International Journal of Research in Engineering and Science (IJRES)
ISSN (Online): 2320-9364, ISSN (Print): 2320-9356
www.ijres.org Volume 09 Issue 10 ǁ 2021 ǁ PP. 61-76

Design and Analysis of Fluid Flow Inconvergent-Divergentnozzleusing ANSYS Fluent Software

TADESSE WASIE (ROLL NO. 1809017)
TILAHUN GETNET (ROLL NO.1809050)
School of Mechanical Engineering (Automobile) from Kalinga Institute of Industrial Technology, Deemed to be University
Bhubaneswar,odisha - 751024

ABSTRACT
In a fluid flow, computational fluid dynamics(CFD) are used to solve and analyze problem which is come from fluid flow based. CFD analysis in this paper used to solve and analysisthe flow pattern of supersonic,sonic,subsonic nozzle at various degree of divergent angle (19°, 29°) and at different inlet pressure, mach numbers etc. This paper aims to study the behavior of flow in convergent divergent and throat section nozzle by analyzing various parameters like pressure, temperature, density and velocity using CFD software, to get maximum velocity at the exit of the nozzle and to check mach number at convergent section is less than one, at the throat is equal to one and at divergent section is greater than one using this software. Solid work software are used to design a C-D nozzle in this project.what are the various parameter properties in divergent section ,throat section, and divergent section contour and streamline velocity also studded, material used to manufactured nozzle, practical application of nozzle, concept of De laval nozzle, simulation of CD nozzle using ANSYS software. during simulation creating geometry,study the use of face split, generate structural mesh, studded the effect of mach number at each section of the nozzle,using density based set up solver. result in different parameters are also plotted in chart, after analysis of different parameters in each section we try to summarize at which divergent angle inlet pressure and velocity obtain maximum and minimum value for each parameter and which is better

KEY WORD
Meshing, Mach Number, back pressure, converging angle,diverging angle, isotropic, Venturi effect

---

Date of Submission: 04-10-2021 Date of acceptance: 18-10-2021

I. INTRODUCTION

Nozzle is a device used to convert the chemical or thermal energy generated inside the combustion chamber into kinetic energy. The nozzle transforms the combustion chamber's low velocity, high pressure, high temperature gas into a high velocity, lower pressure gas and temperature.Swedish engineer of French descent who, trying to develop a more efficient steam engine, designed a turbine that was turned by jets of steam. it is used to accelerate a hot pressurized gas passing through it to a higher supersonic speed in the axial direction and also it is used to control the direction and characteristic of fluid flow to increase the velocity when it exit the chamber. decreasing area from the entrance to the throat, The flow velocity increases as the nozzle region decreases, with the highest velocity occurring at the throat. There is an increase in velocity and a decrease in pressure in a convergent nozzle, but we know that pressure is inversely proportional to area. De laval nozzle.

The nozzle was developed by Swedish inventor Gustaf de Laval in the 19th century and this CD nozzle named as De laval nozzle. It is used to accelerate the flow of a gas that passes through it. The invention of Computational Fluid Dynamics (CFD) has solved this problem while also revolutionizing the engineering industry. In CFD, a problem is simulated in software, and the problem's transport equations are mathematically solved with the help of a computer. As a consequence, we will be able to predict the outcome of a dilemma before attempting to solve it . In CFD simulation of a CD nozzle the properties of velocity, pressure, effect of mach number, density and temperature can study using ANSYS software, this simulation and analysis can be performed at various divergent angle and taken different inlet pressure , can help to predict and for better efficiency.

II. LITERATURE REVIEW

Numerical analysis was utilized to calculate the greatest divergence angle for the nozzle, as described by BijuKuttan P et al., in order to produce the largest outlet velocity while still meeting thrust requirements. The intake dimensions and boundary conditions are held constant while the divergent angles are adjusted in order to understand how the divergent angle effects the flow pattern through the nozzle. R. Boyanapalli and colleagues, 2013.Deshpande and associates, The purpose of this technical report was to investigate and evaluate the De-Laval nozzle. It's a converging-diverging nozzle that generates supersonic jet velocity as it exits the nozzle. It was established in an experiment that velocity is lowest at the inlet and climbs near the nozzle's exit, whereas temperature is highest at the inlet and lowers toward the outlet.

According to Pandey et al., as high velocity and temperature air and fuel enter the combustion chamber, they started to burn, causing the pressure in the combustion chamber to rise, then fall to the convergent component, where the total pressure was a negative number. Suyash s. Deshpande, Nikhil d. DeshpandeVidwans, Pratik r.Surya Narayana and K.Sadhashiva Reddy found that the size and dimensions of the divergent component are crucial in determining the nozzle efficiency.

According to David Gretrix, the most efficient conversion of pressure to kinetic energy happens when the nozzle is first narrowed, bringing the jet's speed down to that of sound. The magnitude of neck velocity is the same for all divergence degrees of angles, according to this article.

OBJECTIVE

- ✓ Under stand the Concept of C-D nozzle
- ✓ About De laval nozzle
- ✓ Creating the axis symmetric nozzle geometry
- ✓ Study use of face split
- ✓ Creating structural mesh
- ✓ Study the effect of mach number
- ✓ Solver setup(density based),solve using it
- ✓ Study and analysis Temperature,pressure,mach number,velocity variation etc, at different divergent angle and inlet pressure.
- ✓ CFD processing of setup and solution

C-D nozzle

The aim of a converging diverging nozzle is to provide supersonic flow near the exit of CD nozzle. The flow would be isotropic within the nozzle and supersonic at the nozzle exit if the rear pressure is ready. The recent exhaust gas exits the combustion chamber and converges all the way down to the nozzle's minimum region at the throat during a CD nozzle. To choke the flow and set the mass rate through the system, the throat size is chosen. Since supersonic flows (mach numbers greater than one) are impossible to attain with convergent nozzles, convergent-divergent nozzles are commonly used for supersonic flows.

What is back pressure in nozzle ? The fluid flow in a nozzle is caused by a variation in pressure between . Then the pressure at the exit is named as the back pressure, and the pressure at the entry is that the stagnation pressure. And therefore the ratio of those two pressure is the back-pressure ratio, which might be wont to control flow of velocity. The pressure drops in a nozzle is due to the Bernoulli Principle , As fluid enters the smaller cross-section, it's speed up thanks to the conservation of mass. The fluid must flow quicker to take care of a gradual amount of fluid flowing through the restricted portion of the nozzle. **Bernoulli's principle**, in keeping with Bernoulli's theorem, a decrease in static pressure or a decrease within the fluid's P.E. occurs concurrently with a rise within the fluid's pace. The sum of mechanical energy, P.E., and internal energy must remain constant for this to happen.

In converging section of a nozzle

From the entrance to the throatthe region decreases, As the nozzle area decreases, the flow velocity increases, with the maximum velocity occurring at the throat. In a convergent nozzle, velocity increases while pressure decreases, but we know that pressure is inversely proportional to area.The fluid must flow quicker to maintain a steady amount of fluid flowing through the restricted portion of the nozzle.The velocity of a fluid flowing through a constricted region of a nozzle is increases while the static pressure decreases. The Venturi effect is the case for this concept or effect. Convergent nozzles accelerate the subsonic fluids. f the nozzle pressure ratio is high enough, the flow can reach sonic velocity at the narrowest point (i.e. the nozzle throat). Divergent nozzles slow subsonic fluids, however they accelerate sonic or supersonic fluids.

compressible fluid flow. The volume or density of a fluid does not change under normal temperature and pressure conditions. Gases, on the other hand, change volume (and thus density) in response to even minor changes in temperature or pressure. When a pressure or force is applied to a compressible fluid, it can indicate a significant change in density.The term "incompressible fluid" refers to a substance that cannot be compressed by applying external pressure.

Application of C-D nozzle

Steam turbine nozzles and rocket nozzles both use the C-D nozzle. The reaction engine nozzle is usually utilized in some models of steam turbines and jet engine nozzles. used as a propelling nozzle (propelling nozzle is that the component of a reaction-propulsion engine that operates to constrict the flow, to make an exhaust jet and to maximise the rate of propelling gases from the engine), in automobile and jets. Also employed in power plants, Rockets (such as for providing sufficient thrust to maneuver upwards).

De laval nozzle

The nozzle was developed by Swedish inventor Gustaf de Laval in the 19th century and this CD nozzle named as De laval nozzle. it's accustomed accelerate the flow of a gas that passes through it. It's utilized in a spread of steam turbines and could be a crucial component in modern rocket engines and supersonic jet engines. Its activity is predicated on the properties of subsonic and supersonic gases moving at different speeds. Since the mass rate is constant, the speed of a subsonic gas flow increase if the nozzle carrying it constricted . The gas flow through a de Laval nozzle is isentropic (gas entropy is sort of constant) and adiabatic (heat loss or gain is sort of zero).Since the gas is compressible in subsonic flow, sound, alittle pressure wave, can spread through it. The gas velocity becomes sonic (Mach number = 1.0) near the nozzle's "throat," where the cross sectional area is at its smallest, a condition referred to as choked flow. The gas begins to expand because the nozzle cross sectional area increases, and also the gas flow increases to supersonic velocities, where a undulation cannot travel backwards through the gas as seen within the remaining portion of the nozzle (Mach number > 1.0). Gas flow in de Laval nozzles is studied.

ideas and assumptions of the appliance for The analysis of gas flow through de Laval nozzles necessitates are :-

a. the combustion gas is believed to be a perfect gas for simplicity's sake.

b. The gas flow is isentropic (such that entropy is constant), friction less, and adiabatic (there is small or no heat gained or lost).

c. The gas flow is along a line from gas inlet to exhaust gas exit (along the nozzle's axis of symmetry).

d. Since the flow is moving at such high velocity, the gas flow activity is compressible.

Material used for nozzle manufacturing are:- Brass, stainless steel, cast iron, copper, tungsten carbide, nickel, silicon carbide, super alloys, titanium, nickel alloys, and tantalum are some of the materials used.

The continuity equation

$A_1V_1\rho_1 = \rho_2A_2V_2$

The steady flow energy equation

$$\frac{Q-W}{m} = h_1 + \frac{V_1^2}{2} + gz_1 + h_2 + \frac{V_2^2}{2} + gz_2$$

Where,
V - Velocity (m/s)
g- Gravitational Acceleration (m/s^2)
z- Height (m)
A- Area (m^2)
h- Enthalpy (kJ/kg)
m- mass flow rate(kg/s)
Q- heat transfer (kw)
W-work transfer(kw)

Mach number

Mach number is a measure of the compressibility characteristics of fluid flow, the fluid (air) behaves under the influence ofcompressibility in a similar manner at a given Mach number. Itis a dimensionless quantity in fluid dynamics representing the ratio of flow velocity to the local speed of sound.

Flow type	Muchnumber(M)
Subsonic	M < 1
Sonic	M = 1
Super-Sonic	M = 1-5
HyperSonic	M = 5-10

Table 1, flow type and mach number

M = *u*/c where M is mach number, *u* is the local flow velocitywith respect to the boundaries, *c* is the speed of sound in the medium

Simulation using CFD fluent
It is an engineering tool that helps us to conduct Experimentation, To perform the geometry or simulation of C-D nozzle and to study the parameter (density based) such as velocity , temperature, pressure , mach number .

◆ for the simulation we have to consider :-

✓ Air is flowing

✓ it is pressure inlet

✓ At the outlet pressure is lower than the inlet

✓ At the throat of C-D nozzle velocity and pressure decreased.

Modeling
Dimensional modelling of the C-D nozzle was done using solid work software as shown in (fig a&b) and The standard dimensions of the nozzle that we have been used in modeling are Total length of nozzle 190 mm,Inlet diameter 96 mm ,Throatdiameter 40 mm , outlet diameter 136 mm , Convergent angle 39 degrees , at Divergent angle 19 degree and 29°, we are using stainless steel for this design.

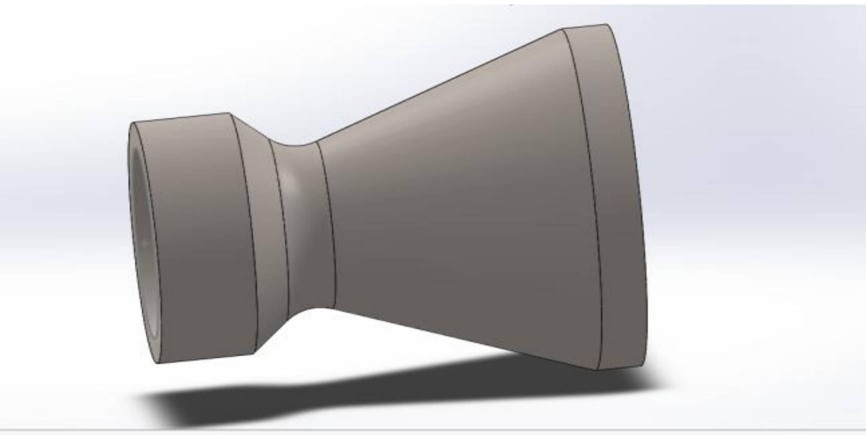


Figure 1 (a) C-D modeling at 19° divergent angle

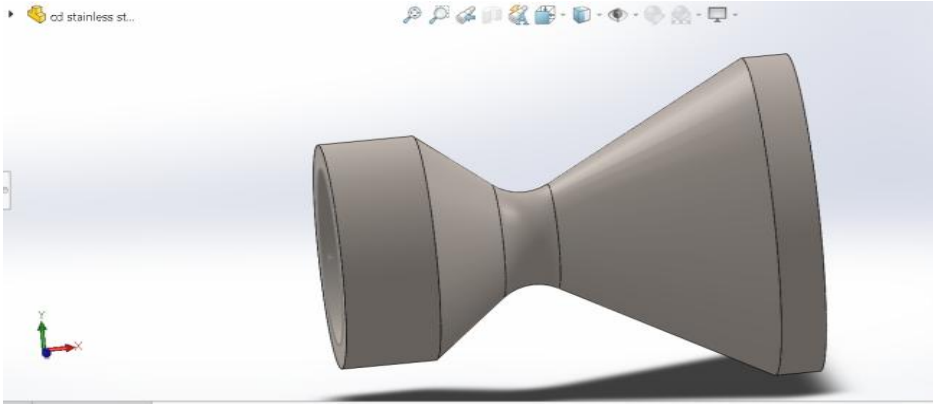


Figure 1 (b) C-D Modeling at 29° divergent angle

Geometry

In the ANSYS Workbench , by double-click the Geometry cell in the elbow fluid flow analysis system. And this displays the ANSYS DesignModelapplication, since we are going to make axis symmetry geometry so, have part of the nozzle as shown in figure 2 and also sizing was done and behavior of flow type set as hard,for curved section set size function as curvature.figure 2 represent geometer at 19 ° and figure 2b show Cd nozzle geometry at 29° divergent angle.

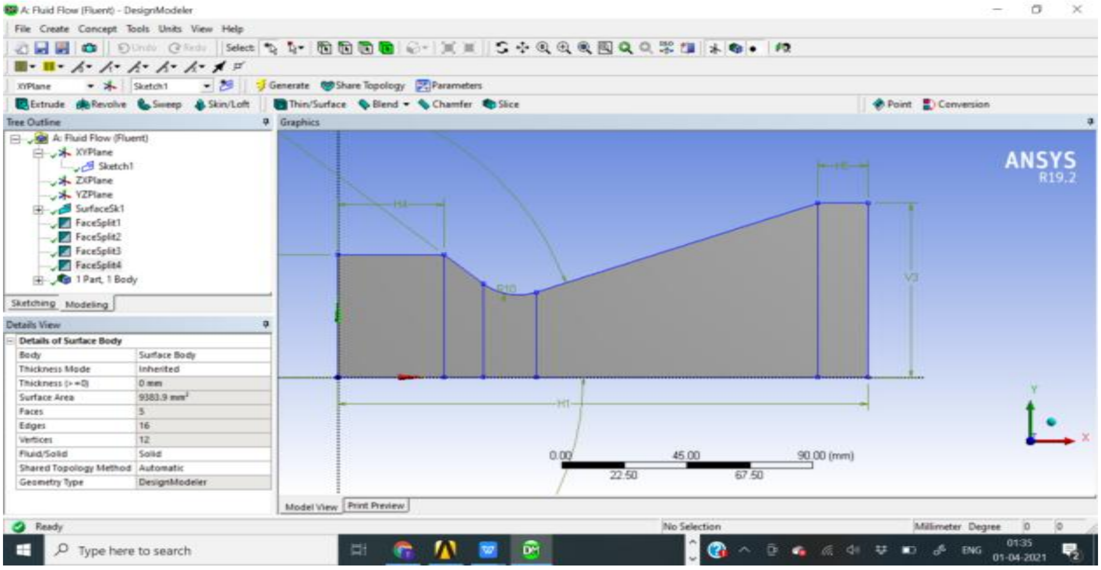


Fig 2a, 2D C-D Nozzle Geometry at 19 °

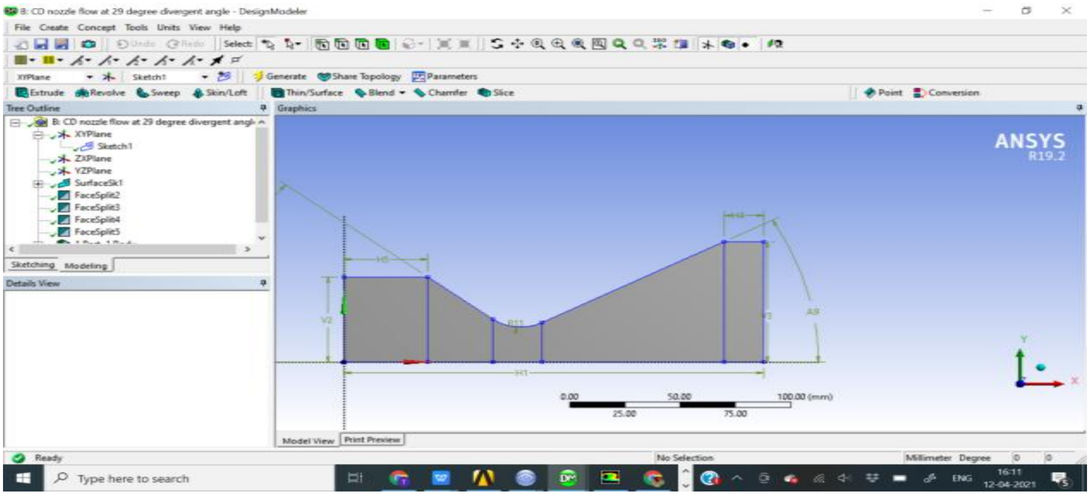


Figure 2b CD nozzle geometer at 29°divergent angle.

Meshing

It represents the geometric body as a set of finite elements, next to geometry, modelling of the nozzlewas done using ANSYS CFD software by clicking generate mesh and face meshing was apply for structural mapping mesh as shown in figure 3 a&b at 19° divergent angle and figure 3c, represent 2D geometry at 29° divergent angle.

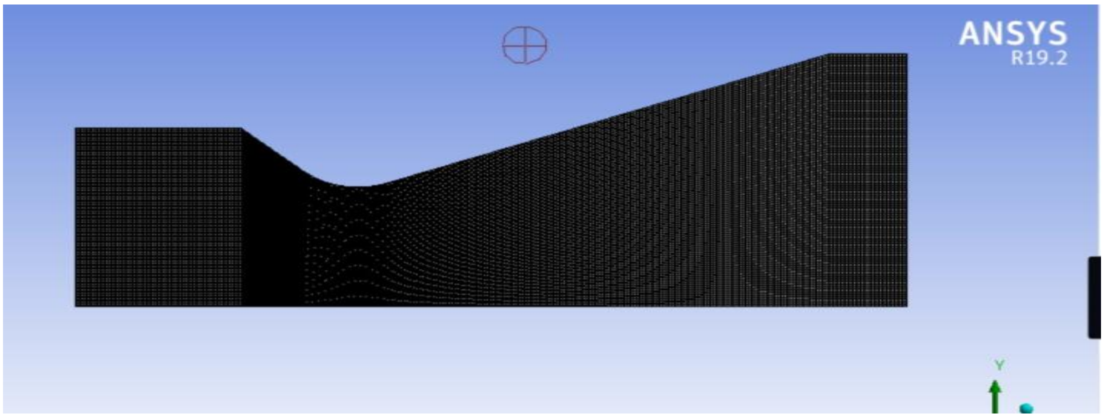


Figure 3 a, 2D meshing C-D nozzle

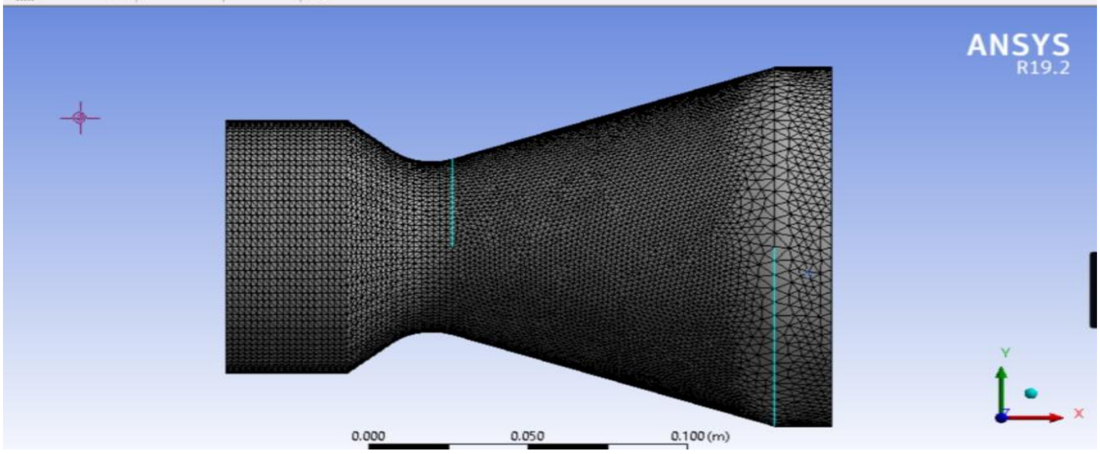


Figure 3b,3D meshing of C-D nozzle 19° divergent angle.

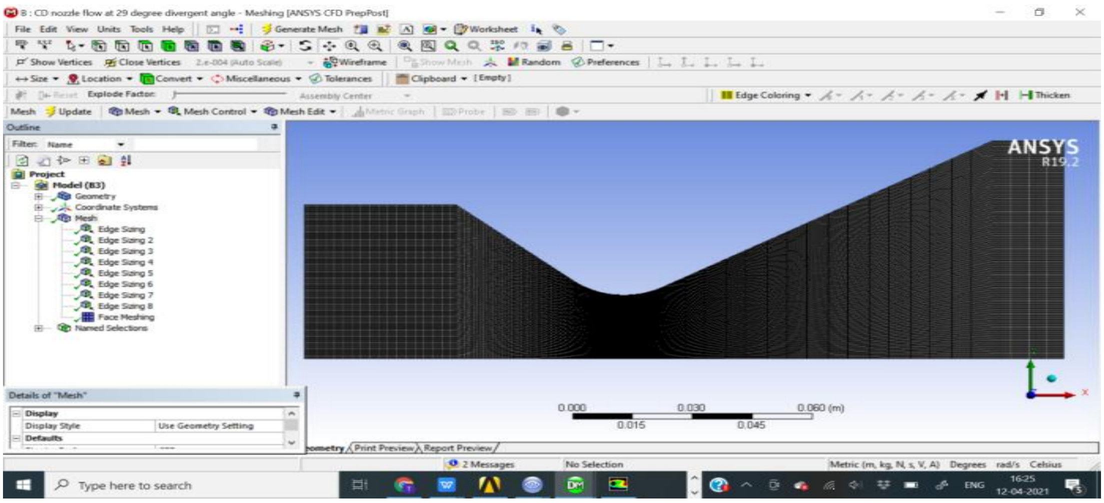
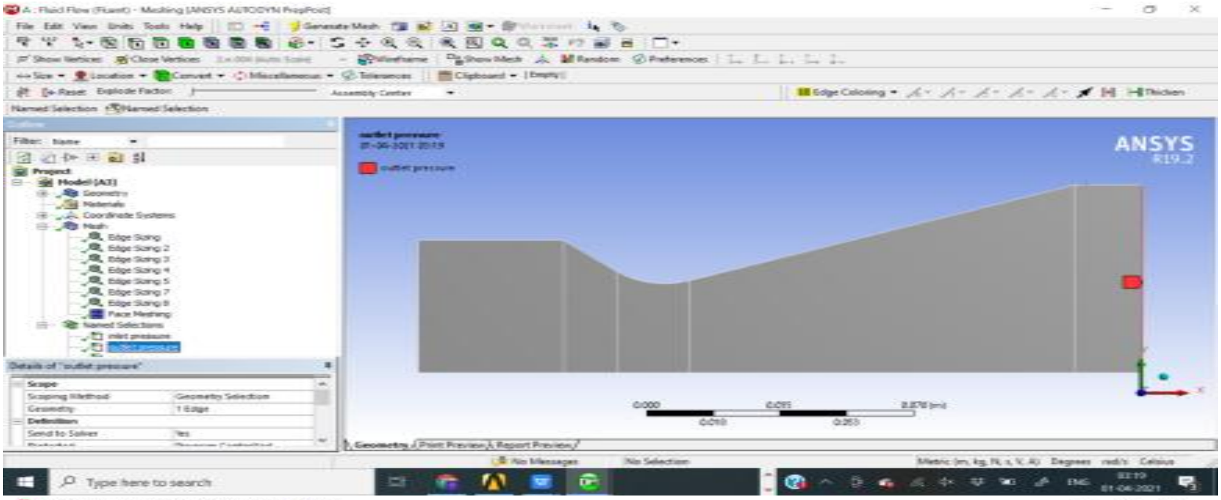
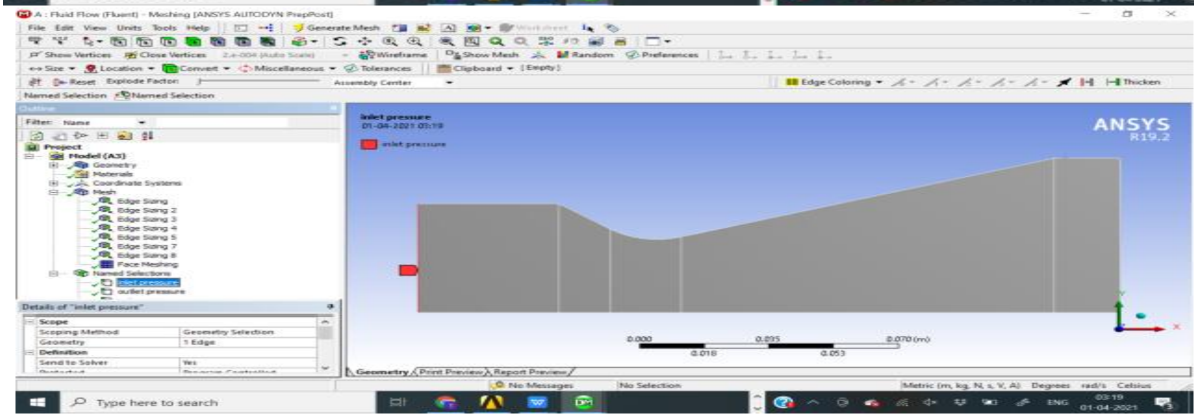


Figure 3c , CD nozzle meshing at 29° diverging angle.

BOUNDARY CONDITIONS(BC)

After meshing boundary condition was done using ANSYS software such as inlet pressure, outlet pressure, axis and wall as shown in the figure 4 checked the the correct boundary naming(fig4a,b,d,c respectively). Inlet pressure is atmospheric pressure (101235 pa), Inlet temperature 300k, for first test, and 102325(pascal) pressure, 62.8m/sec velocity , at 300k,for the third test.figure 4 given below shows boundary condition named selection.





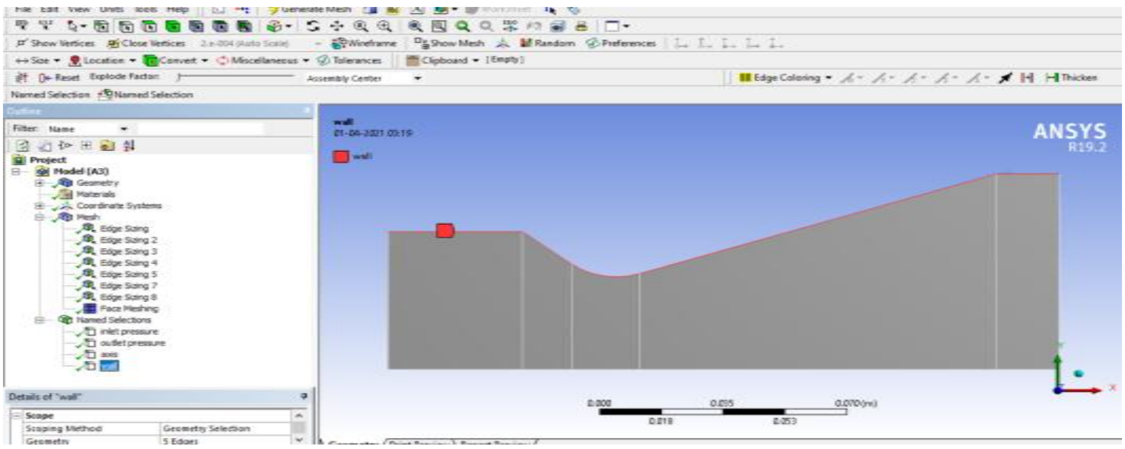


figure 4c

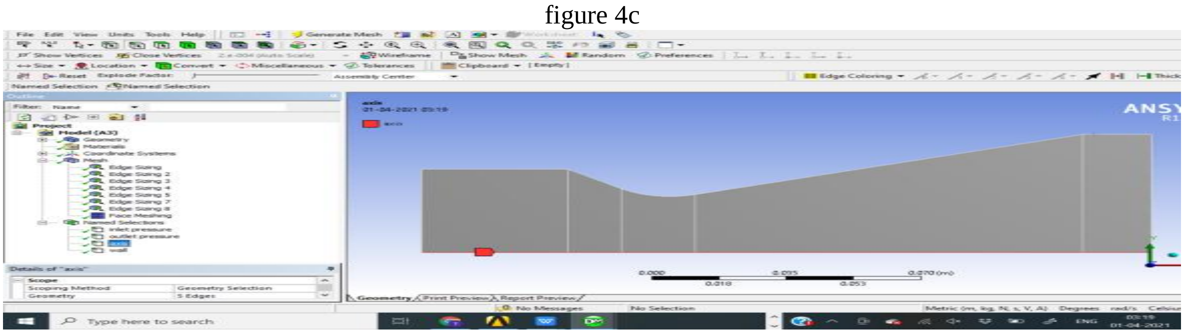


Figure 4,boundary condition d)

Solver Setup, CFD post processing

The solver type is density based because for compressible flow analysis gives more accurate result. 2D space and time is steady state and density is ideal gas, for compressible flow.

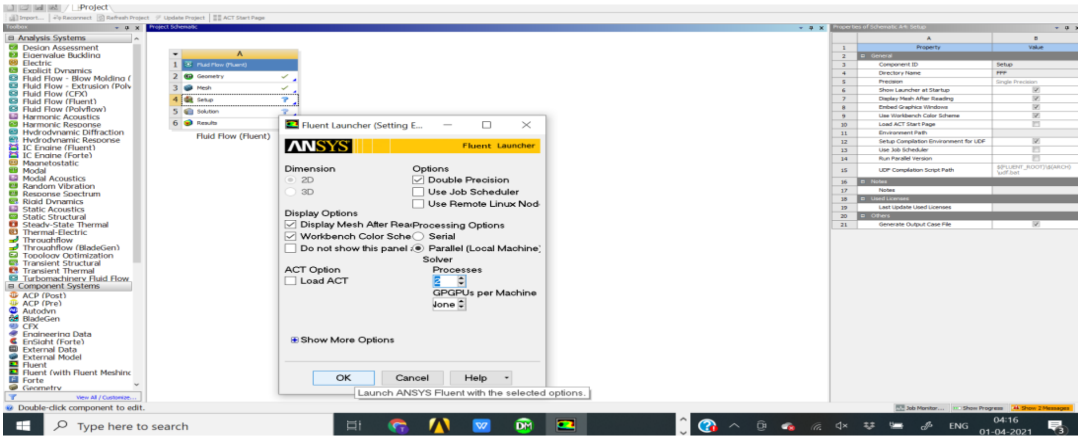


Figure 5, solver setup

The material here selected as air , to run the calculation add the output parameters that used in result such as static pressure, total pressure,mach number, axial velocity.before run calculation,reference value should be set as inlet, number of iteration taken as 1400, then calculate we got figure 6, convergent graph.

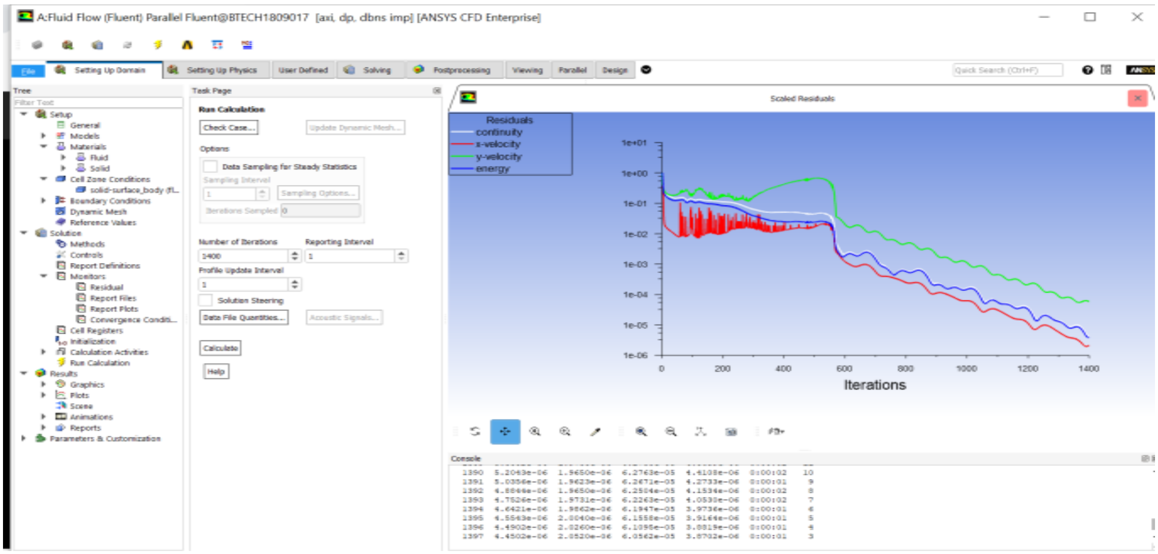


Figure 6a

Figure 6a, convergent graph at 19°divergent angle and atmospheric pressure inlet (101325 pascal).

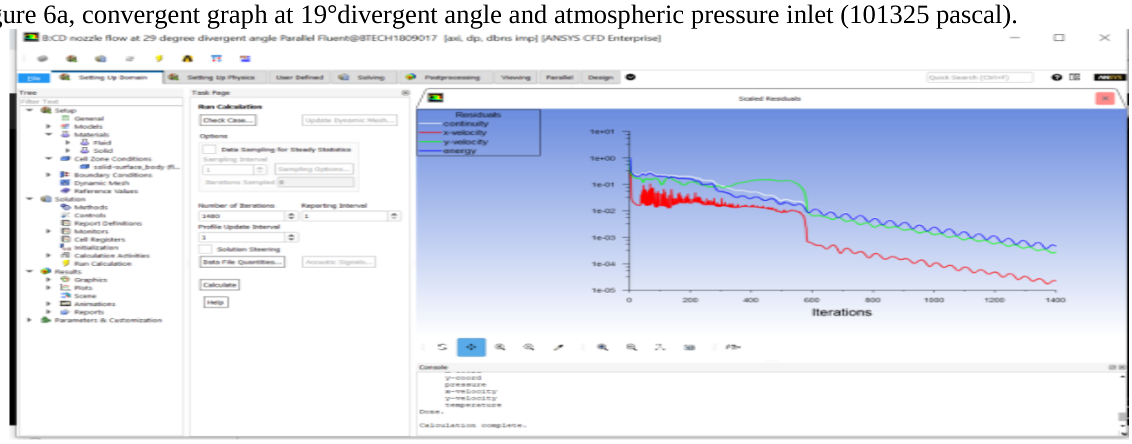


Figure 6b, convergent graph at 29°divergent angle, and at 102325 (pascal) pressure inlet.

The flow was subsonic (mach number < 1) at converging section,the flow becomes sonic (mach number =1) near to the throat section,and the flow becomes supersonic (mach number >1) at diverging section, see the figure below (figure 7a).

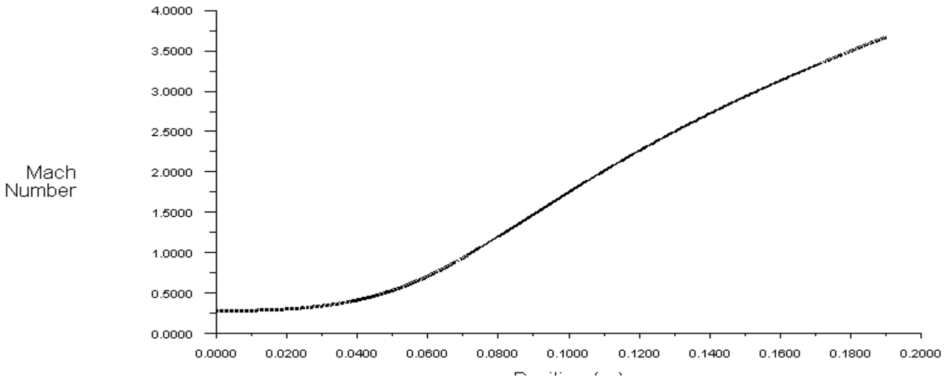


Figure 7a

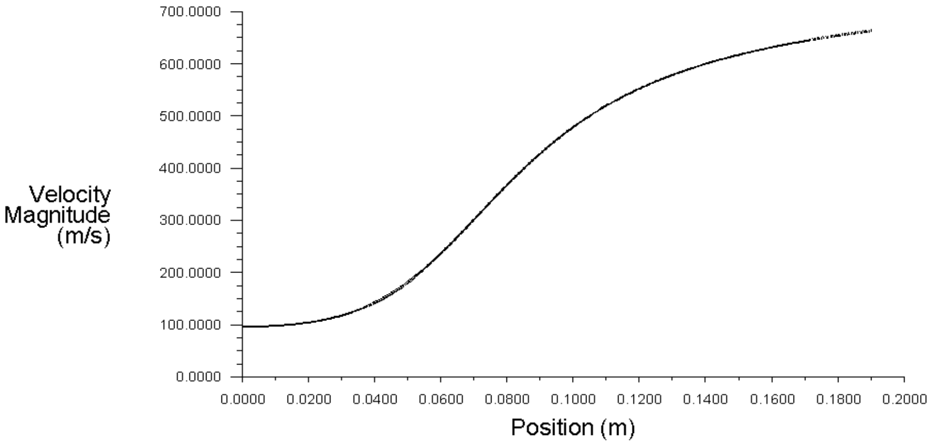


Figure 7b

When the cross section decrease, the velocity was increase and static pressure decrease at that section and this is due to venturi effect as shown in figure 7,b and c respectively

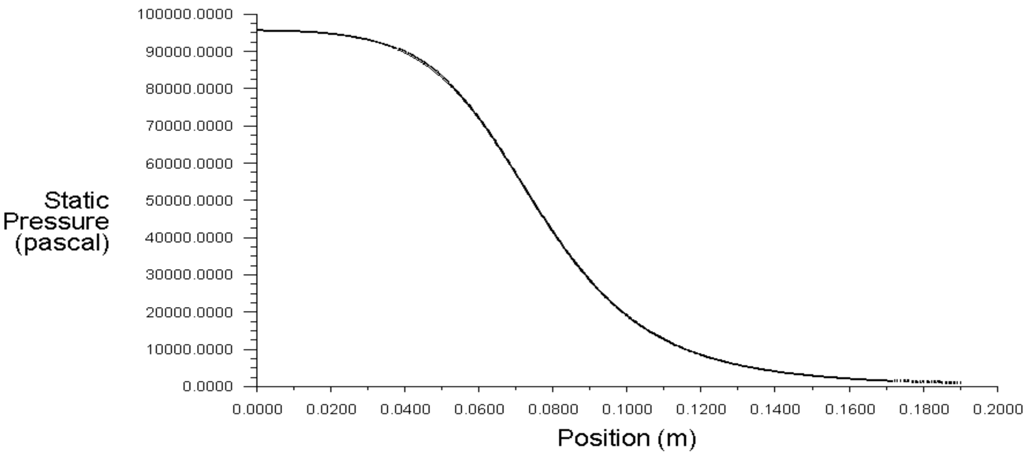


Figure 7c

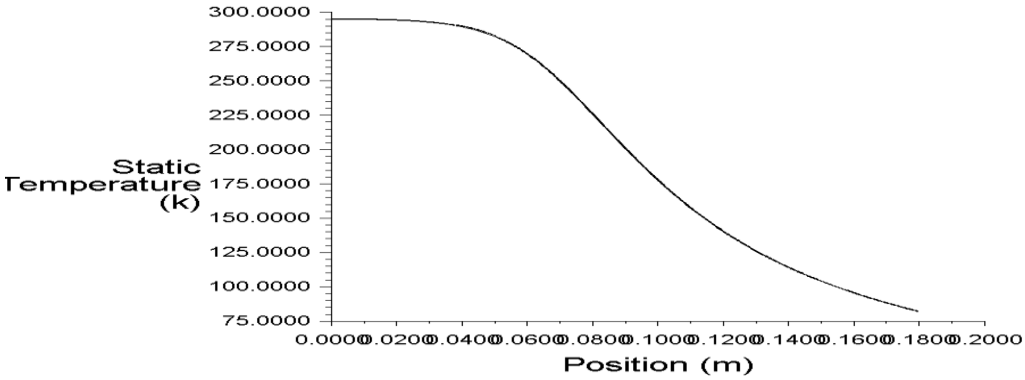


Figure 7d

III. RESULT AND DISCUSION

in Aerospace,steam turbine and rocket engine, Convergent and divergent nozzle plays a major role For controlling the direction and speed of fluid flow. The analysis of C-D nozzle is carried out in ANSYS to obtain the flow properties such as temperature, mach number velocity, pressure and density.

IV. CONTOURS

Velocity contours, pressure contour,density and temperature contour flow properties was determined as follows.

PRESSURE CONTOUR

Figure 8, pressure contour at 19° divergent angle and atmospheric pressure.

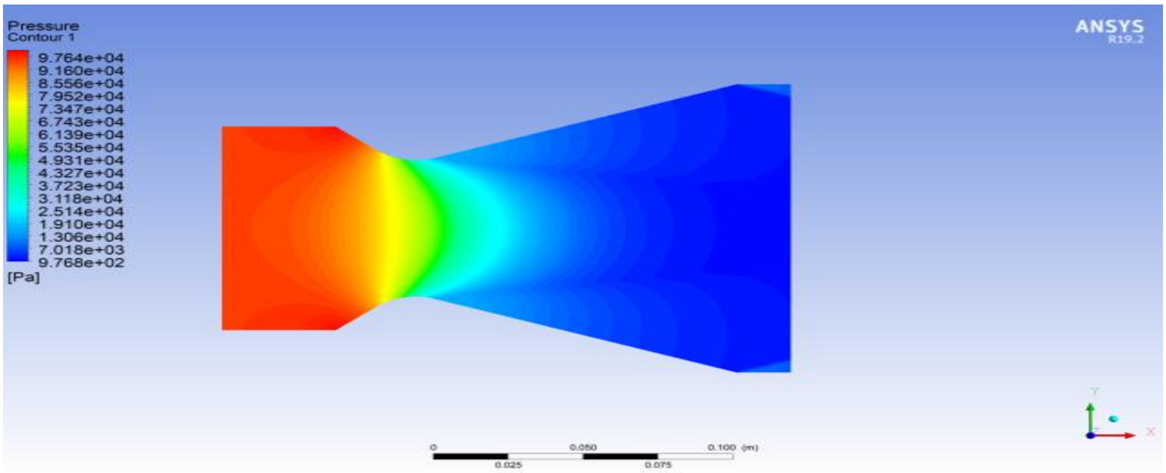


Figure 8a, pressure contour for full nozzle at 19° divergent angle.

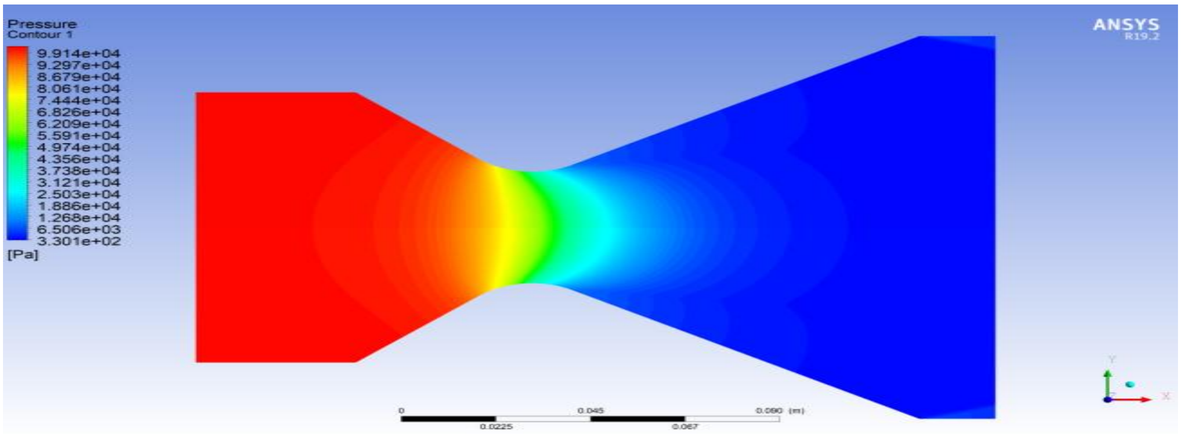


Figure 8b, pressure contour at 29° divergent angle and 102325 pascal pressure inlet.

In figure 8a, we can observed that the maximum static pressure is at the converging section or at he inlet, which is indicated by red colour (9.764e^+04) and minimum value at the diverging section or at outlet (9.77e^+02).but at a divergent angle(29°),the maximum pressure contour is 9.914e^+04 indicated by red colour as figure 8b and has minimum value of 3.301e^+02 at divergent section.

VELOCITY CONTOUR

In figure 9a velocity contour at 19°divergent angle, shows that the velocity is or diverging section represent by red colour having the value 6.464e^+02 , just at at the throat the velocity start to increase.with the minimum value in convergent section or at the inlet (3.106e^+01). and figure 9b,shows that velocity contour at 29°divergent angle, and at 102325(pascal) inlet pressure.with the maximum value at the divergent section 6.766e^+02 m/sec and minimum value 1.182e^+01 m/sec at convergent section. We can see that the velocity is maximum at the 29°

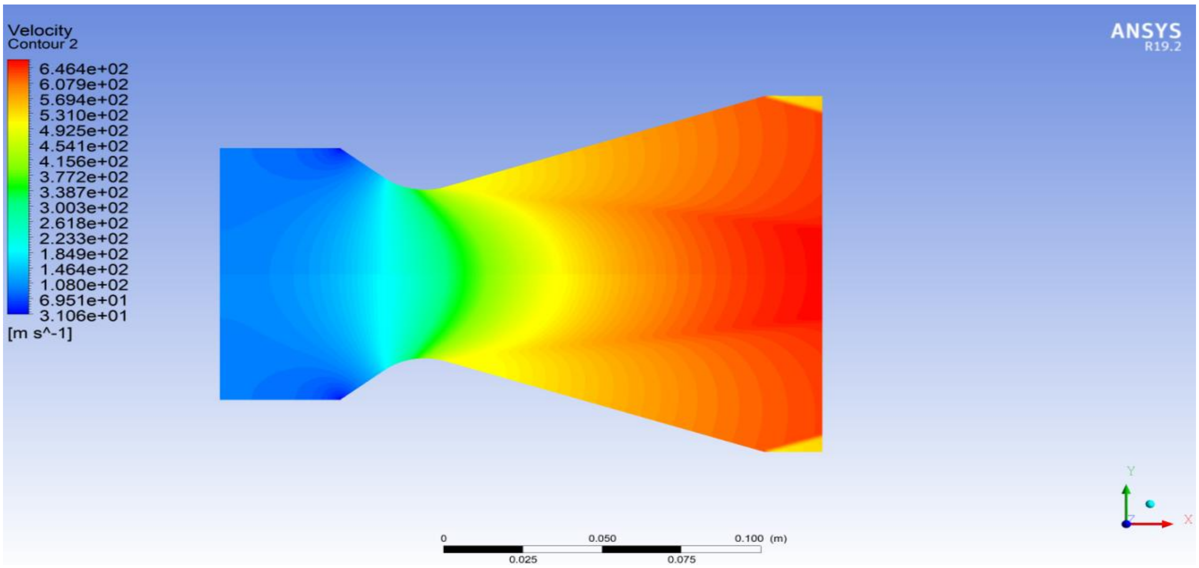


Figure 9a, Velocity contour at 19° divergent angle, velocity of 45m/sec, at inlet pressure 101325 (pascal).

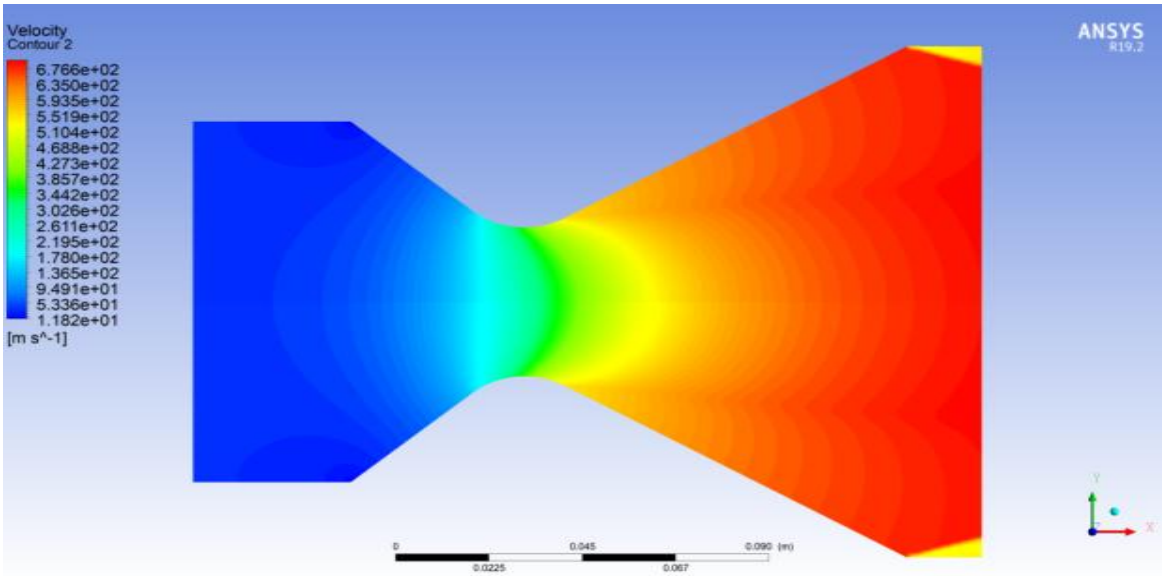


Figure 9b, velocity contour at 29°divergent angle,62.8 m/sec and at 102325(pascal) inlet pressure.

TEMPERATURE CONTOUR

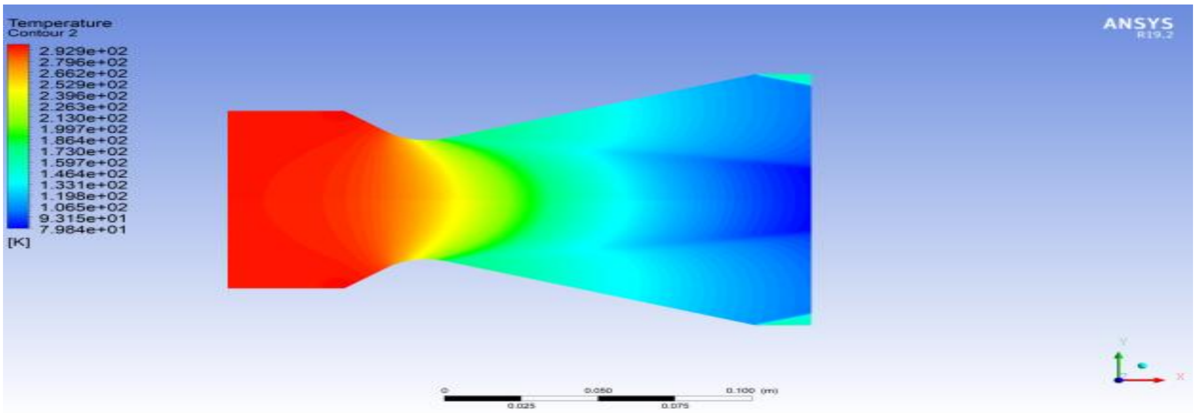


Figure 10 a, temperature contour at 19° divergent angle and atmospheric pressure.

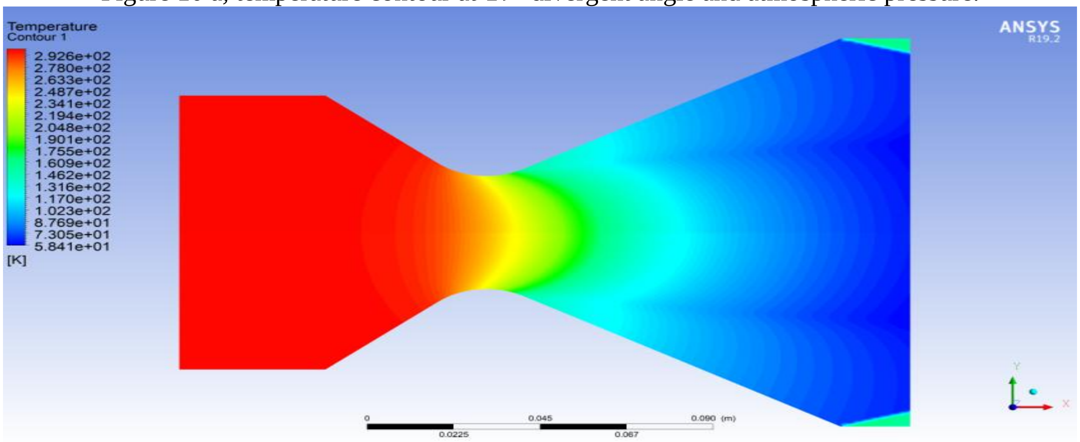


Figure 10 b, temperature contour at 29° divergent angle.

We can observed that in figure 10a , the Red color indicates the maximum temperature contour of (2.929e^+02)k, at the inlet or converging section and the blue color indicates minimum temperature at the outlet or diverging section (7.984e^+01)k. And figure 10 b, shows temperature contour at 29° divergent angle has maximum value indicated by red colour(2.926e^+02)k and minimum value 5.841e^+01 k at diverging section.

Mach number contour

As we discussed in figure 7a graph, The flow was subsonic (mach number < 1) at converging section, the flow becomes sonic (mach number =1) near to the throat section,and the flow becomes supersonic (machnumber >1) at diverging section.Here also we can observe that in figure11a.

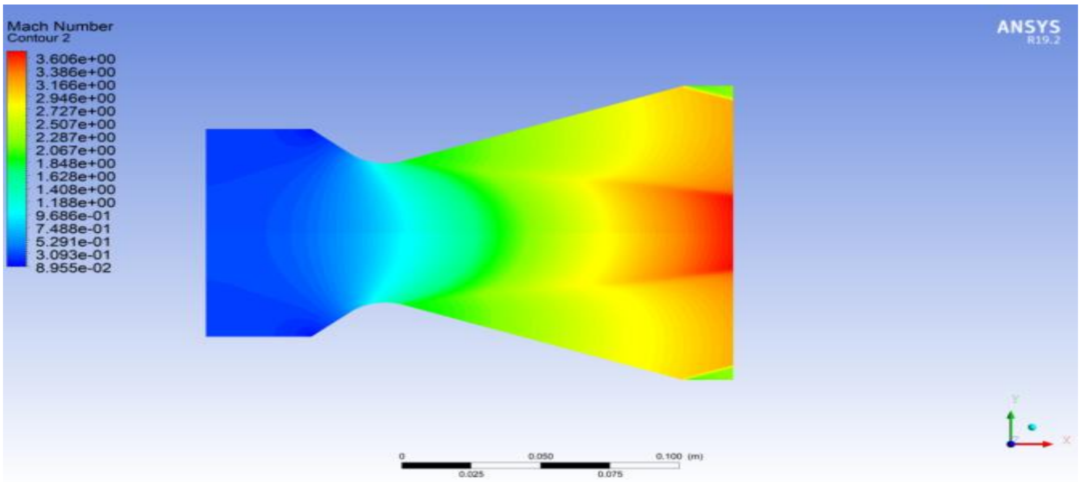


Figure 11 a, mach number contour at 19° divergent angle.

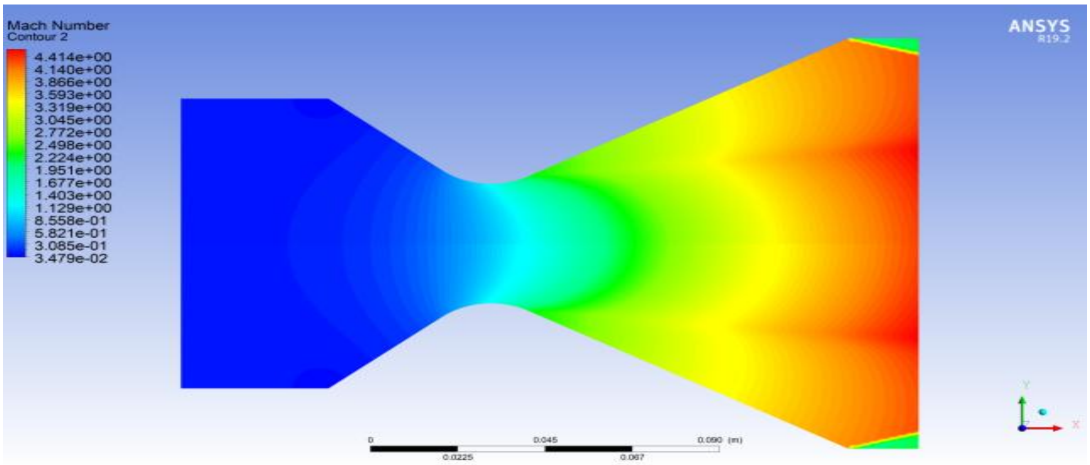


Figure 11 b, mach number contour at 29° divergent angle.

In Figure 11 a, we can seen that mach number at the convergent was less than one and the flow was sonic (between 8.955e^-02 to 9.686e^-01), at the throat nearly one (between 9.686e^-01 to 1.129e^00), so that the flow is sonic, and at the divergent section mach number grater than one(grater than 1.129e^00), the flow becomes hyper sonic.in figure11 b, above observe that mach number at 29° divergent angle,at inlet pressure of 102325 pascal. subsonic flow mach no value between 3.479e^-02 to 8.558e^-01 near to throat, sonic (between8.558e^-01 to1.129e^+00), and greater than 1.129e^00 at the divergent section.

Density contour

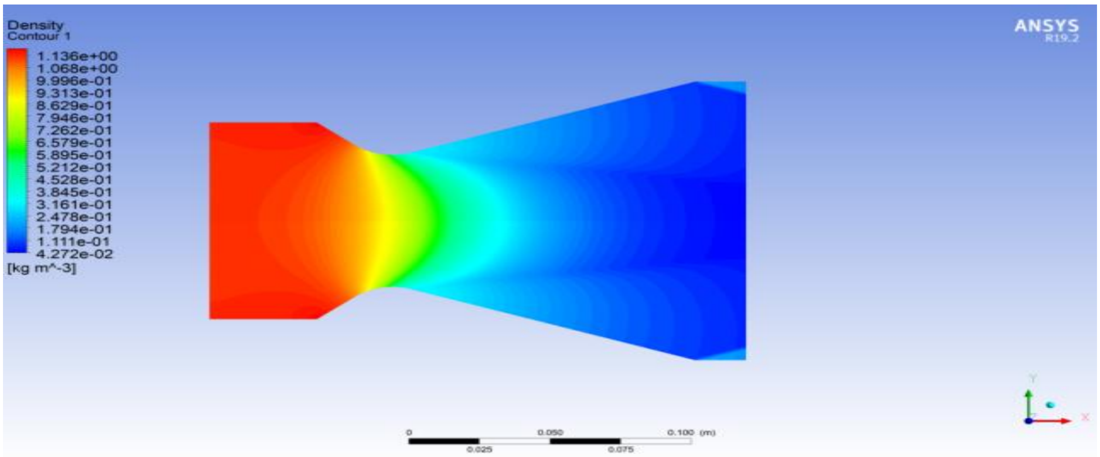


Figure 12 a, density contour at 19° divergent angle.

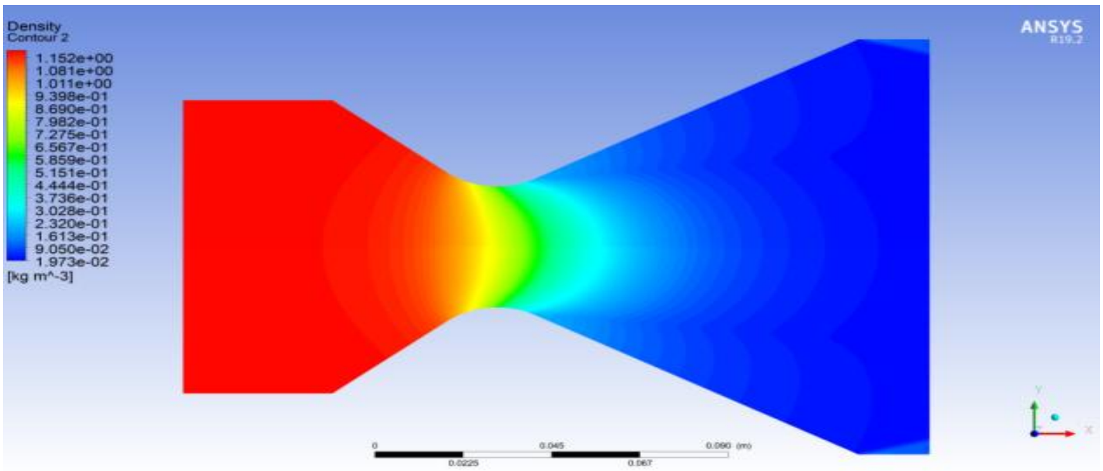


Figure 12 b, density contour at 29° divergent angle.

The density of the fluid flow through the C-D nozzle changes from inlet to outlet gradually,figure 12 a, shows that the changes of density, 4.272 e^-02 at the outlet and 1.136 e^+00 at the inlet. Figure 12 b, shows density contour at 29° divergent angle have minimum value 1.973e^-02 kg/m^3 and maximum 1.152e^+00 kg/m^3.

VELOCITY STREAMLINE

A streamline is a straight line that is tangential to the instantaneous velocity direction (velocity is a vector with magnitude and direction). Streamlines cannot overlap since the velocity at any point in the flow has a single value (the flow cannot go in more than one direction at the same time).Velocity streamline is observed as the figure shown below(figure 13 a).

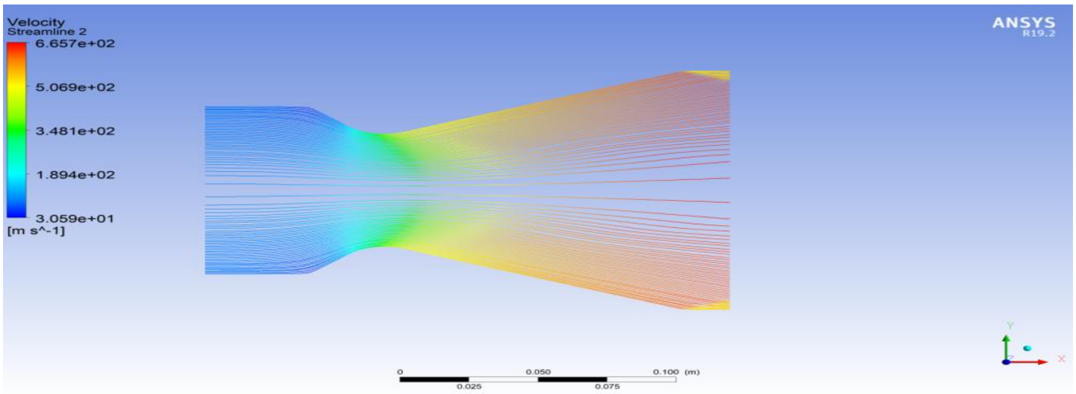


Figure 13 a, velocity streamline at inlet velocity of 45m/sec,101325 inlet pressure(pascal).

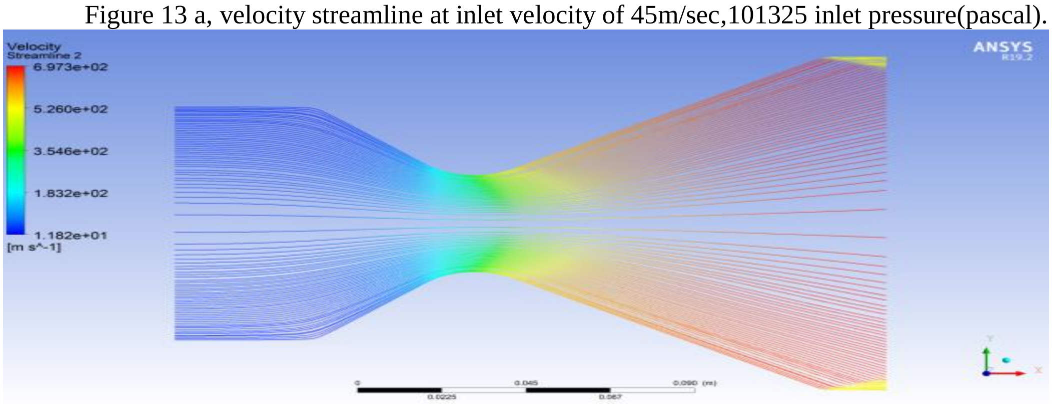
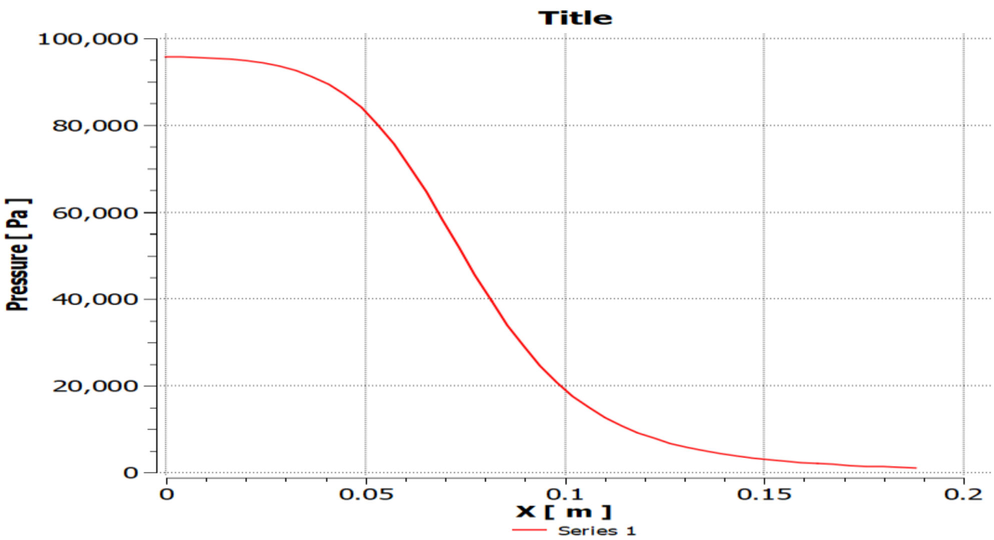
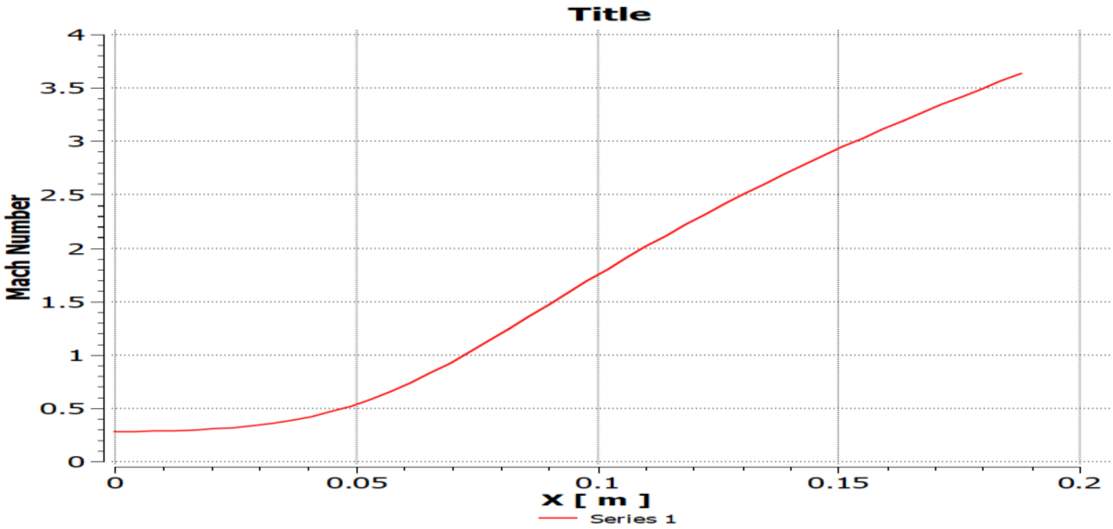


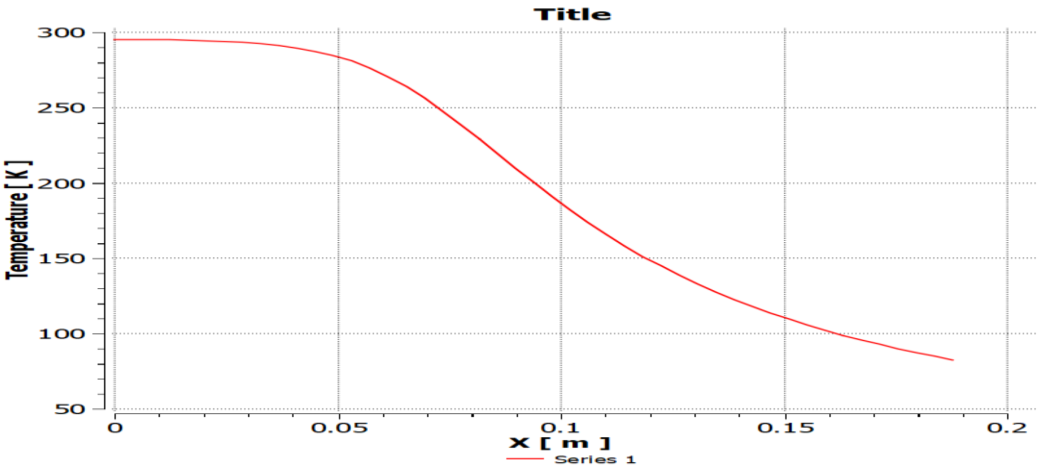
Figure 13 c, velocity streamline at 29° divergent angle,at 102325 inlet pressure(pascal) and a velocity of 62.8m/sec.



figure,14a



figure, 14(b)



Figure,**14**(C)

Fig 14, Chart graph representation of temperature,pressure,density,mach number,and velocity.

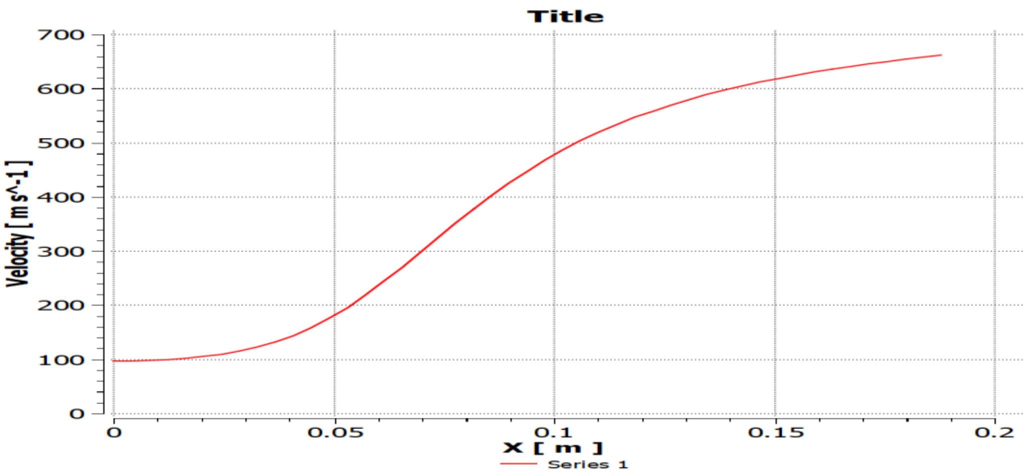


Figure 14(d)

V. CONCLUSION

A nozzle is a device crafted by engineers to control the characteristics of the fluid. It is mostly used to increase the velocity of the fluid which normally consists of convergent portion, Throat section and Divergent section. In this project, design and analysis of a convergent- divergent nozzle is carried out using solid work and

ANSYSFLUENT in order to study the flow field analysis considering input variables as pressure inlet and pressure outlet. From this study and analysis various flow properties like pressure, temperature, velocity and density are found. and The aim of a converging diverging nozzle is to produce supersonic flow near the exit plane and increase the outlet velocity.

In the convergent section the pressure is maximum and minimum at the divergent section ,velocity increase as the area increase and maximum velocity is at the divergent section or exit. using CFD simulation has been done the geometry , use of face split,face meshing, and set up also done using density based set up because of that we are doing a compressible fluid flow and air is taken as a material of fluid. the result for contour temperature, velocity contour, mach number contour, density contour, and pressure contour was obtained at a 19° and 20°divergent angle and 101325pa and 102325pa inlet pressure, velocity streamline also resulted as shown Figure 13a, 101325 inlet pressure(pa) the velocity is 45m/sec, with inlet pressure 101325(pa), Figure 13c, velocity streamline at 29° divergent angle,at 102325 inlet pressure(pascal) and a velocity of 62.8m/sec. Here we can concluded that at grater divergent angle of a nozzle we can obtained the highest velocity in this experiment by 10° changes in the divergent angle we got 17.8 m/sec velocity by keeping convergent angle constant. A de Laval nozzle's gas flow is isentropic (gas entropy is nearly constant) and adiabatic (heat loss or gain is nearly zero). Since the gas is compressible in subsonic flow, sound, a small pressure wave, can spread through it.

SCOPE OF FURTHER WORK

this work can be further analyzed using CFD ANSYS fluent software by changing both the convergent and divergent angle at different inlet pressure , science in this work we are consider and changing the divergent angle by keeping convergent angle constant at 39°. for future investigation of the properties and effect of mach number on the CD nozzle can be study to get better precise result more than this work, to get supersonic flow and maximum velocity at the exit of the CD nozzle.

REFERENCE

[1]. International Journal of Engineering Technology Science and Research IJETSR www.ijetsr.com ISSN 2394 – 3386 Volume 5, Issue 4 April 2018.

[2]. Ali, A., Neely, A., Young, J., Blake, B., & Lim, J. Y. (2010). Numerical Simulation of Fluidic Modulation of Nozzle Thrust. In 17th Australasian Fluid Mechanics Conference (pp. 5–8).

[3]. G. Satyanarayana, Ch. Varun, S.S. Naidu, "CFD analysis ofconvergent-divergent nozzle" in an International Journal of Scientific and Research.

[4]. Gustaf De-Laval, "Concepts and CFD analysis of De- Laval nozzle" in International Journal of Mechanical Engineering and Technology (IJMET) Volume 7, Issue 5, September 1991.

[5]. TrongBui, Bogdan-AlexandruBelega, "CFD analysis of flow in Convergent-divergent nozzle", international conference of scientific paper AFASES 2015 Brasov, 28-30 May 2015.

[6]. note labelSuttoncitebook , Sutton, George P,Rocket Propulsion Elements, An Introduction to the Engineering of Rockets 6th Edition pages636 publisher, Wiley-Interscience , year1992 , id0471529389.

[7]. international conference of scientific paper afases 2015 brasov, 28-30 may 2015.

[8]. A. Khan, Shembhakar, 2008, "Viscous Flow Analysis in a convergent-divergent nozzle", International Conference on Aerospace Science and Technology, Bangalore.

[9]. R. Boyanapalli et al., 2013, "Analysis of Composite De-Laval Nozzle Suitable for rocket applications", International Journal of Innovative Technology and Exploring Engineering, 2, pp. 336-344.

[10]. V. Lijo, H.D. Kim, T. Setoguchi, S. Matsuo, 2010, "Numerical simulation of transient flows in a rocket propulsion nozzle", International Journal of Heat and Fluid Flow, 31, pp. 409-417.

[11]. Q. Xiao, H.M. Tsai, D. Papamoschou, 2007, "Numerical investigation of supersonic flow separation", AIAA Journal, 45, pp. 532-541.

[12]. In Convergent -Divergent Rocket Engine Nozzle Using Computational Fluid Dynamics, AFASES,2015.

[13]. International Journal of Innovative Technology and Exploring Engineering (IJITEE) ISSN: 2278-3075, Volume-9 Issue-6, April 2020.

[14]. Sudhakar BVVN, Sekhar BPC, Mohan PN, Md Ahmad T (2016) Modeling simulation of convergent-divergent nozzle using computational fluid dynamics. Int Res J Eng Technol (IRJET) 03:2395–0072.

[15]. Biju Kuttan P, M Sajesh - *O*ptimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics - The International Journal And Science, volume 2, pages 196-207, 2013, ISSN: 2319 – 1813.