FLOW ANALYSIS OF CONVERGENT AND DIVERGENT NOZZLE USING COMPUTATIONAL FLUID DYNAMICS (CFD)

A project report submitted in partial fulfilment of the requirement for

the award of the degree of

BACHELOR OF TECHNOLOGY

IN

MECHANICAL ENGINEERING

BY

BALAGAM RAMBABU (317126520120)

VADAPALLI PRAVALIKA RANI (318126520L30)

IBRAHIM BASHA (318126520L35)

MOHAMMED ABDUL KHADIRZILANI (317126520153)

MUDDAM SETTY S V BHARGAV KRISHNA SAI (317126520154)

Under the esteemed guidance of

Dr. B.NAGARAJU,

B.E.,M.Tech.,M.E.,Ph.D.,MISTE,MIE,MSAE.

HOD & Professor

DEPARTMENT OF MECHANICAL ENGINEERING

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES (A)

(Affiliated to Andhra University, Accredited By NBA and NAAC with 'A' Grade)

SANGIVALASA, VISAKHAPATNAM (District) – 531162

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES (A)

(Affiliated to Andhra University, Approved by AICTE, Accredited by NBA & NAAC with A grade)
SANGIVALASA, VISAKHAPATNAM (District) – 531162

CERTIFICATE

This is to certify that the Project Report entitled "**ANALYSIS OF CONVERGENT AND DIVERGENT NOZZLE USING COMPUTATIONAL FLUID DYNAMICS (CFD)**" being submitted by BALAGAM RAMBABU (317126520120), VADAPALLI PRAVALIKA RANI (318126520L30), IBRAHIM BASHA (318126520L35), MOHAMMED ABDUL KHADIRZILANI (317126520153), MUDDAM SETTY S V BHARGAV KRISHNA SAI (317126520154) in partial fulfillments for the award of degree of **BACHELOR OF TECHNOLOGY** in **MECHANICAL ENGINEERING, ANITS**. It is the work of bona-fide, carried out under the guidance and supervision of **DR.B.NAGARAJU**, Professor, Department Of Mechanical Engineering, ANITS during the academic year of 2017-2021.

Approved By

PROJECT GUIDE

HEAD OF THE DEPARTMENT

(DR.B.NAGARAJU)
Professor
Mechanical Engineering Department
ANITS, Visakhapatnam.

(Dr. B. Naga Raju)
Head of the Department
Mechanical Engineering Department
ANITS, Visakhapatnam.

ACKNOWLEDGEMENTS

FLOW ANALYSIS OF CONVERGENT AND DIVERGENT NOZZLE USING COMPUTATIONAL FLUID DYNAMICS (CFD)

ABSTRACT

A nozzle is used to give the direction to the gases coming out of the combustion chamber. Nozzle is a tube with variable cross-sectional area. Nozzles are generally used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the exhaust stream that emerges from them. The nozzle is used to convert the chemical-thermal energy generated in the combustion chamber into kinetic energy. The nozzle converts the low velocity, high pressure, high temperature gas in the combustion chamber into high velocity gas of lower pressure and low temperature.

Our study is carried out on Convergent and Divergent nozzle for analysis of flow using computational fluid dynamics (CFD). Numerical study is to be conducted to understand the air flow in a Convergent-Divergent Nozzle at different divergence and convergence angles using three-dimensional models. The present study is aimed at examining the supersonic flow in C-D nozzle at various divergence and convergence angles. The flow is simulated using Ansys fluent. The result shows the variation in the pressure and velocity.

Keywords: CFD, C-D nozzle, Pressure, Velocity

CONTENTS

LIST OF TABLES

LIST OF FIGURES

Chapter-I

INTRODUCTION

1.1 NOZZLE

A nozzle is a device designed to control the direction or characteristics of a fluid flow (specially to increase velocity) as it exits (or enters) an enclosed chamber or pipe. A nozzle is often a pipe or tube of varying cross-sectional area, and it can be used to direct or modify the flow of a fluid (liquid or gas). Nozzles are frequently used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that emerges from them. In a nozzle, the velocity of fluid increases at the expense of its pressure energy. Converging nozzles, are tubes with an area that decreases from the nozzle entry to the exit (or throat) of the nozzle. As the nozzle area decreases, the flow velocity increases, with the maximum flow velocity occurring at the nozzle throat. A nozzle whose cross section becomes larger in the direction of flow are called Divergent nozzle. The figure 1.1 shows the convergent and divergent nozzle.

A C-D nozzle includes three main elements: a converging section, a throat, and a diverging section. The combustion exhaust gas first enters the converging section. The gas moves at subsonic speeds through this area, accelerating as the crosssectional area decreases. In order to reach supersonic speeds, the gas must first pass through an area of minimum crosssectional area called the throat. From here, the supersonic gas expands through the converging section and then out of the nozzle. Supersonic flow accelerates as it expands. The following are the features of nozzle,

- Nozzle produces thrust.
- Convert thermal energy of hot chamber gases into kinetic energy and direct that energy along nozzle axis
- Exhaust gases from combustion are pushed into throat region of nozzle.
- Throat is smaller cross-sectional area than rest of engine; here gases are compressed to high pressure.
- Nozzle gradually increases in cross-sectional area allowing gases to expand and push against walls creating thrust.
- Mathematically, ultimate purpose of nozzle is to expand gases as efficiently as possible so as to maximize exit velocity.
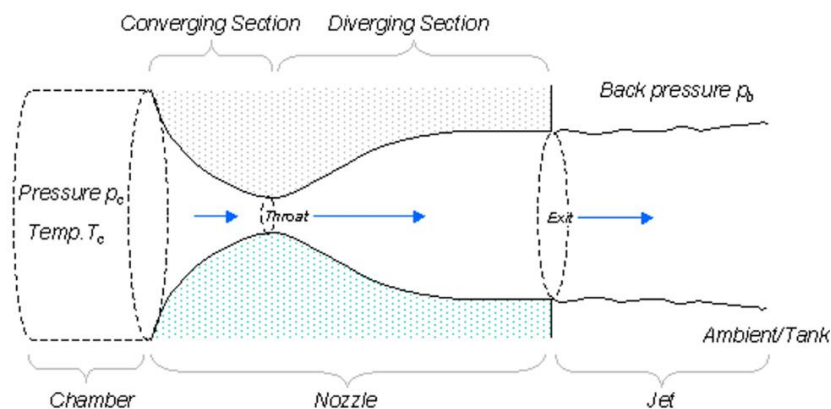

Fig 1.1 C-D nozzle

1.2 TYPES OF NOZZLES

1.2.1 High velocity Nozzle

Frequently, the goal of a nozzle is to increase the kinetic energy of the flowing medium at the expense of its pressure and internal energy Nozzles can be described as convergent (narrowing down from a wide diameter to a smaller diameter in the direction of the flow) or divergent (expanding from a smaller diameter to a larger one). A de Laval nozzle has a convergent section followed by a divergent section and is often called a convergent-divergent (CD) Nozzle ("con-di nozzle"). Convergent nozzles accelerate subsonic fluids. If the nozzle pressure ratio is high enough, then the flow will reach sonic velocity at the narrowest point (i.e., the nozzle throat). In this situation, the nozzle is said to be choked. The figure 1.2 shows the high velocity nozzle.
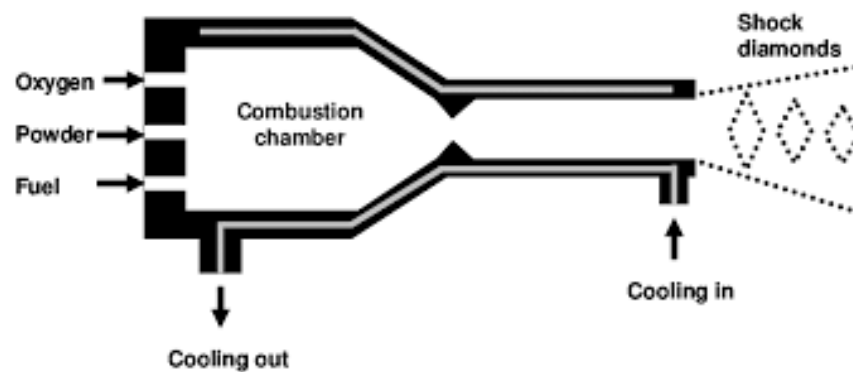
Fig 1.2 High velocity nozzle

Increasing the nozzle pressure ratio further will not increase the throat Mach number above one. Downstream (i.e., external to the nozzle) the flow is free to expand to supersonic velocities; however, Mach 1 can be a very high speed for a hot gas because the speed of sound varies as the square root of absolute temperature. This fact is used extensively in rocketry where hypersonic flows are required and where propellant mixtures are deliberately chosen to further increase the sonic speed. Divergent nozzles slow fluids if the flow is subsonic, but they accelerate sonic or supersonic fluids. Convergent-divergent nozzles can therefore accelerate fluids that have choked in the convergent section to supersonic speeds. This CD process is more efficient than allowing a convergent nozzle to expand supersonically externally. The shape of the divergent section also ensures that the direction of the escaping gases is directly backwards, as any sideways component would not contribute to thrust.

1.2.2 Propelling Nozzle

A jet exhaust produces a net thrust from the energy obtained from combusting fuel which is added to the inducted air. This hot air passes through a high-speed nozzle, a propelling nozzle, which enormously increases its kinetic energy. Increasing exhaust velocity increases thrust for a given mass flow, but matching the exhaust velocity to the air speed provides the best energy efficiency. However, momentum considerations prevent jet aircraft from maintaining velocity while exceeding their exhaust jet speed. The engines of supersonic jet aircraft, such as those of fighters and SST aircraft (e.g., Concorde) almost always achieve the high exhaust speeds necessary for supersonic flight by using a CD nozzle despite weight and cost penalties; conversely, subsonic jet engines employ relatively low, subsonic, exhaust velocities and therefore employ simple convergent nozzle, or even bypass nozzles at even

lower speeds.Rocket motors maximize thrust and exhaust velocity by using convergent-divergent nozzles with very large area ratios and therefore extremely high-pressure ratios. Mass flow is at a premium because all the propulsive mass is carried with vehicle, and very high exhaust speeds are desirable. The figure 1.3 shows the propelling nozzle.
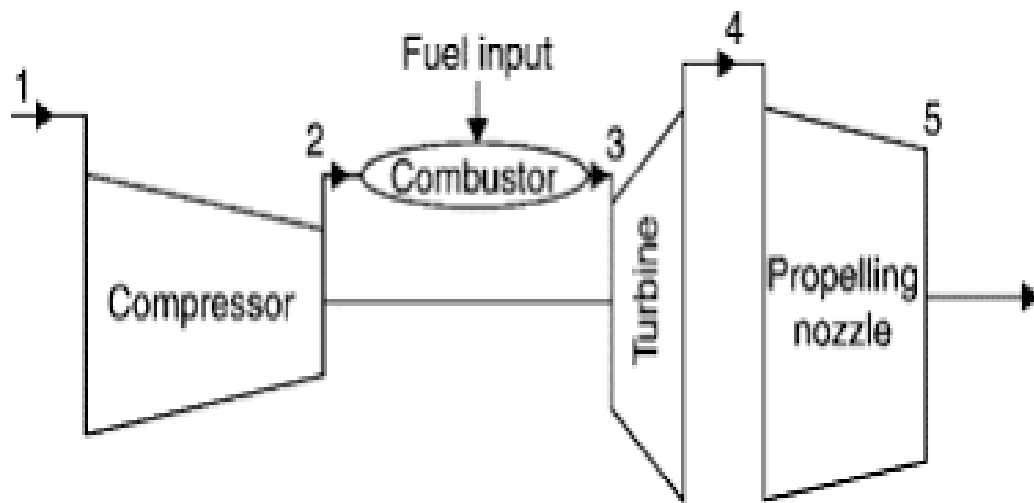
Fig 1.3 Propelling nozzle

1.2.3 Magnetic Nozzle

Magnetic poles have also been proposed for some types of propulsion, such as vasimr, in which the flow of plasma is directed by magnetic fields instead of walls made of solid matter. The figure 1.4 shows the magnetic nozzle.
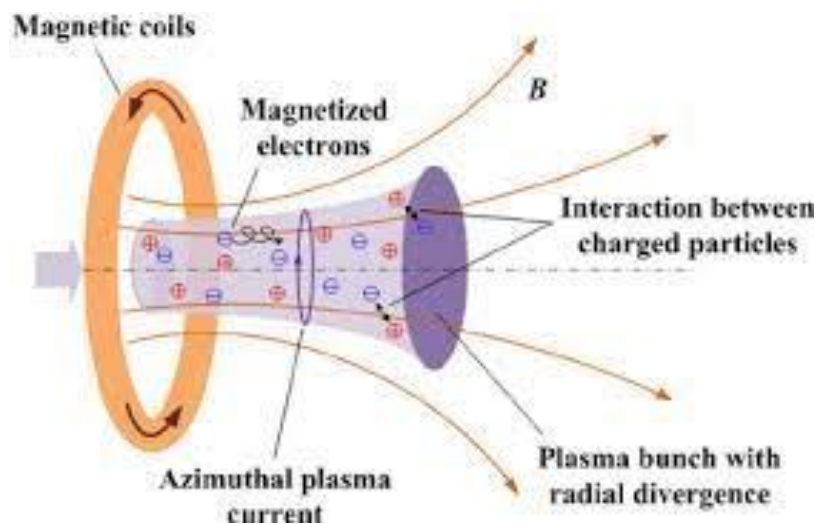
Fig 1.4 Magnetic nozzle

1.2.4 Spray Nozzle

Many nozzles produce a very fine spray of liquids. Atomizer nozzles are used for spray painting, perfumes, carburetors for internal internal combustion engines, spray on deodorants, antiperspirants and many other similar uses. The figure 1.5 shows the spray nozzle.

Air-Aspirating Nozzle uses an opening in the cone shaped nozzle to inject air into a stream of water-based foam (CAFS/AFFF/FFFP) to make the concentrate "foam up". Most commonly found on foam extinguishers and foam handlines.

- **Swirl nozzles** inject the liquid in tangentially, and it spirals into the center and then exits through the central hole. Due to the vertexing this causes the spray to come out in a cone shape.
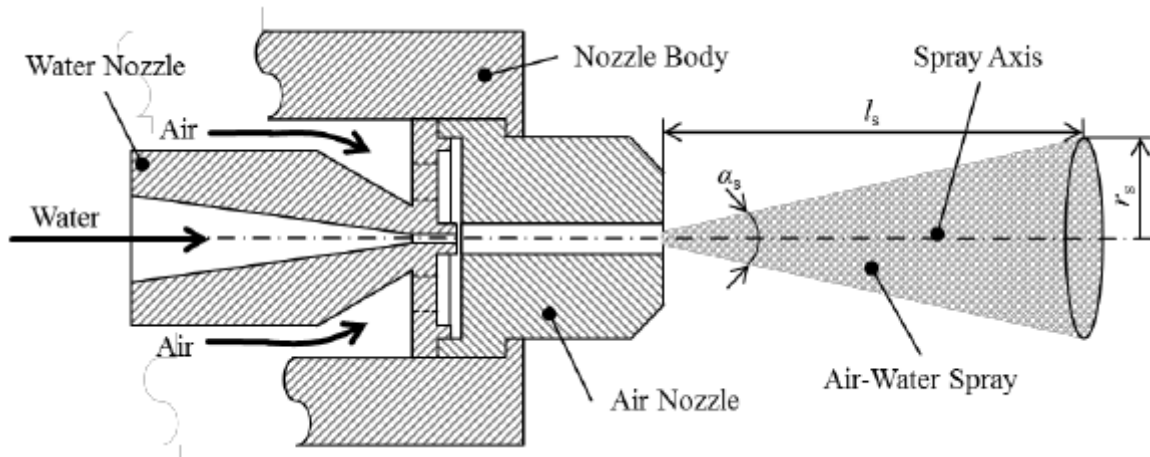
Fig 1.5 Spray nozzle

1.2.5 Vacuum Nozzle

Vacuum cleaner nozzles come in several different shapes. Vacuum Nozzles are used in vacuum cleaners. The figure 1.6 shows the vacuum nozzle.
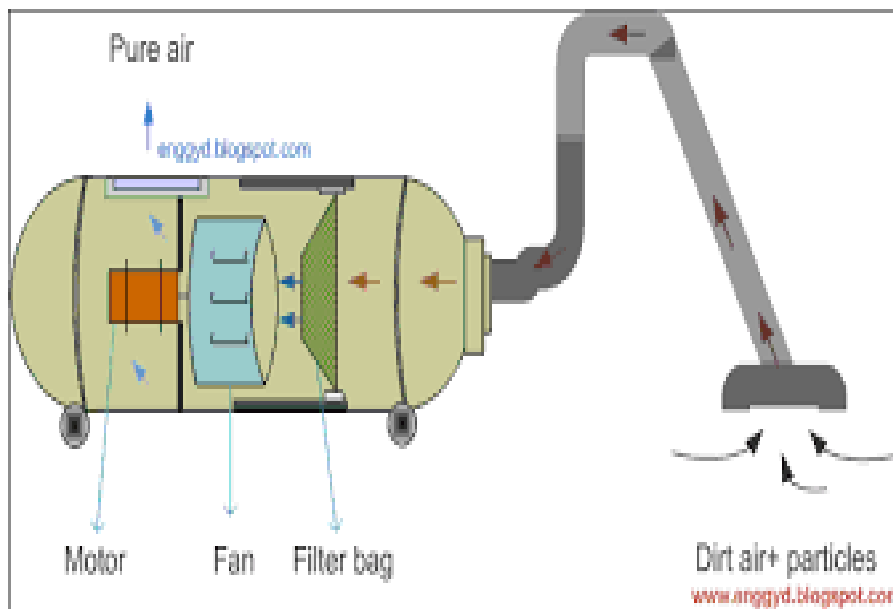
Fig 1.6 Vacuum nozzle

1.2.6 Shaping Nozzle

Some nozzles are shaped to produce a stream that is of a particular shape. For example, extrusion modelling is a way of producing lengths of metals or plastics or other materials with a particular cross-section. This nozzle is typically referred to as a die

1.2.7 Plug Nozzle

The plug nozzle is a type of nozzle which includes a center body or plug around which the working fluid flows. Plug nozzles have applications in aircraft, rockets, and numerous other fluid flow devices. The figure 1.7 shows the plug nozzle. The shaping of the plug and the nozzle opening allows the angle of the ring to be adjusted. Normally this is shaped so that when the plug is pulled back toward the opening it both partially cuts off the water flow, as well as causing it to spread out to the widest possible angle. This can be used for "misting" plants. When the trigger is pushed down further, the plug moves away from the opening, causing less blockage and disruption of the flow, ultimately allowing the water to form back into a stream.
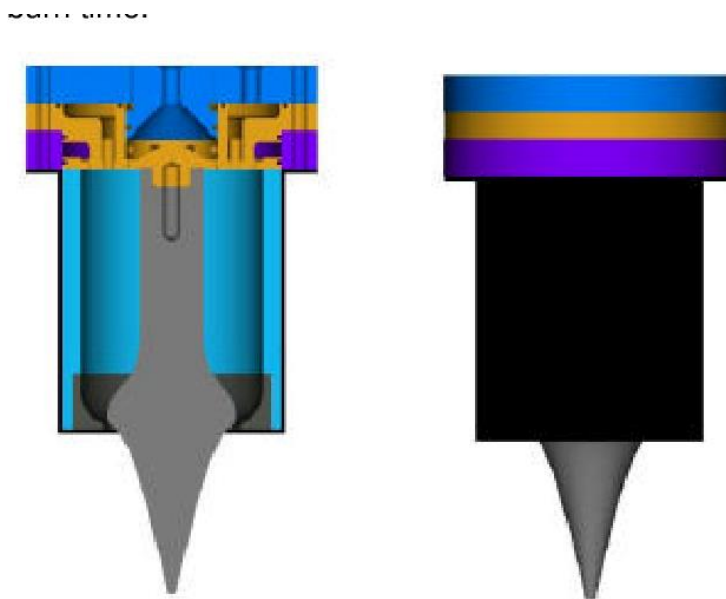
Fig 1.7 Plug nozzle

1.2.8 CONICAL NOZZLE

conical nozzle was often used in early rocket applications because of its simplicity and ease of construction. The cone gets its name from the fact that the walls diverge at a constant angle. A small angle produces greater thrust, because it maximizes the axial component of exit velocity, but minimizing the angle makes the nozzle longer and heavier, which leads to complexities while building them. At the other extreme, size and weight are minimized by a large nozzle wall angle. Unfortunately, large angles reduce performance at low altitude, because the high ambient pressure causes overexpansion and flow separation. Here, the exit flow can be achieved with some angularity, which reduces the axial thrust of the vehicle. The figure 1.8 shows the conical nozzle.

Fig 1.8 Conical nozzle

1.3 ANSYS:

ANSYS is a general purpose finite element modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

In general, a finite element solution may be broken into the following three stages. This is a general guideline that can be used for setting up any finite element analysis.

1. **Preprocessing**: defining the problem

   the major steps in preprocessing are given below:

   - Define keypoints/lines/areas/volumes
   - Define element type and material/geometric properties
   - Mesh lines/areas/volumes as required

   The amount of detail required will depend on the dimensionality of the analysis (i.e. 1D, 2D, axi-symmetric, 3D).

   Solution: assigning loads, constraints and solving; here we specify the loads (point or pressure), contraints (translational and rotational) and finally solve the resulting set of equations.

2. **Postprocessing:**
   further processing and viewing of the results; in this stage one may wish to see:
   - Lists of nodal displacements
   - Element forces and moments
   - Deflection plots
   - Stress contour diagrams

   ANSYS uses certain inputs and evaluates the product behavior to the physics that you are testing it in. It is a general purpose software used to simulate the interactions between various physics like dynamics, statics, fluids, electromagnetic, thermal, and vibrations. ANSYS

typically creates the user an opportunity to create a virtual environment to simulate the tests or working conditions of the products before manufacturing the prototypes. This would certainly reduce the cost of producing prototypes and mainly the time. In this competitive world the accuracy and time are the most deciding factors for the company or the organization to sustain. ANSYS helps in increasing the accuracy and decreasing the time of outcome of the final product.

The Ansys finite element solvers enable a breadth and depth of capabilities unmatched by anyone in the world of computer-aided simulation. Thermal, Structural, Acoustic, Piezoelectric, Electrostatic and Circuit Coupled Electromagnetics are just an example of what can be simulated. Regardless of the type of simulation, each model is represented by a powerful scripting language . The Ansys Parametric Design Language (APDL). APDL is the foundation for all sophisticated features, many of which are not exposed in the Workbench Mechanical user interface. It also offers many conveniences such as parameterization, macros, branching and looping, and complex math operations. All these benefits are accessible within the Ansys Mechanical APDL user interface.

The ANSYS Workbench platform automatically forms a connection to share the geometry for both the fluid and structural analysis, minimizing data storage and making it easy to study the effects of geometry changes on both analyses. In addition, a connection is formed to automatically transfer pressure loads from the fluid analysis to the structural analysis. Figure 1.9 shows the steps and levels of ansys workbench.
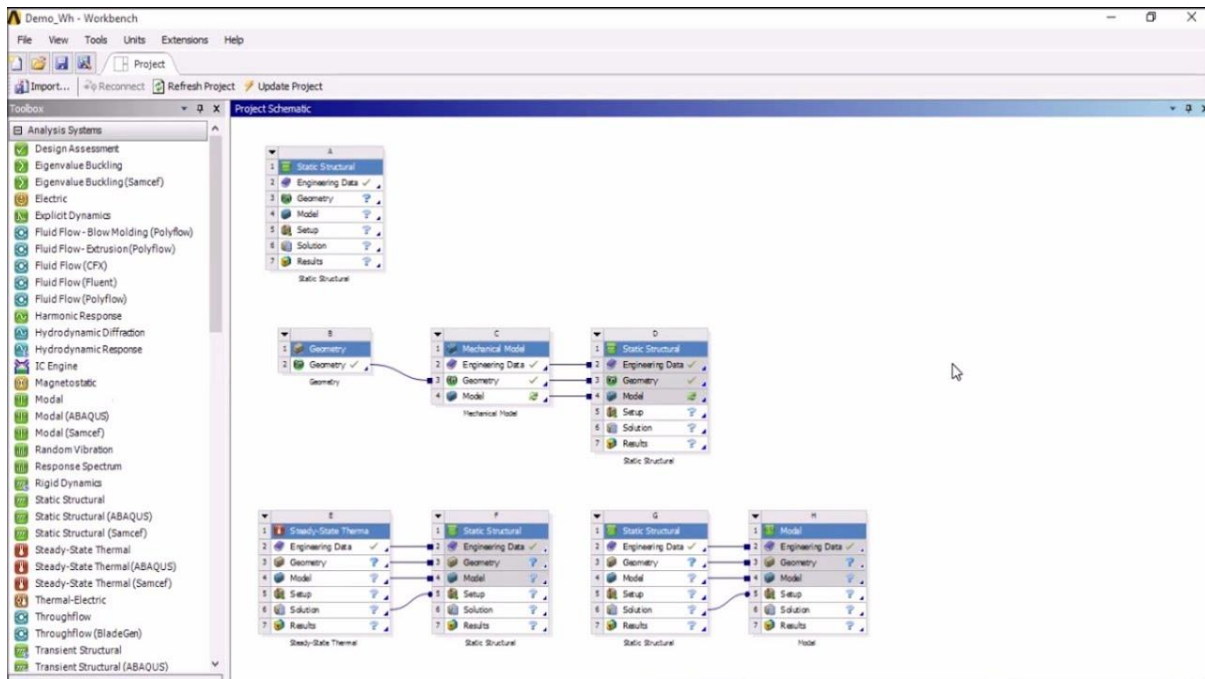

Fig 1.9: Steps and levels of anysys workbench

1.3.1 ANSYS Workbench Features:

Bidirectional, parametric links with all major CAD systems.

- Integrated, analysis-focused geometry modelling, repair, and simplification via ANSYS Design Modeler.
- Highly automated, physics-aware meshing.
- Automatic contact detection.
- Unequalled depth of capabilities within individual physics disciplines.
- Unparalleled breadth of simulation technologies.
- Complete analysis systems that guide the user start-to-finish through an analysis.
- Comprehensive multi physics simulation with drag-and-drop ease of use.
- Flexible components enable tools to be deployed to best suit engineering intent.
- Innovative project schematic view allows engineering intent, data relationships, and the state of the project to be comprehended at a glance.
- Complex project schematics can be saved for re-use.
- Pervasive, project-level parameter management across all physics.
- Automated what-if analyses with integrated design point capability. 44
- Adaptive architecture with scripting and journaling capabilities and API's enabling rapid integration of new and third-party solutions.

ANSYS provides an interface to the LS-DYNA explicit finite element program and is used to calculate fast solutions for large deformation dynamics and complex contact problems.

- In addition to the above analysis types, several special-purpose features are available
- Fracture mechanics
- Composites
- Fatigue
- p-Method

1.3.2 Advantages:

- ANSYS can import all kinds of CAD geometries (3D and 2D) from different CAD software's and perform simulations, and also it has the capability of creating one effortlessly. ANSYS has inbuilt CAD developing software's like Design Modeler and Space Claim which makes the work flow even smoother.
- ANSYS has the capability of performing advanced engineering simulations accurately and realistic in nature by its variety of contact algorithms, time dependent simulations and non linear material models.
- ANSYS has the capability of integrating various physics into one platform and perform the analysis. Just like integrating a thermal analysis with structural and integrating fluid flow analysis with thermal and structural, etc.,
- ANSYS now has featured its development into a product called ANSYS AIM, which is capable of performing multi physics simulation. It is a single platform which can integrate all kinds of physics and perform simulations.
- ANSYS has its own customization tool called ACT which uses python as a background scripting language and used in creating customized user required features in it.

- ANSYS has the capability to optimize various features like the geometrical design, boundary conditions and analyse the behavior of the product under various criterion's

1.4 COMPUTATIONAL FLUID DYNAMICS(CFD)

Fluid (gas and liquid) flows are governed by partial differential equations which represent conservation laws for the mass, momentum, and energy. Computational Fluid Dynamics is the art of replacing such PDE systems by a set of algebraic equations which can be solved using digital computers. Computational Fluid Dynamics provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of mathematical modeling (partial differential equations) numerical methods (discretization and solution techniques) software tools (solvers, pre- and post-processing utilities) CFD enables scientists and engineers to perform 'numerical experiments' in a 'virtual flow laboratory'. CFD gives an insight into flow patterns that are difficult, expensive or impossible to study using traditional (experimental) techniques. The figure 1.9 shows the CFD analysis of a C-D nozzle.
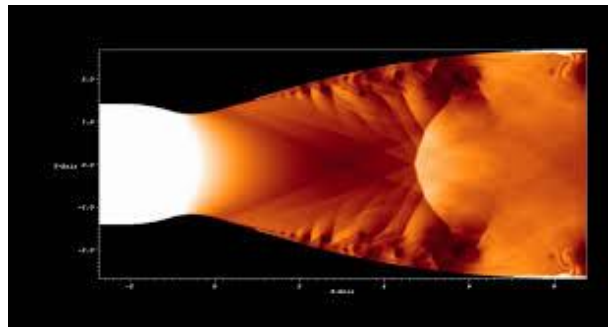
Fig 1.10 CFD on a nozzle

1.4.1 CFD analysis process

1. Problem statement   information about the flow

2. Mathematical model   IBVP = PDE + IC + BC

3. Mesh generation   nodes/cells, time instants

4. Space discretization   coupled ODE/DAE systems

5. Time discretization   algebraic system Ax = b

6. Iterative solver   discrete function values

7. CFD software   implementation, debugging

8. Simulation   runs parameters, stopping criteria

9. Post processing   visualization, analysis of data

10. Verification model   validation / adjustment

1.4.2 Mathematical model

- Choose a suitable flow model (viewpoint) and reference frame.

- Identify the forces which cause and influence the fluid motion.

- Define the computational domain in which to solve the problem.

- Formulate conservation laws for the mass, momentum, and energy.

- Simplify the governing equations to reduce the computational effort:

- Add constitutive relations and specify initial/boundary conditions

1.4.4 Discretization process

The PDE system is transformed into a set of algebraic equations

1. Mesh generation
   - Structured or unstructured, triangular or quadrilateral?
   - CAD tools + grid generators (Delaunay, advancing front)
   - mesh size, adaptive refinement in 'interesting' flow regions
2. Space discretization
   - Finite differences/volumes/elements
   - High- vs. low-order approximations
3. Time discretization
   - Explicit vs. implicit schemes, stability constraints
   - Local time-stepping, adaptive time step control

1.4.5 Iterative solution strategy

The coupled nonlinear algebraic equations must be solved iteratively. The coefficients of the discrete problem are updated using the solution values from the previous iteration so as to get rid of the nonlinearities by a Newton-like method and solve the governing equations in a segregated fashion. The resulting sequence of linear sub-problems is typically solved by an iterative method because direct solvers are prohibitively expensive. It is necessary to check the residuals, relative solution changes and other indicators to make sure that the iterations converge.

1.4.6 CFD simulations

The computing times for a flow simulation depend on the choice of numerical algorithms and data structures, linear algebra tools, stopping criteria for iterative solvers, discretization parameters, cost per time step and convergence rates for outer iterations, programming language and many other things. The quality of simulation results depends on the mathematical model and underlying assumptions, approximation type, stability of the numerical scheme, Mesh, time step, error indicators, stopping criteria etc.

1.4.7 Verification of CFD codes

Verification amounts to looking for errors in the implementation of the models by examining the computer programming by visually checking the source code, documenting it and testing the underlying subprograms individually. Examining iterative convergence by monitoring the residuals, relative changes of integral quantities and checking if the prescribed tolerance is attained. Examining grid convergence: as the mesh and/or and the time step are refined, the spatial and temporal discretization errors, respectively, should asymptotically approach zero. At last by comparing the computational results with analytical and numerical solutions for standard benchmark configurations.

Chapter-2

Literature review

This chapter provides the information about the works carried out by different authors on flow analysis on various nozzles using CFD.

BijuKuttan P [1] Conducted numerical analysis to determine an optimum divergent angle for the nozzle which would give the maximum outlet velocity and meet the thrust requirements. The inlet dimensions and the boundary conditions are kept constant and the divergent angles are varied in order to understand how the variation in divergent angle affects the flow pattern through the nozzle. Among the various models available in Fluent, the k-ε model was selected for their work. A two-dimensional axi-symmetric geometrical model of the nozzle was used for the analysis purpose. Divergence angles chosen were 4°, 7°, 10°, 13° and 15°

C A Hunter [2] conducted experimental, theoretical, and computational study of separated nozzle flows. Experimental testing was performed at the NASA Langley 16-Foot Transonic Tunnel Complex. As part of a comprehensive static recital examination, moment, force and pressure measurements were made and Schlieren flow visualization was obtained for a subscale, non axi-symmetric, two-dimensional, C-D nozzle. In addition, two-dimensional numerical simulations were run using the computational fluid dynamics code PAB3D with two-equation turbulence closure and algebraic Reynolds stress modeling. For reference, experimental and computational results were compared with theoretical predictions based on one-dimensional gas dynamics and an approximate integral momentum boundary layer method

Nazar Muneam Mahmood [3] simulated steady flow of a gas through a C-D nozzle which has a varying cross sectional area and showed that the nature of the flow can be explained by considering how the flow and its characteristics in the nozzle changes as the back pressure decreases. The characteristics of gas flow were simulated using the ANSYS Fluent 12.1 software to solve the quasi-one dimensional nozzle flow. According to the study, the reduction in the back pressure cannot affect conditions upstream of the throat, hence the nozzle is choked. The shock wave increases the pressure, density and temperature and reduces the velocity and Mach number to a subsonic value and as back pressure is further reduced to a certain value, the extent of the supersonic flow region increases, the shock wave moving further down the divergent portion of the nozzle towards the exit plane

Sibendu Soma et al [4] studies were carried out works on Effect of nozzle orifice geometry on spray, combustion, and emission characteristics under diesel engine conditions. The flame structure and stabilization are also noticeably influenced by orifice geometry. The flame lift-off lengths are the highest and lowest for the hydroground and conical nozzles, respectively the amount of soot produced is highest with a conical nozzle, while the amount of NOx produced is the highest with a hydroground nozzle

Padmanathan and vaidya [5] computational analysis were carried out a computational analysis for shock waves on C-D nozzle. It was proven computational that increase in the static pressure, density and the static temperature across the shock. Mach number decrease across the shock

Narayana and R.S Reddy [6] were compared five different mesh structures while investigating a converging diverging nozzle using CFD software. The first mesh is an unstructured mesh while the other four are structured meshes. Meshes are different from each other by their finer dimensions. The nozzle geometry is similar to the one in the undergraduate work. The medium mesh is seen to be convenience for the tradeoff between computational cost and accuracy of the results. However the applicant ignored this mesh independency study too and used one mesh configuration

Sudhakar et al [7] reported a CFD simulation of a converging diverging nozzle while they compared their CFD results with analytical results. Their findings are consistent with the analytical ones

David Gretrix and Dutton [8] were reported that, the most efficient conversion of pressure to kinetic energy occurred when nozzle is first narrowed thereby increasing the speed of jet to speed of sound. In the article it is said that at throat velocity magnitude is same for all divergence degrees of angles and nearby wall Mach number is decreasing for nozzles because of viscosity and turbulence in fluid. With an increase in divergence angle at nozzle exit Mach number increases. He concluded by saying that efficiency of nozzle increases by increasing the divergence angle of nozzle up to certain limit

Trong Bui and Bogdan [9] were conducted experiments on the CFD analysis of Nozzle jet plume effects on a supersonic aircraft. It is observed that the effect of flow viscosity, size of the grid and type of grid are significant in the jet signature plume. The under expanded nozzle shows more reduction in the strength of the tail stock of the nozzle while compared to the over expanded flow

Gustaf De Laval [10] established the concept of impulse steam turbine after that he built a model of steam turbine to demonstrate that such devices can be set up on scale. Results shows that a nozzle to increase steam jet to supersonic speed and used in rocket engine nozzles.

Deshpande et al [11] presented a technical paper was to study and find the analysis of De-Laval nozzle. It is a converging-diverging type of nozzle, which is used to provide supersonic jet velocity at the exit of nozzle

Nathan spots [12] Performed a CFD analysis on compressible flow through convergent conical shaped nozzle and investigated that the coefficient of discharge increase on decreasing nozzle angle and also confirmed that the choked nozzle pressure ratio low for a small nozzle angle

Shembharkar et al [13] Studied studied 2-dimendional convergent Divergent nozzle for nozzle pressure ratio(NPR) corresponding to presence of shock inside the diverging section

of nozzle and concluded that computed solution using fluent differed from the simple theory as structure of shock their location and after shocks are concerned.

2.1 Scope of the work from Literature survey

The extensive literature survey by the authors identifies some of the gaps in the area of flow analysis of convergent & divergent nozzle by varying the convergence angles and divergence angles. Hence the authors made an attempt to study the influence of flow parameters on convergent and divergent nozzle using CFD analysis.

Chapter-3

Problem Statement

The main objective of the present work is to simulate supersonic flow through convergent and divergent nozzle to precisely understand the flow dynamics and variation of flow parameters namely Pressure and velocity using computational fluid dynamics (CFD)

For this study a nozzle is designed of varying convergence angles and divergence angles, analysed using CFD for various performance parameters. The flow conditions were selected based on the pressure and temperature of the air at the inlet of the nozzle.

Chapter-4

MODELLING AND SIMULATION

4.1 modelling

4.1.1 Geometry:

The geometry of the nozzle was created using the Geometry workbench of ANSYS. A two-dimensional geometry of the nozzle was created. The Dimensions of the nozzle is reported in table 4.1.

Table 4.1 dimensions of the nozzle

Length of nozzle	200mm
Inlet width about axis	50mm
Outlet width about axis	80mm
Radius of curvature of throat	15mm
Divergence angle	30°
Convergence angle	30°

The figure 4.1 shows the symmetric figure of a 2-D C-D nozzle
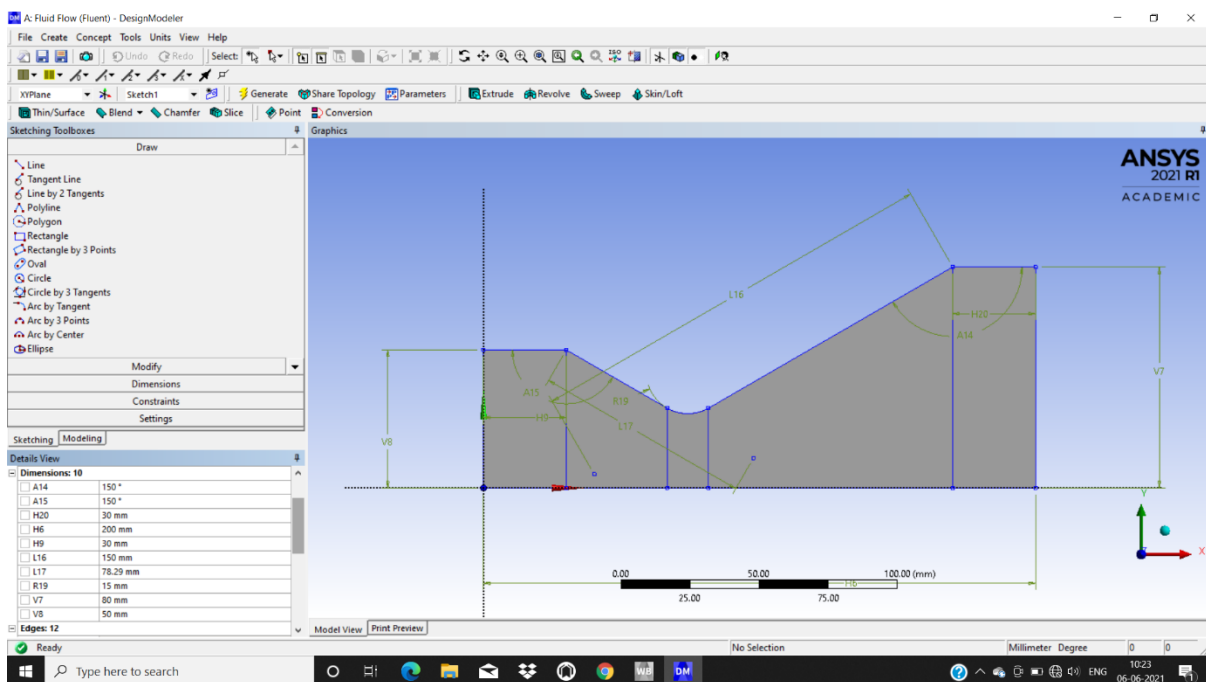
Fig 4.1 2-d Model of C-D nozzle

4.1.2 Meshing

Ansys provides general purpose, high-performance, automated, intelligent meshing software that produces the most appropriate mesh for accurate, efficient multiphysics solutions from easy, automatic meshing to highly crafted mesh. Smart defaults are built into the software to make meshing a painless and intuitive task, delivering the required resolution to capture solution gradients properly for dependable results.

Ansys meshing solutions range from easy, automated meshing to highly crafted meshing. Methods available cover the meshing spectrum of highorder to linear elements and fast tetrahedral and polyhedral to highquality hexahedral and mosaic.

Ansys meshing capabilities help reduce the amount of time and effort spent to get to accurate results. Since meshing typically consumes a significant portion of the time it takes to get simulation results, Ansys helps by making better and more automated meshing tools.

Whether performing a structural, fluid or electromagnetic simulation, ansys can provide us with the most appropriate mesh for accurate and efficient solutions.The below image gives us a glimpse of ansys meshing

The created geometry was imported to the meshing workbench. The mesh used was tetrahedral mesh. Figure 4.2 shows the initial mesh of C-D nozzle.
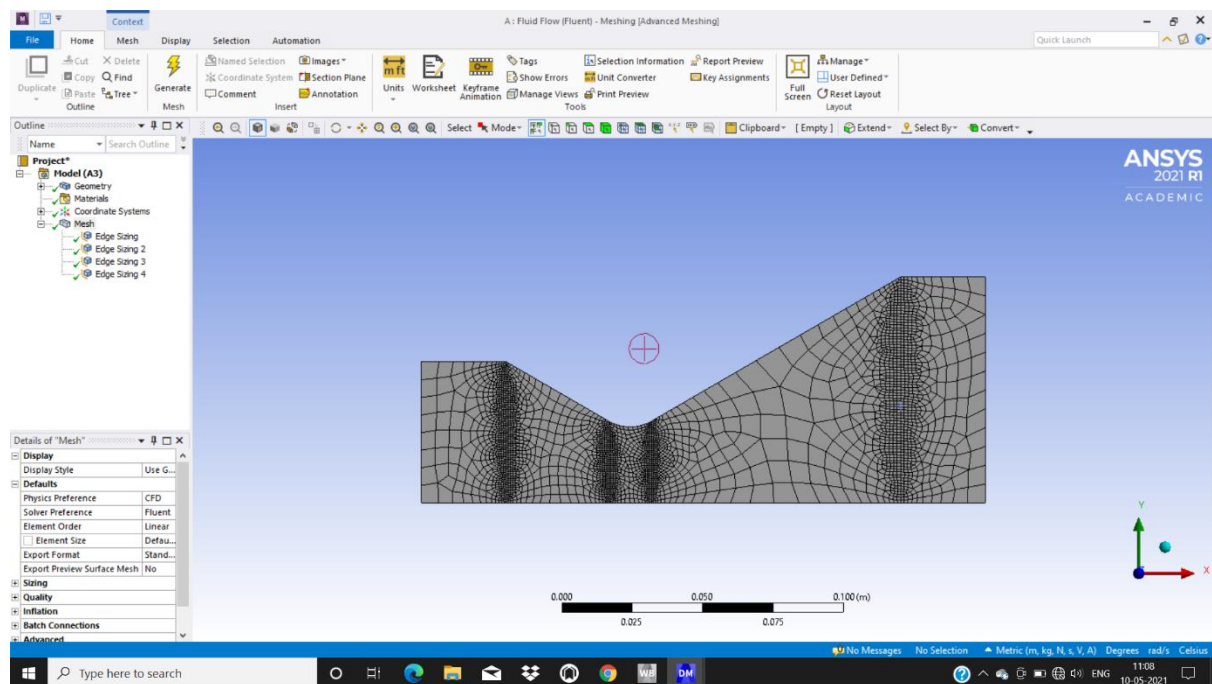

Fig 4.2 Initial mesh of nozzle

The face-mapped meshing option was employed to the geometry in order to avoid the resolution errors. The mesh was refined to the third degree using the refinement option of the workbench. After meshing, the inlet, the axis and the outlet boundaries were named. Edge

sizing, face sizing and face meshing are the meshing techniques adopted for the meshing of C-D nozzle. Figure 4.3 shows the final mesh of a C-D nozzle after applying meshing conditions.
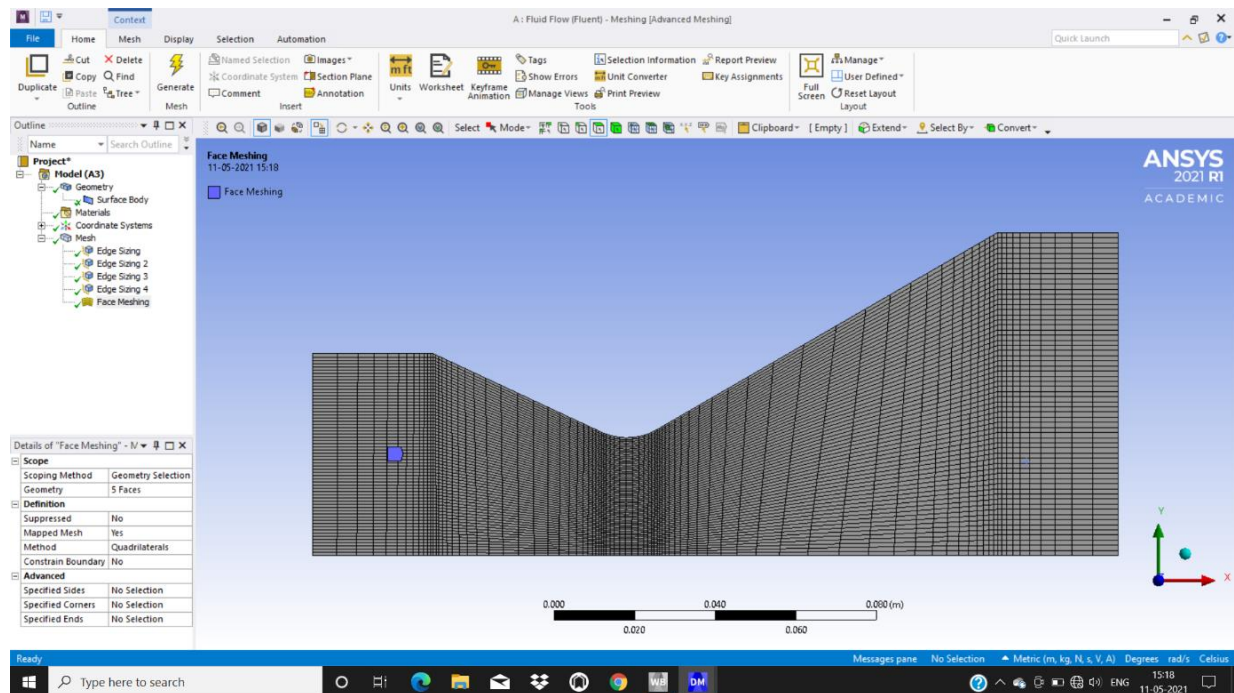

Fig 4.3 Final mesh of the nozzle

4.2 Simulation

The meshed geometry is now imported to the FLUENT workbench Setup module.Certain boundary conditions applied for simulation process, the solver type selected as inviscid to eliminate viscosity and 2-d space as axisymmetric. The figure 4.4 shows the solver setup for ansys simulation.
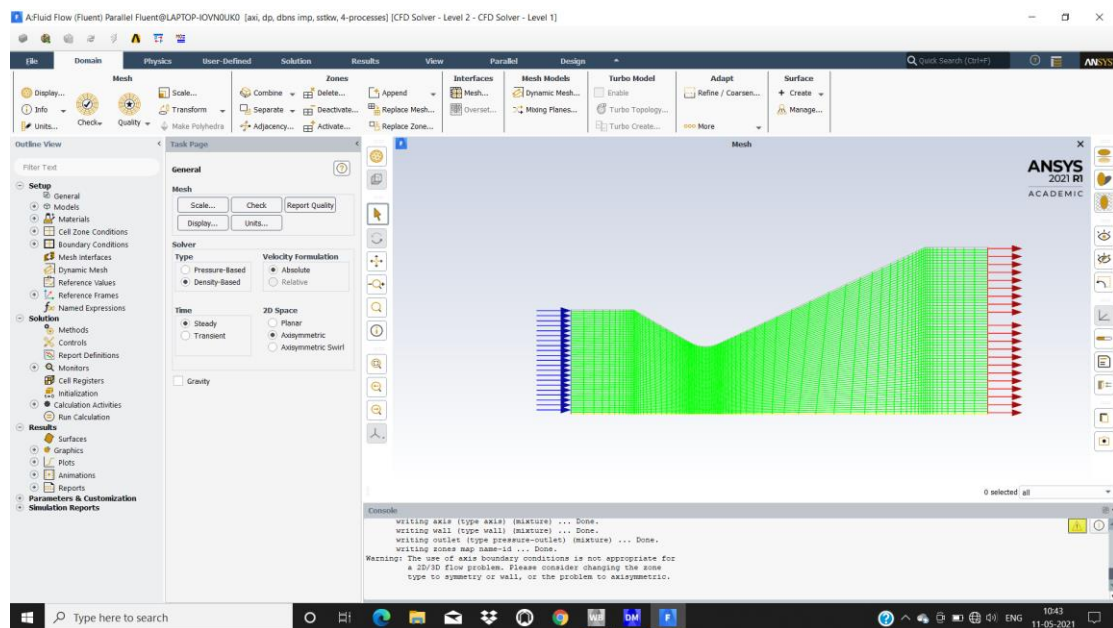
Fig 4.4: Solver setup for ansys simulation

In the setup module the following settings are made

Table 4.2 Problem setup

General	Solver type: density based 2D Space: Axi-symmetric
Models	Energy equation: On Viscous model: inviscid
Materials	Air-Ideal gas
Boundary Conditions	Inlet- pressure Total gauge pressure 101325 pascal Outlet – pressure Gauge pressure 2800 pascal Axis – axis boundary
Operating conditions	Operating pressure 0 pascal

Table 4.3 solver setup

Solution initialization	Compute from: inlet
Run calculation	Check case, Enter number of iterations: 1500 Click Calculation

Table 4.4 Contour setup

Graphics and animation	Use contour option to get the pressure contour and velocity contour
Plots	Use XY plots to plot the velocity and pressure

Chapter 5

RESULTS AND DISCUSSIONS

5.1 Case 1: Convergence 30° Divergence 25°

The pressure contour & velocity contour were analysed using CFD by considering convergence 30° and Divergence 25°
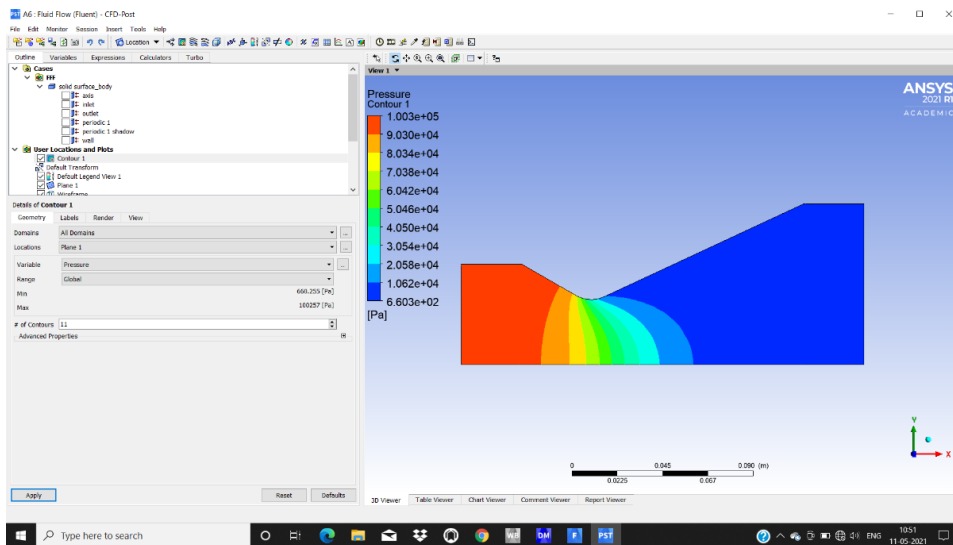
5.1.1: Pressure Contour

Fig 5.1 Pressure contour

From the figure 5.1 it can be observed that the pressure is decreasing form inlet to the outlet. At throat the pressure is $7.038*10^4$ pa, at outlet the pressure is $6.603*10^2$ pa
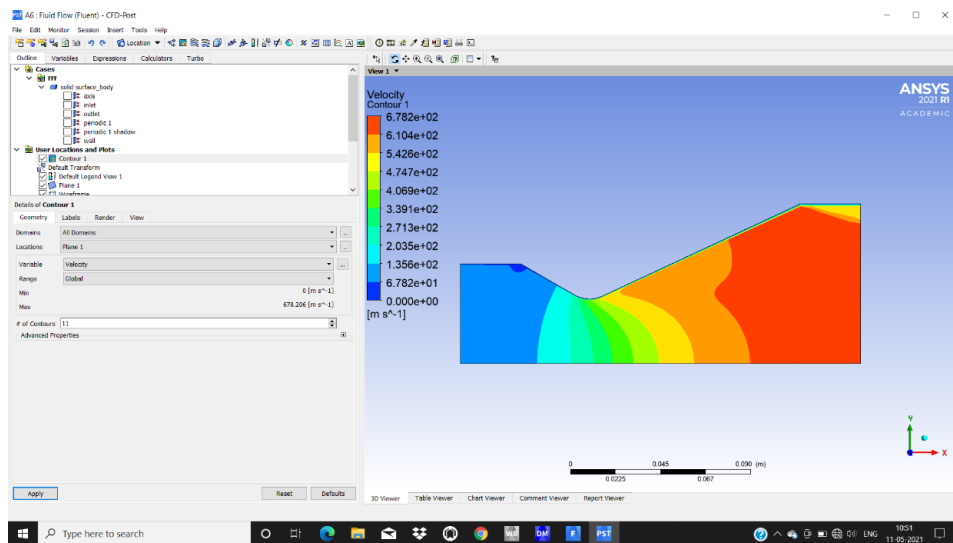
5.1.2: Velocity Contour

Fig 5.2 Velocity contour

From fig 5.2 velocity Contour, it is clearly shown that the velocity is increasing from inlet to outlet. At throat the velocity is nearly 4.069*10^2 m/s at outlet it is nearly 6.782*10^2 m/s.

5.1.3: Velocity vs Pressure plot

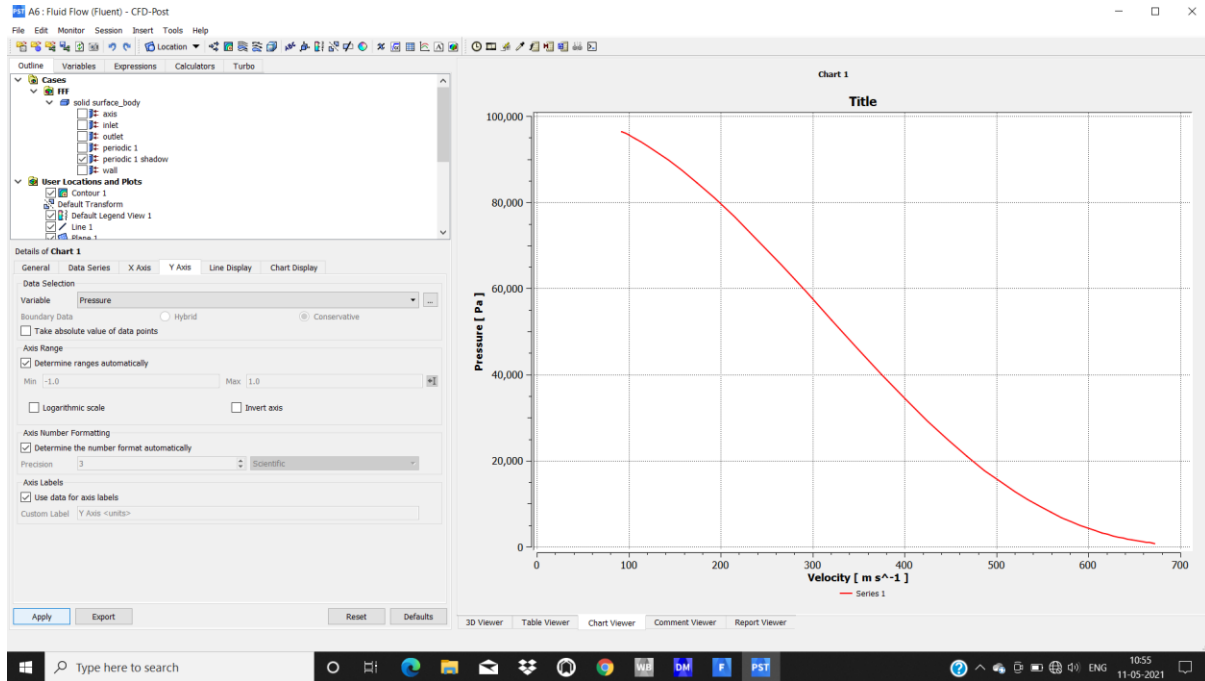The graph shows the velocity vs pressure plot at Convergence 30° & Divergence 25°

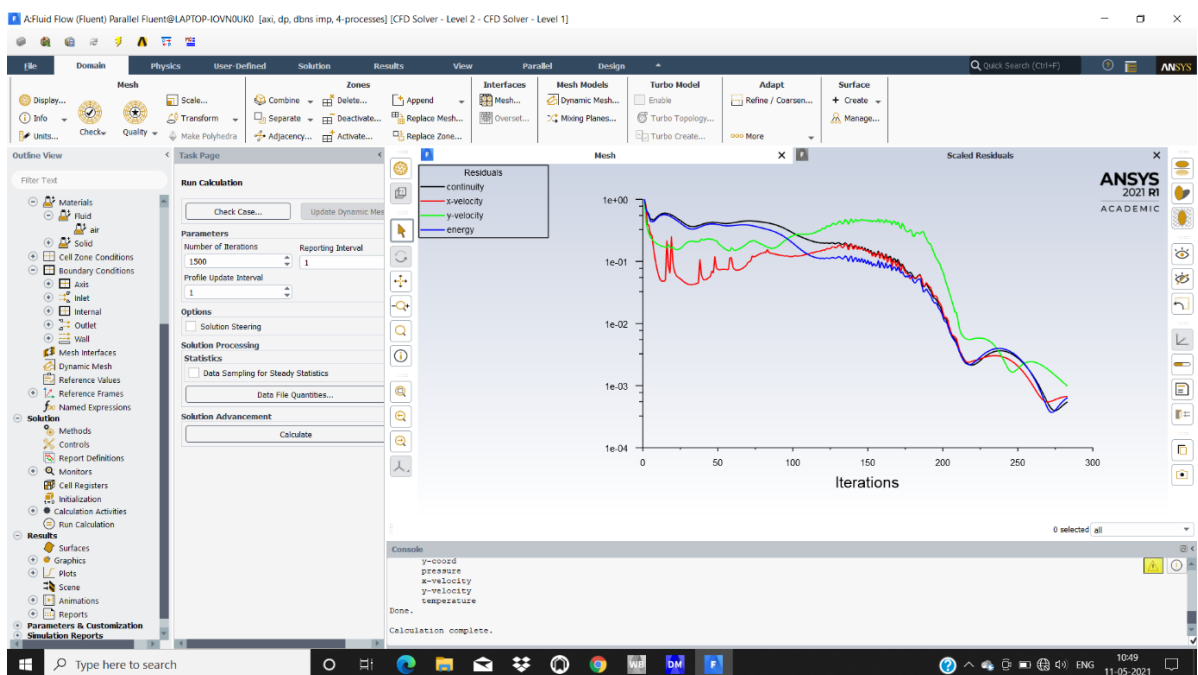Fig 5.3 Velocity vs Pressure

5.1.4: Residual velocities

Fig 5.4 Residual Velocities

Figure 5.4 shows the residual velocities of a C-D nozzle at Convergence angle 30° and Divergence angle 25°

5.2 Case 2: Convergence 30° Divergence 30°

The pressure contour & velocity contour were analysed using CFD by considering convergence 30° and Divergence 30°
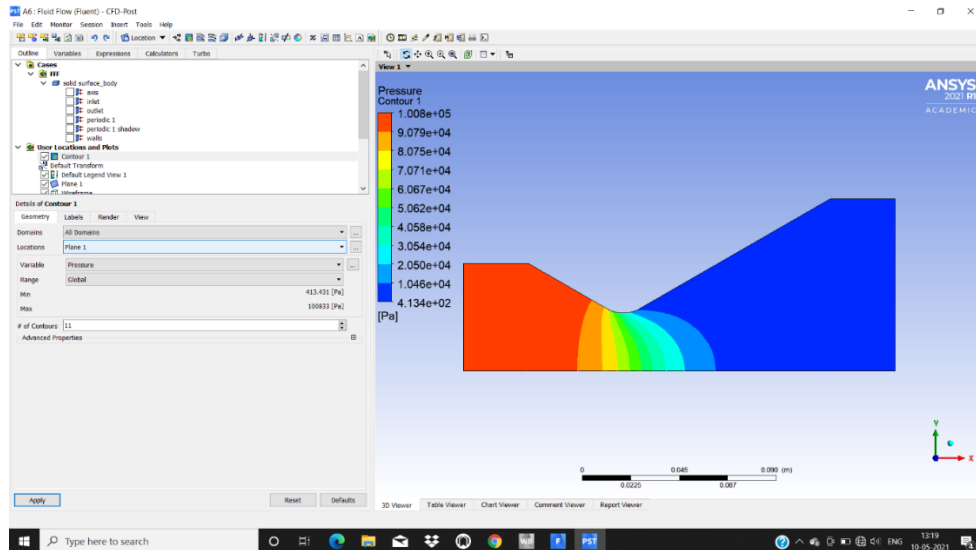
5.2.1: Pressure Contour

Fig 5.5 Pressure Contour

From the figure 5.5 pressure contour, it can be observed that the pressure is decreasing form inlet to the outlet. At throat the pressure is 7.071*10^4pa, at outlet the pressure is 4.134*10^2 pa
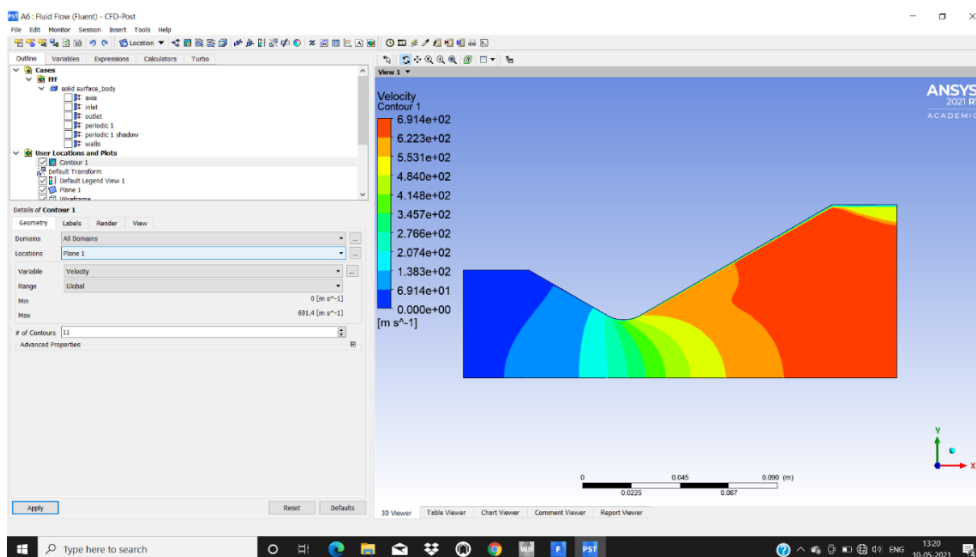
5.2.2: Velocity Contour

Fig 5.6 velocity contour

From fig 5.6 velocity contour it is clearly shown that the velocity is increasing from inlet to outlet. At throat the velocity is nearly 4.148*10^2 m/s at outlet it is nearly 6.914*10^2 m/s.
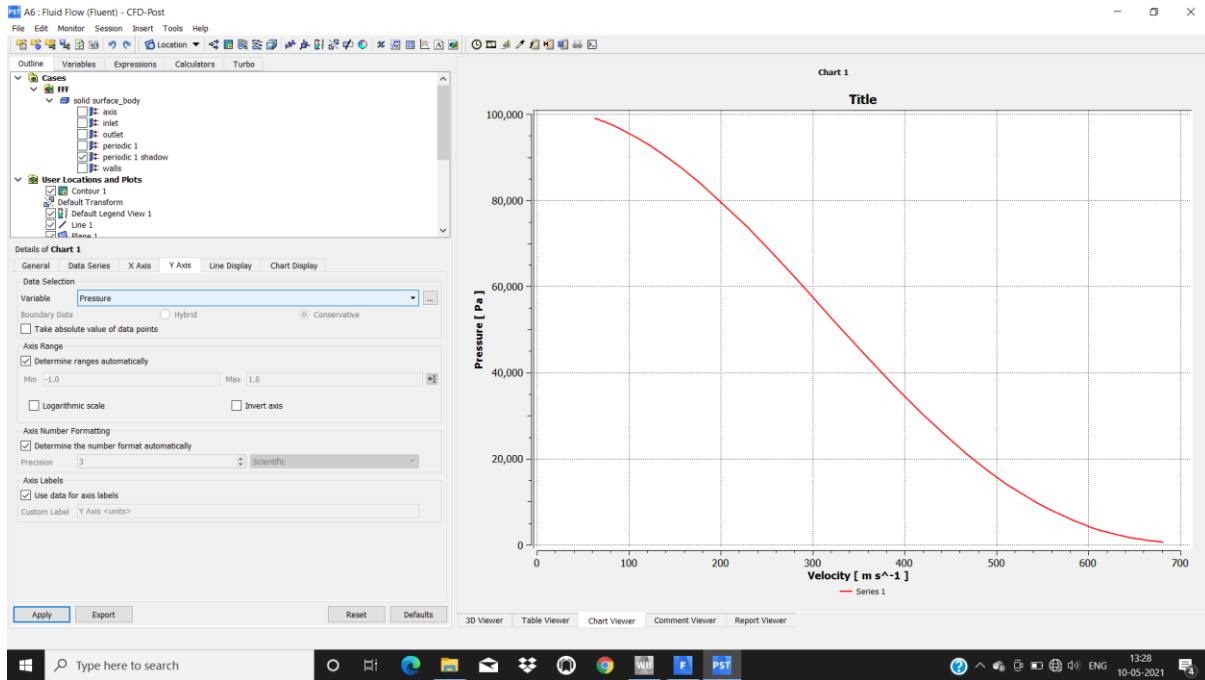
5.2.3: Velocity vs Pressure plot

Fig 5.7 Velocity vs Pressure

The above graph shows the velocity vs pressure plot at Convergence 30° & Divergence 30°

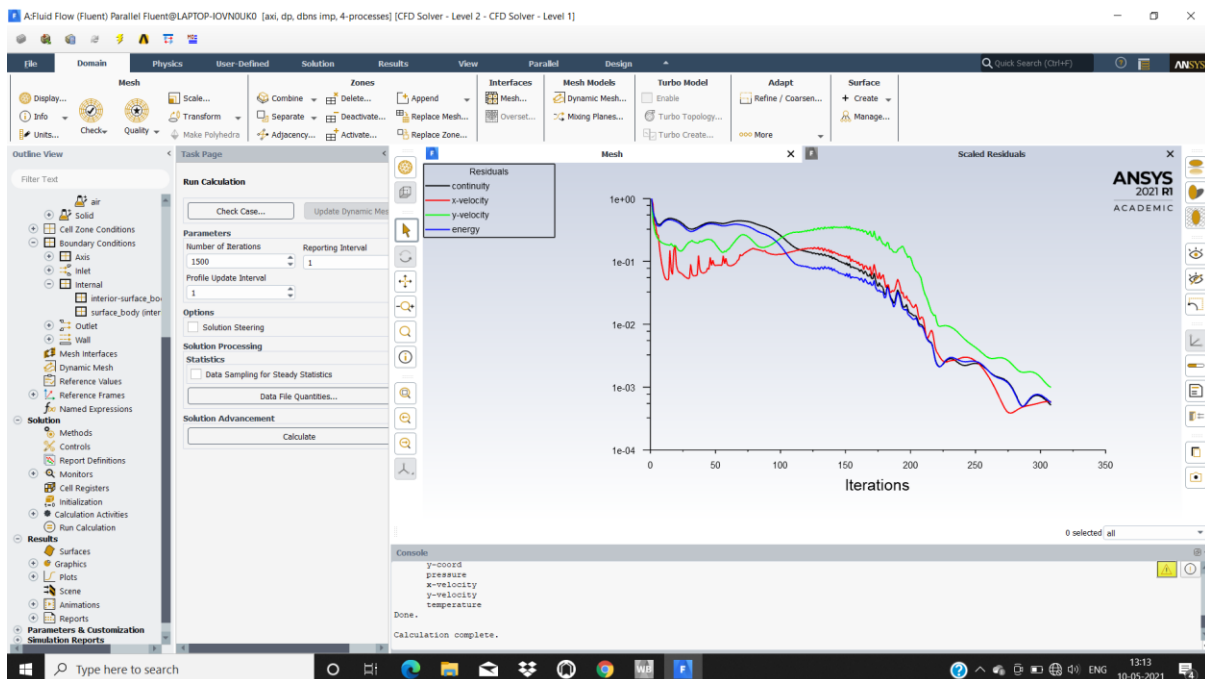### 5.2.4: Residual velocities

Fig 5.8 Residual velocities

Figure 5.8 shows the residual velocities of C-D nozzle Convergence 30° and Divergence 30°

5.3 Case 3: Convergence 30° Divergence 35°

The pressure contour & velocity contour were analysed using CFD by considering convergence 30° and Divergence 35°
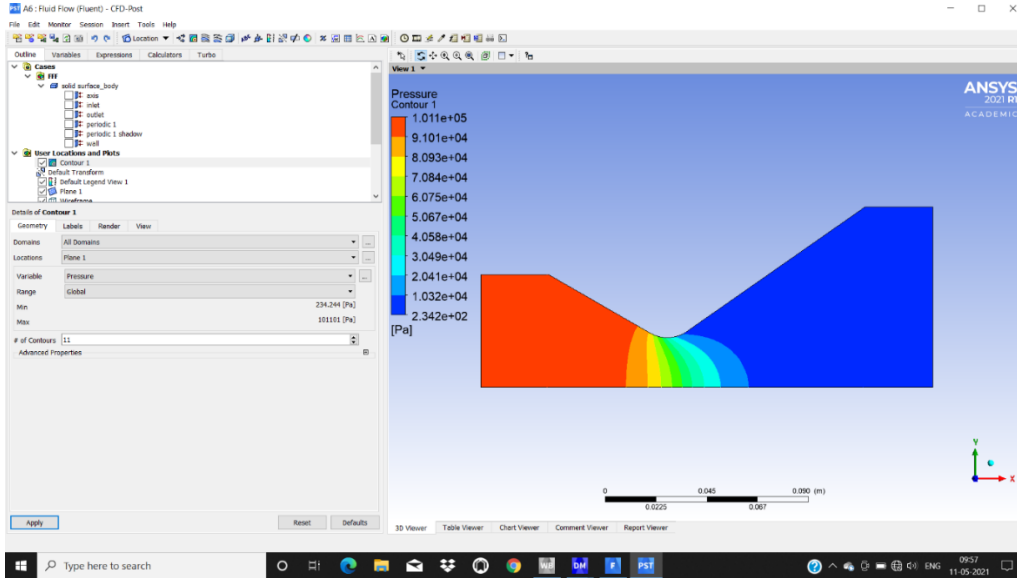
5.3.1: Pressure Contour

Fig 5.9 Pressure contour

From the figure 5.9 pressure contour, it can be observed that the pressure is decreasing form inlet to the outlet. At throat the pressure is 7.084*10^4pa, at outlet the pressure is 2.342*10^2 pa
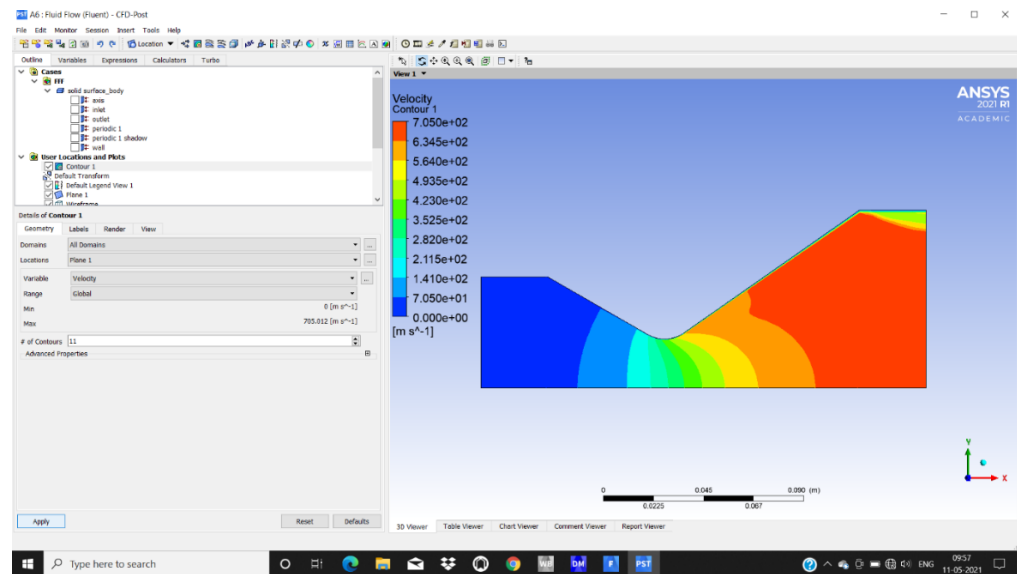
5.3.2: Velocity Contour

Fig 5.10 Velocity Contour

From fig 5.10 it is clearly shown that the velocity is increasing from inlet to outlet. At throat the velocity is nearly 4.230*10^2 m/s at outlet it is nearly 7.050*10^2 m/s
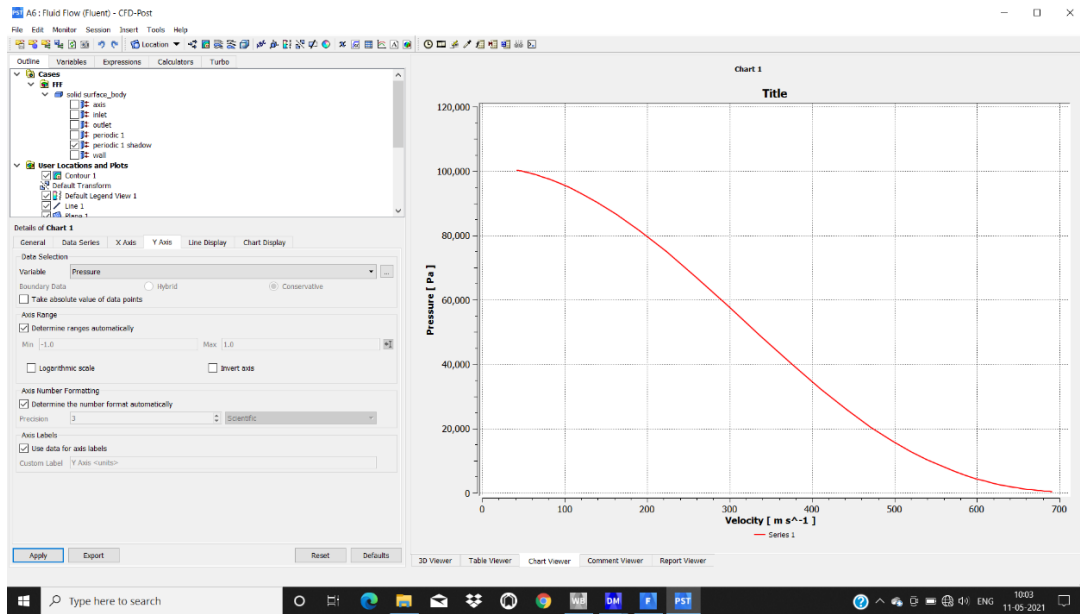
5.3.3: Velocity vs Pressure plot

Fig 5.11 Velocity vs Pressure

The above graph shows the velocity vs pressure plot at Convergence 30° & Divergence 35°
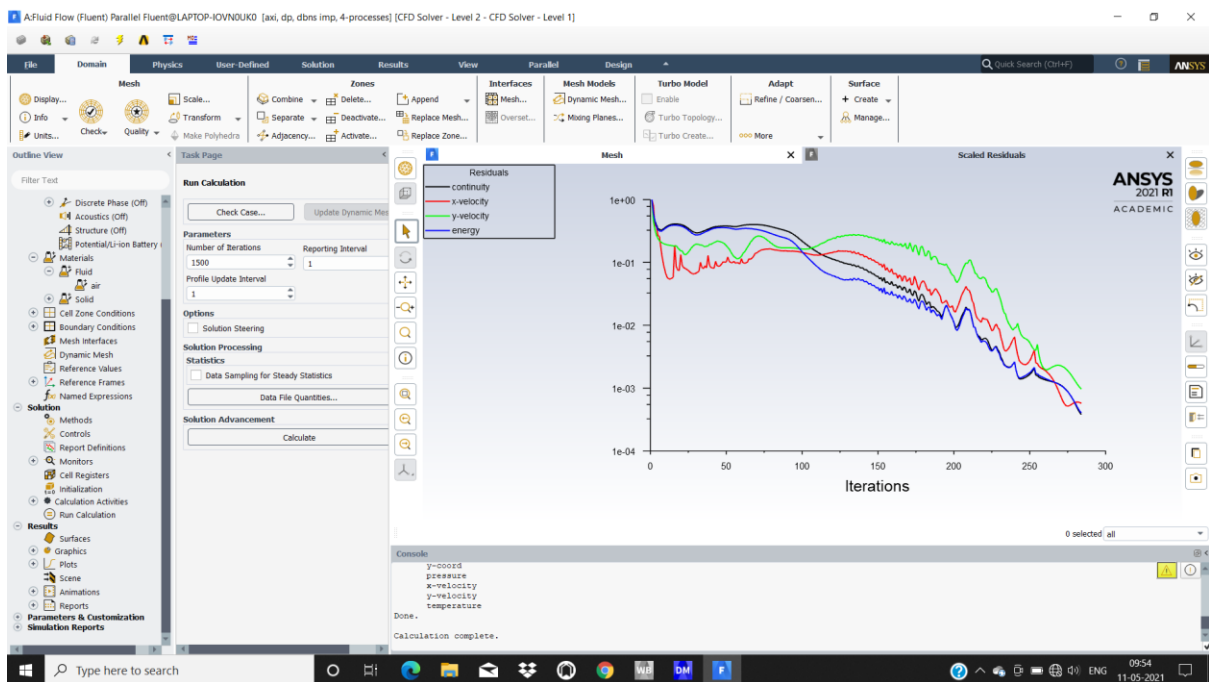
5.3.4: Residual velocities

Fig 5.12 Residual Velocities

The figure 5.12 shows the Residual velocities of C-D nozzle of convergence 30° & divergence 35°

5.4 Case 4: Convergence 25° divergence 30°

The pressure contour & velocity contour were analysed using CFD by considering convergence 25° and Divergence 30°
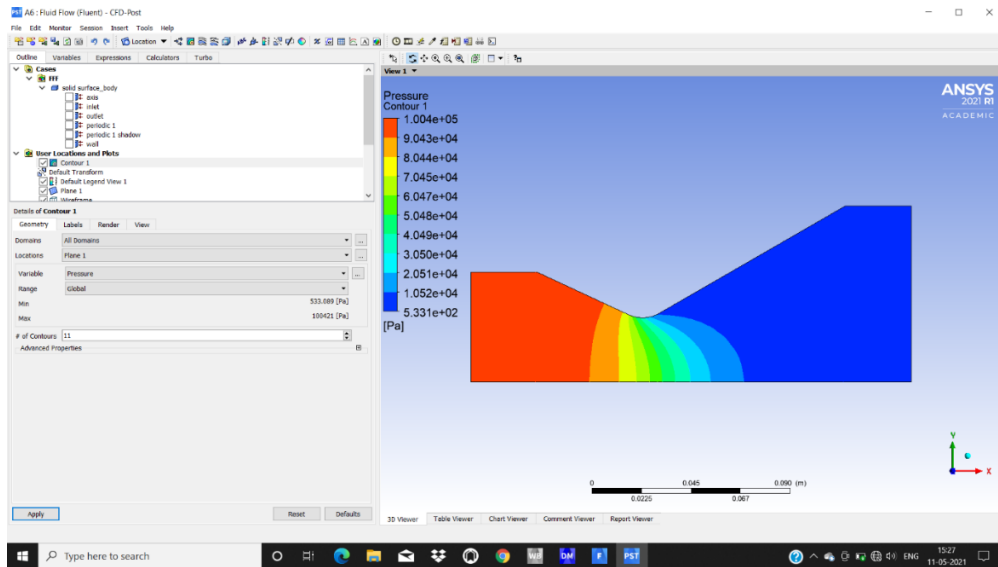
5.4.1: Pressure Contour

Fig 5.13 Pressure Contour

From the figure 5.13 it can be observed that the pressure is decreasing form inlet to the outlet. At throat the pressure is 6.047*10^4 pa, at outlet the pressure is 5.331*10^2 pa

5.4.2: Velocity Contour

Fig 5.14 Velocity Contour

From fig 5.14 it is clearly shown that the velocity is increasing from inlet to outlet. At throat the velocity is nearly 4.107*10^2 m/s at outlet it is nearly 6.845*10^2 m/s

5.4.3: Velocity vs Pressure plot

Fig 5.15 Velocity vs Pressure

The above graph shows the velocity vs pressure plot at Convergence 25° & Divergence 30°

5.4.4: Residual velocities

Fig 5.16 Residual velocities

Figure 5.16 shows the residual velocities of C-D nozzle convergence 25° and divergence 30°

5.5 Case 5: Convergence 35° Divergence 30°

The pressure contour & velocity contour were analysed using CFD by considering convergence 35° and Divergence 30°

5.5.1: Pressure Contour
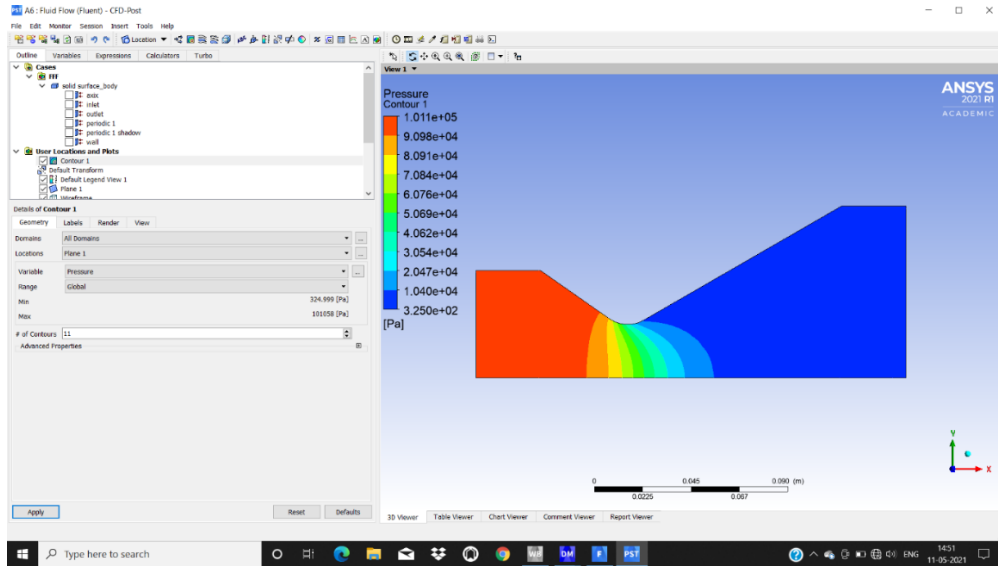
Fig 5.17 Pressure Contour

From the figure 5.17 it can be observed that the pressure is decreasing form inlet to the outlet. At throat the pressure is 6.076*10^4 pa, at outlet the pressure is 3.250*10^2 pa
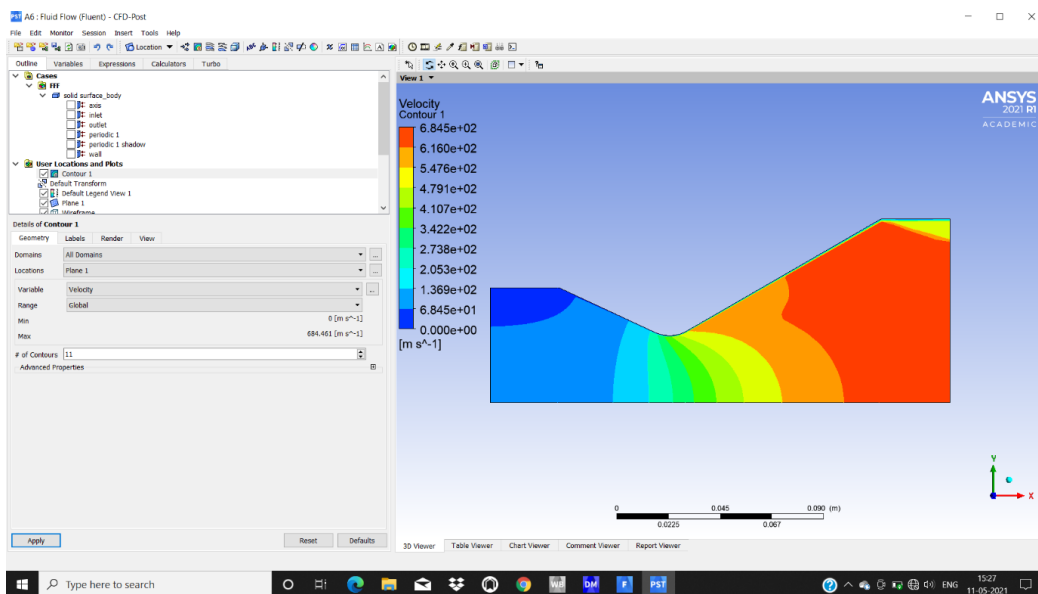
5.5.2: Velocity Contour
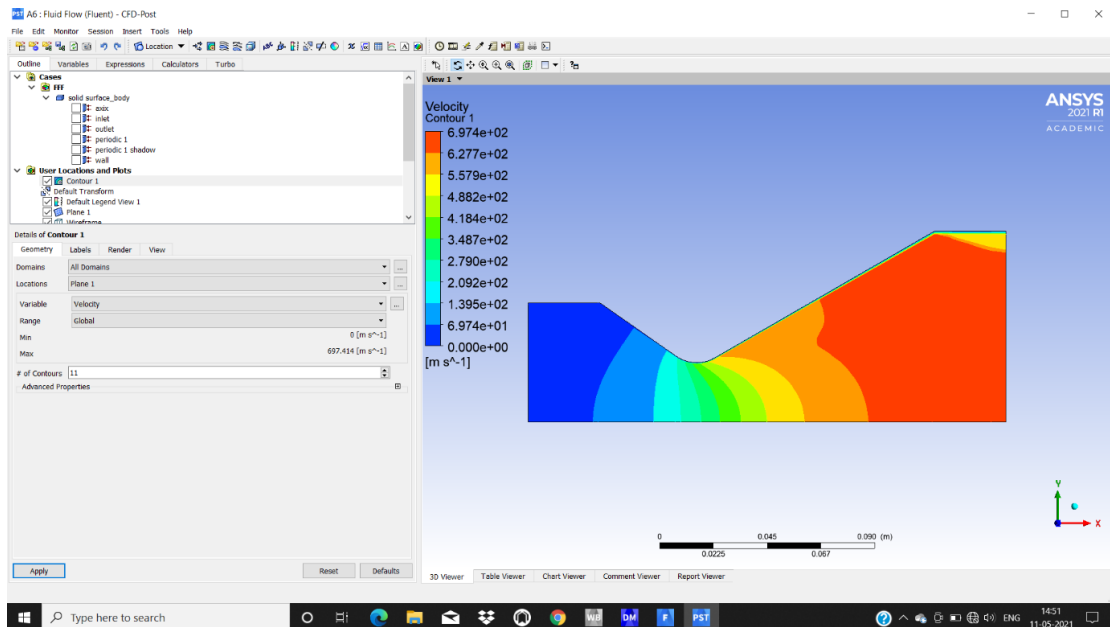
Fig 5.18 Velocity Contour

From fig 5.18 it is clearly shown that the velocity is increasing from inlet to outlet. At throat the velocity is nearly 3.487*10^2 m/s at outlet it is nearly 6.974*10^2 m/s

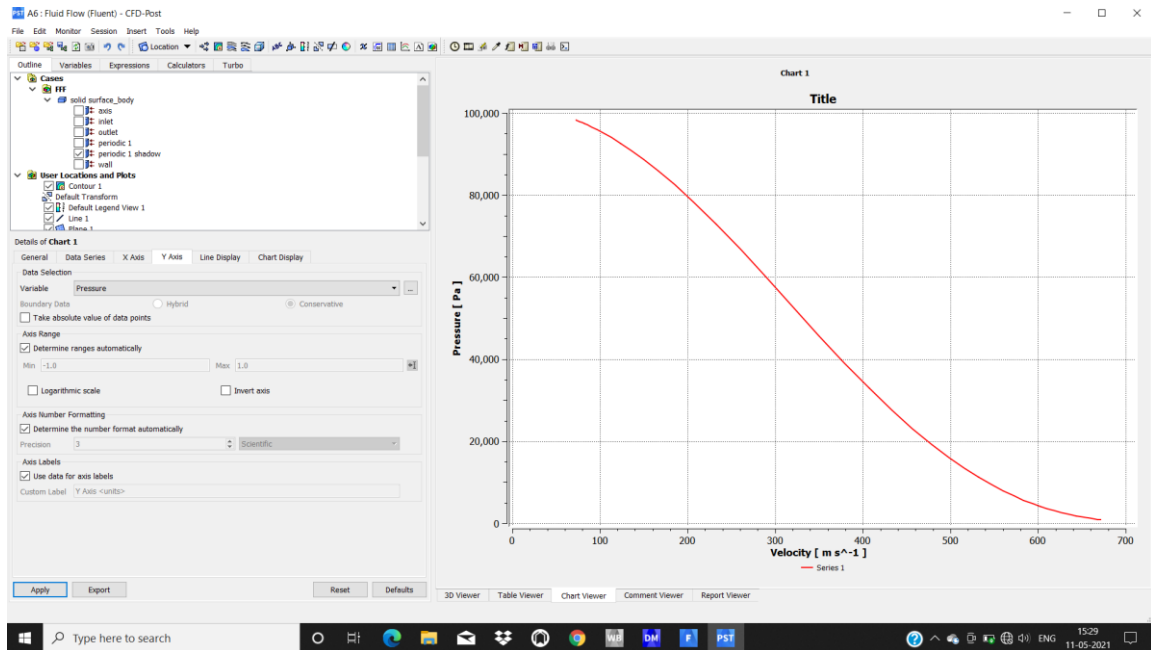### 5.5.3: Velocity vs Pressure plot

Fig 5.19 Velocity vs Pressure

The above graph shows the velocity vs pressure plot at Convergence 35° & Divergence 30°
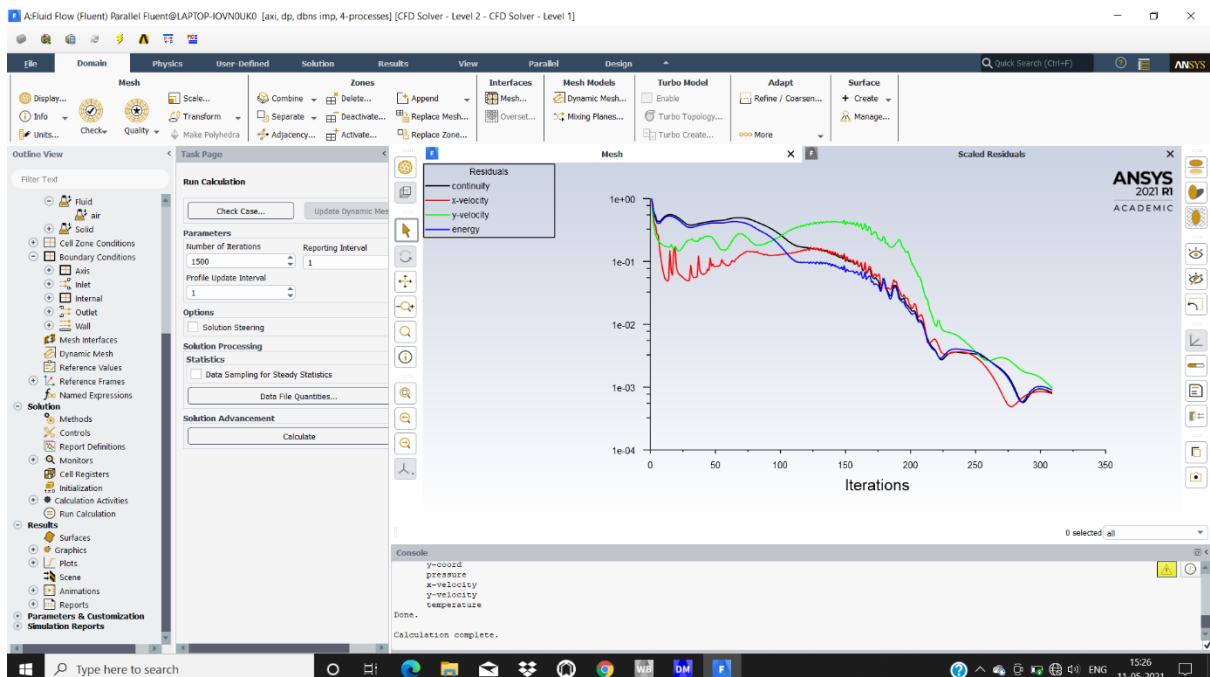
5.5.4: Residual velocities
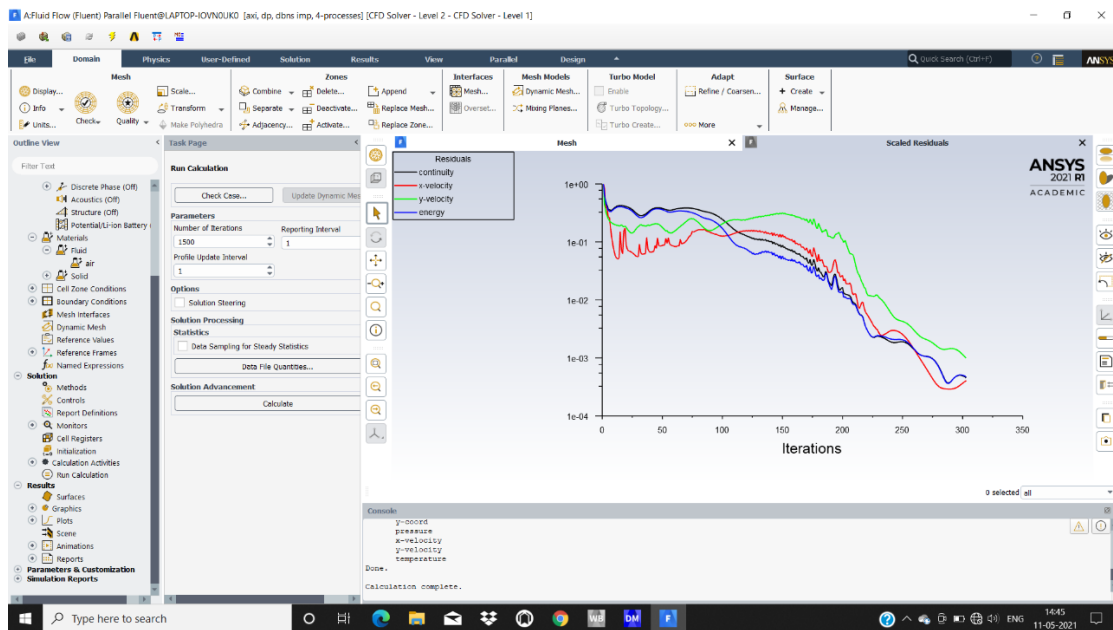
Fig 5.20 Residual Velocities

Figure 5.20 shows the Residual velocities of C-D nozzle at Convergence 35° & Divergence 30°

Table 6.1 At inlet of the nozzle

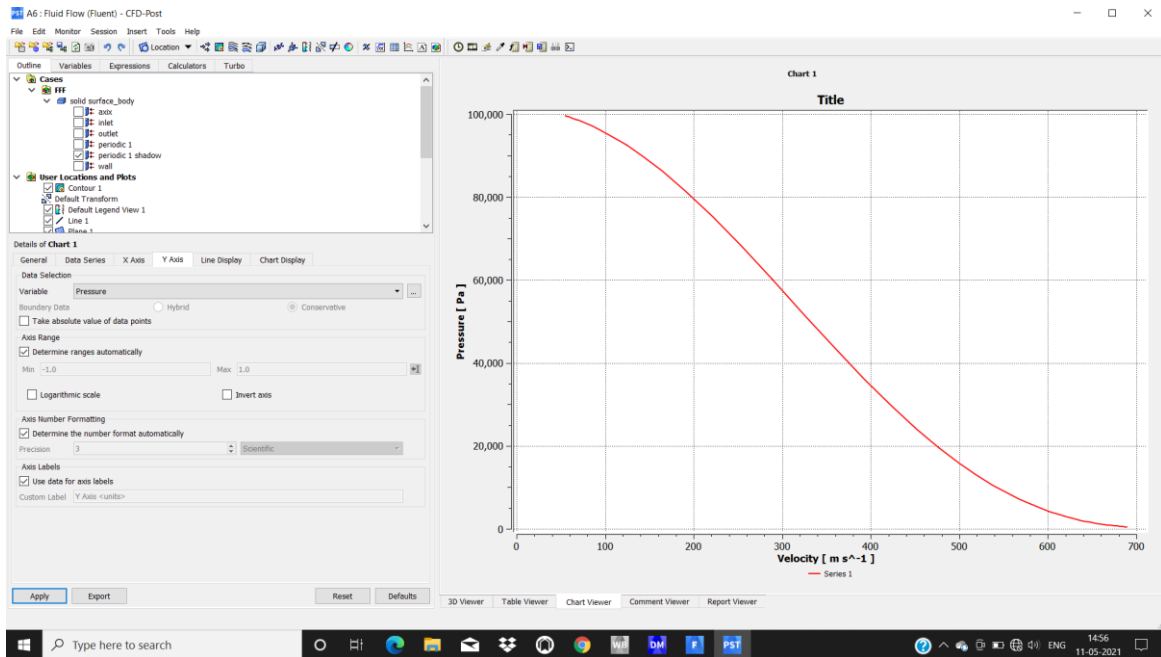| Case	Convergence angle	Divergence angle	Pressure(pa)	Velocity(m/s)
1	30°	25°	1.003*10^5	0
2	30°	30°	1.008*10^5	0
3	30°	35°	1.011*10^5	0
4	25°	30°	1.004*10^5	0
5	35°	30°	1.011*10^5	0

The table 6.1 shows the values of pressure and velocity at inlet of the C-D nozzle at various cases

Table 6.2 At throat section

Case	Convergence angle	Divergence angle	Pressure(pa)	Velocity(m/s)
1	30°	25°	7.038*10^4	4.069*10^2
2	30°	30°	7.071*10^4	4.148*10^2
3	30°	35°	7.084*10^4	4.230*10^2
4	25°	30°	6.047*10^4	4.107*10^2
5	35°	30°	6.076*10^4	3.487*10^2

The table 6.2 shows the values of pressure and velocity obtained at throat of a C-D nozzle from all the cases

Table 6.3 Varying divergence angle

Case	Convergence angle	Divergence angle	Pressure(pa)	Velocity(m/s)
1	30°	25°	6.603*10^2	6.782*10^2
2	30°	30°	4.134*10^2	6.914*10^2
3	30°	35°	2.342*10^2	7.050*10^2

The table 6.3 shows the values of pressure and velocity at outlet of a C-D nozzle at Convergence angle 30° varying divergence angles.

Table 6.4 Varying convergence angle

Case	Convergence angle	Divergence angle	Pressure(pa)	Velocity(m/s)
4	25°	30°	5.331*10^2	6.845*10^2
2	30°	30°	4.134*10^2	6.914*10^2
5	35°	30°	3.250*10^2	6.974*10^2

The table 6.4 shows the values of pressure and velocity at outlet of a C-D nozzle at Divergence angle 30° varying Convergence angles.

Chapter-6

CONCLUSIONS

The following observations were found in the nozzle by varying convergence and divergence angles

- When we increase the divergence angle keeping the convergence angle as constant, pressure decreases and velocity increases in great magnitude.
- By keeping convergence angle 30° and varying divergence angle, the maximum velocity is obtained at 35° divergence angle.
- When we increase the convergence angle keeping the divergence angle constant, we observed there is decrease in pressure and increase in velocity but the variation is less compared to earlier case.
- 35° convergence angle is giving maximum velocity when compared to other angles by keeping divergence angle constant at 30°.

Therefore, we can conclude that in a Convergent-Divergent nozzle, variation in divergence angle affects the velocity and pressure effects compared to variation in convergence angle.

FUTURE SCOPE OF WORK:

1. The present work may be extended to optimization of convergence and divergence angle for more outputs.
2. The analysis may be extended using ANN & Fuzzy Techniques.

List of references

1. BijuKuttan P: "Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics" The International Journal Of Engineering And Science (Ijes),Volume 2,Issue2, Issn: 2319 – 1813 Isbn: 2319 – 1805, 2013, pp 196-207

2. C.A. Hunter: "Experimental, Theoretical, and Computational Investigation of Separated Nozzle Flows-AIAA 98-3107".

3. Nazar Muneam Mahmood Simulation Of Back Pressure Effect On Behavior Of Convergent Divergent Nozzle Diyala Journal of Engineering Sciences ISSN 1999-8716Vol. 06, No. 01, March 2013 pp. 105-120

4. Sibendu Soma, Anita I. Ramirez , Douglas E. Longman , Suresh K. Aggarwal, "Effect of nozzle orifice geometry on spray, combustion, and emission characteristics under diesel engine conditions" journal homepage: www.elsevier.com/locate/fuel

5. P. Padmanathan, Dr. S. Vaidya Nathan, "Computational Analysis of Shockwave in Convergent Divergent Nozzle" International Journal of Engineering Research and Applications, Vol. 2, Issue 2,Mar-Apr 2012, pp.1597-1605

6. K.P.S.S. Narayana and K.S. Reddy, "Simulation of Convergent Divergent Rocket Nozzle using CFD Analysis," IOSR Journal of Mechanical and Civil Engineering, vol. 13, iss. 4, pp. 58–65, 2016.

7. B.V.V.N. Sudhakar, B.P.C. Sekhar, P.N. Mohan, M.D.T. Ahmad, "Modeling and simulation of Convergent-Divergent Nozzle Using Computational Fluid Dynamics," International Research Journal of Engineering and Technology, vol. 3, iss. 8, pp. 346-350, 2016.

8. David Getrix, Dutton, J.C "Swirling Supersonic Nozzle Flow," Journal of Propulsion and Power, vol.3, July 1987.

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

10. 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

11. Nikhil D. Deshpande, Suyash, S. Vidwans, "Theoretical and CFD analysis of De-Laval nozzle" in International Journal of Mechanical and Production Engineering, Volume-2, Issue- 4, April-2014

12. Nathan spotts "A CFD analysis of compressive flow through convergent conical nozzle " first AIAA Propulsion Aerodynamics Workshop, Atlanta Georgia, July 29, 2012

13. T.R. Shembharkar and A.A. Khan, 2008, " viscous Flow Analysis .In A Convergent Divergent Nozzle" Proceeding of the International conference on Aerospace Science and Technology 26-28 June 2008, Banglore, India