A 3-D NUMERICAL INVESTIGATION AND PARAMETRIC_CFD_ANALYSIS OF FLOW THROUGH CONVERGENT-DIVERGENT NOZZLE USING ANSYS_CFX

Tesfaye Barza Zema
Assistant Lecturer, Mechanical Engineering (Thermal) Wolaita Sodo University, Ethiopia
Email id: tesfayebarza@gmail.com
*Corresponding Author: Tesfaye Barza Zema

ABSTRACT

A nozzle is a device used to control the characteristics of fluid. It is mostly used to increase the velocity of fluid. Computational Fluid Dynamics (CFD) is a method which depends on various numerical analysis and algorithms for finding solutions and analyzing the complex problems that concern fluid flow and arbitrary of Fluid Mechanics. This paper is concerned about to determine A 3-D Numerical investigation and parametric_CFD Analysis of flow through Convergent-Divergent Nozzle using Ansys_CFx, which encompasses the variation of flow parameter like velocity, pressure, temperature and density is visualized using Computational Fluid Dynamics. The simulation of convergent divergence Nozzle through ANSYS FLUENT CFX is also done. Furthermore, A 3-D flow simulation of turbulent_numeric the flow pattern in-terms of velocity, streamline and pressure distribution density and eddy viscosity are graphically interpreted. Furthermore, the simulation time was steady state (Total energy) and turbulence (shear stress transport) and moving reference frame was used to consider the C-D under high resolution. And Fluid Flow (CFX) _CFX Solver with global Run setting, Run Mode[Platform MPI Local Parallel] and the parametric results are published in appendix. Besides, the geometry is done in ANSYS 16.0. Finally, CFD analysis was performed in finally, the discretization process and CFD analysis are carried out in Ansys Fluent_16.0.

Keywords: Nozzle, Parametric CFD analysis, ANSYS_CFX, Computational Fluid Dynamics (CFD), Convergent Divergent Nozzle

1. Introduction

Nozzle is a device used to control the speed of the fluid flow, directions and flow characteristics. like pressure, density, temperature and velocity. The major function of a nozzle is to increase flow velocity by converting pressure and heat into kinetic energy. It is mostly used to generate thrust in rockets or air breathing engines to gain lift. A rocket engine nozzle can accelerate fluid at subsonic speeds to supersonic speeds [12]. The magnitude of thrust produced by the nozzle is the most important requirement for designing a nozzle. The altitude at which it operates, as well as the working fluid properties that govern the flow, are its molecular weight, specific heat at constant pressure or volume, and specific heat ratio. Convergent nozzles are those in which the area of cross section decreases as the length of the duct increases. The pressure drops due [1]to the random motion of the particles at the inlet converging point are restricted to fast forward stream line motion. Thus the kinetic energy of the medium increases at the throat or outlet section of the nozzle. Convergent nozzles are widely used for high velocity jets, sprays, injections etc.

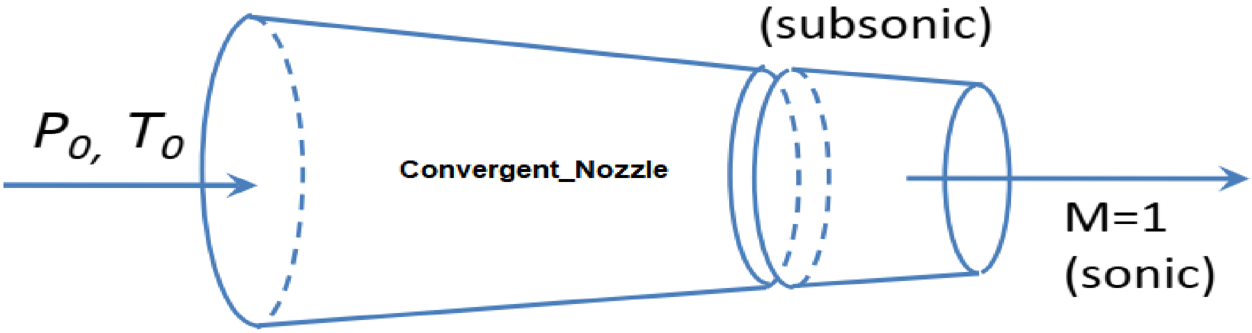


Fig1: Converging Nozzle

On the other hand, The Convergent-Divergent Nozzle was invented in 1888 by Carl Gustaf De Laval, a Swedish engineer. CD Nozzle is another name for De Laval Nozzle. The De Laval nozzle is divided into three major sections: converging, throat, and diverging.Flow through C-D Nozzles is typically assumed to be adiabatic because heat transfer per unit mass is much less than the difference in enthalpy between the inlet and outlet. The flow from the inlet to the throat can be assumed to be isentropic(Stagnation property remains constant).But the flow from the throat to the exit may not be due to the possibility of shocks.[13] The nozzle is designed as a pipe with varying cross sectional areas along its length to change and control the mass flow rate, velocity, direction of flow, pressure ratio, and other parameters. The pressure difference between the inlet and outlet of the nozzle section causes a change in flow characteristics. Convergent-divergent supersonic nozzles are widely used in rocket and high-speed missile nozzles. Various shapes are used depending on the nozzle application. A similar type of CD nozzle was usedfor experimental studies and observed the wall pressure performance with and without control usingthe microjet controller for different Mach number and area ratios of duct [2].

The divergent portion of the nozzle is critical to the expansion characteristics. The CD nozzle's function is to convert thermal energy into kinetic energy in order to expel high-speed exhaust. The major purpose of a CD nozzle in industry is to enhance the kinetic energy of the flow medium at the expense of internal energy and pressure. Nozzles are classified as convergent, divergent, or both convergent and divergent.[3]

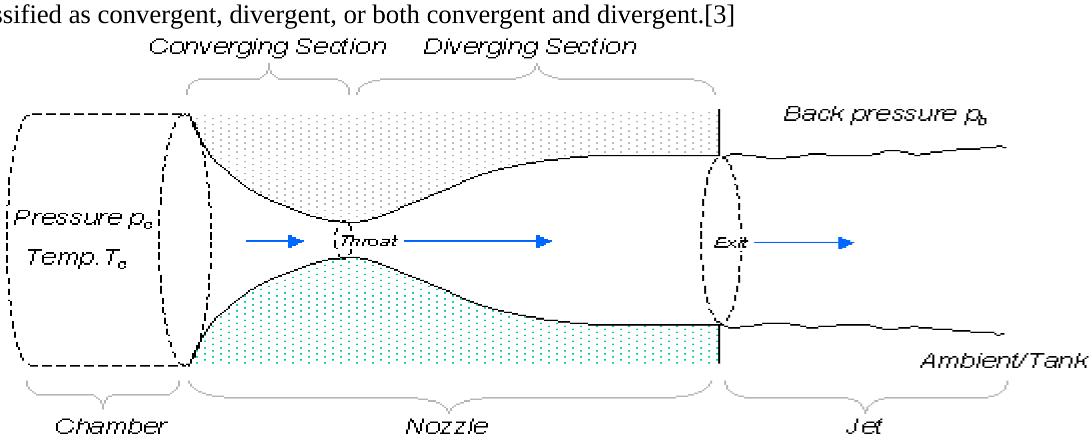
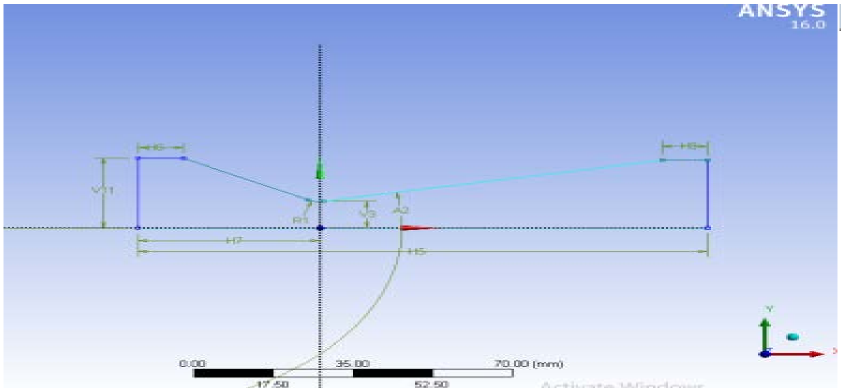


Fig_2: Converging _Diverging NozzleConfiguration

Computational fluid dynamics (CFD) is a versatile technique for communicating relevant results about an object's flow characteristics through the modeling and simulation of flow fields. The solution to the Reynolds averaged Navier-Stokes (RANS) equations is transitory, which complicates computational studies for flow fields through suddenly expanded convergent-divergent (CD) nozzles and the execution of a suitable turbulence model for RANS equation closure.[4] The numerical simulations and experimental measurements show discrepancies in the compressible flow regions of the CD nozzle, which are influenced by intense pressure gradients and complex secondary flows. The main objectives of this study are to compute A 3-D Numerical investigation and parametric_CFD_Analysis of flow through Convergent-Divergent Nozzle using Ansys_CFx. The Preliminary Convergent-Divergent Nozzle Design Specifications by Ansys_CFx mentioned below, Which encompasses thattotal length of Nozzle, Inlet &Outlet radius, Convergent length , Convergent angle(deg) and Nozzle perspective where as A2 is Throat_ Radius & V3 is Nozzle _Angle.[15].



Details View

⊟	**Details of Sketch1**	
	Sketch	Sketch1
	Sketch Visibility	Show Sketch
	Show Constraints?	No
⊟	**Dimensions: 8**	
	D A2	191.17 °
	H5	125 mm
	H6	10 mm
	H7	40 mm
	H8	10 mm
	R1	5 mm
	V11	25 mm
	D V3	9.5894 mm

Table_3: Dimensions of the Convergent - Divergent Nozzle with Sectional view of the angle

2. METHODOLOGY

2. 2.1: CFD Model

In order to create the CFD model of divergent convergent nozzle first, we need to create the project schematic in Ansys workbench at CFx analysis. CFD fluent workbench used for the modelling and analysis of nozzle. First of all, we need to create the 3D sketch of nozzle geometry according to the setup and convert it to the 3D plain geometry. The modelling has done in geometry workbench by using sketch and modelling tools.[5] There are several commands available in geometry workbench by which we are able to create any complex geometry also same as other modelling software.

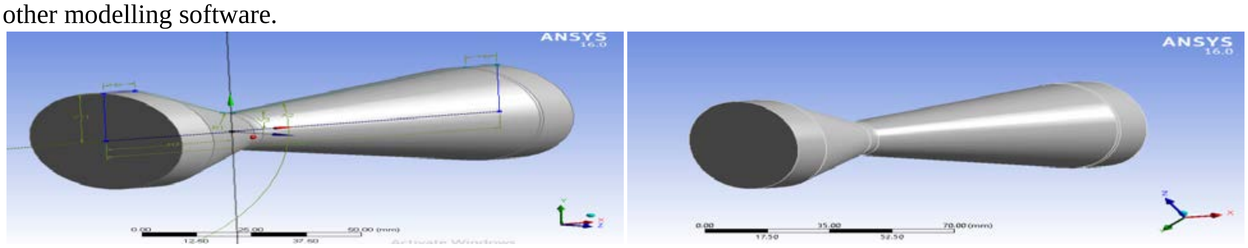


Fig_4: Ansys model of nozzle

Computer simulation of nozzle was done using computational fluid dynamics (CFD). CFD is method tosolve complex problems involving fluid flow.The above conditions were taken as mentioned in the previous section and the computer simulation thatis the analysis was done using Ansys-Fluent. When we performed this analysis, we found out the variation of parameters as we did in the case of theoretical treatment.[6]The CFD Analysis was done in the following steps:

- Modelling
- Meshing
- Pre processing
- Solver/Processing
- Post processing

2.2. Meshing

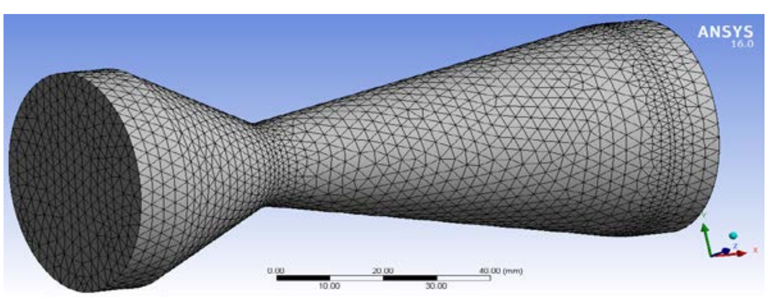


Fig_5:TetrahedraMeshmodel of nozzle

The model created by the above dimensions was meshed in mesh mode of Ansys component systems.Meshing is nothing but converting an infinite number of particles model of finite number of particles. Thedetails of mesh were Physics preference: CFD. The relevance of meshing was set to 100. The proximityand curvature option was selected, since nozzle is a curved object.[7] The smoothing was high and num cellsacross gap were set to 50. The meshing was fine. Therefore number modes were calculated as 12760 andnumber of elements was 12427. Mapped face meshing was used for the two faces. The relevance centerwas set to fine and the elements used were of quadrilateral shape.

2.3: Quality of Mesh:
While defining mesh in the body, we generally look at two mesh metrics

1. Skewness
2. Orthogonality

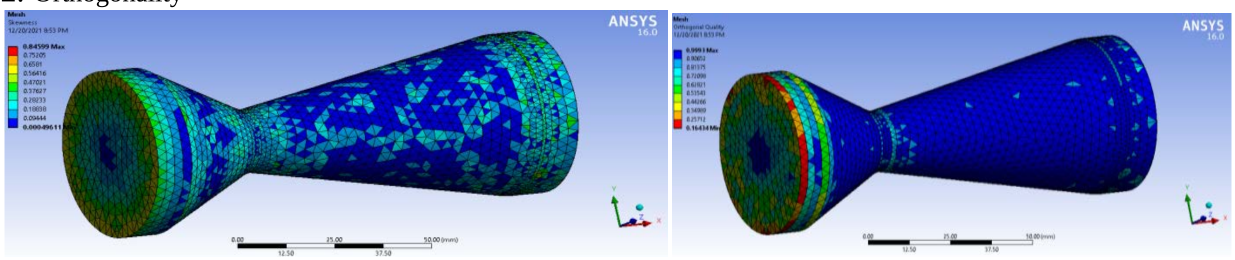


Fig_6:Tetrahedra Mesh Skewness Vrs Orthogonal quality

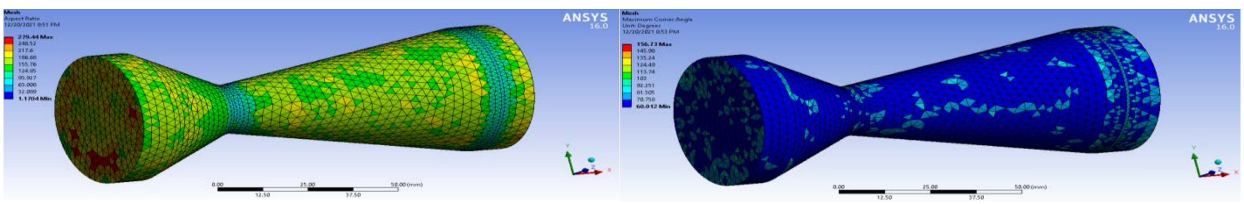


Fig_7: Tetrahedra MeshAspect Ratio Vrs Maximum Corner angle

TetrahedraMesh Report

Table 1: Mesh Information for CFX

Domain	Nodes	Elements	
Default Domain	25675	79268	

The fundamental law of fluid flow

I. conservation of mass or Continuity equation
II. Conservation of momentum
III. Conservation of energy equation

The governingequation Continuity equation,Conservation of momentum And energy equation can be expressed as follows.

$$\nabla.\left(\rho\overline{V}\right)=0 \quad (1)$$

y-momentum:

$$\nabla.\left(\rho v\overline{V}\right)=\frac{\partial p}{\partial y}+\frac{\partial\tau_{yy}}{\partial y}+\frac{\partial\tau_{zy}}{\partial z} \quad (2)$$

z-momentum:

$$\nabla.\left(\rho w\overline{V}\right)=\frac{\partial p}{\partial z}+\frac{\partial\tau_{yz}}{\partial y}+\frac{\partial\tau_{zz}}{\partial z} \quad (3)$$

2.4: Boundary conditions of Convergent Divergent Nozzle

Table 2: Boundary conditions

Boundary	Type
Inlet	Gauge Total Pressure 90[kPa], Temperature 288.15K
Outlet	Gauge Pressure
Wall	Wall function
Surface	Interior surface

Table 3 : Details and sizing inputs of a mesh

Physics Preference	CFD
Solver Preference	CFX
Sizing	
Relevance Center	Coarse
Smoothing	Medium
Transition	Slow
Span Angle Center	Fine
Curvature Normal Angle	Default (18.0 °)
Min Size	Default (7.1583e-002 mm)
Max Face Size	Default (7.15830 mm)
Max Size	Default (14.3170 mm)
Growth Rate	1.40
Minimum Edge Length	60.2520 mm
Inflation	
Inflation Option	Smooth Transition
Transition Ratio	0.77
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
Defeaturing	
Pinch Tolerance	Default (6.4424e-002 mm)
Defeaturing Tolerance	Default (3.5791e-002 mm)
Statistics	
Nodes	25675
Elements	79268

Table 4: Boundary conditions input for nozzle inlet

Flow Regime	Subsonic
Mass and momentum	Static Pressure as 95[kPa]
Flow Direction	Normal to boundary condition
Turbulence option	Low intensity (=1%)
Static temperature	288.15K

Table 5: Boundary condition inputs for nozzle outlet

Flow regime	Subsonic
Mass and Momentum	Static pressure 90[kPa]

Table 6: Boundary conditions inputs for nozzle wall/surface

Mass and Momentum	No Slip Wall
Wall Roughness	Smooth
Heat Transfer	Total Energy

Table 7: Solver control setting for solution

Flow Analysis	
Advection Scheme	High Resolution
Turbulence Numerics	High Resolution
Convergent control	
Min Iteration	1
Max Iteration	200
Convergent criteria	
Residual Type	RMS
Residual Target	1E-6

3. PRE PROCESSING

The next step of the CFD after meshing is pre processing. In pre processing appropriate boundryconditions are applied to the meshed model. The pre processing was done in Ansys Fluent.[8]

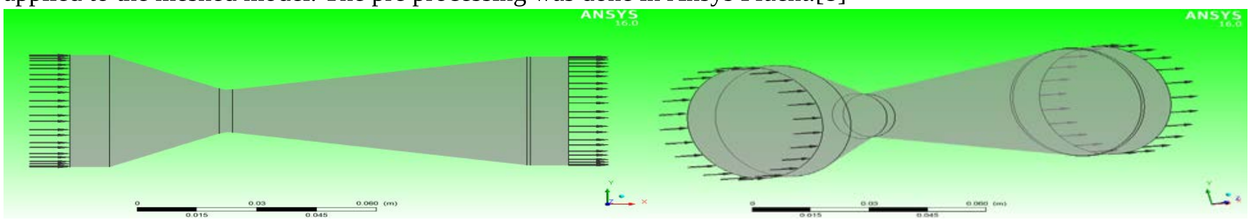


Fig_8: **ENERGY GENERATION MODE (A4; Fluid Flow(CFX)_CFX_ Pre Boundary(Inlet outlet &wall)**

The following graphs present the PreprocessingResults of mass, momuntum and Turbulence(KO) Fluid Flow(CFX) _CFX Solver with global Run setting, Run Mode[Platform MPI Local Parallel results below.

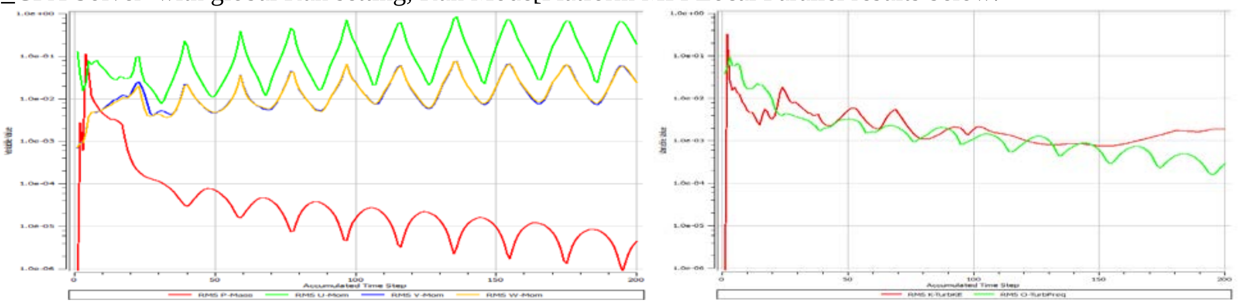


Fig_9:Mass and Momuntum vrs Turbulence(KO)

Table 8: Pre Processing Fluid Flow(CFX) _CFX Solver results

Global Length	5.1654E-02
Minimum Extent	5.0000E-02
Maximum Extent	1.2500E-01
Density	5.7942E-01
Dynamic Viscosity	1.8310E-05
Velocity	1.1816E+00
Advection Time	4.3716E-02
Reynolds Number	1.9314E+03
Speed of Sound	3.4181E+02
Mach Number	3.4569E-03
Thermal Conductivity	2.6100E-02

Specific Heat Capacity at Constant Pressure	1.0044E+03
Specific Heat Capacity at Constant Volume	7.1730E+02
Specific Heat Ratio	1.4003E+00
Prandtl Number	7.0462E-01
Temperature Range	7.4304E+00
Minimum Extent	5.0000E-02
Maximum Extent	1.2500E-01

4. Post Processing/Result

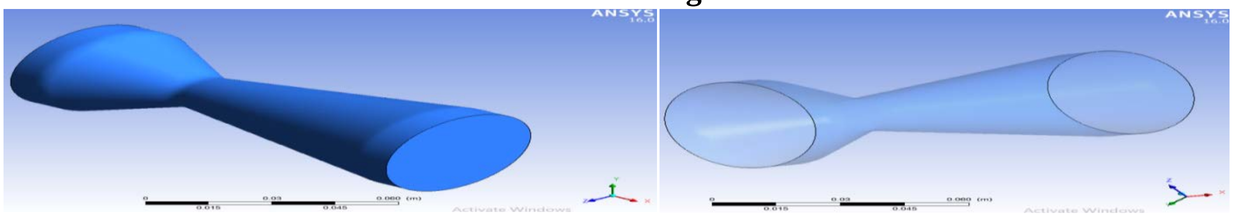


Fig_10: CFX_Post Processing wall Vrs Render Transparency 0.8(Smooth shading)

4.1. Velocity Variation

Velocity goes on increasing from left to right. It is minimum at the inlet section and maximum at the outlet section. Velocity is equal to sonic velocity at the throat. Minimum and maximum velocity detailed.

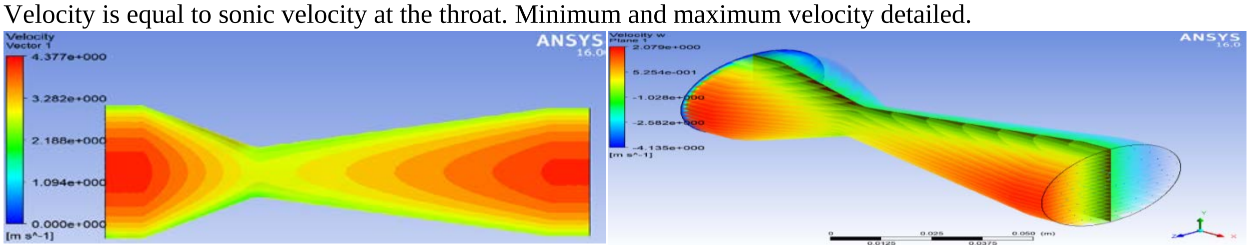


Fig_11: Velocity Variation

4.2. Pressure Variation

The total Pressure gradient in streamline is maximum at the Throat section and minimum at the inlet section gradually increase outlet sectionwhichrevealed below.[9]

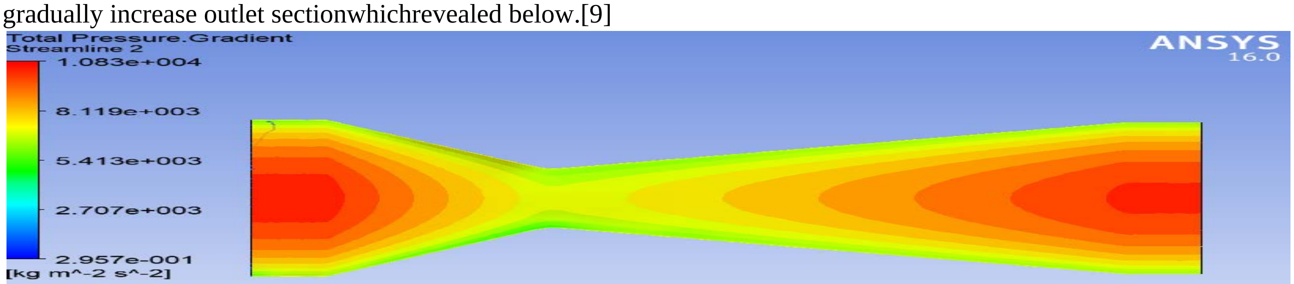


Fig_12: Pressure gradient in streamline

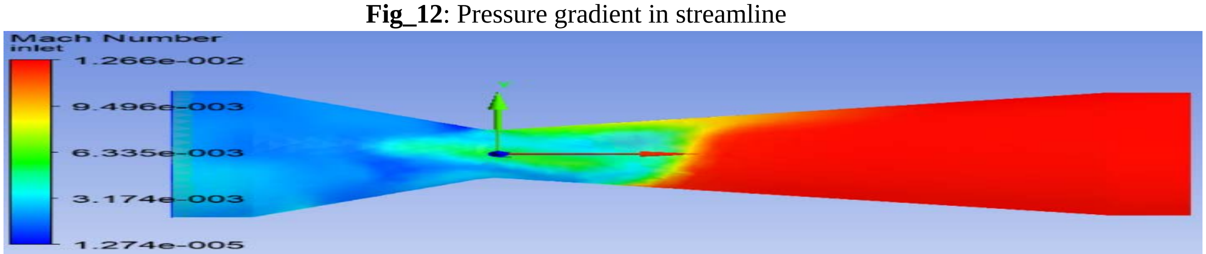


Fig_14: Mach number

The mach number is in the range of 1.266-1.274, I.e , the mach number is subsonic at the inlet and exit portion, at the throat portion, the flow becomes supersonic.[10]

4.3. Temperature Variation

Temperature variation is similar to pressure variation. It is maximum at the inlet and minimum at the outlet. Maximum value of temperature is 294.90K and the minimum value of temperature 288.15K.

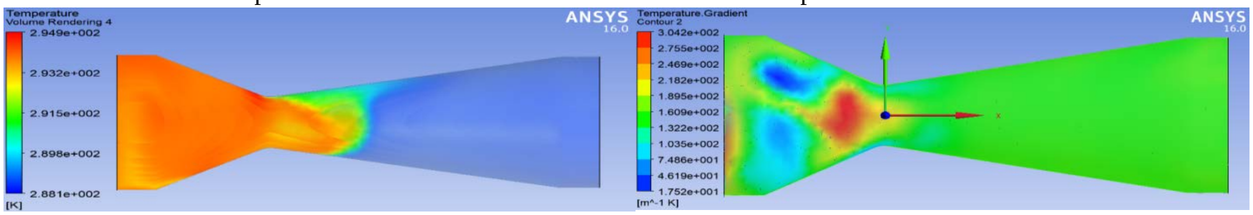


Fig_13: Temperature volume rendering Vrs Temperature Gradient contour

5.SOLVER

The next step is solver. In solver the solution is initialized and calculation is proceeded with the desired number of iterations. it is the most important port of CFD analysis.[11] Using ANSYS -FLUENT (CFX) it is possible tosolve the governing equation related to flow physical properties.

Table 9:**Solver Details**

Solver Details	
Solution control	Courant number = 5
Solution Initialization	Compute form = inlet
Run Calculation	Number of Iteration = 200

The solution was terminated after 164 iterations

6.ANALYSIS

The present study based on the computational fluid dynamics (CFD) method which is applicable[11] for the solution of different types of fluid flow problems. The fluid flows performances the internal flow of the fluid model and the external flow of the solid model. The fluid flow problem simulates with appropriate boundary conditions by using fundamental equations for the solutions. CFD method is applicable for different types of solutions such physics, science, mathematical and engineering problems. The fundamental equations of fluid flow analysis are the continuity equation, momentum equation, and energy equation.[12].These equations can be expressed as Eq. (1)-(3), respectively.

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \qquad (1)$$

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_i} = \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i}\tau_{ij} \qquad (2)$$

$$\frac{\partial}{\partial t}[\partial(\mathrm{e} + \frac{\mathrm{V}^2}{2})] + \frac{\partial}{\partial x_i}[\rho \mathrm{u_j}(\mathrm{e} + \frac{\mathrm{V}^2}{2}) + \mathrm{p} + q_j - u_i u_j] = 0 \qquad (3)$$

Table10. Domain Physics for CFX

Domain - Default Domain	
Type	Fluid
Materials	
Air Ideal Gas	
Settings	
Reference Pressure	0.0000e+00 [atm]
Heat Transfer Model	Total Energy
Include Viscous Work Term	True
Turbulence Model	SST

Domain	Boundaries	
Default Domain	**Boundary – inlet**	
	Location	Inlet
	Settings	
	Flow Direction	Normal to Boundary Condition
	Flow Regime	Subsonic
	Heat Transfer	Total Temperature
	Total Temperature	2.8815e+02 [K]
	Mass And Momentum	Total Pressure
	Relative Pressure	1.0133e+02 [Pa]
	Turbulence	Medium Intensity and Eddy Viscosity Ratio
	Boundary – outlet	
	Type	OUTLET
	Location	Outlet
	Settings	
	Flow Regime	Subsonic
	Mass And Momentum	Average Static Pressure
	Pressure Profile Blend	5.0000e-02
	Relative Pressure	BackPressure
	Boundary – wall	
	Location	Wall
	Settings	
	Heat Transfer	Adiabatic
	Mass And Momentum	No Slip Wall
	Wall Roughness	Smooth Wall

Table 11. Boundary Flows for CFX_(Solution Report)

	Type	Mass Flow	Momentum		
			X	Y	Z
inlet	Boundary	0.0000e+00	9.3838e+01	0.0000e+00	0.0000e+00
outlet	Boundary	0.0000e+00	-8.9710e+01	0.0000e+00	0.0000e+00
wall	Boundary	0.0000e+00	-4.6073e+00	-2.9542e-03	4.1557e-04

CONCLUSIONS

A nozzle is a device crafted by engineers to controlthe characteristics of the fluid. It is mostly used toincrease the velocity of the fluid which normallyconsists of convergent portion, Throat section andDivergent section.The main contribution of this paper was to demonstrate a 3-D Numerical investigation and parametric_CFD_Analysis of flow through Convergent-Divergent Nozzle using Ansys_CFx, with design software, ANSYS CFD to generate the high quality of Mesh (Tetrahedra) and CFx as the CFD solver. Besides, the vector plot, contour plot, streamline plot and volume rendering are generated for the better understanding of flow through CD nozzles. Finally, the air is found to performbetter in low divergent angle C_D nozzles are found simulation is done using Ansys_CFx under steady phenomenon.

Reference:

[1] Mohammed Faheem, Mohammed Kareemullah, Abdul Aabid, Imran Mokashi, and Sher Afghan Khan. "Experimenton of Nozzle Flow with Sudden Expansion at Mach 1.1."International Journal of Recent Technology and Engineering8, no. 2S8 (2019): 1769–1775.

[2] Dr.A.Paulmakesh, Gizachew Markos Makebo September2021, Failure Modes Of Cold Formed Steel Angle Sections," Journal of University of Shanghai for Science and Technology", Volume 23, Issue 9, - 2021 Page-604-613.

[3] Sher Afghan Khan, Abdul Aabid, Imran Mokashi, and Zaheer Ahmed. "Effect of Micro Jet Control on the Flow Filedof the Duct at Mach 1. 5."International Journal of Recent Technology and Engineering (IJRTE) 8, no. 2S8 (2019):1758–1762.

[4] A. Paulmakesh and Gizachew Markos Makebo, 2021, Numerical Analysis of Cold rolled structural steel angle sections subjected to tension members,"Journal of Physics: Conference Series, Volume 2040, 012025, International Conference on Physics and Energy 2021 (ICPAE 2021), Pg no 1-13

[5] Sher Afghan Khan, Zaheer Ahmed, Abdul Aabid, and Imran Mokashi. "Experimental Research on Flow Developmentand Control Effectiveness in the Duct at High Speed."International Journal of Recent Technology and Engineering(IJRTE) 8, no. 2S8 (2019): 1763–1768.

[6] Akhtar, Mohammad Nishat, Elmi Abu Bakar, Abdul Aabid, and Sher Afghan Khan. "Effects of Micro Jets on the FlowField of the Duct with Sudden Expansion."International Journal of Innovative Technology and Exploring Engineering(IJITEE) 8, no. 9S2 (2019): 636–640.

[7] Paul Makesh A , Arivalagan S, June 2017, Experimental Analysis Of Coldformed Steel Members Subjected To Tension Load, "International Journal of Civil Engineering and Technology" Volume 8, Issue 6, June 2017, Pages 621-629.

[8] Akhtar, Mohammad Nishat, Elmi Abu Bakar, Abdul Aabid, and Sher Afghan Khan. "Numerical Simulations of a CDNozzle and the Influence of the Duct Length."International Journal of Innovative Technology and ExploringEngineering (IJITEE) 8, no. 9S2 (2019): 622–630.

[9] Akhtar, Mohammad Nishat, Elmi Abu Bakar, Abdul Aabid, and Sher Afghan Kha. "Control of CD Nozzle Flow UsingMicrojets at Mach 2.1."International Journal of Innovative Technology and Exploring Engineering (IJITEE) 8, no. 9S2(2019): 631–635.

[10] Dr.A.Paulmakesh, **,** Gizachew Markos Makebo September**,**2021**,** Analysis Of Light Gauge Steel By Using Fem Subjected To Tension Load," Journal of University of Shanghai for Science and Technology", Volume 23, Issue 9, - 2021 Page-590-603

[11] Sher Afghan Khan, Imran Mokashi, Abdul Aabid, and Mohammed Faheem. "Experimental Research on WallPressure Distribution in C-D Nozzle at Mach Number 1.1 for Area Ratio 3.24."International Journal of RecentTechnology and Engineering 8, no. 2S3 (2019): 971–975.

[12] Azami, Muhammed Hanafi, Mohammed Faheem, Abdul Aabid, Imran Mokashi, and Sher Afghan Khan."Experimental Research of Wall Pressure Distribution and Effect of Micro Jet at Mach."International Journal ofRecent Technology and Engineering 8, no. 2S3 (2019): 1000–1003.

[13] Azami, Muhammed Hanafi, Mohammed Faheem, Abdul Aabid, Imran Mokashi, and Sher Afghan Khan. "Inspectionof Supersonic Flows in a CD Nozzle Using Experimental Method."International Journal of Recent Technology andEngineering 8, no. 2S3 (2019): 996–999.

[14] Sher Afghan Khan, Abdul Aabid, and Zakir Ilahi Chaudhary. "Influence of Control Mechanism on the Flow Field ofDuct at Mach 1.2 for Area Ratio 2.56."International Journal of Innovative Technology and Exploring Engineering 8,no. 6S4 (2019): 1135–1138.