*International Journal of Application or Innovation in Engineering & Management (IJAIEM)*
**Web Site: www.ijaiem.org Email: editor@ijaiem.org**
**Volume 11, Issue 5, May 2022** **ISSN 2319 – 4847**

# A REVIEW PAPER ON OPTIMIZATION FLOW ANALYSIS OF SUPERSONIC NOZZLE USING CFD

**Mr. B. Phanindra Kumar[1], A.Manasa[2], B.Sai[2], K. Sangeetha[2]**

[1]Assistant Professor, Mechanical Engineering, GNIT

[2]Student, Mechanical Engineering, GNIT

## ABSTRACT

A Nozzle is a device which is used to control the fluid flow and directions. The design is considered based on the pipe cross sectional area & length of the pipe. It is used for converting the pressure energy into kinetic energy. There are different types of Nozzles used according to the requirement. Most commonly used nozzles are Convergent nozzle, Divergent nozzle and Convergent Divergent Nozzle. Convergent Nozzle is used to increase the pressure of the fluid and it decreases the cross-sectional area. Divergent Nozzle is quite opposite to the convergent nozzle it increases the cross-sectional area of the tube to decrease the pressure. Convergent-Divergent Nozzle is the combination of both convergent and divergent nozzles. Nozzles are frequently used to control the rate of flow, speed, direction, mass, shape and the pressure of the stream that emerges from them. Nozzle is used in steam turbines, gas turbines, water turbines and in jet engines, jet propulsion. Nozzles are used for flow measurement as it is used in Venturi meter. Nozzles are used to remove air from a condenser, injectors for pumping feed water to feed water to boilers and also used in artificial fountains. A convergent divergent nozzle with a variable area and profile through which the relative airflow is supersonic is known as supersonic nozzle. The expansion of supersonic flow causes the static pressure and temperature to decrease from the throat to the exit, so the amount of the expansion also determines the exit pressure and temperature. Supersonic nozzles are used in a variety of engineering applications to expand a flow to desired supersonic conditions. Here Supersonic jet nozzle is considered for optimization flow analysis. Aluminum alloy is the metal considered for analysis. Catia is used for designing the nozzle and Ansys is used for analysis of the nozzle. Computational Fluid Dynamics (CFD) is used for flow analysis of the nozzle. Thermal analysis and flow analysis are done using CFD. Boundary conditions like pressure and temperature are applied on the nozzle. As supersonic nozzle has wide variety of applications it is necessary to optimize the flow through a supersonic nozzle with minimum losses. By CFD analysis of supersonic nozzle high concentration of stress, strain and deformation of nozzle is observed. By thermal analysis expansion of high temperature area and deformation due to high temperature is observed. This analysis helps us to improve the nozzle design and to optimize the flow of the fluid through supersonic nozzle. Further, this helps in improvement of efficiency of supersonic nozzle.

## 1. INTRODUCTION

A Nozzle is a device which is used to control the fluid flow and directions. The design is considered based on the pipe cross sectional area & length of the pipe. It is used for converting the pressure energy into kinetic energy.

### Types of Nozzles:

Convergent Nozzle is used to increase the pressure of the fluid and it decreases the cross-sectional area.

Divergent Nozzle is quite opposite to the convergent nozzle it increases the cross-sectional area of the tube to decrease the pressure.

Convergent-Divergent Nozzle is the combination of both convergent and divergent nozzles.

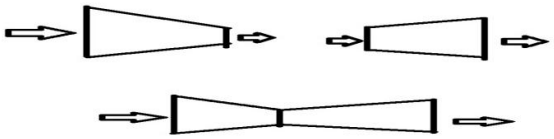

Fig: Types of nozzles

Mach number is the ratio of the speed velocity to the speed of the sound at surrounding medium.
Sonic nozzle is defined as the inlet flow is equals to the outlet flow of the region.
Supersonic flow is the speed of an object that is greater than the speed of the sound. It ranges up to 1.3 to 5.0.
Hypersonic means that travels that above 5X(times) speed sound. Its ranges up to greater than 5 (>5)

## 2. Literature Survey:

**Pathan Khizar Ahmed** This paper says that intensity of velocity is found to have an increasing trend with increment in divergent angle thereby obtaining an optimum divergent angle, eliminates instabilities due to shock and satisfy the thrust requirement from the nozzle. CFD analysis is conducted to analyze flow pattern. At different divergent angles velocity magnitude, Static pressure, turbulent intensity and static temperature are analyzed using CFD. The author says that oblique shocks are formed during flow through the nozzle, when the divergent angle was low and as the divergent angle is increased the shock is completely eliminated from the nozzle. This could be considered as good design for the nozzle. [1]

**Snehil Varghese** In this paper the convergent-divergent nozzle with varying cross- section are designed and are investigated by numerical simulations. By keeping all input parameters constant divergent angle is varied, Velocity, pressure and Mach number contours are compared for every nozzle configuration using CFD. It is found that the study says that 9.5degrees nozzle angle is the best suited for the maximization outlet thrust and velocity. Small variation of nozzle angle leads to reverse flow in the divergent section which is not acceptable because high negative pressure was observed in the whole flow region. The limitation of this study is that the flow is taken to be turbulent thus the study is not applicable for certain applications. [2]

**Bovas Herbert Bejaxhin ALPHONSE Mohamed K**. In this study, the compressed air is used considering it as perfect gas, the desired nozzle is designed and evaluated. The author says that the internal compressible flow along a passage is usually affected by a number of different conditions like shear stress, heat transfer, cross-sectional area change and shock and expansion waves. The performance of aircrafts depends on the improved and properly integrated propulsion systems which rely on the design of the nozzle. This study can be utilized in creating software that is used for the analysis of compressible flow and also utilized for different gases and different operating conditions. This involves reducing the required driving pressure by modifying the geometry of the passage. [3]

**Dushyant Kalihari** In this paper, for two-dimensional study, MATLAB program is developed for scheming minimum length of the supersonic nozzle for the optimum ratio at the nozzle exit with uniform flow at the divergent section of the nozzle. The planning issues square measure targeted at the radiating section. This can be obtained by contacting the growth section and reducing the entire length of the nozzle. The study says that the essential feature needed to realize steady, sustained supersonic flow is that the nozzle contour. The required exit ratio is achieved [4]

**Brijesh Patel** This paper consists of Computational Analysis of a Convergent Divergent Nozzle. Ejector mainly uses the principle of fluid dynamics for pumping. Software SED is used for calculations. 3D models are developed using AutoCAD. CFD analysis in ANSYS is done, Mach number, pressure and a temperature variation at different fluid positions along the

flow is applied. Unlike pressure and Mach number the value of inlet temperature decreases at throat and increases gradually. At the end the convergence is achieved with the designed shape of the CD nozzle to drain out the low-pressure exhaust gases to the ambient. This says that the rate of decrease of temperature is directly proportional to the inlet fluid temperature. [5]

**S. A. Khan** In this paper has been administered with AN aim to introduce an additional economical and higher activity nozzle. In increasing procedure Fluid Dynamics connected explorations in industries; the implementation of additional economical and better activity nozzles is going to be essential. This project's objective is to bring nearer to the fact. This includes style of various shapes of nozzles on the idea of reference knowledge that had given a practical impact to the analysis of the nozzle and conjointly evaluated their performance. CFD has given a practical approach of testing of those nozzles by applying normal, constant and ranging conditions to the various parameters module through analysis results are obtained and evaluated which is able to be primary supply for the more exploration within the style and analysis of nozzles. These can kind the bottom for more analysis in style and development of various shapes of nozzle. [6]

**P. S. Dabeer** In this paper propulsion installations have a major impact on the general potency of engine systems, significantly for supersonic and hypersonic flight vehicles. To assess the impact of a nozzle style on World Wide Web thrust and specific fuel consumption for a given engine style, either the nozzle performance characteristics should be known beforehand, analyzed to work out performance. This report describes a series of analyses that are developed into a performance for engine nozzle systems. The methodology will be accustomed predict performance for a given nozzle geometric style. [7]

**Mihir Baranwal** This paper presents the CFD analysis to check the result of space and nozzle pressure on the flow of axe-symmetric fast enlargement from convergent-divergent nozzles to a circular duct of larger cross-sectional space than that of nozzle exit space. The study is that specialize in the pressure and rate distribution on its length. The analysis is completed by variable the realm ratios and nozzle pressure ratios. Zone-type specifications explain the physical and operational characteristics of the model at its boundaries and among specific regions of its domain. To research all the cases, the boundary conditions outlined in annoys fluent square measure pressure recess at recess and pressure outlet at outlet. [8]

**Vikky Chobey** In this paper Nozzle part of a rocket has been below constant development and analysis for higher potency and performance. A nozzle is essentially a 2-Dpipe visualized in 3-Dwith varied cross- sectional space accustomed direct and accelerates the flow of gases created by the combustion chamber. This analysis includes study of style of many nozzles and examination them with the present ones as within the advancing years varied organizations square measure going to explore different planets for analysis functions to check their habitat and to seek out an improved home. There would be issue in operative nozzles wherever the atmosphere is totally different than on Earth. There square measure several crucial planets wherever the atmosphere consists of the many different gases like gas, helium, gas and dioxide wherever the nozzle efficiency should be maintained. So, the current study incorporates varied parameters like Mach speed, temperature and pressure. A series of CFD simulations square measure carried out to know the nozzle potency at totally different conditions of temperature by critical appraisal of nozzles in ANSYS Fluent and planning in CatiaV5. [9]

**G. Susheel Narayan** A rocket needs high thrust with that it might fly within the sky and location that is achieved by convergent divergent nozzle. A convergent-divergent nozzle could be an outstanding a part of rocket such to produce optimum thrust to drive it into the skies with high rate. Exhaust gases setting out of the nozzle have supersonic flow i.e., their Mach number is larger than one. The target of this paper is to interpret and analyze however the modification in throat diameter changes the thrust of focused divergent nozzle. This study computes the flow of gases within the focused divergent nozzle victimization commercially offered procedure fluid dynamics (CFD) tool ANSYS FLUENT. Totally

different completely different models are taken that differ in their throat diameter and also the material used and also the outcomes have been compared to the quality ones presently used. [10]

**P. Mani Kiran** In this paper CFD may be a branch of hydraulics that admit numerical ways and algorithms to resolve and analyze downside that involves fluid flow. CFD analysis has been wont to analyze flow pattern of supersonic rocket nozzle at varied degree of divergent angle, physicist numbers etc. This paper aims to check the behavior of flow in convergent divergent nozzle by analyzing varied parameters like static pressure, temperature and speed in physicist No mistreatment procedure fluid dynamics computer code (C.F.D). These results were additional premeditated comparison them with analytical values. During this section we tend to prove the continuity Equation within the C-D Nozzle with necessary knowledge. [11]

**Kumar Uttam et al** The author says that the experimental and CFD results of mach number for Divergent Convergent Nozzle shows very good agreement for same boundary conditions. Pressure contour is maximum at inlet and it is falling down with the axial length of the nozzle and the velocity gradient is low at inlet and it goes up with the axial length of the nozzle. Computational Fluid Dynamics software is used to obtain the values of pressure and Mach number in Ansys software. The author used CVM (Computational Volume Method) to solve the governing equation of the fluid flow under boundary conditions. Divergent angle is varied and the nozzle is analyzed using CFD. Optimization of divergent convergent angle is also done by the author.[12]

**MD. Safayet Hossain** The author compared flow analysis of two different nozzles. The input parameters are kept constant. Modelling is done and Computational fluid Dynamics software is used for analysis, then the results are compared. The difference between the two nozzles is the divergent angle. It is observed that the nozzle with more outlet divergence angle gives higher velocity at outlet than the nozzle with low divergence angle, the nozzle with high divergence angle has high mach number and the nozzle with low divergence angle has low mach number compared to the nozzle with high divergence angle. Pressure for the nozzle with high divergence angle is less at the outlet and the pressure at the outlet is more for the nozzle with the low divergence angle. The author also says that the variations in different parameters and properties of nozzles are occurred due to change in divergence angle from throat towards the expansion zone.[13]

**Venkatesh C Jaya pal Reddy** This paper provides a discussion about the design procedure of supersonic convergent divergent nozzle. The Convergent Divergent nozzle both conical and contour is designed by the author. He assumed that the gas flowing through the supersonic nozzle is ideal gas, the flow is isentropic, frictionless and adiabatic, the gas flow is constant, the gas flow is along a straight line from inlet to exhaust of the nozzle, and the flow is compressible since the flow is at very high velocities. Fluent is utilized to solve the equations and the author observed that the counter nozzle gives a greater expansion ratio comparatively to a conical nozzle. The conical nozzle has a simple geometry and easy to fabricate whereas the Contour nozzle has a complex geometry and is difficult to fabricate.[14]

**Tapas Kumar Nandi and Prosun Roy** The author says that we have accurately model the inertia force for compressible flow simulation. The compressible flow in convergent divergent nozzle is investigated by using Finite Volume Method (FVM) for solving partial differential equations; the normal shock had been obtained by using gas table. The flow is considered a two dimensional compressible and supersonic nozzle, characterized by the presence of walls requires the consideration of viscosity effects. It is observed that oblique shocks are formed during flow through the nozzle and the shock is completely eliminated when the divergent angle is increased. The static pressure decrease with increasing divergent angle, the shock goes on increasing by decreasing the operating pressure ratio. Near the wall the Mach number is decreasing for all the nozzles, this is due to the viscosity and turbulence in the fluid.[15]

**Yesu Ratnam.Maddu Md. Azeem** The author concentrated on the determination of the variation of flow parameters like pressure, temperature, velocity and density. Computational Fluid Dynamics software is used for analysis. In this paper the author used Finite Volume Method to study flow through the convergent divergent nozzle. The author used PTC Creo Parametric 3.0 for modelling of the nozzle geometry. Boundary Conditions like pressure and temperature at inlet and outlet are applied. The analysis of Convergent Divergent nozzle is carried out in ANSYS to obtain the flow properties like temperature, pressure, velocity and density. The Author says that velocity is increased rapidly and at the exhaust velocity is very high, the density of the fluid flow through the nozzle changes from inlet to the outlet gradually. He says that the flow is supersonic before the shock and subsonic after the shock by observing the shock waves, the higher the Mach number the higher shock will be generated.[16]

**C.SATHEESH A.ARULMURUGU** The author observed flow properties like pressure, velocity, temperature and Area by varying the Mach number value. He says that there is no way to reach Mach number value of 1. CAD modelling software Solid works 2012 is used by the author to generate model of convergent divergent nozzle. It is observed that by increasing the convergent area increases the velocity in a convergent divergent nozzle. Analysis of the nozzle is done by using the Ansys fluent 12. He says that at end of the experiment the pressure of the nozzle is increased from convergent duct to divergent duct, velocity of the nozzle is increased from convergent duct to divergent duct. When two phase flow in convergent-divergent nozzle is carried out the feed water temperature is as low as possible below the boiling point at the inlet of the convergent divergent nozzles.[17]

**Vishwajeet Yadav, Pawan Kumar Tiwari** The author observed the effects of convergence and divergence half angles on the performance of a nozzle at different pressure ratios and is investigated numerically. The baseline geometry was modified by changing the convergence and divergence angles. The author kept the parameters constant by changing the convergent and divergent angles. ANSYS is used for simulation of the nozzle; CFD solver is used to carry out the Analysis work is ANSYS CFX. CFD study of the compressible through convergent conical nozzles to investigate the effect of the nozzle performance and observed that the Mach number increases from inlet to outlet and the shock is observed at the throat of the convergent divergent nozzle and also the velocity increase from inlet to outlet continuously.[18]

**Anup Singh Nikita Shukla** The author analyzed the Rocket Engine Nozzle usually being used in rocket engine; the convergent divergent nozzle has two major types of contours namely conical and parabolic. CFD is used for analysis of the rocket engine nozzle, the conical and parabolic nozzle contour using ANSYS Fluent. Taguchi Methodology which focuses on quality improvement and product and processes insensitive to manufacturing and environment variations is used for geometrical optimization of the nozzle. The author observed that the parabolic nozzle contour is better by analysis. Software tool Minitab is used for optimization. Author says that the Optimization resulted in maximization of thrust and Mach number considering the area ratio, angle ratio and length ratio as constants. The nozzle geometry is successfully optimized to obtain relatively higher thrust and velocity is found to be increased.[19]

**References:**

1. IOSR Journal of Mechanical and Civil Engineering (IOSR- JMCE) ISSN: 2320–334X PP 05-066
2. The International Journal of Engineering and Science (IJES) ISSN (online): 2321-0613
3. Research Gate (Mohamed K. Elshebani)
4. IJERTV ISSN: 2IS1-10026
5. International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
6. AEGAEUM JOURNAL ISSN NO: 0776-3808
7. Lewis Research Center under Contract NAS3-27186 NASA Contractor Report 20412
8. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320–334X PP 05-07
9. International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181

10. International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 p-ISSN: 2395-0072
11. International Journal of Aerospace and Mechanical Engineering ISSN (O): 2393-8609.
12. International Journal of Advance Research and Development © 2018
13. IJRET: International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321-7308
14. International Journal of Novel Research in Interdisciplinary Studies ISSN 2394-9716
15. African Journal of Science and Research, 2016, (5)4:61-66 ISSN: 2306-5877
16. International Journal of Engineering Technology Science and Research IJETSR www.ijetsr.com ISSN 2394 – 3386 Volume 5, Issue 4 April 2018
17. Scientific Journal Impact Factor (SJIF): 1.711 International Journals of Modern Trends in Engineering and Research e-ISSN: 2349-9745 p-ISSN: 2393-8161
18. International Journal of Science, Engineering and Technology (An open access journal) ISSN (Online): 2348-4098 ISSN (Print): 2395-4752
19. International Journal of Innovative Research in Science, Engineering and Technology ISSN (Online): 2319-8753 ISSN (Print): 2347-6710