THOMSON REUTERS ENDNOTE

[Allaboina*, 5(5): May, 2016]

ISSN: 2277-9655
Impact Factor: 3.785

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

CFD ANALYSIS OF C-D 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

A.V.Ramana Rao, I.Akhil

[*] Assistant Professor, Department of Mechanical engg., Pragati Engg. College, Surampalem, INDIA
B.Tech Student, Department of Mechanical engg., Pragati Engg. College, Surampalem, INDIA

DOI: 10.5281/zenodo.51446

ABSTRACT

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. In this paper, CFD analysis of pressure, velocity (mach number) and temperature of flow for a Convergent-Divergent nozzle of two different diverged angles (50 & 100) has been performed, because the fluid properties like pressure, velocity, and temperature are dependent on the cross section of the nozzle which affects the flow within the nozzle. A nozzle is a device designed to control the direction or characteristics of a fluid flow as it exits or enters. The analysis has been performed according to the diverged angle of the C-D nozzle and keeping the same input conditions. Our objective is to investigate theoretically what we have studied about sonic (M=1), sub-sonic (M<1) and super-sonic (M>1) and flow parameters, we are proving that by CFD simulation.

Initially the 2-D model of the C-D nozzles is taken in the present research work. The analysis type is 2-D PLANER. The modeling of the C-D nozzles has been done and later on mesh generation and analysis have been carried out in Ansys-Fluent 14.0 and various contours, plots and vectors of pressure, velocity and temperature have been taken and their variation according to different divergent angle nozzles has been studied.

KEYWORDS: CFD, C-D nozzle, divergent angle, Mach number, Sub-sonic, Super-sonic, Sonic, Ansys-Fluent.

INTRODUCTION

CD nozzle is used as a means of accelerating the flow of fluid passing through it to a supersonic speed. It is widely used in steam turbine and essential part of the modern rocket engine and supersonic jet engines. The nozzle was developed by Swedish engineer Gustaf de Laval in 1897 for use on an impulse steam turbine. This principle was used in a rocket engine by Robert Goddard.

CFD has become an integral part of the engineering design and analysis to predict performance of new designs or processes before they are manufactured. The critical requirements for CFD tool used in thermal applications is the ability to analyze flows along nozzles and turbines. Such features have pressure gradients, mach numbers, velocity distribution, temperature distribution, velocity vectors etc….

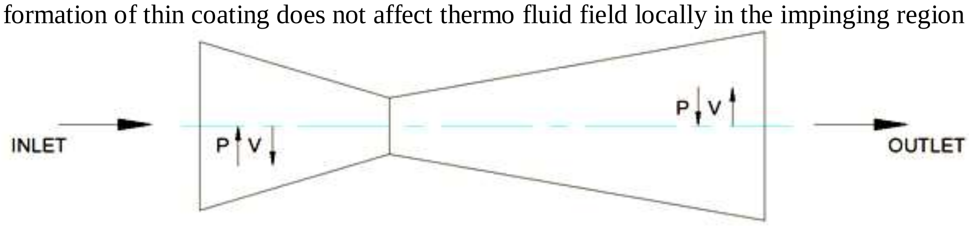
The current study aims analysis of flow through the CD nozzle and prediction of theoretically what we have studied about sonic, sub-sonic, super-sonic and flow parameters, we are proving that by CFD simulation.

ASSUMPTIONS

The assumptions for the calculation are as follows:

i. The gas obeys the ideal gas law.
ii. There is no friction impeding the gas flow.
iii. The gas flow is adiabatic, i.e., no heat exchange occurs with the surroundings.

iv. Steady-state conditions exist.
v. Expansion of the gas occurs in a uniform manner without shock or discontinuities.
vi. Flow through the nozzle is one-dimensional.
vii. Particles do not influence gas conditions.
viii. Collision between particles is neglected.
ix. The effect of the presence of particles does not affect the space charge.
x. The formation of thin coating does not affect thermo fluid field locally in the impinging region



Principle of C-D Nozzle

MODELING PROCEDURE FOR C-D NOZZLE

The procedure for modeling of C-D nozzle is done by using the ANSYS 14.0 [Module CFD Fluent] workbench software as described in following steps:

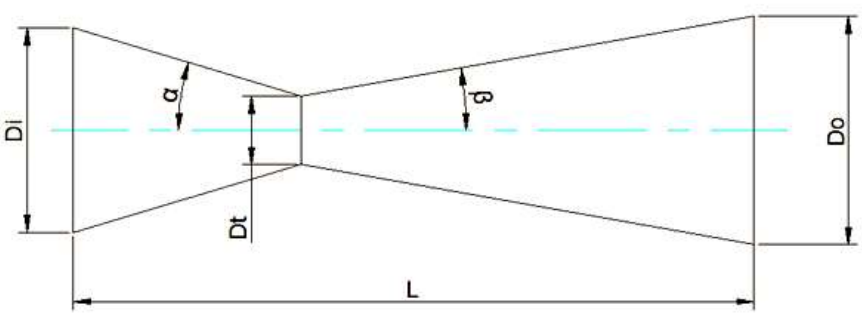
STAGE-1Nozzle configurations:

Two different types of nozzles are under investigation in this research work. In all these configurations the ideal gas is injected axially of the nozzle. The inlet operating conditions for the two nozzle configurations are same to measure the output conditions.

The inlet pressure (Pi) at the nozzle is 3 bar. The nozzle surface is assumed to be the wall surface having no slip. The pressure Po, velocity Vo and temperature To is to be measured at the outlet of the nozzle.

STAGE-2: Determination of Problem Domain

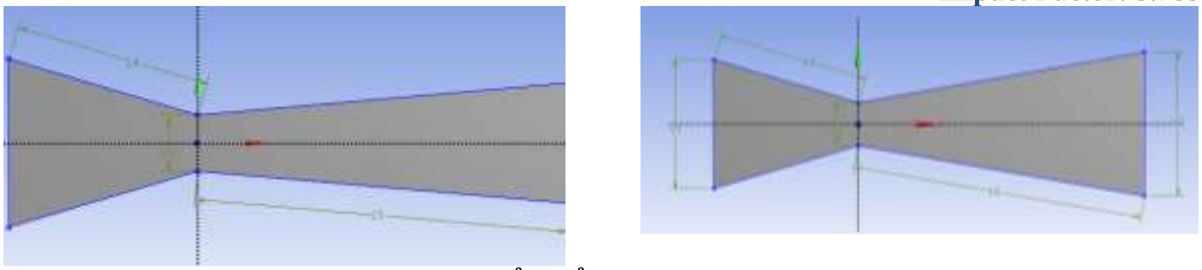
In this stage, the geometry of the problem is developed. The 2-D model for the problem is taken in this project work. The analysis type is 2-D PLANER. In this work, two different Nozzles under investigation named as Nozzle-1 and Nozzle-2 were considered. The various dimensions for these nozzles as shown in Table.



Nozzle Configurations

Table: Nozzle Configurations (All Dimensions in mm):

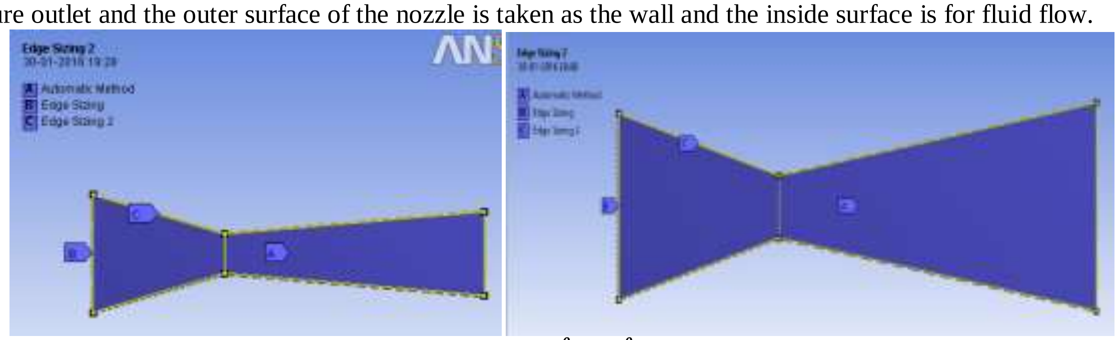
s.no	Configuration (mm)	Nozzle-1	Nozzle-2
1	Inlet diameter Di	60	
2	Throat diameter Dt	20	
3	Outlet diameter Do	43	67
4	Convergent angle α	17^0	
5	Divergent angle β	5^0	10^0
6	Nozzle length L	200	



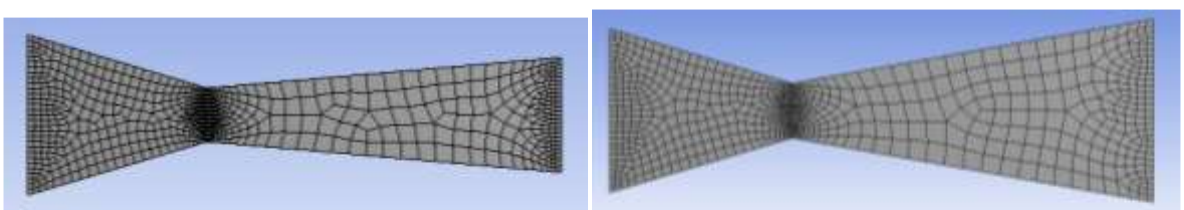
2-D model for 5^0 & 10^0 divergent C-D nozzle

***STAG- 3: Meshing and Boundary Conditions**:*

The total surface is meshed using ANSYS meshing. The mapped free meshing contains the quadrilateral surface element shown in Figure. After the meshing is done, the boundary conditions are defined as given. In the meshing the surface name is created as inlet, outlet and walls. Nozzle inlet is taken as pressure inlet, outlet is taken as pressure outlet and the outer surface of the nozzle is taken as the wall and the inside surface is for fluid flow.



Edge sizing for meshing for 5^0 & 10^0 divergent C-D nozzle



Meshing model for 5^0 & 10^0 divergent C-D nozzle

Boundary conditions:

The boundary conditions described for flow through the C-D nozzle in the process are as given below:

1. At the inlet of the nozzle:

Pressure (Pi) = 3 Bar

2. The outlet of nozzle:

Pressure (Po) = 0 Bar

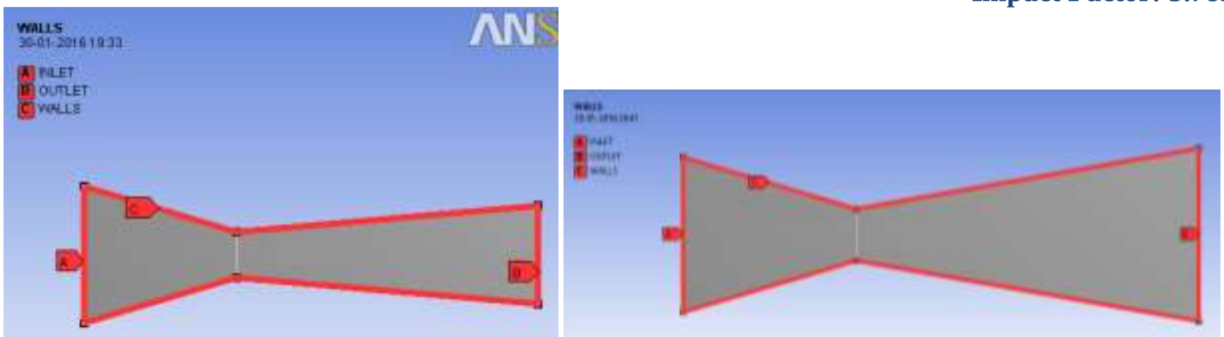
3. At the nozzle surface:

Vx1 =0

4. Fluid is – air (Ideal Gas)

5. Ideal gas viscosity- Sutherland

ISSN: 2277-9655
Impact Factor: 3.785



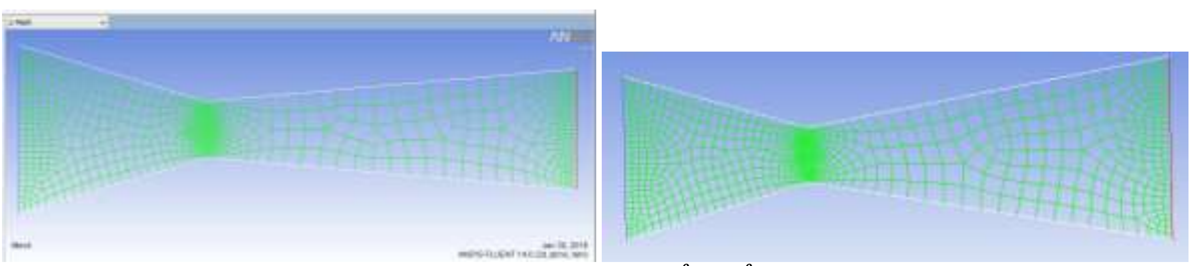
Boundary conditions for 5^0 & 10^0 C-D nozzle

ANALYSIS PROCEDURE FOR C-D NOZZLE:

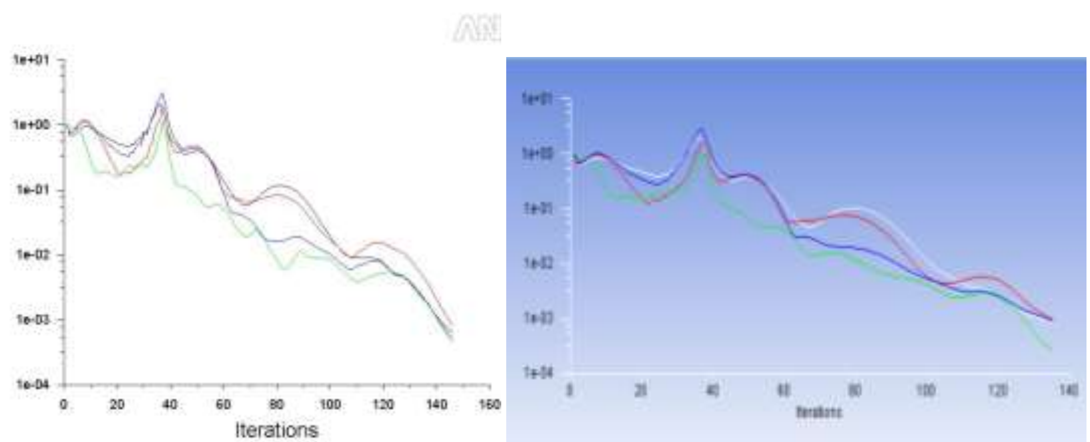
The Analysis is carried out in the ANSYS 14.0 [Module CFD FLUENT]. The steps that are followed are given below which include all the conditions and the boundaries values for the problem statement.

STEP 1: Checking of Mesh and Scaling

The Fluent solver is opened where 2DDP is selected. The meshed file then undergoes a checking where the number of nodes for different nozzles are displayed. After this grid check is done following which smoothing and swapping of grid is done. Following this the scaling is done. Scale is scaled to mm. Grid created was changed to mm. After this defining of various parameters are done.



Imported model Mesh checking for 5^0 & 10^0 C-D nozzle



Scaled residuals for 5^0 & 10^0 C-D nozzle

STEP 2: Solvers and Material Selection

The solver is defined first. Solver is taken as density based and formulation as implicit, space as 2D PLANER and time as steady. Velocity formulation as absolute and gradient options as least square Cell based are taken. Energy equation is taken into consideration. The viscous medium is also taken. The ideal gas is used as carrier gas .The properties of ideal gas are mentioned in Table.

Table: Various properties of ideal gas (Air)

s.no	Property	value
1	Density, ρ (kg/m^3)	1.225 (Ideal gas)
2	Specific heat, C_p (J/kg K)	1006.43
3	Thermal conductivity, h (W/m K)	0.0242
4	Viscosity, μ (kg/m s)	1.7894×10^{-5}

STEP3: Boundary Conditions

- Nozzle inlet:
  The Nozzle inlet is taken as the pressure inlet.
  Input the following values as: Pressure (Pi) = 3 Bar.
  Temperature (Ti) = 300 K
- Outlet :
  The Nozzle was set as pressure outlet.
  Output the following values as: Pressure (Po) = 0 Bar.
  Temperature (Ti) = 300 K
- Wall:
  T = Room temperature TR on the nozzle surface
  Type: stationary wall
  Slip: no-slip

The above mentioned boundary conditions used to analyze the C-D nozzle.

STEP 4: Controls Set Up

The solutions controls are set as listed below in Table. The under relaxation factor was set as given.

Table: Control set-up

s.no	Property	value
1	Density, ρ (kg/m^3)	1.225 (Ideal gas)
2	Enthalpy H J/kg	629541.35
3	Velocity V m/s	699.8541
4	Viscosity, μ (kg/m s)	1.7894×10^{-5}
5	Ratio of specific heats	1.4

Pressure Velocity Coupling was taken as SIMPLE
Discretization Equation selected is as given
Flow is second order uowind

STEP 5: Initialization

The standard initialization is done. Initial. Temperature is taken as 300 K. Residual monitorization is done and convergence criteria are set up. The convergence criteria of various parameters are listed below.
Gauge pressure- 0 Pascal
X-Velocity m/s – 699.8541
Y-Velocity m/s - 0
The number of iterations is then set up and iterations starts. The iteration continues till the convergence is reached.

STEP 6: Plotting of contours and graphs

After the solution of the problem, next step is to plot the contours of velocity, pressure and temperature on the urface and the graphs for same. The different shades of the colors in the contour represent the different value of velocity, pressure and temperature at the different positions.
The graph is plotted along the line. The beginning of the line from the starting of nozzle, convergent section till the end, divergent section.

RESULTS AND DISCUSSION

The simulation procedure involves the determination of velocity, temperature and pressure through the nozzle. The velocity, temperature and pressure distribution were analyzed by the finite element method using the ANSYS 14.0

having module FLUENT. The resultant pressure, temperature and the velocity of flow during the process cycle were determined.

At the first instant, a model has been built using ANSYS 14.0 software considering the pressure inlet for the air. The mesh file for the nozzle is also created in ANSYS 14.0 and defining the boundary conditions. Then CFD software FLUENT (ANSYS 14.0) was used for the solving process. Since in the finite element analysis, the results are available only on the nodes, the geometry of the problem is meshed such that the nodes can be found on the locations where the values of the degree of freedom (flow velocity, temperature and resultant pressure) are needed. A finer meshing is required in the regions where the changes in gradients of the degree of freedom are higher. For fulfilling all these requirements, the quadrilateral meshing is used for meshing the geometry.

For nozzle-1 (5^0 divergences):

Velocity distribution:

The patterns of velocity distribution obtained by using FLUENT (ANSYS 14.0). The fig. shows the distribtion of the Mach number and velocity distribution in the nozzle. Different shades indicate different values of the velocity. The velocity at the inlet of nozzle is low as compared to throat section and the velocity at the outlet of nozzle is maximum as compared to throat section. The plot also shows the graphical representation of the velocity through the nozzle. The velocity at the outlet of the nozzle is about 5.60e+02 m/s.

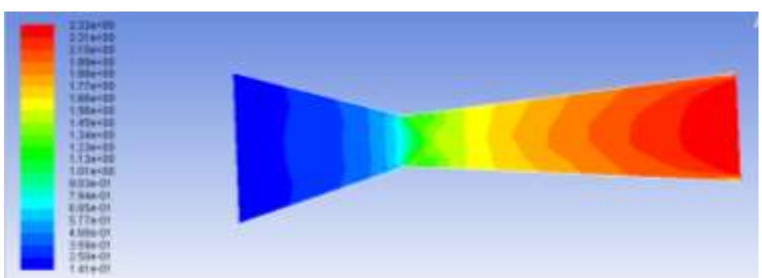
Velocity distribution table

S.No	Boundary of Nozzle	Velocity. m/s	
1	Inlet	4.89e+01	Low
2	Throat	2.79e+02	Moderate
3	Outlet	5.60e+02	High

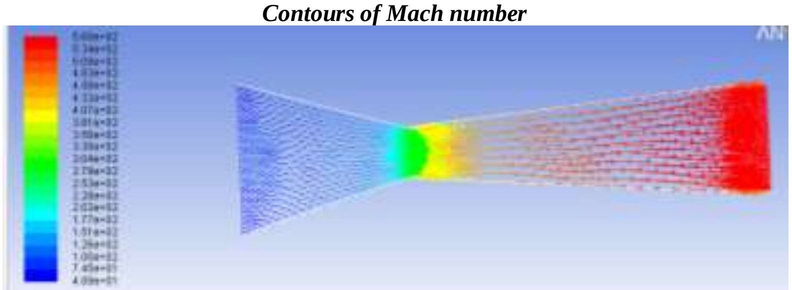
The Mach number at the inlet of nozzle is less than 1 (M<1), at throat is equal to 1 (M=1) and at outlet is greater than 1 (M>1).

Mach number table

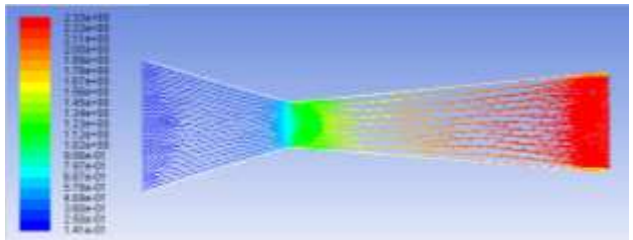
S.No	Boundary of Nozzle	Mach No.	
1	Inlet	1.41e-01	~ < 1
2	Throat	1.01e+00	~ = 1
3	Outlet	2.32e+00	~ > 1



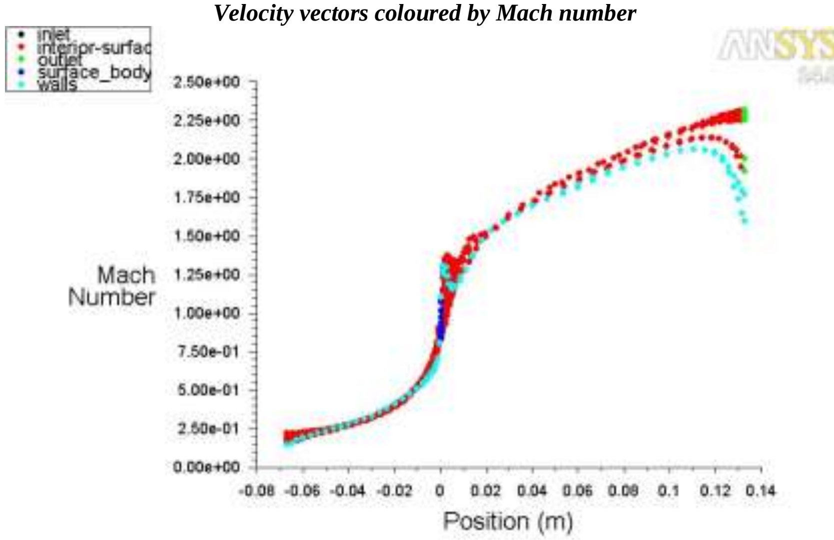
Contours of Mach number



Velocity vectors coloured by velocity magnitude



Velocity vectors coloured by Mach number



Plot for Mach number distribution

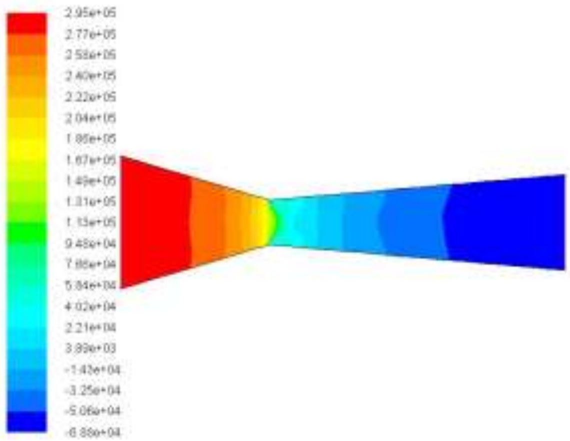
Pressure distribution

Figure shows the resultant pressure contours through the nozzle along the total length of the nozzle. The resultant pressure at the inlet of the nozzle is maximum and decreases along the length towards the outlet of the nozzle. Different shades of contour indicate different values of the resultant pressure. The resultant pressure near about the powder feeder is below the atmospheric pressure. The pressure at the outlet of the nozzle is about -6.88e+04 Pa.

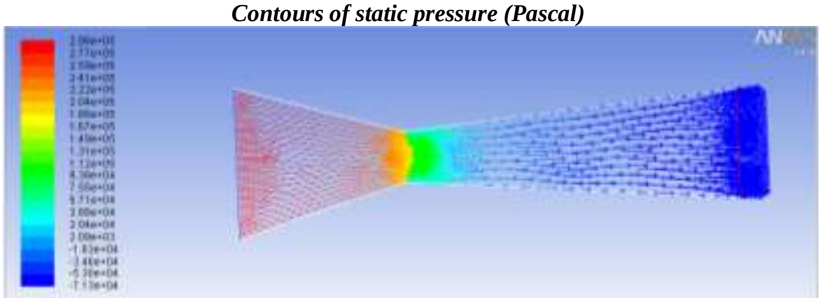
Table: Pressure distribution table

S.No	Boundary of Nozzle	Pressure Pa	
1	Inlet	2.95e+05	High
2	Throat	1.67e+05	Moderate
3	Outlet	-6.88e+04	Low

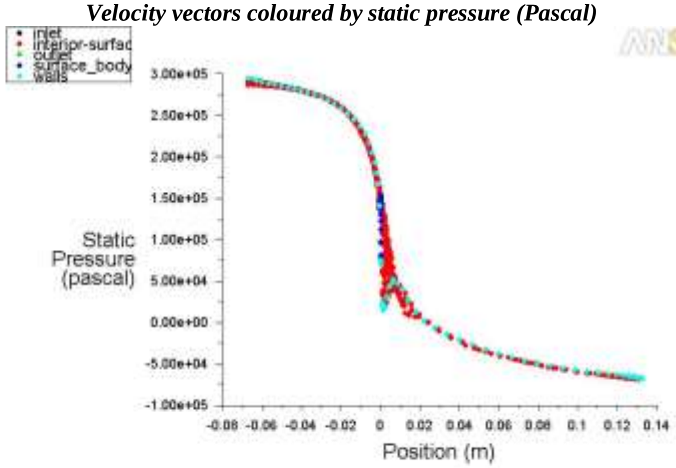
ISSN: 2277-9655
Impact Factor: 3.785



Contours of static pressure (Pascal)



Velocity vectors coloured by static pressure (Pascal)



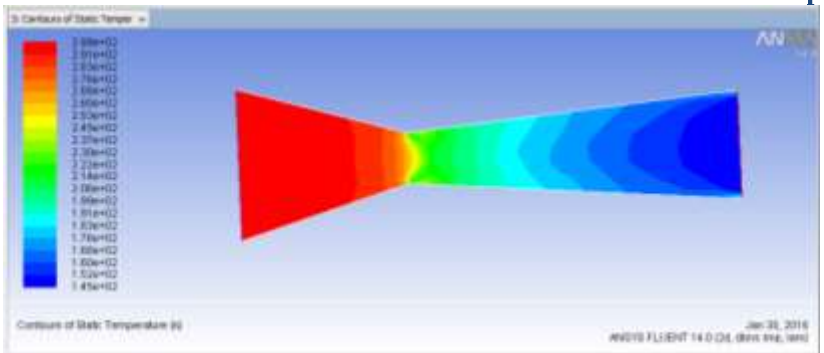
Plot for static pressure distribution

Temperature distribution

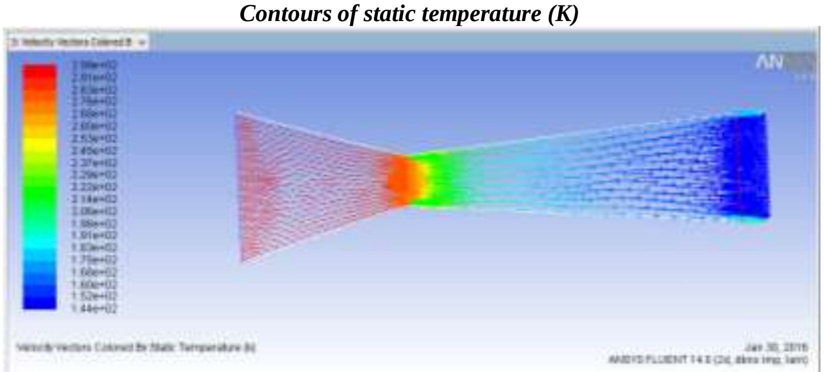
Figure shows the temperature distribution through C-D nozzle. The contours of the temperature are shown through the nozzle. It shows that the temperature at inlet is maximum. The measured temperature at the outlet of the nozzle is 1.45e+02 K.

Temperature distribution table

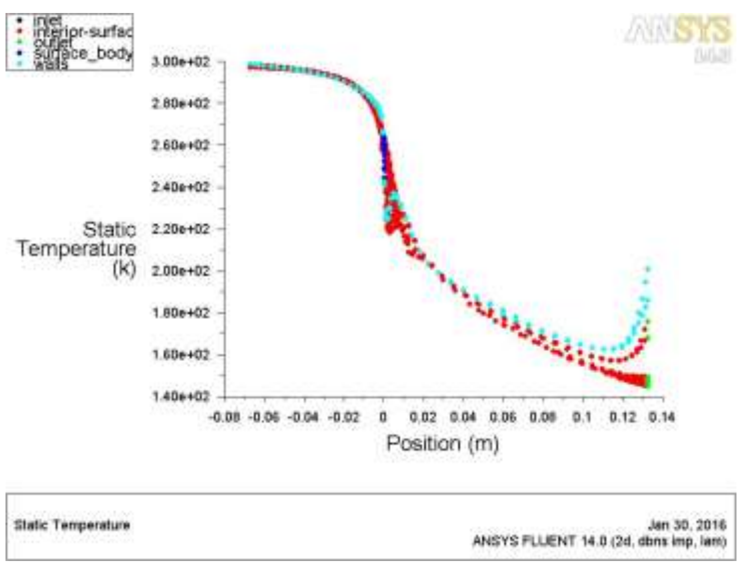
S.No	Boundary of Nozzle	Temperature K	
1	Inlet	2.99e+02	High
2	Throat	2.53e+02	Moderate
3	outlet	1.45e+02	Low



Contours of static temperature (K)



Velocity vectors coloured by static temperature (K)



Plot for static temperature distribution

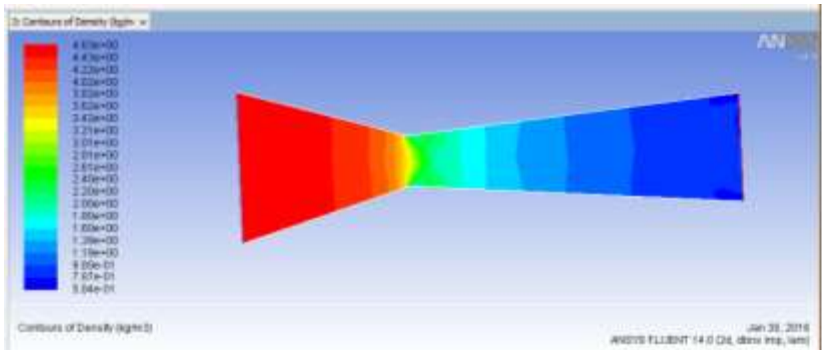
Density distribution

Figure shows the density distribution through C-D nozzle. The contours of the density are shown through the nozzle. It shows that the density at inlet is maximum. The measured density at the outlet of the nozzle is 5.84e-01 kg/m^3.

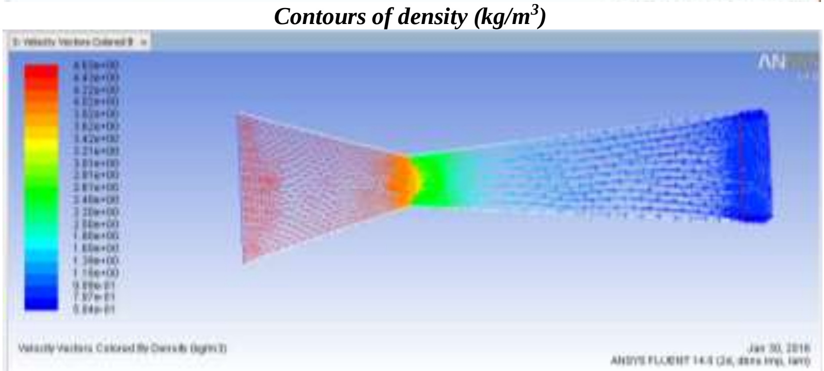
Density distribution table

S.No	Boundary of Nozzle	Density Kg/m^3	
1	Inlet	4.63e+00	High
2	Throat	3.21e+00	Moderate

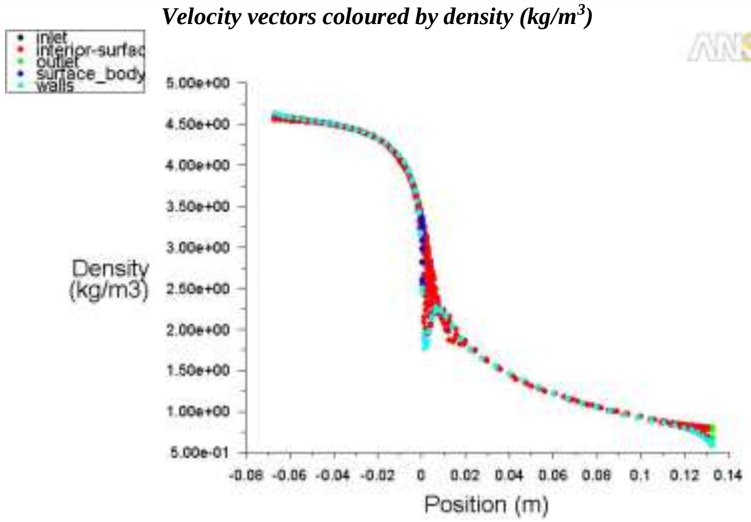
3	Outlet	5.84e-01	Low



Contours of density (kg/m^3)



Velocity vectors coloured by density (kg/m^3)



Plot for density (kg/m3)

Table: Result analysis

S.No	Boundary of Nozzle	Nozzle-1 (5^0 divergence)				
		Mach No.	Velocity. m/s	Pressure Pa	Temperature K	Density Kg/m^3
1	Inlet	1.41e-01	4.89e+01	2.95e+05	2.99e+02	4.63e+00
2	Throat	1.01e+00	2.79e+02	1.67e+05	2.53e+02	3.21e+00
3	outlet	2.32e+00	5.60e+02	-6.88e+04	1.45e+02	5.84e-01

ISSN: 2277-9655
Impact Factor: 3.785

For nozzle-2 (10^0 divergences)

Velocity distribution

Figure shows the velocity distribution through the nozzle-2 having configuration in the below Table. The velocity at the outlet of the nozzle is about 6.10e+02 m/s.

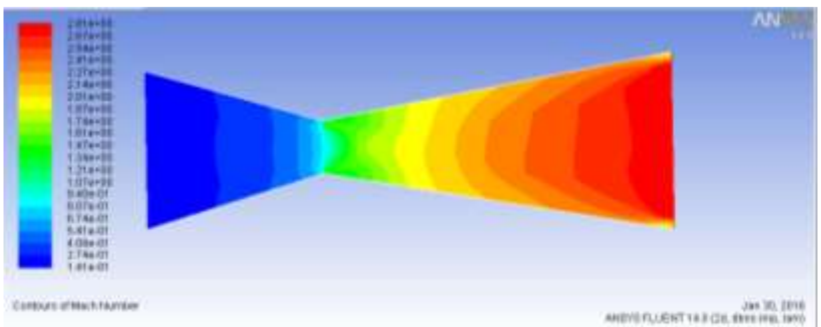
Table: Velocity distribution table

S.No	Boundary of Nozzle	Velocity. m/s	
1	Inlet	4.89e+01	Low
2	Throat	2.73e+02	Moderate
3	Outlet	6.10e+02	High

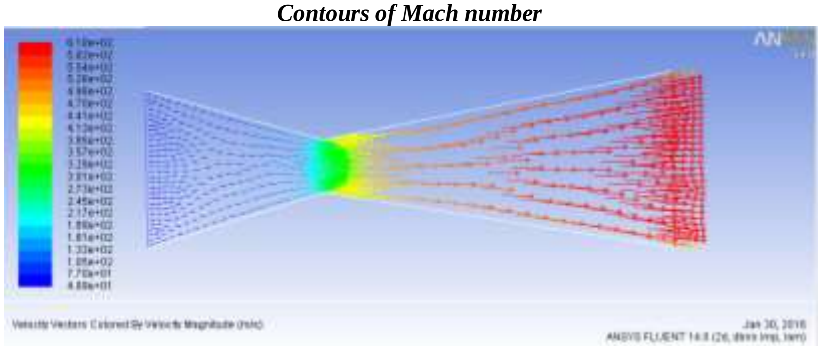
The Mach number at the inlet of nozzle is less than 1 (M<1), at throat is equal to 1 (M=1) and at outlet is greater than 1 (M>1).

Mach number table

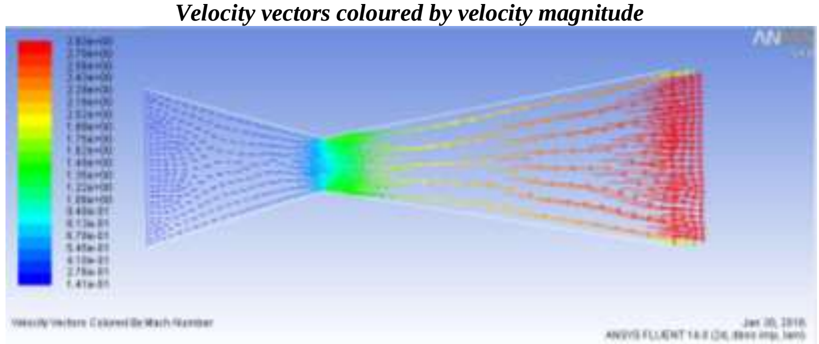
S.No	Boundary of Nozzle	Mach No.	
1	Inlet	1.41e-01	~ < 1
2	Throat	1.07e+00	~ = 1
3	Outlet	2.81e+00	~ > 1



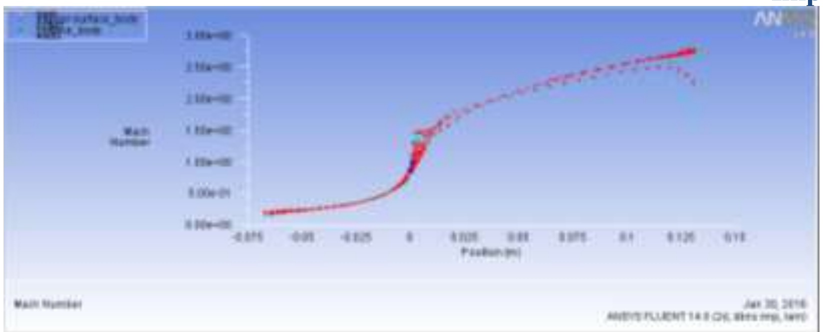
Contours of Mach number



Velocity vectors coloured by velocity magnitude



Velocity vectors coloured by Mach number



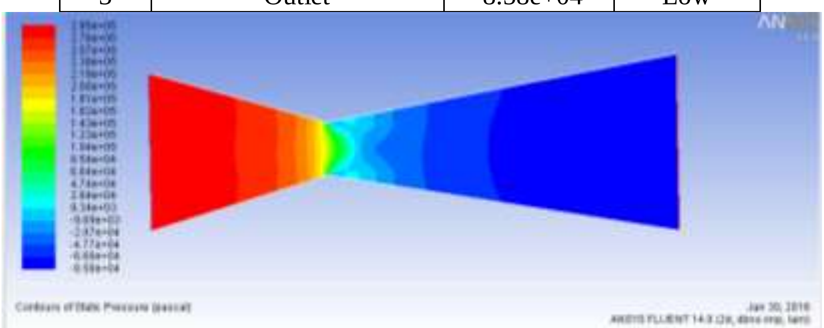
Plot for Mach number distribution

Pressure distribution

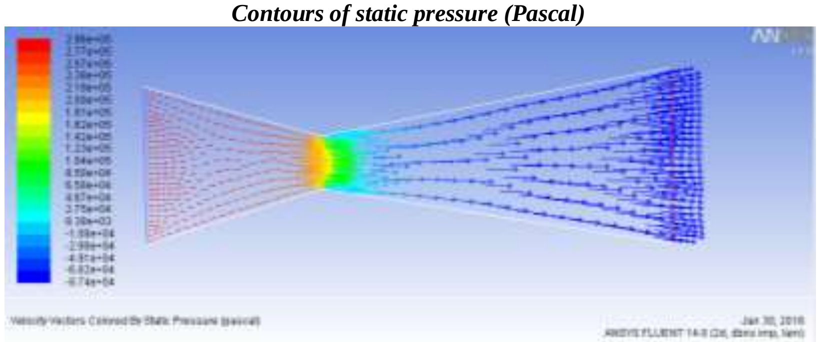
Figure shows the distributions of the pressure through the nozzle. The different shades show the different values of pressure into the nozzle. The pressure at the outlet of the nozzle is about -8.58e+04 Pa.

Pressure distribution table

S.No	Boundary of Nozzle	Pressure Pa	
1	Inlet	2.95e+05	High
2	Throat	1.62e+05	Moderate
3	Outlet	-8.58e+04	Low

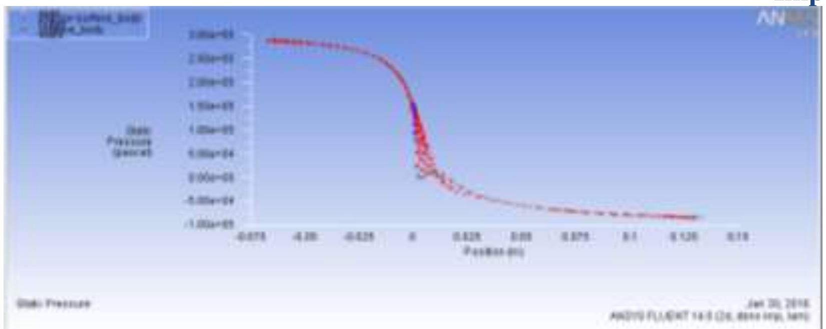


Contours of static pressure (Pascal)



Velocity vectors coloured by static pressure (Pascal)

ISSN: 2277-9655
Impact Factor: 3.785



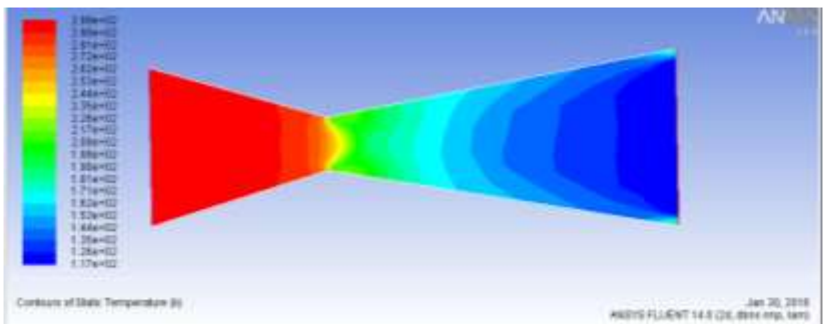
Plot for static pressure distribution

Temperature distribution

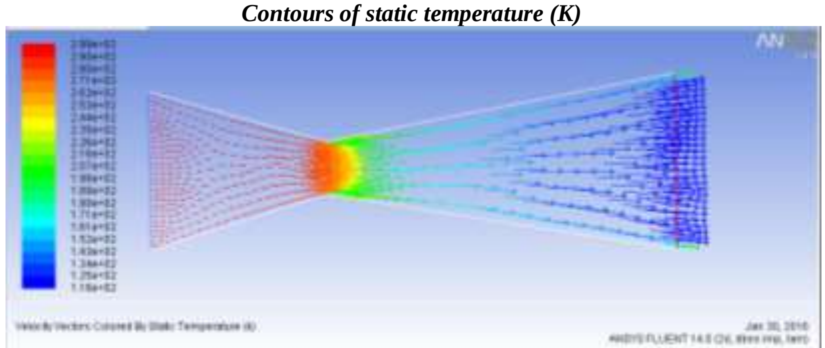
Figure shows the distribution of temperature through the nozzle along the length. The temperature at the inlet of nozzle is about 2.99e+02. The outlet temperature for the nozzle is 1.17e+02 K.

Temperature distribution table

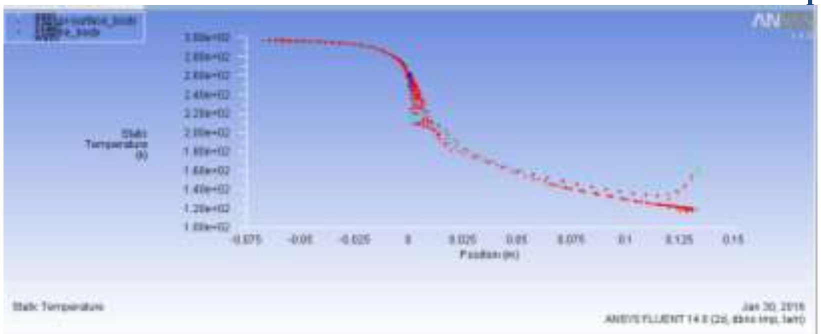
S.No	Boundary of Nozzle	Temperature K	
1	Inlet	2.99e+02	High
2	Throat	2.44e+02	Moderate
3	Outlet	1.17e+02	Low



Contours of static temperature (K)



Velocity vectors coloured by static temperature (K)



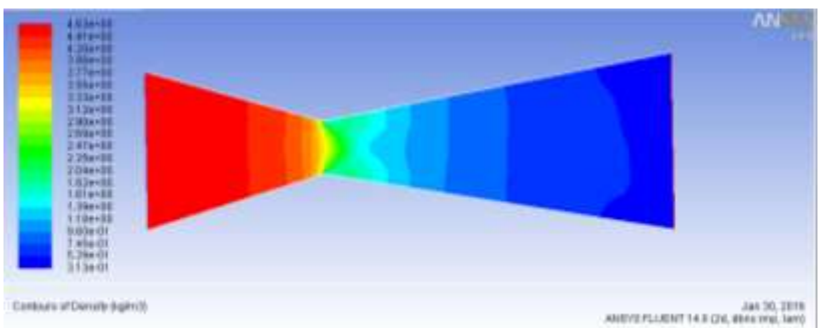
Plot for static temperature distribution

Density distribution

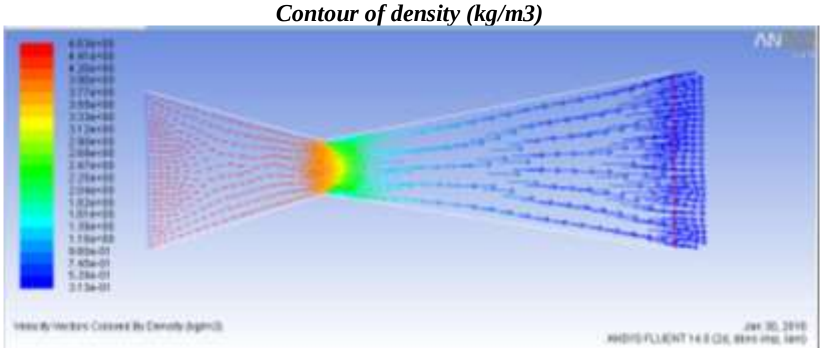
Figure shows the density distribution through C-D nozzle. The contours of the density are shown through the nozzle. It shows that the density at inlet is maximum. The measured density at the outlet of the nozzle is 3.13e-01kg/m^3.

Density distribution table

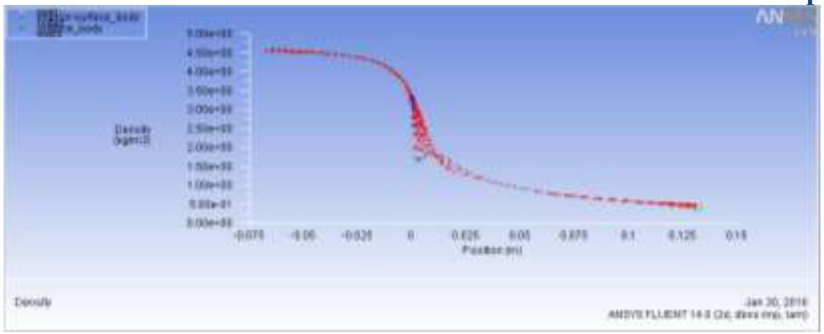
S.No	Boundary of Nozzle	Density Kg/m^3	
1	Inlet	4.63e+00	High
2	Throat	3.33e+00	Moderate
3	Outlet	3.13e-01	Low



Contour of density (kg/m3)



Velocity vectors coloured by density (kg/m3)



Plot for density distribution

S.No	Boundary of Nozzle	Nozzle-1 (5^0 divergence)				
		Mach No.	Velocity. m/s	Pressure Pa	Temperature K	Density Kg/m^3
1	Inlet	1.41e-01	4.89e+01	2.95e+05	2.99e+02	4.63e+00
2	Throat	1.01e+00	2.79e+02	1.67e+05	2.53e+02	3.21e+00
3	outlet	2.32e+00	5.60e+02	-6.88e+04	1.45e+02	5.84e-01

S.No	Boundary of Nozzle	Nozzle-1 (10^0 divergence)				
		Mach No.	Velocity. m/s	Pressure Pa	Temperature K	Density Kg/m^3
1	Inlet	1.41e-01	4.89e+01	2.95e+05	2.99e+02	4.63e+00
2	Throat	1.07e+00	2.73e+02	1.62e+05	2.44e+02	3.33e+00
3	outlet	2.81e+00	6.10e+02	-8.58e+04	117e+02	3.13e-01

CONCLUSION

The following observations were found in the **convergent divergent nozzle at two different diverged angles:**

Mach number:

1. The Mach number at the inlet, throat and outlet of C-D nozzle at 5^0 divergences is found to be 0.141 Mach (sub-sonic), 1.01 Mach (sonic) and 2.32 Mach (super-sonic).
2. The Mach number at the inlet, throat and outlet of C-D nozzle at 10^0 divergences is found to be 0.141 Mach (sub-sonic), 1.07 Mach (sonic) and 2.81 Mach (super-sonic).

From the above two simulation processes we investigated theoretically what we have studied about sonic, sub-sonic and super-sonic; we are proved that through this research work.

Static Pressure: The static pressure is increases at inlet and gradually it decreases throughout the length of nozzle.

Velocity: The velocity value of Nozzle increases inlet to exit.

Static Temperature: The temperature value of Nozzle decreases inlet to exit.

Density: The density value of Nozzle decreases inlet to exit.

ACKNOWLEDGEMENT

The authors acknowledge the valuable suggestions from Dr. S.Sambhu Prasad, Professor, Mr.P.Sukumar Associate Professor, Pragati engineering college, Surampalem, E.G.Dist. A.P. India.

REFERENCES

1. Kunal Pansari, S.A.K Jilani "ANALYSIS OF THE PERFORMANCE AND FLOW CHARACTERISTICS OF CONVERGENT DIVERGENT (C-D) NOZZLE" International Journal of Advances in Engineering & Technology, July 2013, Vol. 6, Issue 3, pp. 1313-1318, ISSN: 22311963.
2. K.M.Pandey, Member, IACSIT and S.K.Yadav "CFD Analysis of a Rocket Nozzle with Four Inlets at Mach 2.1" International Journal of Chemical Engineering and Applications, Vol. 1, No. 4, December 2010, ISSN: 2010-0221.
3. G. SATYANARAYANA, Ch. VARUN, S.S. NAIDU, "CFD ANALYSIS OF CONVERGENT-DIVERGENT NOZZLE" ACTA TECHNICA CORVINIENSIS-Bulletin of Engineering Tome VI (Year 2013)-FASCICULE3 [July-September], ISSN 2067 – 3809.
4. Balaji Krushna.P, P. SrinivasaRao, B. Balakrishna "ANALYSIS OF DUAL BELL ROCKET NOZZLE USING COMPUTATIONAL FLUID DYNAMICS" International Journal of Research in Engineering and Technology Nov-2013, Volume: 02 Issue: 11, eISSN: 2319-1163, pISSN: 2321-7308.