# ANALYSIS OF CONVERGENT DIVERGENT NOZZLE WITH RESPECTIVE TO THE POSITION OF FLUID FLOW IN STANDARD AND REALIZABLE TURBULENT MODEL

**Devendra Kumar Lohia[a,*], Navneet Singh[b]**

[a,*] Head of Mechanical Engineering, Institute of Technology And Management, GIDA, Gorakhpur, INDIA

[b] Research Scholar, Lovely Professional University, School of Mechanical Engineering, Jalandhar, INDIA

## ABSTRACT

A branch of fluid mechanics, which uses numerical method and algorithms for solving and analysing the fluid related problems, is known as CFD (Computer Fluid Dynamics). And for the simulation of interaction of fluid and gases to the boundary regions, there is the need of computer.A mechanical device which is used to convert the chemical energy of fluid extracted from the chamber into the kinetic energy, is called as nozzle. In this research paper, ANSYS modelling is used to discuss about that how the static pressure of nozzle and Mach no is dependent to the position of the fluid flow. Static pressure is the pressure, when fluid is at rest position and measurement is taken along the fluid flow. It is measured with the gauges and transmitters, whether in the fluid dynamics the Mach number represents the ratio between the flow velocity past the boundary to the local speed of sound. This paper shows the comparison of the standard turbulent model to the realizable turbulent model. At the initial position of the fluid flow, the maximum static pressure is $3.00*10^5$pascal, in the case of standard turbulent model and realizable turbulent model. As the length or position of the nozzle in which fluid is flowing, is increases, the static pressure is simultaneously decreases.

Keywords: Computer Fluid Dynamics (CFD), C-D nozzle, Mach Number, Static Pressure.

## INTRODUCTION

For the production of thrust in aircraft and rocket engines, nozzle is a critical component. Nozzle can be classified as following - convergent, divergent, convergent-divergent (C-D) and single expansion ramp nozzle (SERN). As the convergent – divergent nozzle, word describe its meaning itself, to itself accelerate the flow of fluid of passing through it to a supersonic speed. It is used in steam turbine and the modern rocket engine and supersonic jet engine. For the use of impulse steam turbine, Swedish engineer Gustaf de Laval developed the nozzle in 1897. And its principle is used in the rocket engine by Robert Goddard. CFD (Computer Fluid Dynamics) has become a part of the engineering design. The critical requirements for CFD tool used in thermal applications are the ability to analyse flows along nozzles and turbines. For analysing these flows there are such features have pressure gradients, mach numbers, temperature distribution and velocity vectors etc. For increasing the performance of vehicle, it is essential to optimize the shape, length and contour of the nozzle. For the subsonic engine, the convergent nozzles are very suitable, whether the supersonic speed, the convergent-divergent (C-D) nozzles are very suitable. By increasing the kinetic energy of flow in expense of its pressure, nozzles are used to modify the flow of the fluid. For the supersonic flow, the convergent and divergent nozzles are very useful, because it is impossible to create supersonic flows (mach number is more than one) in convergent type of nozzle.

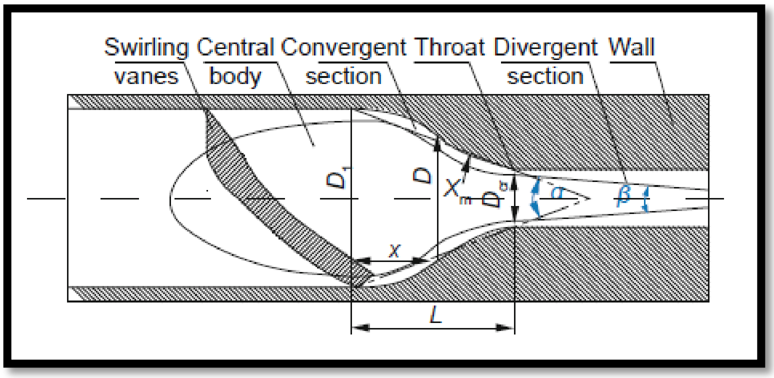


Fig 1: Structure of the nozzle [1]

The channel between the wall and the central body, forms a convergent – divergent nozzle, which is composed of three sections: convergent (subsonic zone), throat (critical zone) and divergent sections (supersonic zone), as shown in above figure1 [1].

Many of the studies have done for optimizing the ejector design and to increase the thrust. Huang et al. developed the one-dimensional model for prediction of supersonic ejector performance at critical mode. For the optimal value of mixing chamber diameter of a supersonic ejector, an integral method for design is presented by Antonio et al. It included a thermodynamic model based on isentropic flow of perfect gases with the addition of model of losses. The model was validated with three hundred steam ejectors and less than 5% relative errors. Croquer et al. did the comparison between the predictions of a thermodynamic model and CFD(Computer Fluid Dynamics) model. Thongtip and Aphornratna did the comparison between the six ejector nozzles and their effects on the performance of a refrigerator ejector system. In which two nozzles were having same throat diameter but different nozzle area ratio, whether four of them having same nozzle ratio and different throat diameter. They found that, operating at a lower generating temperature, using a bigger nozzle throat diameter gives a highly COP (coefficient of performance). This issue is solved by the use of C-D (Convergent – Divergent) nozzle, where the mass flow rate of the motive steam is changing with a throat diameter of nozzle.

## ASSUMPTIONS

The flow model uses isentropic relationships and linear nozzle geometry. For the calculations there are following assumptions [2], such as –

i. The gas follows the ideal gas law.
ii. For the gas flow there is no friction.
iii. The gas flow is adiabatic.
iv. Steady state conditions are to be implemented.
v. Gas expands inuniform manner and without shock or any discontinuities.
vi. Flow is one dimensional through nozzle.
vii. Particles do not influence gas conditions.
viii. Collision isneglected between the particles.
ix. Space charge does not affect by the effect of the presence of particles.
x. In the impinging region thermo fluid field does not affect by the formation of the thin coatings. [2]

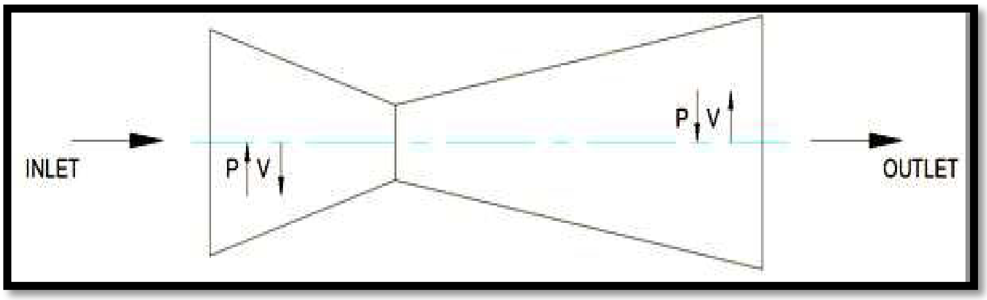


Fig 2: Principle of C-D Nozzle [2]

## DESIGN CONSIDERATIONS OF NOZZLE

The modelling of C-D nozzle is done by using ANSYS 14.0 (CFD Fluent) workbench software, as following steps: -

Step 1: Nozzle Configuration

In this paper, there are two types of nozzle are in under investigation. The ideal fluid is injected axially of the nozzle. The inlet operating conditions for the two nozzle configurations are same to measure the output conditions. The inlet gauge pressure at the nozzle is $3*10^5$Pa, at 300° K. There is assumption that nozzle surface is to be the wall surface having no slip. At the outlet of the nozzle, pressure $P_o$, velocity $V_o$, temperature $T_o$ is measured.

Step 2: Determination of Problem Domain

The geometry of problem is developed in this step. In this paper, the 2-D model of C-D nozzle is taken.

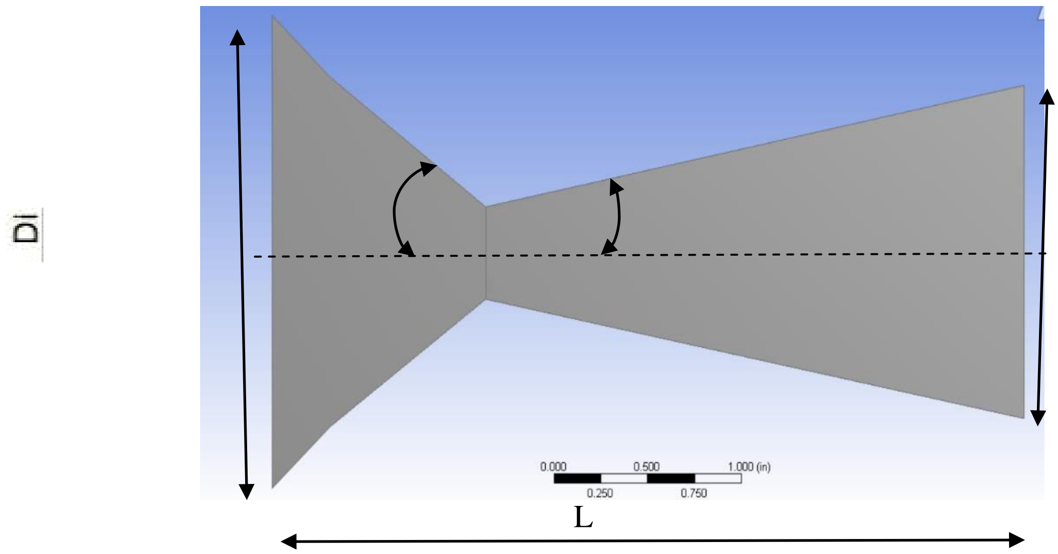


Fig 3: Nozzle Configurations

Step 3: Meshing and apply the boundary conditions

The total surface is meshed using ANSYS meshing. And the nozzle of inlet is known as pressure inlet, outlet is as pressure outlet and outer surface of the nozzle is as the wall and the inner surface is for fluid flow.

Boundary Conditions:

The boundary conditions described for flow through the density based k-ε standard turbulent and realizable C-D nozzle model in the process are given below: -

At the inlet of the nozzle

Pressure (Pi) = $3*10^5$ Pa

T =300° K

At the nozzle surface, $V_{x1} = 0$

Fluid = air (ideal gas)

Ideal gas viscosity = Sutherland

**ANALYSIS FOR C-D NOZZLE –**

---

The analysis is carried out in the ANSYS 14.0 (CFD Fluent). There are the following steps as shown in the following figure4.

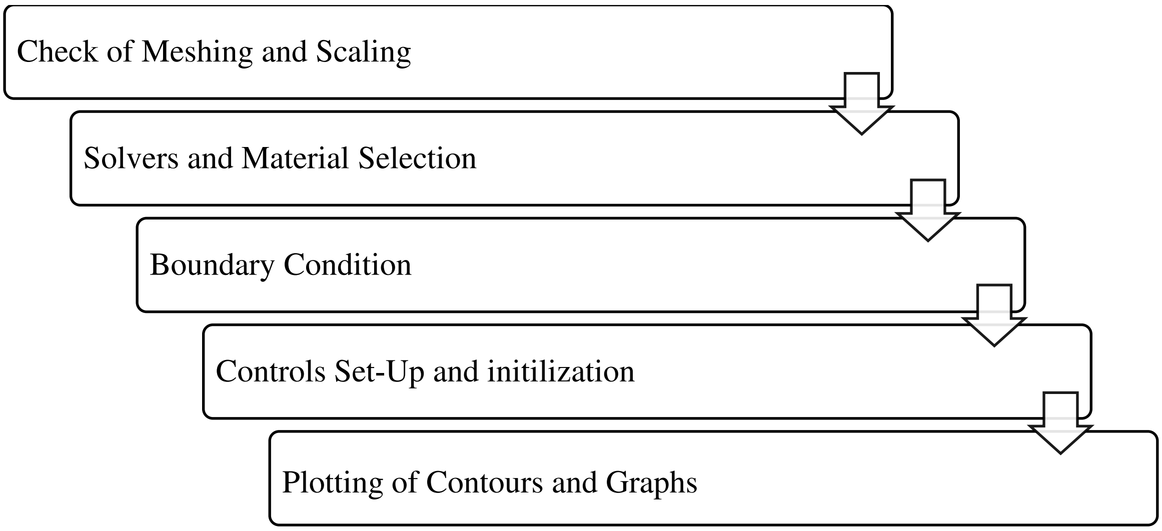


Figure 4: Flowchart for analysis procedure of C-D nozzle

The total time rate of change of mass in a fixed region is zero, according to the law of conservation of mass. From the laminar flow using the Reynolds averaging (time averaging) procedure, the conservation equation for turbulent flow is to be obtained [3]. It is as following-

$$\left(\frac{\delta}{\delta xj}(\rho v)\right) = 0$$

Where ρ = density of gas and v = velocity vector in $j^{th}$ direction.

For the isentropic flow, through a de Laval nozzle with throat area, a transcendental equation named Stodola relates the area ratio $A_t/A$ and the mach number M at any cross section of surface area A [4,5]:

$$\left(\frac{A}{A_t}\right)^2 = \frac{1}{M^2}\left\{\frac{2}{\gamma+1}\left[1+\frac{1}{2}(\gamma-1)M^2\right]\right\}^{\frac{\gamma+1}{\gamma-1}}$$

The graph is plotted between static pressure (Pa)vs position and mach number vs position for the standard turbulent and realizable model along the line. The beginning of the line from the starting of the nozzle, convergent section till the end, divergent section.

**FINDINGS –**

Case1: For Standard Turbulent Model

The graph of standardturbulent model for the static pressure vs position (fig.5) shows that the maximum inlet pressure ($3*10^5$ Pa) at the starting of the nozzle (from convergent section) to the end point of the nozzle (divergent section) decreases.

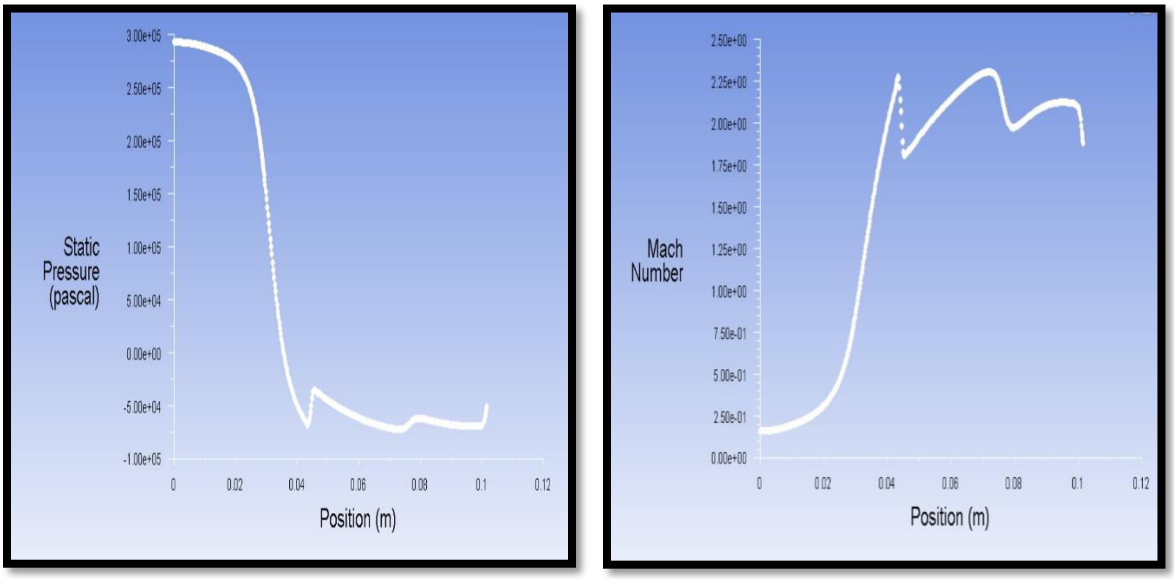


Figure 5: Graph of standard turbulent model for the (a) static pressure vs position (m) and (b) mach number vs position (m)

Case 2: For Realizable turbulent model

The graph of realizable turbulent model for the static pressure vs position (fig 6) follows the standard turbulent model till convergent nozzle and same as on the case of mach number and position.

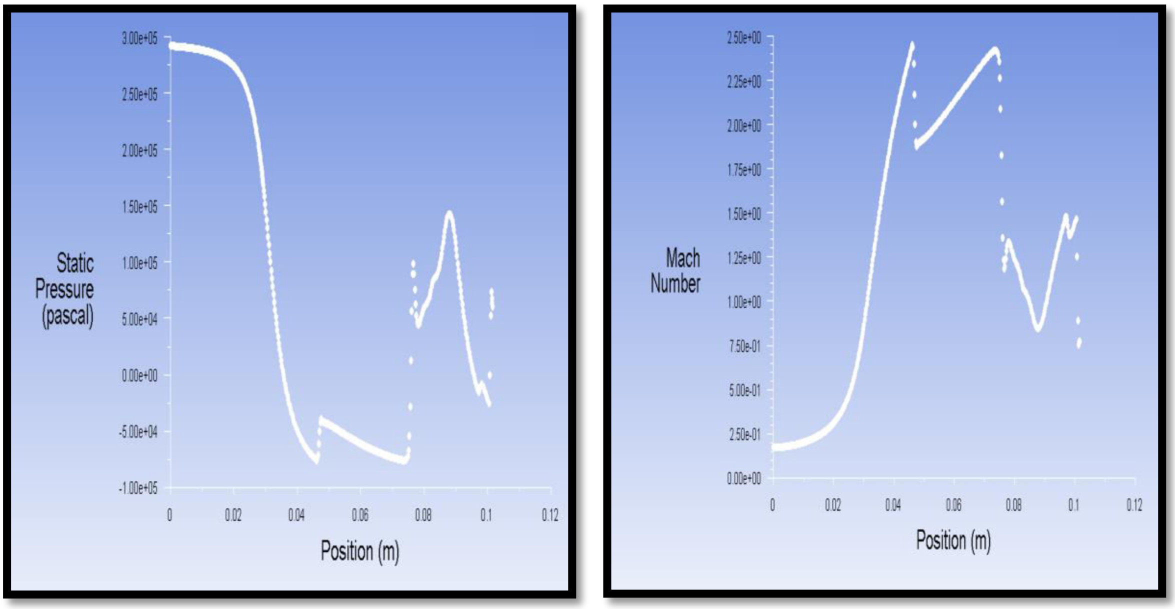


Figure 6: Graph of realizable turbulent model for the (a) static pressure vs position (m) and (b) mach number vs position (m)

## RESULTS AND DISCUSSION -

After comparing the standard and realizable turbulent model, it is find that in the convergent section of nozzle, the flow is subsonic and in the divergent section of nozzle, the flow is supersonic flow. And the graph of static pressure and mach number vs position is same in the convergent section, whether it changes from the starting point of the divergent section.

## REFERENCES –


1. Wen, Chuang, Xuewen Cao, Yan Yang, and Jing Zhang. "Supersonic swirling characteristics of natural gas in convergent-divergent nozzles." *Petroleum Science* 8, no. 1 (2011): 114-119.
2. Rao, AV Ramana, and I. Akhil. "INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY CFD ANALYSIS OF CD NOZZLE AT TWO DIVERGENT ANGLES FOR PREDICTION OF MACH NUMBER (M< 1, M= 1 & M> 1) AND FLOW PARAMETERS OF NEWTONIAN FLUID BY USING ANSYS-FLUENT 14.0."
3. Goyal, Tarun, S. Prince, R. S. Walia, and T. S. Sidhu. "Effect of nozzle geometry on exit velocity, temperature and pressure for cold spray process." *Int. J. Mat. Sci. Eng* 2 (2011): 65-72.

4. Majdalani, Joseph, and Brian A. Maicke. "Direct calculation of the average local Mach number in converging–diverging nozzles." *Aerospace Science and Technology* 24, no. 1 (2013): 111-115.
5. Sushma, L., A. UdayaDeepik, Sathish Kumar Sunnam, and M. Madhavi. "CFD Investigation for different nozzle jets." *Materials Today: Proceedings* 4, no. 8 (2017): 9087-9094.
6. Petrovic, Andrija, JelenaSvorcan, AleksandarPejcev, DarkoRadenkovic, and AleksandarPetrovic. "Comparison of novel variable area convergent-divergent nozzle performances obtained by analytic, computational and experimental methods." *Applied Mathematical Modelling* 57 (2018): 206-225.
7. Li, Wen-Ya, Hanlin Liao, Hong-Tao Wang, Chang-Jiu Li, Ga Zhang, and C. Coddet. "Optimal design of a convergent-barrel cold spray nozzle by numerical method." *Applied Surface Science* 253, no. 2 (2006): 708-713.
8. Boyanapalli, Raviteja, Raja Sekhara Reddy Vanukuri, PrudhviGogineni, JanakinandanNookala, Goutham Kumar Yarlagadda, and V. Gada. "Analysis of composite De-Laval nozzle suitable for rocket applications." *International Journal of Innovative Technology and Exploring Engineering* 2 (2013): 336-344.
9. Shih, Tsan-Hsing, William W. Liou, AamirShabbir, Zhigang Yang, and Jiang Zhu. "A new k-ϵ eddy viscosity model for high reynolds number turbulent flows." *Computers & Fluids* 24, no. 3 (1995): 227-238.
10. Behera, Upendra, P. J. Paul, S. Kasthurirengan, R. Karunanithi, S. N. Ram, K. Dinesh, and S. Jacob. "CFD analysis and experimental investigations towards optimizing the parameters of Ranque–Hilsch vortex tube." *International Journal of Heat and Mass Transfer* 48, no. 10 (2005): 1961-1973.