Mane and Manikant Raman, Concepts and CFD Analysis of De-Laval Nozzle. International Journal of Mechanical Engineering and Technology, 7(5), 2016, pp. 221–240.

[6] Sher Afghan Khan, Abdul Aabid, Maughal Ahmed Ali Baig. "CFD Analysis of CD Nozzle and Effect of Nozzle Pressure Ratio on Pressure and Velocity for Suddenly Expanded Flows", International Journal of Mechanical and Production Engineering Research and Development (IJMPERD) ISSN(P): 2249-6890; ISSN(E): 2249-8001 Vol. 8, Issue 3, Jun 2018, 1147-1158

[7] Prapti Joshi, Tarun Gandhi, Sabiha Parveen. "Critical Designing and Flow Analysis of Various Nozzles using CFD Analysis", International Journal of Engineering Research & Technology (IJERT), Vol. 9 Issue 02, February-2020

[8] Mustafa Atmaca; Berkay Çetin; Cüneyt Ezgi; Ergin Kosa; (2021). CFD analysis of jet flows ejected from different nozzles. International Journal of Low-Carbon Technologies https://doi.org/10. 1093/ijlct/ ctab022

[9] Uttam Kumar, Sudhir Singh Rajput, Dr. Praveen Borkar. "CFD analysis and parameter optimization of Divergent Convergent Nozzle", International Journal of Advance Research and Development, Volume 3, Issue 10, pp. 9-13

[10] Rajat Mishra, Devendra Lohia. "CFD Analysis of Convergent and Divergent Nozzle", International Research Journal of Engineering and Technology (IRJET), Volume: 07 Issue: 05 | May 2020, pp. 7639-7643

[11] Khalid, M. W., & Ahsan, M. (2020). Computational Fluid Dynamics Analysis of Compressible Flow Through a Converging-Diverging Nozzle using the k-ε Turbulence Model. Engineering, Technology & Applied Science Research, 10(1), 5180–5185. https://doi.org/10.48084/etasr.3140

[12] Munipally Prathibha, M. Satyanarayana Gupta, Simhachalam Naidu. "CFD Analysis on a Different Advanced Rocket Nozzles", International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249 – 8958, Volume-4 Issue-6, August 2015, pp. 14-22.

[13] Benzenine, Fadela; Seladji, Chakib; Darfilal, Djamal (2019). [IEEE 2019 9th International Conference on Recent Advances in Space Technologies (RAST) - Istanbul, Turkey (2019.6.11-2019.6.14)] 2019 9th International Conference on Recent Advances in Space Technologies (RAST) - CFD Analysis of Two Supersonic Nozzles Designed for a High-Test Peroxide Thruster, 207 213. https://doi.org/10. 1109/RAST. 2019. 8767802

[14] Pankaj Kumar Singh, Amrendra Tripathi. "CFD Analysis of De-Laval Nozzle", International Journal of Advance Research and Innovative Ideas in Education, Vol-3 Issue-2 2017, pp. 4390-4411

[15] Bhawarker, Yogesh Mr.; Patidar, Ritik; and Katdare, Prakash (2022) "CFD Analysis of Nozzle Flow with Sudden Expansion in Aerospace Engineering," Graduate Research in Engineering and Technology (GRET): Vol. 1: Iss. 7, Article 15. https://doi.org/10.47893/GRET.2022.1119

[16] Khan, A., Aabid, A., & A. Khan, S. (2018). CFD analysis of convergent-divergent nozzle flow and base pressure control using micro-JETS. International Journal of Engineering & Technology, 7(3.29), 232-235. http://dx.doi.org/10.14419/ijet.v7i3.29.18802

[17] Gandikota Babu Vishnu Kishore, Kaparthy Akash. "CFD Analysis of a Rocket Nozzle with one Inlet at Mach 0.6", Dissertation, Malla Reddy College of Engineering and Technology, 2014-15

[18] Pardhasaradhi Natta, V. Ranjith Kumar, Dr. Y. V. Hanumantha Rao. "Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (CFD)", Engineering Research and Applications (IJERA), Vol. 2, Issue 5, September- October 2012, pp.1226-1235

[19] Vishwajeet Yadav, Prof. Pawan Kumar Tiwari. "A Review on CFD Analysis of Effects of Convergence and Divergence Angles on the Performance of a Nozzle", International Journal of Science, Engineering and Technology, 2020, 8:1

[20] Shaik Khaja Hussain, B V Amarnath Reddy, A V Hari Babu. "Modelling and Computational Fluid Dynamic Analysis on Jet Nozzle", International Journal of Scientific Development and Research (IJSDR), 2016, Volume 1, Issue 9

[21] Xilai ZHANG, Shiping JIN, Suyi HUANG, Guoqing TIAN. Experimental and CFD analysis of nozzle position of subsonic ejector. Front Energ Power Eng Chin, 2009, 3(2): 167–174 https://doi.org/10. 1007/s11708 -009-0001-5

[22] Kaviya sundar, Thanikaivel Murugan. D. "CFD Analysis of Supersonic Nozzle with Varying Divergent Profile", International Journal of Engineering and Technology (IJET), Vol 9 No 3 Jun-Jul 2017

[23] Satheesh Kumar K V, Dharmaraj M, Udhayan M, Surya M R, Vedhasagaran M. (2020). CFD Analysis on Nozzle Inclinations in Forced Air-Cooling System for Industries. International Journal of Advanced Science and Technology, 29 (3), 9670 - 9691. Retrieved from
http://sersc.org/journals/index.php/IJAST/article/view/26940

[24] Anubhav Vishwakarma, Devesh Kumar. "CFD Analysis of Interaction of The Fluid Structure of The Convergent divergent Nozzle", International Journal of Creative Research Thoughts (IJCRT), Volume 6, Issue 1 March 2018

[25] Goyal S., Singh S. (2021). "CFD Analysis of Supersonic C-D Nozzle for Optimization of Divergent Angle", The International Journal of Engineering and Science (IJES), Volume-10, Issue-3, Series-II, PP 33-43

[26] Abdul Aabid; Sher Afghan Khan; Muneer Baig; (2021). A Critical Review of Supersonic Flow Control for High-Speed Applications. Applied Sciences, https://doi.org/10.3390/app11156899

[27] Nicholas J. Georgiadis; James R. DeBonis (2006). Navier–Stokes analysis methods for turbulent jet flows with application to aircraft exhaust nozzles. 42(5-6), 377 https://doi.org/10. 1016/j. paerosci. 2006. 12. 001

[28] C. Nadarajah; D. Mackenzie; J.T. Boyle (1996). Limit and shakedown analysis of nozzle/cylinder intersections under internal pressure and in-plane moment loading. 68(3), 261–272.

https://doi.org/10. 1016/0308-0161(95)00064-x

[29] Wen-Ya Li; Hanlin Liao; G. Douchy; C. Coddet (2007). Optimal design of a cold spray nozzle by numerical analysis of particle velocity and experimental validation with 316L stainless steel powder. 28(7), 2129–2137. https://doi.org/10.1016/j.matdes.2006.05.016

[30] Geng, Lihong; Liu, Huadong; Wei, Xinli (2019). CFD analysis of the flashing flow characteristics of subcritical refrigerant R134a through converging-diverging nozzles. International Journal of Thermal Sciences, 137, 438–445. https://doi.org/10.1016/j.ijthermalsci.2018.12.011

[31] Pothala, S., Raju, M.V.J., Bangarraju, B., Lakshmi Narayana, V. (2022). Computational Analysis of a Convergent Divergent Nozzle. In: Deepak, B.B.V.L., Parhi, D., Biswal, B., Jena, P.C. (eds) Applications of Computational Methods in Manufacturing and Product Design. Lecture Notes in Mechanical Engineering. Springer, Singapore. https://doi.org/10.1007/978-981-19-0296-3_46

[32] Chrystella Jacob; T. Sasipraba; Lilly Mercy Jayaraman; Sanjay Paul Bedford; S. Ganesh; (2020). Simulation of converging-diverging (CD) ejector nozzle and its performance analysis. 3rd International Conference on Frontiers in Automobile and Mechanical Engineering (Fame 2020) AIP Conference Proceedings. https://doi.org/10.1063/5.0034541

[33] P. Vadivelu, G. Senthilkumar, G. Sivaraj, D. Lakshmanan, Nayani Uday Ranjan Goud. “Performance analysis of CD nozzle”, Materials Today: Proceedings, Volume 64, Part 1, 2022, Pages 620-631, ISSN 2214-7853, https://doi.org/10.1016/j.matpr.2022.05.148

[34] Md Mashiur Rahman. “Test and performance optimization of nozzle inclination angle and swirl combustor in a low-tar biomass gasifier: a biomass power generation system perspective”, Carbon Resources Conversion, Volume 5, Issue 2, 2022, Pages 139-149, ISSN 2588-9133,

https://doi.org/10.1016/j.crcon.2022.01.002

[35] Shabani S, Majkut M, Dykas S, Smołka K, Lakzian E, Zhang G. Validation of the CFD Tools against In-House Experiments for Predicting Condensing Steam Flows in Nozzles. Energies. 2023; 16(12):4690.