

ENHANCING MIXING CHARACTERISTICS OF CONVERGENT NOZZLE

Submitted in partial fulfillment of the requirements for the award of

Bachelor of Engineering degree in
Aeronautical Engineering

By

RAMBABU.E (Reg. No. 40260019)

BALAJI.S (Reg. No. 40260005)



**DEPARTMENT OF AERONAUTICAL ENGINEERING SCHOOL
OF MECHANICAL ENGINEERING**

SATHYABAMA

**INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)**

Accredited with grade "A++" by NAAC

JEPPIAAR NAGAR, RAJIV GANDHI SALAI, CHENNAI – 600 119

April 2024



SATHYABAMA

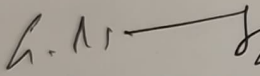
INSTITUTE OF SCIENCE AND TECHNOLOGY
(DEEMED TO BE UNIVERSITY)

Accredited "A++" Grade by NAAC | 12B Status by UGC | Approved by AICTE
www.sathyabama.ac.in

DEPARTMENT OF AERONAUTICAL ENGINEERING

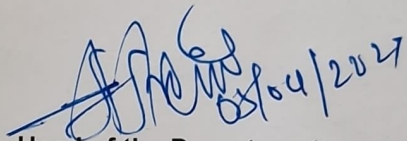
BONAFIDE CERTIFICATE

This is to certify that this Project Report is the bonafide work of **BALAJI.S** (Reg. No. 40260005) and **RAMBABU.E** (Reg. No. 40260019) who carried out the project entitled "**ENHANCING MIXING CHARACTERISTICS OF CONVERGENT NOZZLE**" under my supervision from December 2023 to April 2024.

 4/4/24.

Internal Guide

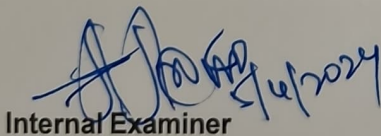
Mr.G.Madhan kumar,M.E., (Ph.D.,)

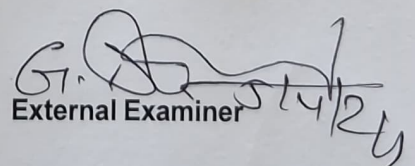
 05/04/2024

Head of the Department

Dr.A.Anderson ,M.E., Ph.D.,

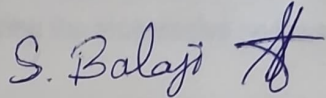
Submitted for Viva voce Examination held on : 05.04.2024

 5/4/2024
Internal Examiner

 5/4/2024
External Examiner

DECLARATION

I **BALAJI.S** (Reg. No. 40260005) hereby declare that the Project Report entitled "**ENHANCING MIXING CHARACTERISTICS OF CONVERGENT**" done by me under the guidance of **Mr.G.Madhan kumar,, M.E., (Ph.D.,)** is submitted in partial fulfilment of the requirements for the award of Bachelor of Engineering degree in Aeronautical Engineering.



DATE:05.04.2024

SIGNATURE OF THE CANDIDATE

PLACE: CHENNAI

ACKNOWLEDGEMENT

I am pleased to acknowledge my sincere thanks to Board of Management of **SATHYABAMA** for their kind encouragement in doing this project and for completing it successfully. I am grateful to them.

I convey my sincere thanks to **Dr.A.Anderson, M.E., Ph.D.**, Head of the Department, Department of Aeronautical Engineering for providing me necessary support and details at the right time during the progressive reviews.

I would like to express my sincere and deep sense of gratitude to my Project Guide **Mr.G.Madhan kumar, M.E., (Ph.D.,)** for his valuable guidance, suggestions and constant encouragement paved way for the successful completion of my project work.

I also express my thanks to all Teaching and Non-teaching staff members of the Department of Aeronautical Engineering who were helpful in many ways for the completion of the project.

ABSTRACT

This project presents a computational analysis for determining the flow properties of novel circular nozzles. In last few decades, circular nozzles have been investigated intensively due to their remarkably superior capabilities in enhancing mass entrainment over non circular nozzles. In this thesis, to increase the amount of ambient fluid entrained in the jet flow, three different geometrical modifications are applied to circular nozzles. In this study, the mixing characteristics of a circular free jet were investigated computationally for different turbulence models. Found that K-epsilon turbulent model deemed to be fit for $M=0.6$ The study found that the variation in length of the nozzle leading to improved mixing characteristics. The flow properties of air emanating from geometrically modified circular nozzles are studied by using Computational Fluid Dynamics (CFD). This computational analysis shows that suitably modified circular nozzles are very effective passive flow conditioning devices and can modify the flow field. Particularly, nozzle with length variation entrains the ambient fluid at a higher rate than the diameter ratio modified nozzles.

TABLE OF CONTENTS

CHAPTER NO.	TITLE	PAGE NO.
	ABSTRACT	i
	LIST OF TABLES	iv
	LIST OF FIGURES	v
	LIST OF SYMBOLS AND ABBREVIATIONS	viii
1	INTRODUCTION	1
1.1	Circular Jets	1
1.2	Turbulent Jets	3
1.3	Flow Control	3
1.4	Objectives	4
2	LITERATURE REVIEW	6
3	AIM AND SCOPE OF STUDY	17
3.1	Aim of the Study	17
3.2	Scope of the Study	17
4	METHODOLOGY OF COMPUTATIONAL STUDIES	18
4.1	Finite Volume Method	18
4.2	Governing Equations	18
4.3	Segregated Flow Model	20
4.4	Turbulence Modeling	21
4.4.1	Two equation models	21
4.4.2	The realizable K- ϵ turbulence model	22
4.5	Boundary Conditions	23
4.6	Mesh Generation	26
4.7	Nozzle geometry	27

CHAPTER NO.	TITLE	PAGE NO.
4.8	Experimental Setup	28
4.8.1	Shadowgraph	29
5	RESULT AND DISCUSSION	31
5.1	Grid Independence Study	31
5.2	Analysis of Diameter Ratio = 2 with Turbulent Model-K EPSILON STD	33
5.3	Analysis of Diameter Ratio = 3 with Turbulent Model-K EPSILON STD	34
5.4	Analysis of Diameter Ratio = 4 with Turbulent Model-K EPSILON STD	35
5.5	Analysis of Diameter Ratio = 2 with Turbulent Model-K EPSILON RNG	36
5.6	Analysis of Diameter Ratio = 3 with Turbulent Model-K EPSILON RNG	37
5.7	Analysis of Diameter Ratio = 4 with Turbulent Model-K EPSILON RNG	38
5.8	Analysis of Diameter Ratio = 2 with Turbulent Model-K EPSILON REALIZABLE	39
5.9	Analysis of Diameter Ratio = 4 with Turbulent Model-K OMEGA STD	40
5.10	Analysis of Diameter Ratio = 3 with Turbulent Model-K OMEGA STD	41
5.11	Centerline Velocity Decay	42
5.12	Experimental Results and Discussion	48
5.12.1	Center line pressure decay	49
5.12.2	Shadow graph images	54
6	CONCLUSION	55
	REFERENCES	56

LIST OF TABLES

TABLE NO.	TITLE	PAGE NO.
4.1	Nozzle configurations	27
5.1	Turbulence Model Comparison for Computational Analysis	32
5.2	Potential Core Length for all models	43
5.3	Comparison Of Potential Core Length	52

LIST OF FIGURES

FIGURE NO.	TITLE	PAGE NO.
1.1	Sketch of a circular turbulent jet [2]	2
4.1	The perspective view of the computational setup	23
4.2	The computational domain after the volume mesh generation	26
4.3	Experimental Jet Setup	29
4.4	Shadow graph System arrangement	30
5.1	Cut planes of the major axis, $X/De=1$, $X/De=5$ and $X/De=9$	31
5.2	Velocity Contour of 10 mm length nozzle with Diameter Ratio of 2	33
5.3	Velocity Contour of 30 mm length nozzle with Diameter Ratio of 2	33
5.4	Velocity Contour of 50 mm length nozzle with Diameter Ratio of 2	33
5.5	Velocity Contour of 10 mm length nozzle with Diameter Ratio of 3	34
5.6	Velocity Contour of 30 mm length nozzle with Diameter Ratio of 3	34
5.7	Velocity Contour of 50 mm length nozzle with Diameter Ratio of 3	34
5.8	Velocity Contour of 10 mm length nozzle with Diameter Ratio of 4	35
5.9	Velocity Contour of 30 mm length nozzle with Diameter Ratio of 4	35
5.10	Velocity Contour of 50 mm length nozzle with Diameter Ratio of 4	35
5.11	Velocity Contour of 10 mm length nozzle with Diameter Ratio of 2	36

FIGURE NO.	TITLE	PAGE NO.
5.12	Velocity Contour of 30 mm length nozzle with Diameter Ratio of 2	36
5.13	Velocity Contour of 50 mm length nozzle with Diameter Ratio of 2.	36
5.14	Velocity Contour of 10 mm length nozzle with Diameter Ratio of 3	37
5.15	Velocity Contour of 30 mm length nozzle with Diameter Ratio of 3	37
5.16	Velocity Contour of 50 mm length nozzle with Diameter Ratio of 3	37
5.17	Velocity Contour of 10 mm length nozzle with Diameter Ratio of 4	38
5.18	Velocity Contour of 30 mm length nozzle with Diameter Ratio of 4	38
5.19	Velocity Contour of 50 mm length nozzle with Diameter Ratio of 4	38
5.20	Velocity Contour of 10 mm length nozzle with Diameter Ratio of 2	39
5.21	Velocity Contour of 30 mm length nozzle with Diameter Ratio of 2	39
5.22	Velocity Contour of 50 mm length nozzle with Diameter Ratio of 2	39
5.23	Velocity Contour of 10 mm length nozzle with Diameter Ratio of 4	40
5.24	Velocity Contour of 30 mm length nozzle with Diameter Ratio of 4	40
5.25	Velocity Contour of 50 mm length nozzle with Diameter Ratio of 4	40
5.26	Velocity Contour of 10 mm length nozzle with Diameter Ratio of 3	41

FIGURE NO.	TITLE	PAGE NO.
5.27	Velocity Contour of 30 mm length nozzle with Diameter Ratio of 3	41
5.28	Velocity Contour of 50 mm length nozzle with Diameter Ratio of 3	41
5.29	The centerline velocity decay in the jets for all turbulent models	44
5.30	The centerline velocity decay for 10mm length nozzle	44
5.31	The centerline velocity decay for 20mm length nozzle	45
5.32	The centerline velocity decay for 30 mm length nozzle	45
5.33	The centerline velocity decay for 40 mm length nozzle	46
5.34	The centerline velocity decay for 50 mm length nozzle	46
5.35	The centerline velocity decay for 60 mm length nozzle	47
5.36	The centerline velocity decay in the different turbulent models by Iovaraju et al	48
5.37	The centerline velocity decay of Length= 10mm nozzle (DR= 3)	50
5.38	The centerline velocity decay of Length= 20mm nozzle (DR= 3)	50
5.39	The centerline velocity decay of Length= 30mm nozzle (DR= 3)	51
5.40	Decay of various length at NPR 1.1- EXPERIMENTAL RESULTS	51
5.4.1	NPR of shadow graph images	54

LIST OF SYMBOLS AND ABBEREVIATIONS

Ae	Axial Depth Of Cut (Mm)
Vc	Cutting Speed (Mm/Sec)
D	Diameter (Mm)
F _n	Feed Per Revolution (Mm/Rev)
L	Length Of Cut (Mm)
T _c	Machining Time (Sec)
MRR	Metal Removal Rate (Q)
V _c	Penetration Rate (Mm/Sec)
A _p	Radial Depth Of Cut (Mm)
N	Spindle Speed (Rpm)
R _a	Surface Roughness Factor
F _x	Tangential Force (N)
F _t	Thrust Force (N)
M _c	Torque (N/M)
CAA	Civil Aviation Authority
DNS	Direct Numerical Simulation
LES	Large Eddy Simulation
RANS	Reynolds Averaged Navier Stokes Equation
FW-H	Ffowcs Williams Hawkings
SPL	Sound Pressure Level
SGS	Sub Grid-Scale
SST	Shear Stress Transport
ASM	Algebraic Stress Model
SIMPLE	Semi Implicit Method for Pressure Linked Equation

WIKA	Werner Wirth GmbH & Co.KG
PCL	Potential Core Length
DR	Diameter Ratio
U_o	Free Stream Velocity
ρ	Density
u	Velocity
f	External Force
I	Identity Tensor
T	Viscous Stress Tensor
u_g	Grid Velocity
a	Face Area Vector
f	Face Quantity
σ	Stress Tensor

CHAPTER 1

INTRODUCTION

Jet flows are widely used in many engineering applications such as gas turbine combustors, engine injectors, water jets, and hydraulic drilling. In most of these engineering applications, main purposes are mixing and mass entrainment enhancement in the flow domain. The efficiency of all those applications is highly dependent on flow properties of the jet flow. In this thesis, novel non-circular nozzles are studied to improve mass entrainment rates of non-circular nozzles.

Axisymmetric jets are the most common jet applications because they are easy and convenient to produce, and they provide stable flows due to their smooth and straightforward geometries. In the past few decades, researchers have begun to look for alternative jet designs that are more suitable than a circular jet for particular applications including combustion, mixing, and cleaning [1]. The search for alternative jet designs in hopes of finding better jet designs eventually led them to investigate non-circular jet flows.

Many researchers reported that non-circular jet flows showed better mixing and mass entrainment over circular jets [13, 15, 23]. Experimental studies suggested that enhancement of mixing and mass entrainment is related to the azimuthal distortion caused by the sharp edges of non-circular jets [6, 8].

It is highly important to understand the flow properties of a jet flow in order to find new ways of control and improvement over common jet configurations.

1.1 CIRCULAR JETS

Circular jets are the most commonly used free shear flows that are formed when the fluid emerges from a circular outlet. Circular jets are often referred to as axisymmetric jets. The axisymmetric jets have been investigated

comprehensively to understand the behaviors of the jet after emerging from an outlet.

Experimental studies on mean velocity fields in circular jets show that in the stream-wise direction, the jet flow could be divided into two different regions where the flow has similar properties [2].

- **Potential core region**, which is also called flow development region. In the potential core region, exit velocity does not diminish. The region has a cone-like shape where the exit velocity has the same magnitude everywhere within the potential core.
- **Fully developed region**, in which the centerline velocity starts to diminish as the jet propagates downstream. The jet interacts with the surrounding fluid and momentum exchanges in the mixing layer. As a result of this exchange, more surrounding fluid is entrained and the jet becomes wider, as the exit velocity diminishes.

A simple sketch of a circular jet is shown in Fig 1.1 to illustrate how a circular jet propagates after emerging from an outlet.

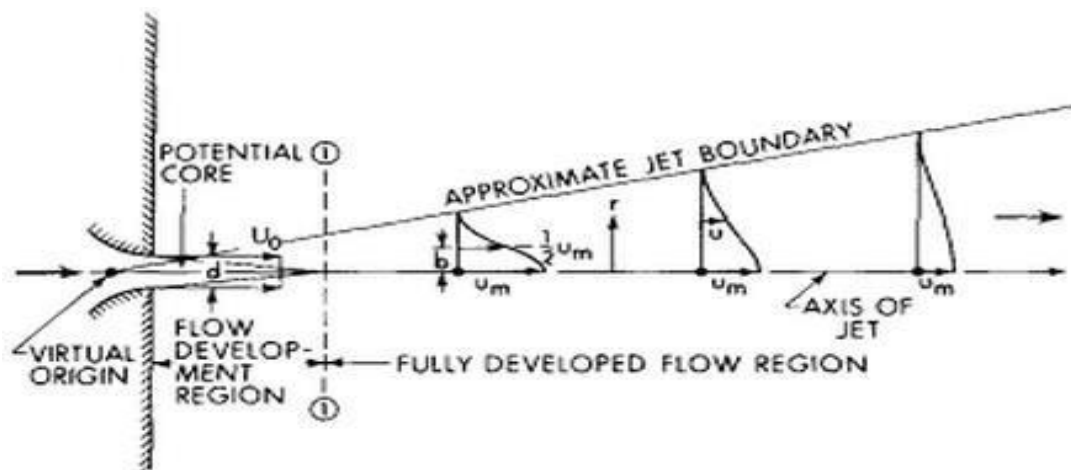


Fig:1.1 : Sketch of a circular turbulent jet

As can be seen from Fig 1.1, the exit velocity, U_0 , is preserved until the axial section 1-1, where the flow development region ends.

1.2 TURBULENT JETS

Turbulent jets are free shear flows driven by a source of momentum and emerging from an orifice or a nozzle into surrounding fluid. As soon as the jet exits the outlet, the fluid penetrates into ambient fluid, and a shear layer is created between the jet and ambient fluid. The shear layer causes the mixing process between the jet and surrounding fluid. The jet entrains surrounding fluid and propagates downstream. As the jet moves downstream, it entrains more fluid, and the mixing region becomes wider. The centerline velocity starts to decay due to momentum exchange in the shear layer.

1.3 FLOW CONTROL

The role of vortex dynamics (large and small coherent structures) and vortex induction in mass entrainment are investigated by many researchers [6, 16]. Manipulating the flow field is the process to control the jet. Controlling of the jet provides perfectly excited flow for engineering applications such as combustion chambers where both small and large scale mixing are needed, and also the aircraft jets where noise reduction is necessary. Flow control can be achieved in two different ways.

- Active Control
- Passive Control

Active control is adapting the jet flow properties by forcing the flow externally. For active flow control, finding an optimum excitation is very important. The excitation mode, the frequency and the amplitude are the primary parameters that have to be set to precise values. An analyst should decide what frequency and level are needed for the purposes of excited jet.

Another way of controlling the flow is passive control. In passive control, the jet is not forced externally to change the flow patterns. Instead, the passive control can be achieved directly by changing the geometry such as the contraction profiles, tabs or internal grooves, and notched edges.

Rather than forcing the flow mechanically or acoustically, it is

preferable to modify the nozzle passively to control the flow. It is also generally cheaper to adjust the nozzle geometry to control the flow.

Gutmark and Grinstein suggested that non-circular jets could be a powerful passive control technique to manipulate the flow field. They argued that the flow properties of non-circular jets mainly depend on the shape of an outlet. Hence, they modified the outlet shape of non-circular nozzle in different ways such as elongated nozzle and sharp-edged nozzle. The modification they made in nozzle enabled them to control the flow properties of the jet at very low-cost. They suggested that controlled non-circular jets could be used at many engineering applications such as jet noise reduction, impinging jet heat transfer, combustion, and mixing control [25].

New [27] used notched nozzles to study the influences of the nozzle outlet geometry on mean flow behavior of axisymmetric turbulent jets. He reported the flow differences observed between V- and A- notches when the high aspect ratios were used. Also, he reported that severe flow distortions occurred when relatively sharper nozzles were used.

Much effort was spent to develop efficient ways of controlling the small and large structures actively or passively to augment mixing and increase mass entrainment, and much more effort will be spent in near future.

1.4 OBJECTIVES

The objective of this thesis is to study the flow properties of different circular nozzles to find the best nozzle design for better mixing and higher mass entrainment rates. In this study, circular nozzles are modified geometrically and the effects of those modifications on improving the flow properties are studied.

The implemented literature study is described briefly in Chapter 2. The aim and scope of the study is explained in chapter 3. The process of how the model is built and the model parameters are selected is explained. The nozzle geometries and the computational domain are defined in Chapter 4. In

Chapter 5, discussion on the important flow properties which are indicators of higher mass entrainment rates. Finally, in Chapter 6, conclusion from the results and give the concluding remarks on the mixing capabilities of circular jets.

CHAPTER 2

LITERATURE REVIEW

Numerical solutions Aircraft noise is a serious environmental concern for people living in the vicinity of the airports. The dominant source of aircraft noise especially during takeoff is the propelling jet. Jet mixing noise in aircrafts can cause permanent hearing damage to human ear. The reduction of jet noise has become a topic of ongoing intense research due to stringent noise regulations around airports. A review of the available literature on experimental and numerical investigation of compressible subsonic jet for the jet aerodynamics and aeroacoustics is presented in this chapter. The review is based on different parameters like the inflow and boundary conditions, the simulation method used etc. The field of aeroacoustics have begun with the first theory on jet noise by Sir James Lighthill (Lighthill, 1952). This theory was developed to predict the intensity of noise produced by turbulent jets. To describe the acoustic waves, Lighthill reformulated the equations of fluid motion as a wave equation and is now referred to as Lighthill's acoustic analogy. An experimental investigation to establish the relationship between instability waves and jet noise was conducted by Moore (1977). It was observed that the instability waves were not a direct source of noise but in a subsonic jet it modifies the downstream turbulence. Thus it was concluded that the accurate prediction of downstream turbulence could lead to an accurate jet noise model. Then the focus of research had been placed on flow characteristics that are directly influencing the jet flow development and mixing process and thereby the jet noise produced. This was possible either through changing the nozzle conditions or by influencing the instability waves in the shear layer. Boersma et al. (1998) confirmed that altering the nozzle condition can not only control the mixing region but can influence the turbulent properties downstream of the initial mixing region. Thus new concepts and designs in the nozzle geometry were established as methods for jet noise reduction. Computational aeroacoustics (CAA) has undergone spectacular progress with the rapid development of computer resources during the last

three decades. Direct Numerical Simulation (DNS), Large Eddy Simulation (LES) and

Unsteady RANS simulations are the simulation methods generally used in computational aeroacoustics. In these methods, the acoustic information can be obtained directly by solving the compressible flow equations or in a secondary step. Various methods like FW-H analogy (Di Francescantonio, 1997; Ffowcs Williams and Hawkings, 1969) and the Kirchhoff formulation (Lyrintzis, 1994) have been developed for compressible simulations which extend the acoustic waves outside the simulation domain. Several numerical studies have been carried out in Computational Aero Acoustics (CAA) to investigate the sound radiated from a jet with different initial conditions like nozzle geometries, Reynold's number, temperature conditions etc. Many of the previous works show that the jet mixing aero dynamics is highly dependent on the nozzle exit shape (Ho and Gutmark, 1987; Gutmark et al., 1989; Hussain and Husain, 1989; Quinn, 1989; Miller et al., 1995). Devices like chevrons, bevel nozzles, lobed mixers, micro jets etc. has been established as a method for jet noise reduction since last few decades. The complex interactions between azimuthal and stream wise vortices resulted in increased entrainment characteristics and better fine-scale mixing of noncircular jets compared to the round nozzle of the same nozzle exit area (Mi et al., 2000). By altering the mean and turbulent flow characteristics through the variation in nozzle geometries a reduction in jet noise can be achieved. Anderson et al. (2003, 2005) investigated the flow field and the radiated sound of an isothermal Mach 0.75 compressible jet with Reynolds number 50000 using Large Eddy Simulation and Kirchhoff surface integration. The Favre filtered Navier Stokes Equations were solved using cell centered finite volume method solver. For the solution of convective fluxes a low dissipation third order upwind scheme and for the viscous fluxes, a second order centered difference approach was applied. A three stage second order Runge-Kutta method was used for time marching. The sub grid scale stresses were computed using Smagorinsky's sub grid scale model. The computational domain contained boundary fitted block structured mesh with a total of 3

million cells. At the inlet of the nozzle, total pressure and total enthalpy were specified and at outlet static pressure was specified. At all free boundaries, absorbing boundary conditions were used. Sound pressure levels in the far-field were evaluated with a hybrid approach. Kirchhoff surface integration was utilized for the propagation of sound to far-field locations. Instantaneous pressure on the

Kirchhoff surface obtained from the LES predicted sound pressure levels which were in excellent agreement with measured values. The aerodynamic results and the sound pressure levels were validated against the experimental data reported by Jordan et al. (2002) and were found in good agreement except for the initial jet spreading and the potential core length.

Large Eddy Simulation of two compressible jets, one an isothermal jet and other a hot jet were carried out by Anderson et al. (2004) for a Mach number of 0.75.

It was found that heating of the jets results in a higher mixing and hence a shorter potential core region. Heated jet was found to be narrower than the isothermal jet.

The aerodynamic results and the sound pressure levels were validated against the experimental data reported by Jordan et al. (2002, 2003). Initial jet spreading and potential core length were not predicted correctly owing to the presence of coarse region in the nozzle outlet. A higher Sound Pressure Level (SPL) was obtained with heated jet in both experimental and numerical work. However, the increase was not sufficient to make any conclusion regarding the relation between the temperature ratio and SPL.

Boersma and Lele (1999) carried out the Large Eddy Simulation of a compressible 0.9 Mach jet, one at a moderate Reynolds number 36×10^3 and other at a high Reynolds number 100×10^3

They used a model based on the Boussinesq hypothesis. Navier-Stokes equations were rewritten in a cylindrical coordinate system, with x , r ,

and θ as the axial, radial, and azimuthal coordinate directions, respectively. In the numerical scheme used, spatial derivatives in axial and radial direction were taken with 6th order compact finite differences. In the azimuthal direction a Fourier method with approximate idealising was used. The time integration was carried out with a fourth order Runge-Kutta scheme. The number of grid points used in simulations for moderate Reynolds numbers was less when compared to the fine grids employed for high Reynolds numbers. The centre line velocity, axial velocity, Reynolds stresses, potential core, density profiles, time series of pressure and power spectra were analysed in these range of Reynolds number.

Bogey et al. (2003) carried out a three-dimensional circular jet simulation with a Mach number of 0.9 and Reynolds number of 65000 using LES to study both flow field and acoustic radiation. The validation of the computation is then carried out by comparing both flow properties and the radiated sound field with experimental data available in the literature. The full three-dimensional Navier-Stokes equations written in a conservative form were solved. To compute sound wave with more accuracy a low dispersive and low dissipative numerical algorithm was used. To accommodate different discretization in the flow field and in the acoustic far field a Cartesian and non-uniform mesh was used. Though Large Eddy Simulation (LES) can be performed for simulating flows at higher Reynolds number, by computing only the larger scales and by taking into account the effects of unresolved ones through a sub grid scale model, the Smagorinsky model was applied to make the problem simpler. The physical part of the computational domain was extended axially up to 20 times the radius of the jet for the flow field, and up to 30 times radius for the acoustic field.

The computational domain was discretized with varying mesh sizes, minimum near to the centerline and increasing in the radially outward direction. The simulated flow field showed that the potential core length obtained was consistent with the experimental value. Flow properties, namely flow development, mean flow parameters and turbulent intensities, were in good

agreement with measurements. The sound field provided by LES compared well with experimental results in terms of directivity, spectra and levels. The sound sources in the jet were found around the end of the potential core and were similar to the experimental observations. Most of the experimental studies on the influence of initial conditions on the jet flow have been conducted for jets with Reynolds numbers about $10^5 < Re_D < 5 \times 10^5$. In this range, the exit shear layer is expected to be transitional, but the use of tripping devices in the nozzle can make it go fully turbulent.

Bogey and Bailly (2005) investigated the effect of inflow conditions on the flow development and sound field of 0.9 Mach circular jet with Reynolds number 4×10^5 . The investigated parameters were the forcing amplitude, the initial shear-layer thickness and the use of the first four azimuthal modes when synthesizing the forcing disturbances. The filtered compressible Navier–Stokes equations were solved using highly accurate numerical schemes with low dispersion and low dissipation properties. A 13-point finite difference scheme was used for spatial discretization, whereas an explicit six-stage Runge–Kutta algorithm was applied for time integration. The mean profiles of velocities, pressure, and density were imposed at the inflow boundary. Both the flow development and the emitted sound were shown to depend appreciably on the initial parameters chosen to model the inflow of the transitional jet. Bodony and Lele (2004, 2005) performed Large Eddy Simulation of three cold and hot jets ($T_j/T_\infty = 1.8, 2.3$ and 2.7) at Mach numbers 0.5, 0.9 and 1.5. In the first part of calculation a near-field large-eddy simulation that captures the sound generating turbulent field and the radiated noise, and in the second part a region that continues the radiated sound to distances far away from the jet plume was adopted. The azimuthal derivatives were evaluated using Fourier bases. The acoustic far field was related to the near field through a Kirchhoff surface in the form of a right circular cylinder that encloses the jet. Pressure fluctuations on the surface of the cylinder were collected as functions of time and of the axial and azimuthal directions. Predictions of flow parameter were found to be in good agreement with experimental data. The ability of LES to accurately predict the radiated sound

was found to be strongly dependent on the jet operating conditions. The noise from the heated jet simulations were found to be over-predicted and the unheated jet sound predictions were in better agreement with the experimental data. A high order accurate three dimensional LES code was developed and tested by Uzun et al. (2003) utilizing a robust dynamic Sub Grid-Scale (SGS) model for a turbulent isothermal jet at a Reynolds number 10^5 . A sponge zone that was attached downstream of the physical domain, damps out the disturbances before the spurious waves reach the outflow boundary. To predict the far field noise accurately they coupled the near field data provided by LES with the Ffowcs Williams-Hawkings method for computing noise propagation to the far field. Favre-filtered unsteady, compressible, non-dimensionalized Navier-Stokes equations formulated in curvilinear coordinates were solved. To compute the spatial derivatives at interior grid points away from the boundaries, a non-dissipative sixth-order compact scheme was employed. Implicit spatial filtering was employed to get rid of the high frequency oscillations resulting from the unresolved scales. The mean flow parameters calculated were found to be in good agreement with the experimental results.

Bogey et al., (2007) experimentally investigated a round, isothermal and cold jet of diameter 38 mm and Mach number between 0.6 to 1.6 and $ReD \geq 5 \times 10^5$. Properties of near-field jet noise particularly at 7.5 diameters from the jet centre line were documented which differ appreciably from properties of far-field noise. The results were presented to form a database that can be used for the validation of the acoustic fields determined numerically by compressible Navier-Stokes computations. Tide and Babu (2009a) performed an unsteady RANS simulation with the Shear Stress Transport (SST $k-\omega$) model for cold and hot jets. Nozzle of 50 mm diameter was used to study the baseline round configuration and 6 lobed chevron configurations with 00 and 50 taper angles along the nozzle edge. The computational domain used for the three-dimensional simulation was 1800 semi-circular cross section. The domain extended 50Dj downstream of the nozzle exit, 10Dj at inlet, 20Dj at outlet and the length of the buffer zone was

40Dj. Favre averaged Navier–Stokes equations along with the SST $k-\omega$ turbulence model equations were solved. The nozzle inlet was modelled as a pressure inlet with static and stagnation pressure specified. The rest of the domain inlet and the upper semi-circular entrainment surface were modelled as pressure inlets at the ambient pressure and temperature. The domain outlet was taken to be a pressure outlet at the ambient pressure. The bottom boundary was treated as a symmetry surface. All solid surfaces were treated as adiabatic surfaces and standard wall functions were used. It was found that heating of the jets results in a higher mixing and hence a shorter potential core region. Heated jet was found to be narrower than the isothermal jet. Comparison of mean actual velocity and fluctuating components showed good agreement with the LES results. The effect of chevrons on noise was also investigated by Tide and Babu (2009b) for $M=0.75$ hot and cold jets. Noise predictions using unsteady RANS calculation was improved by applying periodic boundary conditions and improving the near wall resolution. The 3D calculations were carried out on a hexahedral mesh with 1.1 million cells in a 30° pie sector for the baseline round nozzle. Hybrid meshes of 1.5 million cells were used for simulations with chevron lobes in a view to keep the wall y^+ less than 10. Overall sound pressure levels at far-field observer locations were calculated using Ffowcs Williams-Hawkings equation. Numerical prediction of stagnation pressure, stagnation temperature, turbulent kinetic energy and 90° observer spectral trends were compared with experimental data. Freund et al., (2001) computed the mean flow development and the radiated sound of a 0.9 Mach circular jet by using DNS. Karabasov et al., (2010) developed a hybrid methodology for jet noise prediction using optimised modelling and numerical techniques. Acoustic propagation was modelled by solution of a system of adjoint Linear Euler Equations, capturing convective and refraction effects using a spatially developing jet mean flow provided by a RANS CFD solution. Goldstein's acoustic analogy, including a Gaussian function model for the two-point cross-correlation of the fourth-order velocity fluctuations in the acoustic source was used for modelling sound generation. Uzun and Hussaini (2010) carried out

LES for a round nozzle with overset grid capability to compute the jet flow in detail and the effect of inflow conditions on the jet flow field and far field noise. Simulation of a baseline round nozzle for the flow inside the nozzle and the free jet flow outside were performed simultaneously by a high-order accurate, multi-block, large eddy simulation (LES) code with overset grid capability. For the round nozzle jet the governing equations were solved on a very fine mesh with approximately 370 million grid points. The application of Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES) are more feasible to jet noise prediction methodologies with the recent improvements in the processing speed of computers. The first DNS of a turbulent jet was carried out by Freund et al (2000) for a supersonic jet at Mach number 1.92 with Reynolds number 2000. At similar convective Mach numbers, the computed overall sound pressure levels were found to be in good agreement with experimental data. Freund (2001) simulated a Mach 0.9 turbulent jet using 25.6 million grid points with a Reynolds number of 3,600. The predictions of mean flow and radiated sound showed excellent agreement with experiment data provided by Stormberg (1980). Moore (2007) performed DNS of a low Reynolds number jet at three temperature ratios. Analysis of turbulent kinetic energy contours showed that heating of the jet can reduce jet noise. Even though the results proved DNS to be best choice for jet noise problem, it still restricted to low Reynolds number and simple geometries. This is because of the wide range of length and time scales which necessitates high resolution requirements that are far beyond the reach of computational facilities available. Since most of problems with practical interest have high Reynolds number, an alternative method known as Large Eddy Simulation (LES) was developed. Large Eddy Simulation has been used extensively to compute the flow field of turbulent jets. LES is having less computational cost compared to DNS and suitable for doing simulations for problems of practical interest. In LES the large scales are directly resolved and the effect of the small scales or the sub grid scales on the large scales are modelled. Many successful LES computations for different types of flows have been performed to date by various researchers [Boersma and Lele, 1999; Anderson et al. (2003, 2004,

2005); Bogey et al. (2001, 2003, 2007); Bogey and Bailly, 2005; Bodony and Lele (2004, 2005); Lew et al., 2005; Shur et al., 2007; Rembold and Kleiser, 2004; Uzun and Hussaini, 2010; Tucker, 2008; Viswanathan et al. (2005, 2008)]. One of the first attempts in using LES as a tool for jet noise prediction was carried out by Mankbadi et al. (1994). A high order numerical scheme was employed to perform LES of a supersonic jet flow. The simulation captured the time- dependent flow structure and Lighthill's theory was applied to calculate the far field noise. De Bonis (2006) made a review on the current state of Large Eddy Simulation (LES) for nozzle flows and reported the issues which currently limit its application. LES requires high order numerical schemes which do not lend themselves well to complicated grid topologies. Although Reynolds Averaged Navier Stokes (RANS) calculations greatly depend on turbulence models to model all relevant scales of turbulence, the application of RANS based methods to jet noise prediction is also a subject of ongoing research (Khavaran, 1999; Hunter, 2003). The mean flow properties provided by a RANS solver is used for the noise predictions. Since noise generation is a multiscale problem that involves a wide range of length and time scales, it appears the success of RANS based prediction methods will remain limited unless excellent turbulence models capable of accurately modelling a wide range of turbulence scales are developed and implemented into existing RANS solvers. Nallasami (1999) presented a review of turbulence models that are significant for the jet noise computations. Flow solutions obtained with $k-\epsilon$ model, algebraic Reynold's stress model and Reynold's stress transport equations were reviewed. Algebraic stress model (ASM) and Reynolds stress transport model were implemented in flow solvers to compute the Reynolds stress components. Three dimensional Navier-Stokes code PAB3D was developed by Pao and Hamid (1996) to obtain numerical solutions to the Reynolds averaged Navier-Stokes equations in three-dimensional spatial domain. A unified method for subsonic and supersonic jet analysis was reported. The Navier-Stokes code was used to obtain solutions for axisymmetric jets with on-design operating conditions at Mach numbers ranging from 0.6 to 3.0, supersonic jets containing weak shocks and Mach

disks and supersonic jets with non-axisymmetric nozzle exit geometries. Even though the jet flow contains a variety of complex flow physics, the RANS calculations require only the initial and boundary conditions to be specified. Detailed flow physics developments in the jet were predicted well by the RANS based methods. A more accurate aerodynamic and aeroacoustics prediction tool for high speed high temperature 3D jet flows was provided by Hamid et al. (2003) making modifications to the k- ϵ model. A new temperature gradient correction to the eddy viscosity term was introduced and validated which proved to be a great promise in predicting high and low temperature jet flows. Hamid et al. (2006) reviewed the computational methods, code implementation and computed results for a wide range of nozzle configurations at various operating conditions and made comparisons with the available experimental data. The capabilities of numerical methods applied to compute the underlying turbulent jet fluid dynamics were overviewed by Georgiadis and De Bonis (2006). The article was primarily divided between RANS modelling and unsteady modelling methods like LES. The comparison between RANS based models and LES using practical examples reveal certain issues in flow simulations. Unlike LES, RANS based models can provide only time-averaged information. However, it can easily capture the trends measured in experiments for different nozzle configurations and flow conditions with lesser computational resources. It is obvious that RANS based models are capable of fast evaluation of new nozzle designs and hence immensely used by the designers and manufacturers for better shapes and profiles.

SUMMARY

1. Among the different methods employed by researchers to calculate the flow field and the radiated sound of jets emanating from nozzles, LES appears to be the most preferred one. The small scales of turbulence can be accurately captured using LES, only if proper grid resolution is employed.
2. LES methods require high-order numerical schemes and high-resolution mesh which do not lend themselves well to complicated grid topologies. LES is computationally very expensive which in turn make it difficult to utilize for the fast evaluation of new designs.
3. Even though RANS based methods could only provide time-averaged turbulent information it can capture the trends exhibited by experiments well while altering the geometry or flow conditions of the nozzles. This statement coupled with the fact that RANS solutions are relatively computationally less expensive indicates that RANS based methods still have a place for better nozzle configurations.

CHAPTER 3

AIM AND SCOPE OF THE STUDY

3.1 AIM OF THE STUDY

1. To study the Mixing Characteristics of a Circular Jet in a convergent nozzle in under-expanded condition by varying its length and the diameter ratio through computational studies.
2. To validate the computational results of the optimized model with its experimental investigation.

3.2 SCOPE OF THE STUDY

1. Jet flow studies are mainly motivated by two considerations. In many practical situations enhanced mixing of jet fluid with the ambient fluid is desirable.
2. By suitably modifying or controlling the initial development of the jet shear layer, desirable mixing and acoustic characteristics of the jet can be achieved.

CHAPTER 4

METHODOLOGY OF COMPUTATIONAL STUDIES

Numerical solutions presented in this thesis are obtained by using Star CCM+. The Star CCM+ Computational Fluid Dynamics (CFD) solver is one of the most powerful and commercially available CFD solver for engineering applications. Nozzle geometries are prepared by using SolidWorks 2013 and then imported into Star CCM+ as parasolid files.

4.1 FINITE VOLUME METHOD

The Star CCM+ solver uses the finite volume method, which is a numerical method that divides the problem domain into finite number of volumes and evaluates conservation laws (conservation of mass and momentum) for every finite volume. These volumes are called cells of a computational grid. The next step is to find discrete versions of conservation laws in integral form for every control volume in the domain. The main objective is to obtain a set of linear equations corresponding to the number of cells in the grid [9].

4.2 GOVERNING EQUATIONS

In this thesis, the governing equations are Navier-Stokes equations of incompressible flows. Navier-Stokes equations can be written in tensor form

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} - \mathbf{f} \right) - \nabla \cdot \boldsymbol{\sigma} = 0, \quad (4.1)$$

$$\nabla \cdot \mathbf{u} = 0, \quad (4.2)$$

where ρ , $\mathbf{u}$, and $\mathbf{f}$ are the density, velocity, and external force, respectively. The stress tensor is denoted by $\boldsymbol{\sigma}$ and represented as

$$\boldsymbol{\sigma} (p, \mathbf{u}) = -p\mathbf{I} + \mathbf{T}, \quad (4.3)$$

where $\mathbf{I}$ is the identity tensor and $\mathbf{T}$ is the viscous stress tensor which is also called as the deviatoric part of the stress tensor and p is the pressure. In Star CCM+, the finite volume discretization starts with continuous integral form of Navier-Stokes equations of incompressible flow for continuity and momentum

$$\frac{\partial}{\partial t} \int_V \rho \mathbf{u} dV + \oint_A \rho \mathbf{u} (\mathbf{u} - \mathbf{u}_g) \cdot d\mathbf{a} = - \oint_A p \mathbf{I} \cdot d\mathbf{a} + \oint_A \mathbf{T} \cdot d\mathbf{a} + \int_V \mathbf{f} dV, \quad (4.4)$$

where $\mathbf{u}_g$ is the grid velocity and applicable only if the mesh is moving. The terms on left-hand side of the equation (4.4) are the transient term and the convection term. The terms on right-hand side are the pressure gradient term, the viscous flux, and the external force term. To obtain a discrete form for a single control volume cell-0, above equation is applied to a reference volume cell-0

$$\frac{\partial}{\partial t} (\rho \mathbf{u} V)_0 + \sum_f [\rho \mathbf{u} (\mathbf{u} - \mathbf{u}_g) \cdot \mathbf{a}]_f = - \sum_f (p \mathbf{I} \cdot \mathbf{a})_f + \sum_f \mathbf{T} \cdot \mathbf{a}, \quad (4.5)$$

where $\mathbf{a}$ is the face area vector, and the subscript f indicates the face quantity.

The transient term occurs in Navier-Stokes equations if the problem is time- dependent. It must be included in governing equations. There are several ways to solve transient simulations in STAR CCM+. The transient solver that is used in this thesis is an implicit unsteady solver with a first-order temporal discretization scheme. This scheme is referred as backward Euler method.

$$\frac{d}{dt} (\rho \mathbf{u} V)_0 = \frac{(\rho_0 \mathbf{u}_0)^{n+1} - (\rho_0 \mathbf{u}_0)^n}{\Delta t} V_0. \quad (4.6)$$

In (4.6), the transient term is solved at both current time step "n+1" and previous time step "n". The backward Euler method provides unconditional stability, but solving equations at both current and previous time step increases the computational expenses.

Another important term appears in (4.4) is the convection term. To obtain more accurate and stable numerical solutions to physical problems, it is important to discretize the convection term properly. There are several discretization schemes available in Star CCM+. It should be noted that when working with Reynolds-Averaged Turbulence Models only first and second-order upwind schemes are available in Star CCM+.

4.3 SEGREGATED FLOW MODEL

There are two unsteady flow solvers available in Star CCM+, namely coupled flow solver and segregated flow solver. Coupled flow model solves conservation equations for mass and momentum simultaneously using a time-marching approach. The coupled flow solver is mainly used in compressible flows, natural convection problems, and the flows with large body forces [9].

The segregated flow model solves the flow equations in a segregated manner. A predictor-corrector approach is used to link momentum and continuity equations, which are solved separately. The segregated flow model offers a good compromise between accuracy and computation time for constant density flows. It is also suitable to use segregated flow model for mildly compressible flows. It is highly recommended to use segregated flow where the computational resources are an issue and the physics of the problem is suitable to use segregated flow [9].

There are several algorithms to couple pressure-velocity and derive equations for the pressure from continuity and momentum equations. The default algorithm in Star-CCM+ is "Semi Implicit Method for Pressure Linked Equations (SIMPLE)" algorithm. SIMPLE is not only the default algorithm for the solver, but also the most widely used one to solve segregated flow model. SIMPLE algorithm was introduced by Patankar and Spalding. In SIMPLE algorithm, initial pressure field is defined by using boundary conditions. The gradients of velocity and pressure are computed according to the initial pressure field. The discretized momentum equations are solved to compute intermediate velocity field. The face mass fluxes are checked in every cell to

see if continuity equation is satisfied for every cell. To satisfy continuity equation, the pressure correction is applied and the pressure field is updated by using new corrected pressure values. The face mass fluxes and cell velocities are corrected with respect to updated pressure field. These fundamental steps can be repeated as long as the convergence criteria are satisfied.

4.4. TURBULENCE MODELING

Turbulence is a very complicated physical behavior of the fluid. Although this behavior of turbulent flow is mostly understood today, it is still difficult to approximate a fairly complicated phenomenon because approximation process causes loss of some data in the numerical solution. The process of introducing the lost data is known as "Turbulence Modeling." There are three most common approaches for modeling turbulence that are available in STAR CCM+

- Reynolds- Averaged Navier-Stokes (RANS)
- Large Eddy Simulation (LES)
- Detached Eddy Simulation (DES)

Reynolds-Averaged Navier-Stokes (RANS) equations are divided into three main groups. The number of additional transport equations that is solved in the turbulence model classifies the turbulence models. These models are one-equation, two-equation, and second-moment closure models [10].

4.4.1 Two-equation models

Two-equation models are the most widely used equation models in engineering applications. They provide a good compromise between computational cost and accuracy. They are also well suited to industrial-type applications [9].

The two most commonly preferred turbulence models among two-equation models are the standard k - ϵ and the k - ω model. They are both two-

equation models in which transport equations are solved for the turbulent kinetic energy k and either the dissipation rate ε or the specific dissipation ω [9]. In order to show the capabilities of these two commonly used models in predicting jet flow properties, Berg et al. [5] studied the performance of the k - ε and the k - ω turbulence models to see which model gives the best result that is consistent with the experimental results of turbulent free jet. In their study, the k - ε turbulence model succeeded in predicting the main features of the flow better than the k - ω model.

4.4.2 The realizable k - ε turbulence model

Although the standard k - ε model predicts the jet flow properties better than k - ω turbulence model, it is reported that the standard k - ε turbulence model still does not provide consistent results with the experimental studies for the plane and the round jets [11]. It is also reported that although standard k - ε model demonstrates excellent performance when predicting the flow properties of confined flows, it does not accurately predict the flow properties of unconfined flows, where higher mean shear rates are expected. Also, standard k - ε overpredicts the spreading rate of the axisymmetric jet [28].

To overcome these weaknesses of standard k - ε model in predicting the spreading of jet and the flow properties of unconfined flows, the realizable k - ε turbulence model was developed by Shih et al. [12]. In this turbulence model, they suggested a new transport equation for turbulent dissipation rate ε . Also, unlike the standard k - ε turbulence model, the model coefficient, C_μ , is no longer defined as a constant in this new model. Instead, this coefficient is expressed as a function of mean flow and turbulence properties. The idea of a variable coefficient is also in agreement with the experimental studies. This new turbulence model satisfies the realizability constraints, which are positivity of normal Reynolds stresses and the Schwarz's inequality for turbulent shear stresses [12].

After carrying out several benchmark computations to test the present model, they reported well-known spreading rate anomaly was completely

removed. In addition, this new dissipation rate equation is expected to enhance numerical stability in turbulent flow conditions.

In conclusion, after investigating the previous computational studies carefully, and because of the reasons listed in this section, the realizable k- ϵ turbulence model is selected for this study.

4.5 BOUNDARY CONDITIONS

Boundary conditions are necessary to introduce initial conditions to the numerical simulation. Boundary conditions also tell the flow solver how to treat fluid flow along these boundaries of the computation domain. In this study, three different boundary conditions are introduced into the flow domain.

- Velocity Inlet
- Pressure Outlet
- Wall

The computational setup and boundary conditions are presented in Fig 4.1.

A 2D constant area single jet model was created with a nozzle length of 60mm, an inner diameter 20mm and an outer diameter 10mm in the GAMBIT platform. Due to symmetry only top half of the model was designed. A rectangular block was created and often it was pronounced as the outer/computational domain.

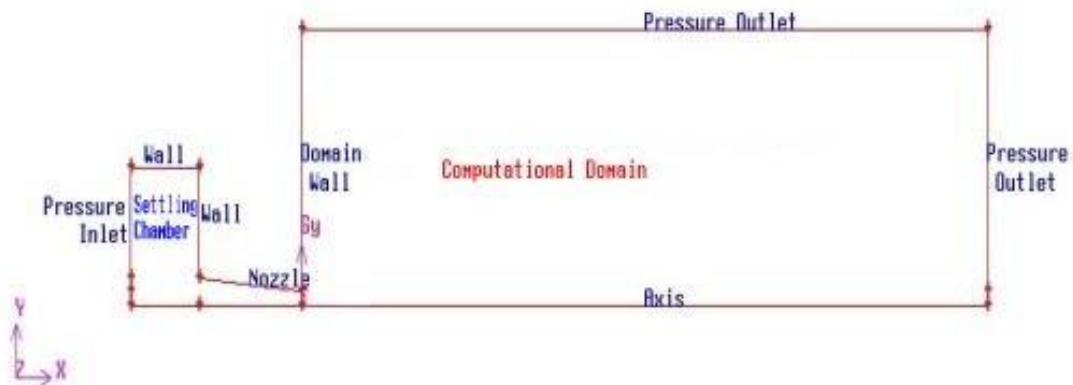


Fig:4.1 : The perspective view of the computational setup

The computational domain consists of a nozzle and a chamber where the jet discharges into ambient air. The computational chamber is cylindrical with a diameter of 1 m and a height of 1.2 m. To make sure that the cylindrical walls were sufficiently far from jet flow, the computation was also carried out with a diameter of 2 m chamber and it is seen that the diameter of 1 m is wide enough not to influence the flow field. It can be seen from Fig 4.1 the GAMBIT model for single free jet with settling chamber, nozzle and the duct. The settling chamber inlet is denoted as pressure inlet, and its walls, nozzle upper wall and domain left side wall are designated to wall boundary condition. The bottom of the domain is designated as axis throughout the length of the domain. The top and right-hand side of the domain are designated as pressure outlet boundary condition.

Assigning of boundary conditions involves identifying the boundary locations (e.g., inlets, walls, axis) and supplying data at the boundaries. The data involved at a boundary depends upon the boundary condition type and the physical models employed.

Poorly defined boundary conditions can have a significant impact on the solution. The primary inlet and the secondary inlet were taken as pressure inlets.

The domain's top and right end was taken as pressure outlet. The duct walls and the left end of the domain were taken as walls. The meshed geometry is to be exported for further analysis. It can be reported as a mesh file or as a neutral file to FLUENT. But generally, it is preferred to export it as a mesh file. Thus, the meshed geometry with the boundary conditions assigned is exported to FLUENT as a mesh file.

The meshed file is imported to FLUENT is read as a case file. The imported grid is checked for skewness and scaled to mm. Once the grid checking and scaling is done, a suitable solver has to be selected. Since the flow is high subsonic and correctly expanded sonic, a compressible, axisymmetric solver is employed for the compressible jet flow symmetric about

its axis. Green gauss cell based gradient option is used. Required solver is selected by selecting solver menu items under the model submenu in define menu option. Material properties can be set and modified in FLUENT for fluids, solids, mixtures, and particles. The fluid used for analysis is air and its properties are set to it by selecting materials under define menu option. The inlet pressure is varied for different flow conditions and the pressure outlet is ambient pressure (101325 N/m²) applied to domain.

Before computing the solution, the solution controls are to be set. The discretized equation which is to be used for solving is selected in the solution control. The discretized conservation equations are solved iteratively and usually a number of iterations are required to reach a converged solution. Convergence is reached when changes in solution variables from iteration to the next iteration are negligible and overall property conservation is achieved. The accuracy of the converged solution depends on accuracy of the physical model, grid resolution and independence. This iteration process for the solution is done by selecting iterate menu item in the solve menu option.

FLUENT (commercial software package for fluid flow predictions) was used to complete this investigation of flows related to co-flowing jets with lip thickness and bypass ratio. The working fluid throughout this investigation was air. Therefore, a robust computational model capable of predicting the effects due to high speed, turbulent, viscous and compressible phenomena is utilized. Formation of the computational grid receives considerable attention when conducting a computational analysis. Grid resolution must be high enough to accurately resolve gradients.

However, CPU memory imposes a limit on the size and resolution of the grid used. Inaccuracies and long solution times can result if the grid is not formed correctly.

4.6 MESH GENERATION

Mesh generation has an important role in dependable and accurate computational solution. An analyst must choose the correct mesh size and type that help to discretize the whole computational domain properly and efficiently. The balance between computational resources and mesh size determines the limitations of numerical simulation. This is because higher mesh density requires more computational resources, which means an increase in the total cost. Therefore, the final mesh must provide necessary conditions to accurately solve the problem, while also staying within reasonable limits in regards to computational cost.

A non-uniform polyhedral mesh was used during simulations. However, a structured mesh was used, as well, where the extruder mesh function was enabled. To capture the essence of the flow properties in the shear layers, the mesh was refined around the centerline of the jet flow. The refined meshes were expanded towards the wall boundaries using a conical-shaped refinement region. The refinement ensured relatively finer mesh in the region where high velocity gradients were expected. The computational domain is presented in Fig 4.2.

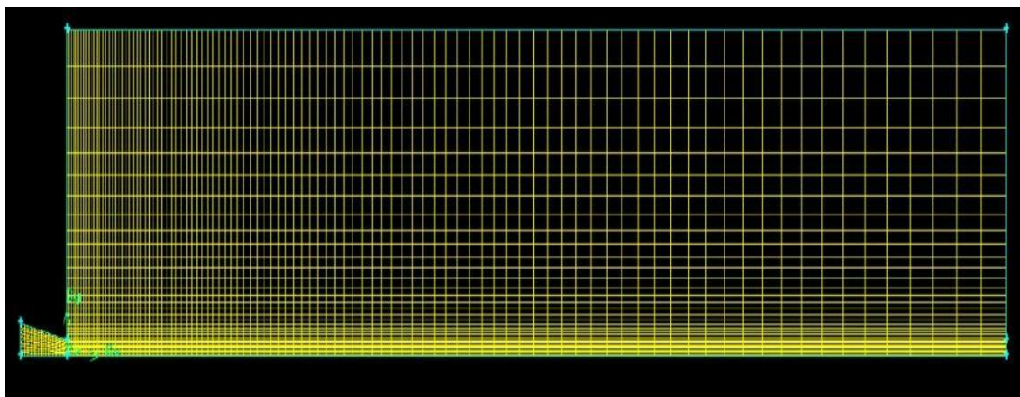


Fig:4.2 : The computational domain after the volume mesh generation

4.7 NOZZLE GEOMETRY

Detailed information regarding nozzle properties is given in Table 4.1 for all nozzle configurations.

Table 4.1 : Nozzle configurations

	INLET DIAMETER (mm)	OUTLET DIAMETER (mm)	LENGTH (mm)	CONVERGENCE ANGLE (in deg)
NOZZLE- 1	20	10	10	26.56
	30	10	10	45.968
	40	10	10	56.81
NOZZLE- 2	20	10	20	14.036
	30	10	20	26.565
	40	10	20	36.87
NOZZLE- 3	20	10	30	9.462
	30	10	30	18.435
	40	10	30	26.565
NOZZLE- 4	20	10	40	7.125
	30	10	40	14.036
	40	10	40	20.556
NOZZLE-5	20	10	50	5.711
	30	10	50	11.31
	40	10	50	16.699
NOZZLE- 6	20	10	60	4.754
	30	10	60	9.452
	40	10	60	14.036

4.8 EXPERIMENTAL SETUP

The experiments of a convergent subsonic jet were conducted at the High-Speed Aerodynamics Laboratory in Chennai, India at Bharath University shown in Fig 4.3. Compressed air was provided to the primary settling chamber from a storage tank via a control valve to achieve the desired Mach number. The isentropic relations were used to set the back pressure for the primary nozzle, while the settling chamber pressure (P_0) was regulated to attain the desired Mach number ranging from 0.4 to 1.0 Mach. The experiment was conducted on the nozzle dimensions of 10,20 and 30mm length of diameter ratio 3.

Firstly, the compressor was switched on and the air was sent into the pressure tank and was stored in it. Then the nozzle was fixed to the high-speed jet facility by using an adapter. WIKA sensor was used to measure the variation of Pitot pressure in the axial directions of the jet at different locations. The sensor has a range of 0–10 bar and an accuracy of $\pm 0.15\%$ full scale after re-zero calibration.

The measurement of total pressure was done using a Pitot tube with inner and outer diameters of 0.4 and 0.6mm respectively. The probe was mounted on a traverse mechanism with a linear translation resolution of 0.1 mm. Then the experiment was started by releasing the air into the settling chamber by rotating the control valve. The pressure values coming out of the jet were calculated by the WIKA sensor using the Pitot tube which was mounted on the traverse mechanism. The pressure values are calculated for every quarter position up to the length of 30 cm by rotating the traverse mechanism in the axial direction of the jet. The experiment was carried out for a while of 16 min and the data that was obtained from the sensor was used for the post-processing work to get accurate results of the total pressure.

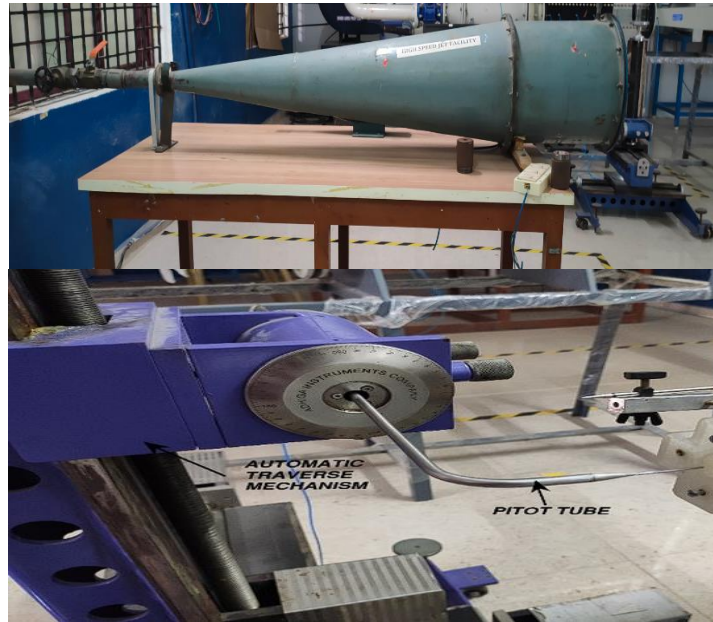


Fig:4.3 : Experimental Jet Setup

4.8.1 Shadowgraph

Shadowgraph is suitable for visualizing the flow field where strong density gradients exist. Sonic under expanded and supersonic (over expanded and under expanded) jet flows produce large density gradients that lead to variations in optical refractive index of the light that passes through the supersonic flow. Shadowgraph system was used in this study for visualizing wave patterns in the jet flow field. The flow pattern was captured using a shadowgraph system with a Helium spark arc light source in conjunction with 150 mm concave mirror. The concave mirror is mounted on a stand. The light source was collimated by the condenser lens and was then brought to the concave mirror. The parallel beam from the mirror was made to pass through the jet flow field and projected on the screen. The photographs of shadowgraph images of shock-train on the screen were taken directly by using a CCD camera. The surface finish of the mirror is $\lambda/6$ and the focal length is 1.6 m. The arrangement of mirror and light source are shown in fig 4.4

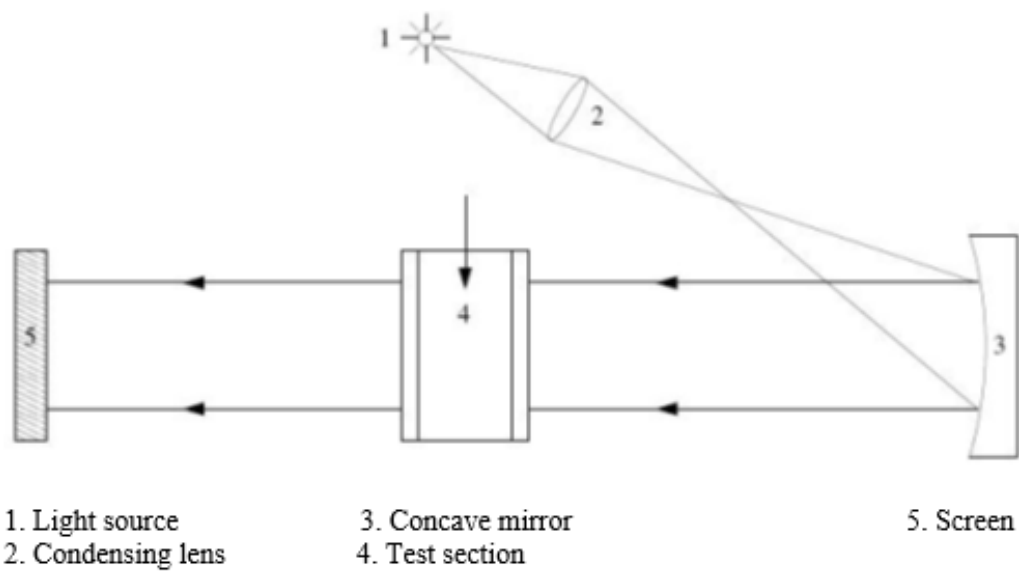


Fig:4.4 : Shadow graph System arrangement

CHAPTER 5

RESULTS AND DISCUSSION

5.1 GRID INDEPENDENCE STUDY

Three sets of grid cells, a coarse grid of about 24,000, a medium grid of about 43,300, and a fine grid of 1,71,800, were used for the grid independence study.

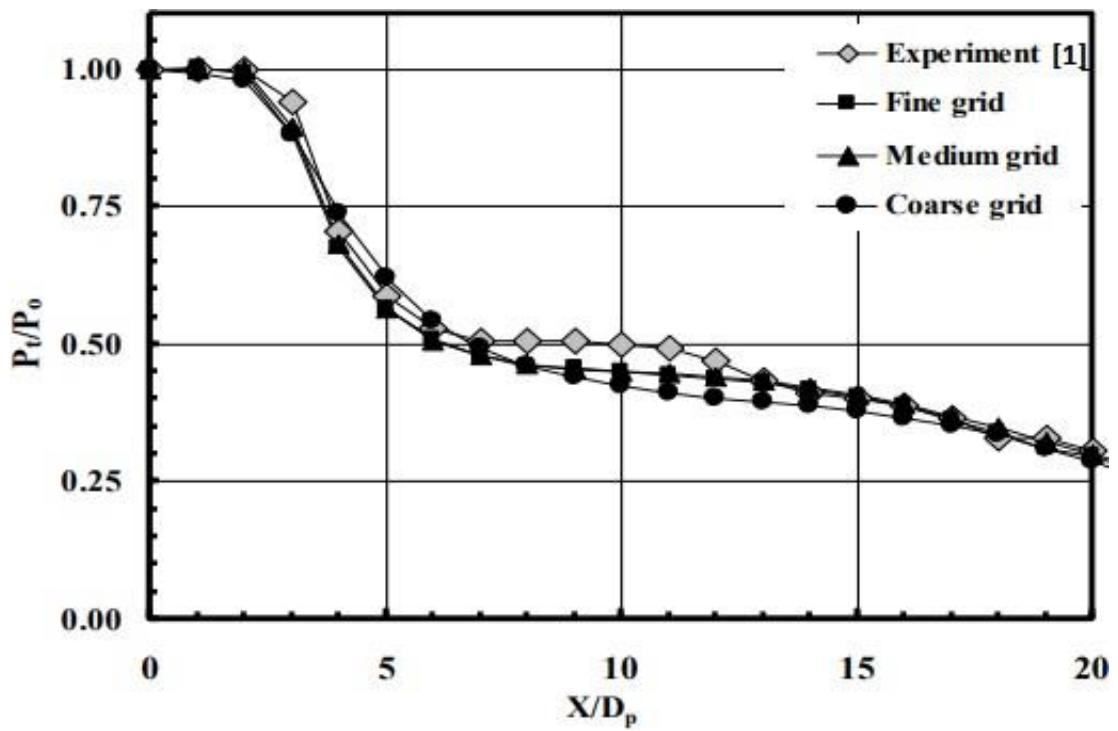


Fig:5.1 : Cut planes of the major axis, $X/De=1$, $X/De=5$ and $X/De=9$

Table 5.1 : Turbulence Model Comparison for Computational Analysis

K Epsilon	Standard
	RNG
	Realizable
K Omega	Standard
	GEKO
	BSL
	SST
Transition K-KL Omega	-
Transition SST	-
Reynolds Stress	Linear Pressure Strain
	Stress - Omega
	Stress - BSL

The SST model employs a k - ϵ formulation in the inner region of the wall boundary layers and switches to a transformed k - ϵ formulation in the outer region of the boundary layers and in the free shear layer/mixing regions.

It should be noted that the intention here is not so much as to make a detailed evaluation of the different models and their different variants in computing the present flow field. Rather, it is to select one that can represent flow among these models.

The primary jet Potential Core Length (PCL) is defined as the axial extension until the primary exit nozzle velocity prevails for subsonic jets (Lovaraju et al., 2011). Mach 0.6, for the seven different turbulence models and for the experiment conducted by lovaraju et al. It is evident that the k - ϵ standard model produces data that agrees well with the experimental data, consistent with its advantage in predicting the flow field characteristics, and the k - ϵ standard model decays slower than the experimental variation in the characteristic decay region. Other turbulence models decay at a slower rate when compared with the experimental decay.

Ansys
2023 R2
STUDENT



5.3 ANALYSIS OF DIAMETER RATIO= 3 WITH TURBULENT MODEL – K EPSILON STD

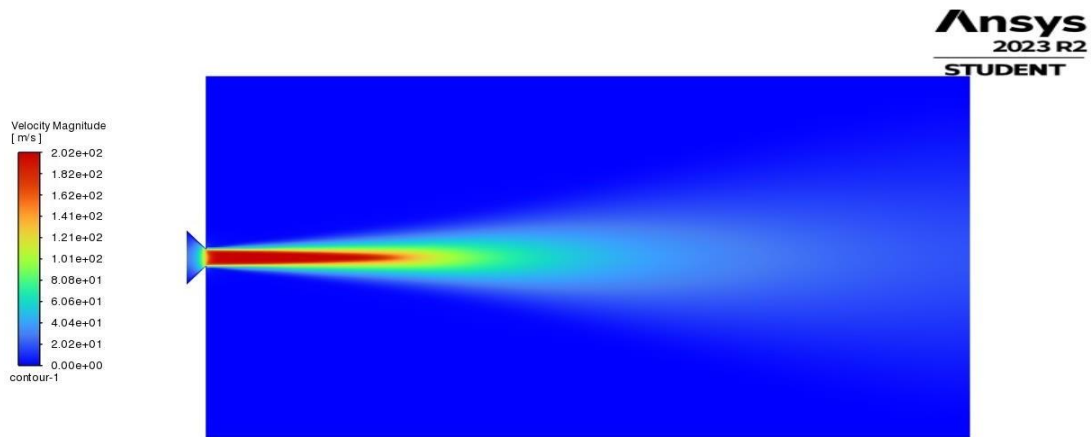


Fig:5.5 : Velocity Contour of 10 mm length nozzle with Diameter Ratio of 3

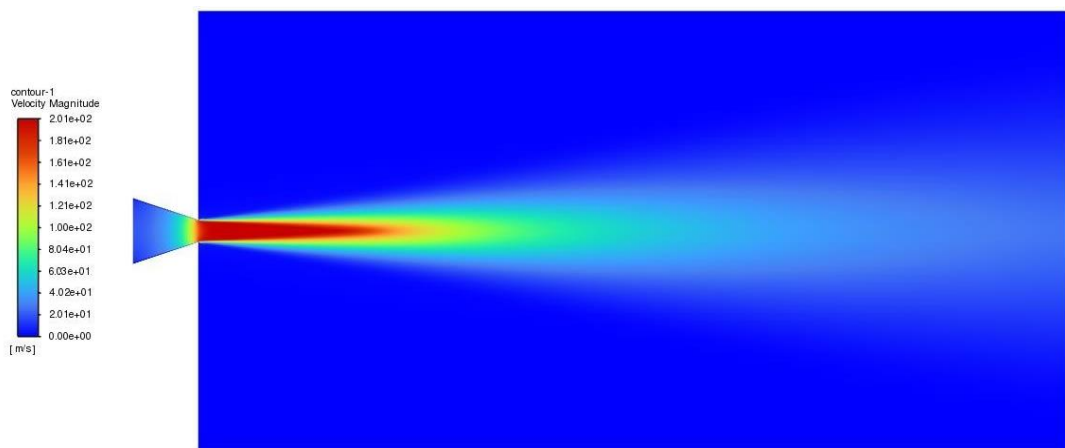


Fig:5.6 : Velocity Contour of 30 mm length nozzle with Diameter Ratio of 3

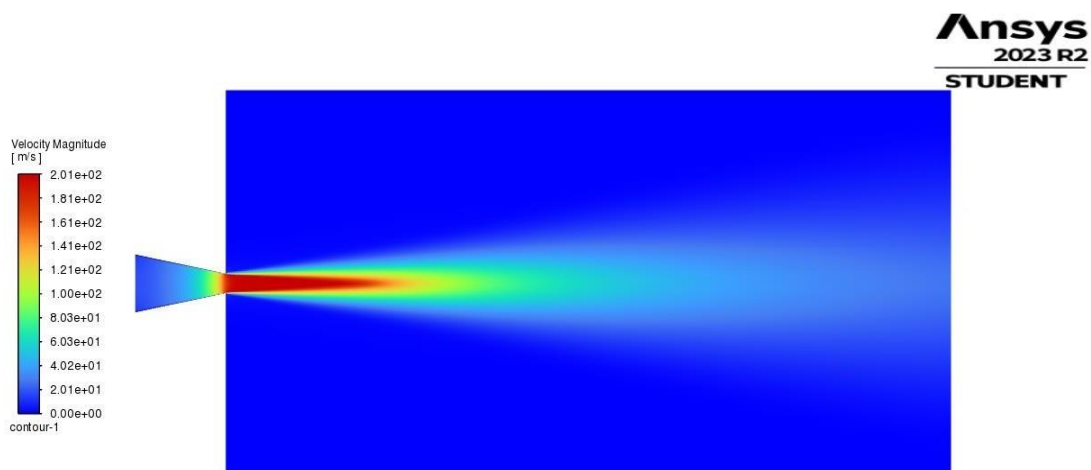


Fig:5.7 : Velocity Contour of 50 mm length nozzle with Diameter Ratio of 3

5.4 ANALYSIS OF DIAMETER RATIO= 4 WITH TURBULENT MODEL – K EPSILON STD

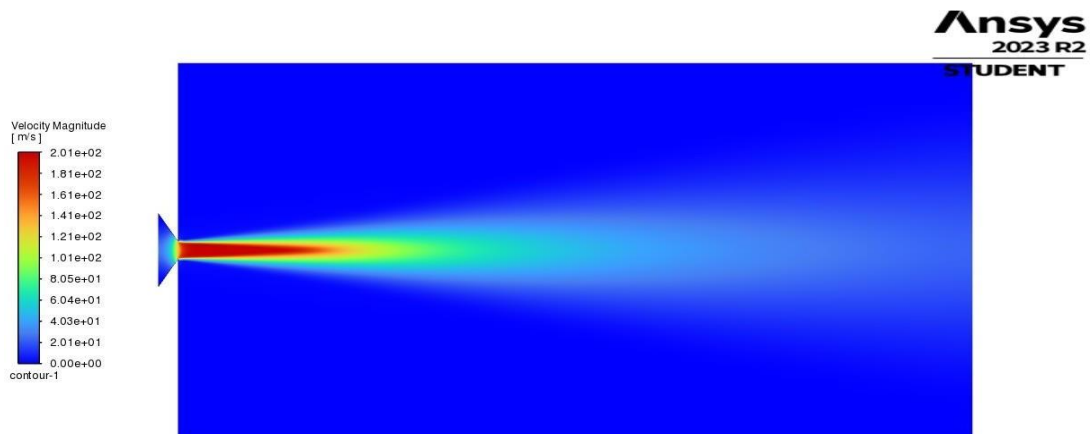


Fig:5.8 : Velocity Contour of 10 mm length nozzle with Diameter Ratio of 4

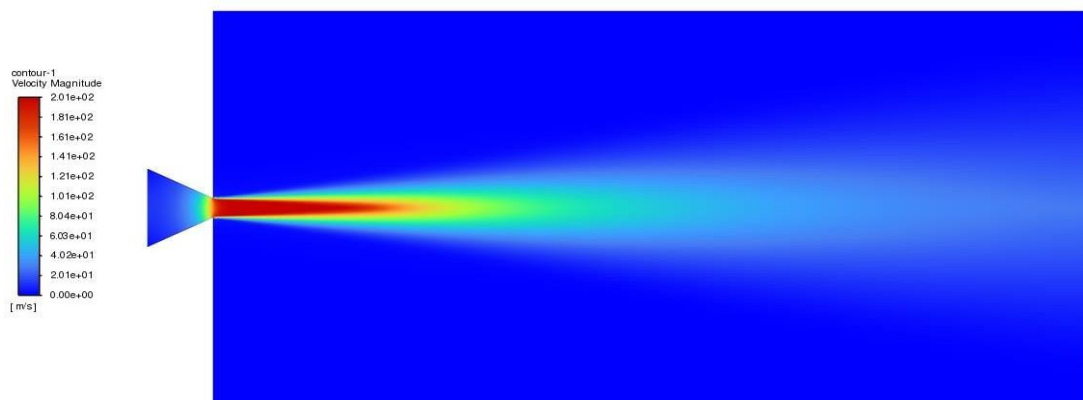


Fig:5.9 : Velocity Contour of 30 mm length nozzle with Diameter Ratio of 4

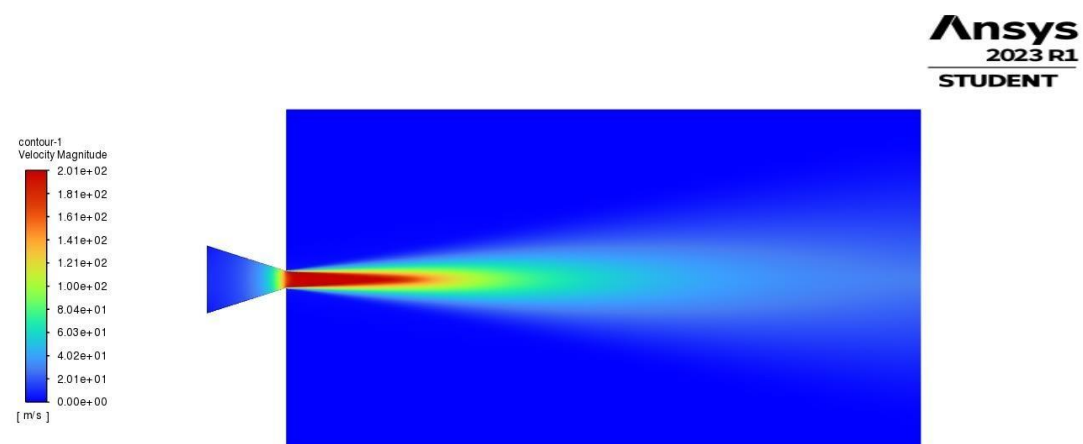


Fig:5.10 : Velocity Contour of 50 mm length nozzle with Diameter Ratio of 4

5.5 ANALYSIS OF DIAMETER RATIO= 2 WITH TURBULENT MODEL – K EPSILON RNG

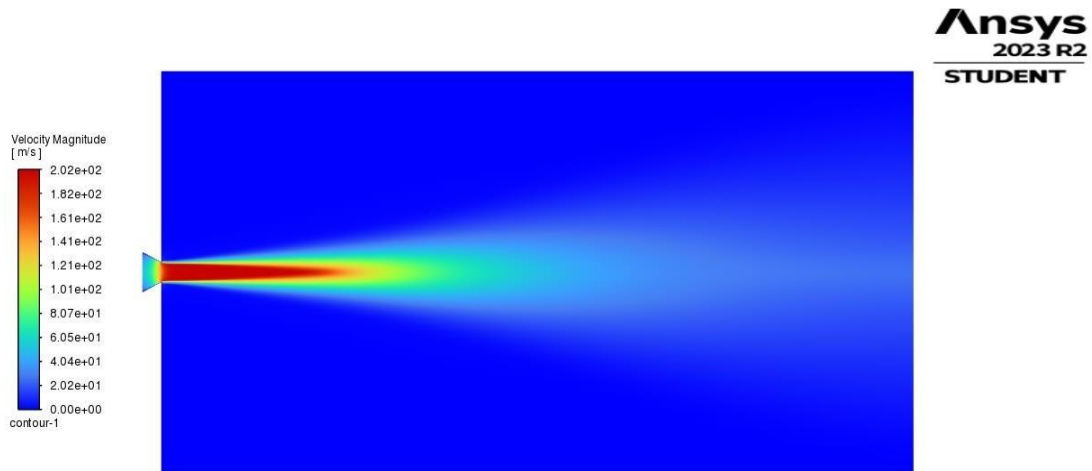


Fig:5.11 : Velocity Contour of 10 mm length nozzle with Diameter Ratio of 2

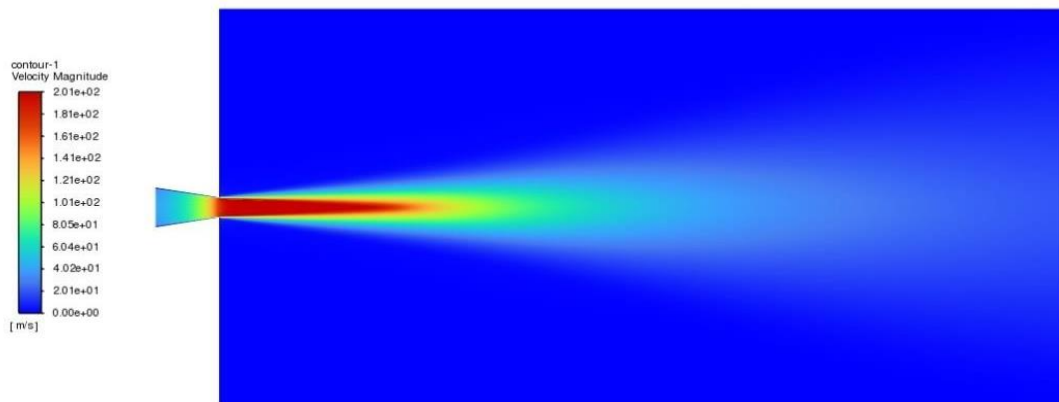


Fig:5.12 : Velocity Contour of 30 mm length nozzle with Diameter Ratio of 2

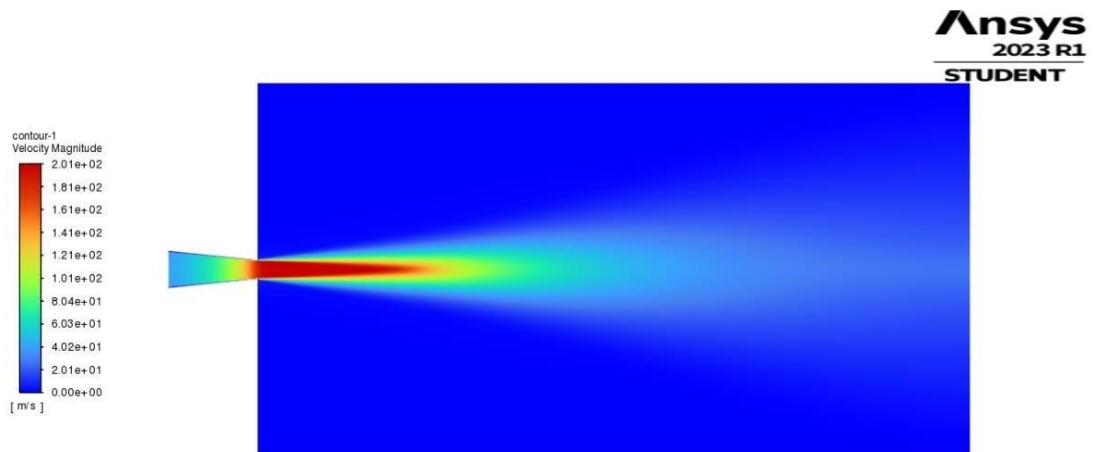


Fig:5.13 : Velocity Contour of 50 mm length nozzle with Diameter Ratio of 2

5.6 ANALYSIS OF DIAMETER RATIO= 3 WITH TURBULENT MODEL – K EPSILON RNG

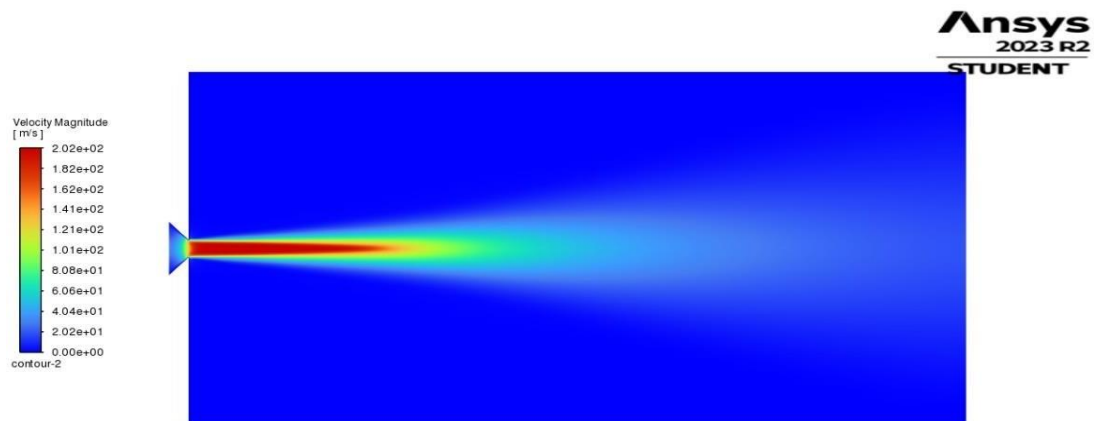


Fig:5.14 : Velocity Contour of 10 mm length nozzle with Diameter Ratio of 3

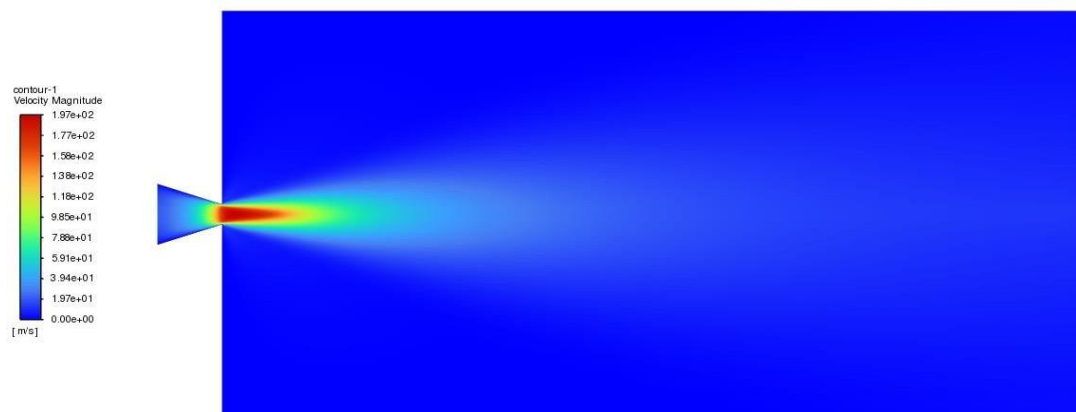


Fig:5.15 : Velocity Contour of 30 mm length nozzle with Diameter Ratio of 3

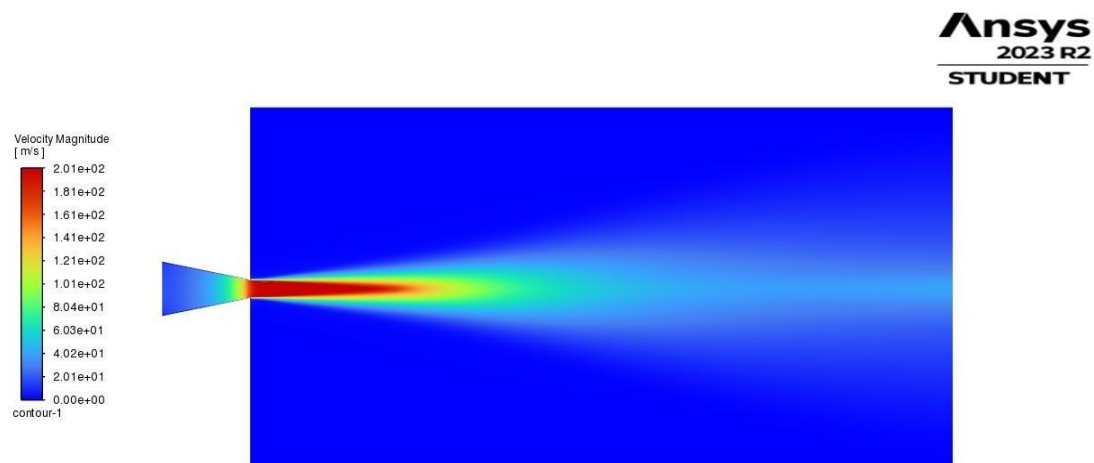


Fig:5.16 : Velocity Contour of 50 mm length nozzle with Diameter Ratio of 3

5.7 ANALYSIS OF DIAMETER RATIO= 4 WITH TURBULENT MODEL – K EPSILON RNG

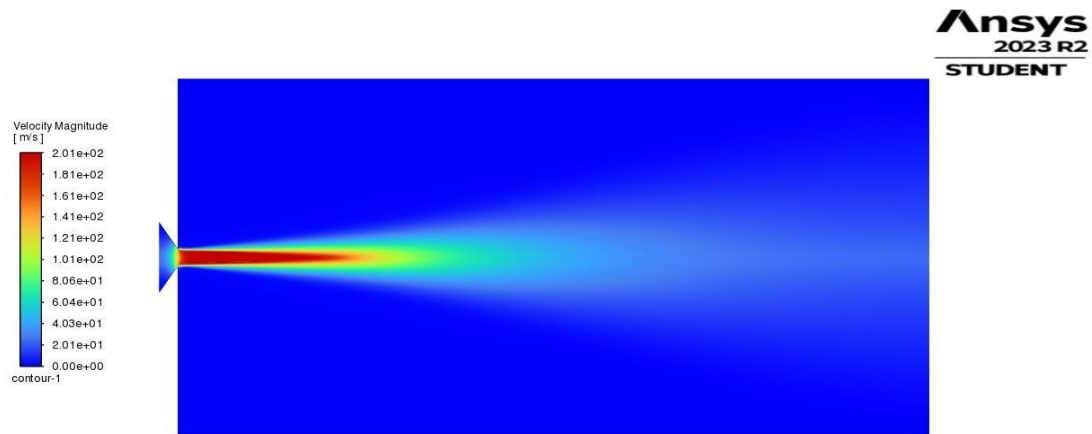


Fig:5.17 : Velocity Contour of 10 mm length nozzle with Diameter Ratio of 4

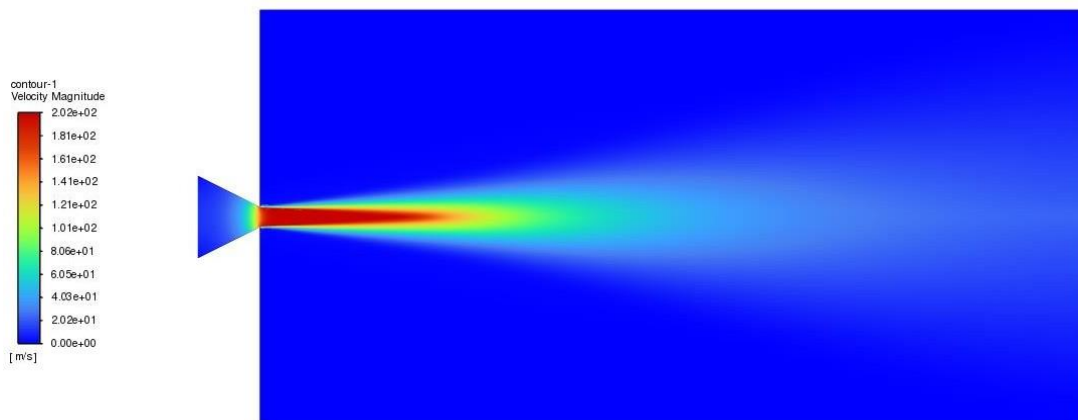


Fig:5.18 : Velocity Contour of 30 mm length nozzle with Diameter Ratio of 4

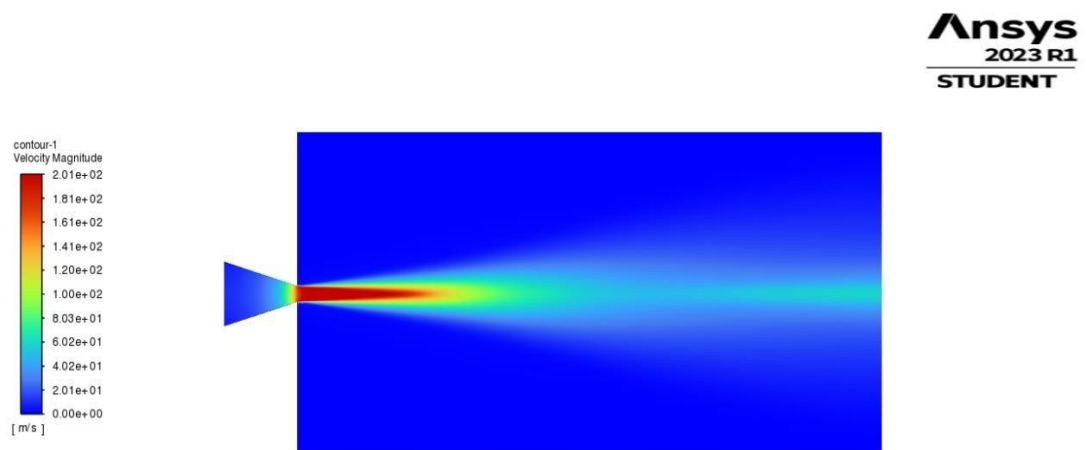


Fig:5.19 : Velocity Contour of 50 mm length nozzle with Diameter Ratio of 4

5.8 ANALYSIS OF DIAMETER RATIO= 2 WITH TURBULENT MODEL –K EPSILON REALIZABLE

Ansys
2023 R2
STUDENT

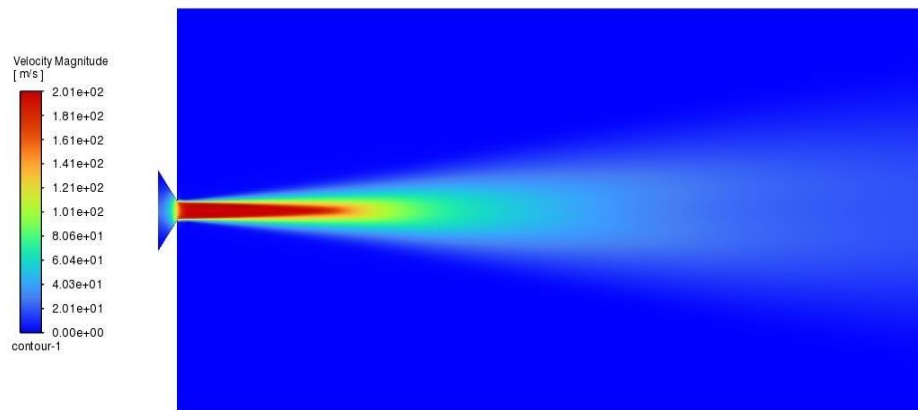


Fig:5.20 : Velocity Contour of 10 mm length nozzle with Diameter Ratio of 2

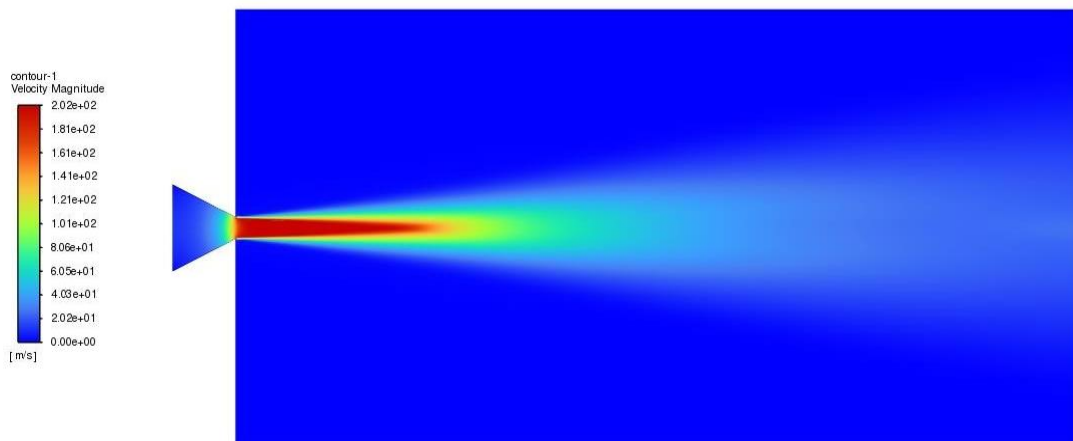


Fig:5.21 : Velocity Contour of 30 mm length nozzle with Diameter Ratio of 2

Ansys
2023 R1
STUDENT

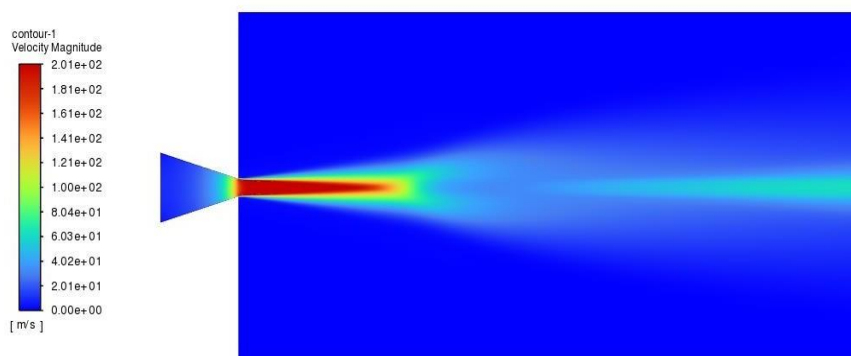


Fig:5.22 : Velocity Contour of 50 mm length nozzle with Diameter Ratio of 2

5.9 ANALYSIS OF DIAMETER RATIO= 4 WITH TURBULENT MODEL – K OMEGA STD

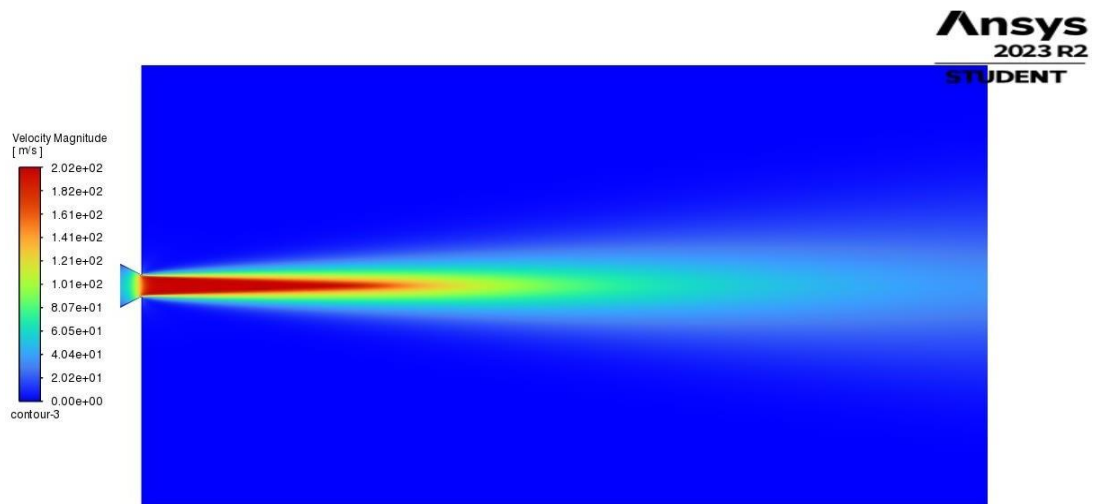


Fig:5.23 : Velocity Contour of 10 mm length nozzle with Diameter Ratio of 4

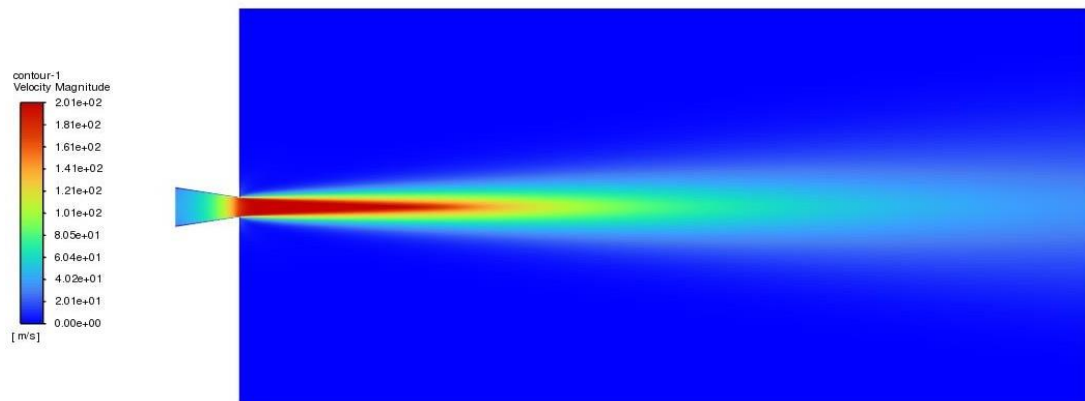


Fig:5.24 : Velocity Contour of 30 mm length nozzle with Diameter Ratio of 4

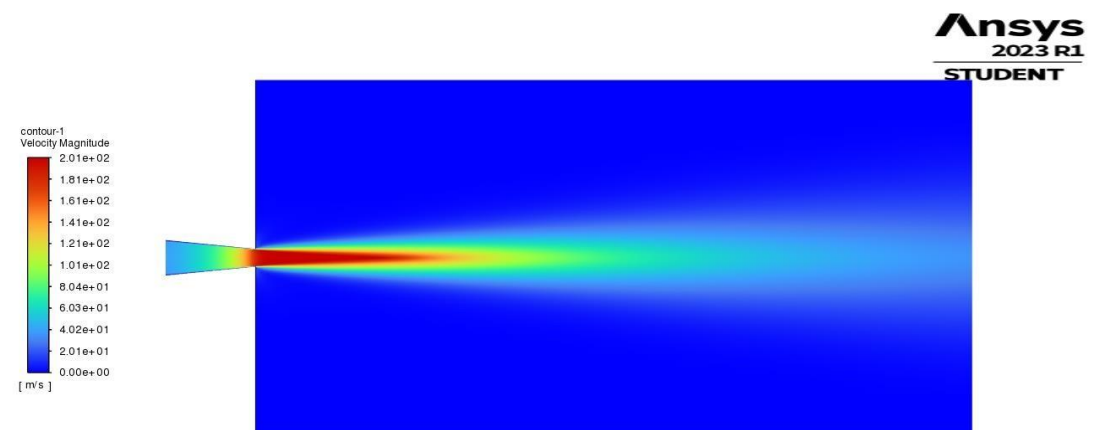


Fig:5.25 : Velocity Contour of 50 mm length nozzle with Diameter Ratio of 4

5.10 ANALYSIS OF DIAMETER RATIO= 3 WITH TURBULENT MODEL – K OMEGA STD

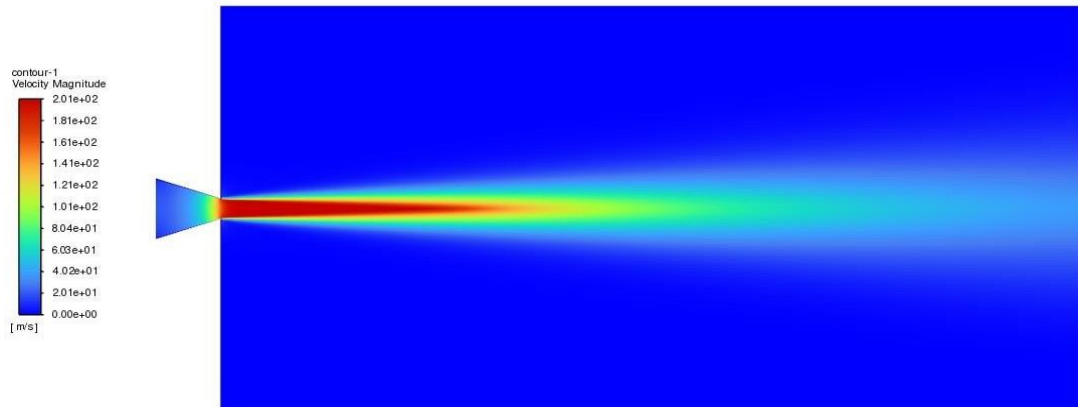


Fig:5.26 : Velocity Contour of 10 mm length nozzle with Diameter Ratio of 3

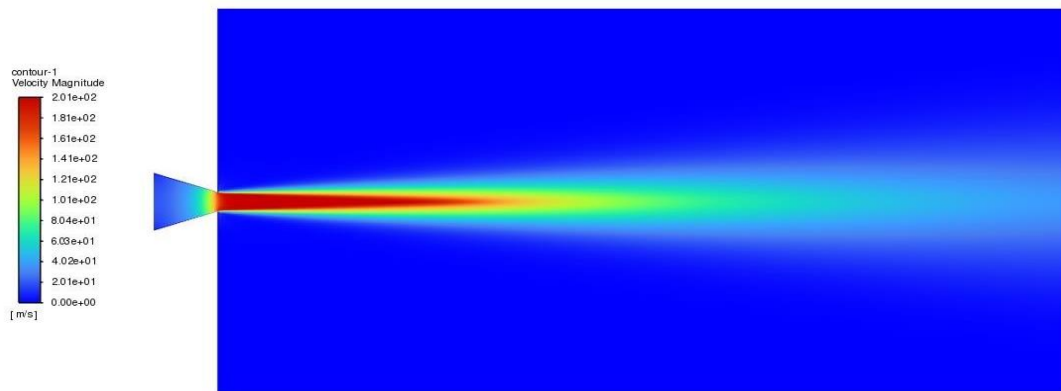


Fig:5.27 : Velocity Contour of 30 mm length nozzle with Diameter Ratio of 3

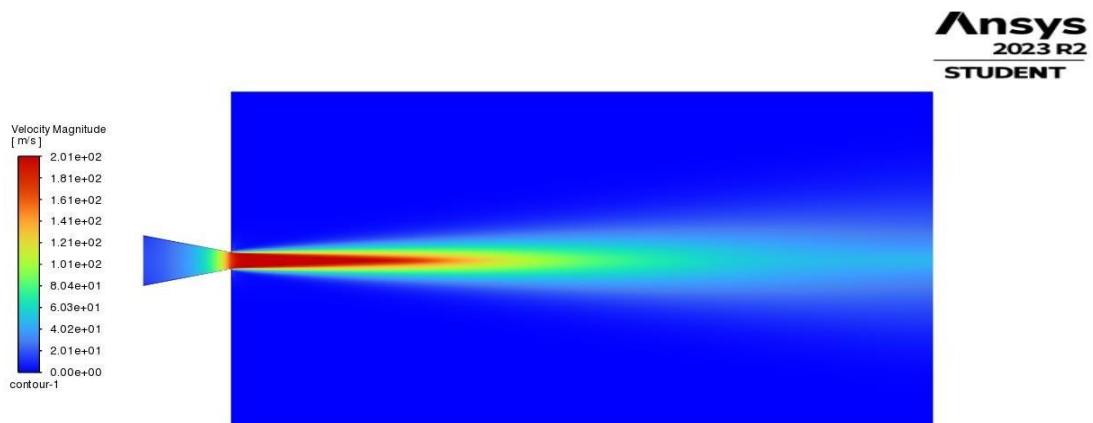


Fig:5.28 : Velocity Contour of 50 mm length nozzle with Diameter Ratio of 3

5.11 CENTRELINE VELOCITY DECAY

The Centerline Velocity Decay The centerline velocity decay rate is a practical indicator to estimate mass entrainment ratios in the jet flows because as the velocity decay rate gets higher, the potential core diminishes earlier; therefore, the jet entrains more fluid from the surroundings. The higher mean stream-wise velocity decay rate of jets issuing from sharp-edged slots implies higher entrainment ratios and consequently better mixing in the jet flows [15]

Relatively shorter potential core in non-circular jets also indicates the superior mixing capabilities of those jets over their circular counterparts.

Table 5.2 : Potential Core Length for all models

Length (in mm)	Diameter Ratio	Mach No.	Turbulent Model	Core Length (in mm)
10	2	0.6	K Epsilon Standard	6.233
	3			5.0
	4			5.298
20	2	0.6	K Epsilon Standard	6.233
	3			6.233
	4			5.99
30	2	0.6	K Epsilon Standard	5.75
	3			5.99
	4			5.75
40	2	0.6	K Epsilon Standard	5.99
	3			6.48
	4			5.99
50	2	0.6	K Epsilon Standard	5.99
	3			5.99
	4			5.99
60	2	0.6	K Epsilon Standard	6.233
	3			6.481
	4			6.5

The centerline velocity decays are plotted as a function of normalized axial distances (X/D_e) for all nozzles in Fig 5.29. This figure is also important to see how the different contraction profiles, the twist angles, and nozzle lengths influence the centerline velocity decay.

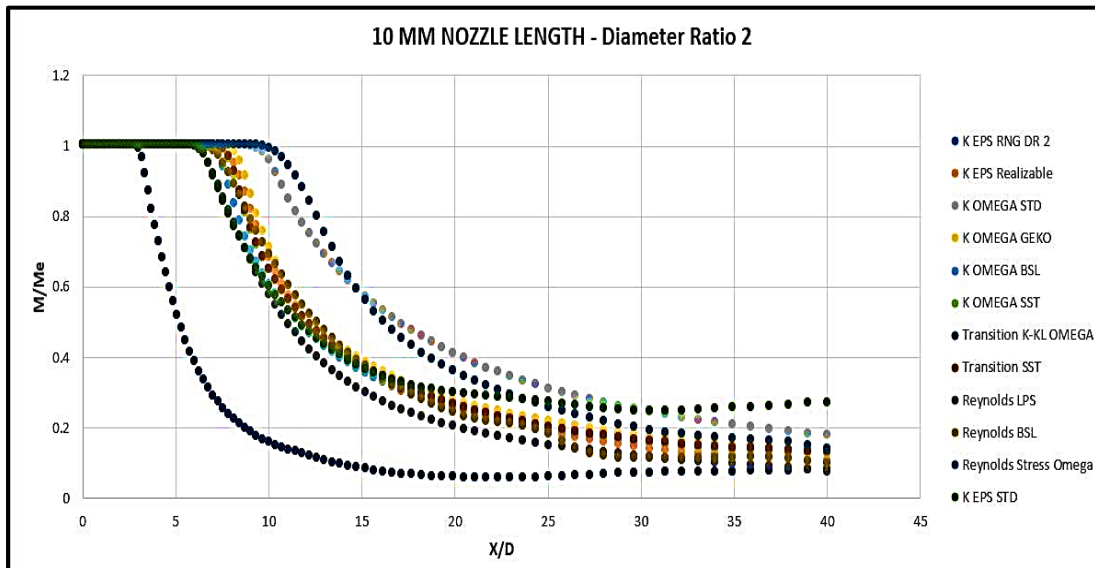


Fig:5.29 : The centerline velocity decay in the jets for all turbulent models

As can be seen in Fig 5.29, the velocity decay rates in circular nozzles are much faster in K-omega model than others. Also, From Fig 5.30, the velocity decay rate is much faster for DR=3 nozzle than other two diameter ratio for 10 mm length. Whereas for 20 mm length the DR=2 performs better than other two DR's (from Fig 5.31)

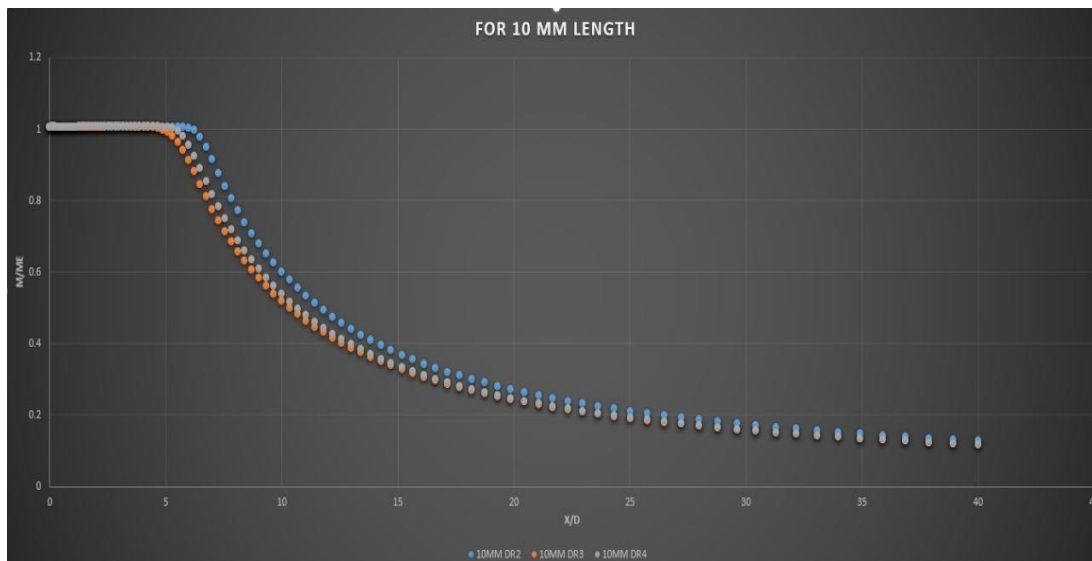


Fig:5.30 : The centerline velocity decay for 10mm length nozzle

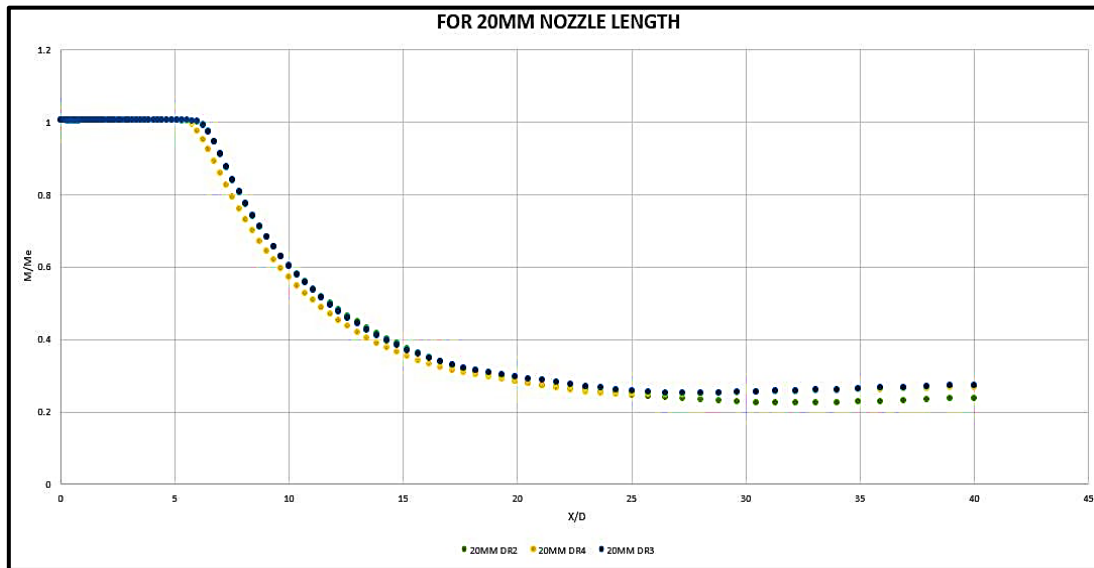


Fig:5.31 : The centerline velocity decay for 20mm length nozzle

Also, From Fig 5.32, the velocity decay rate is much faster for DR=4 nozzle than other two diameter ratio for 30 mm length. Whereas for 40 mm length the DR=4 performs better than other two DR's (from Fig 5.33)

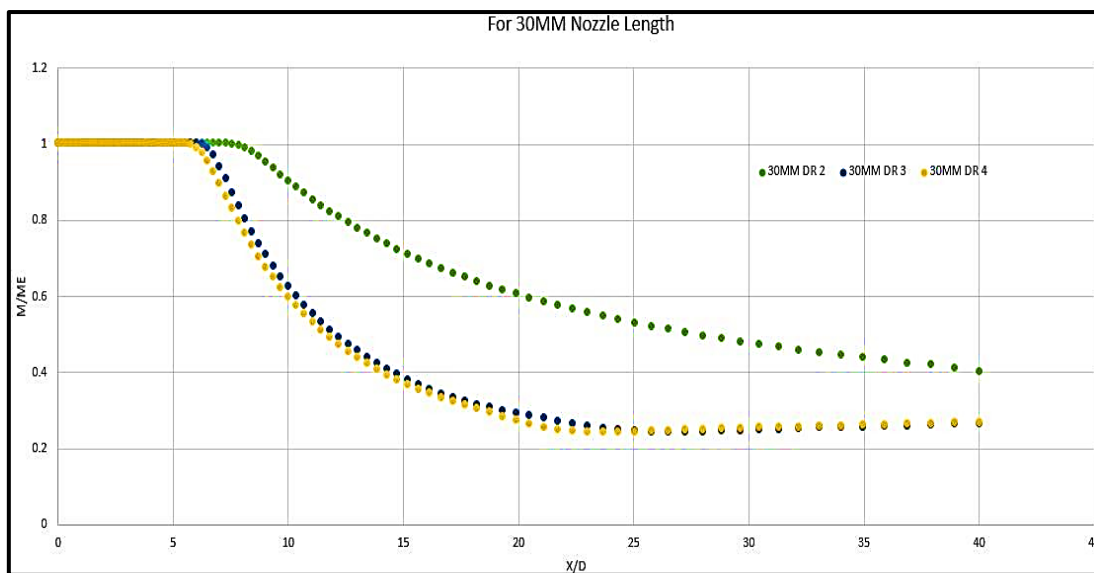


Fig:5.32 : The centerline velocity decay for 30 mm length nozzle

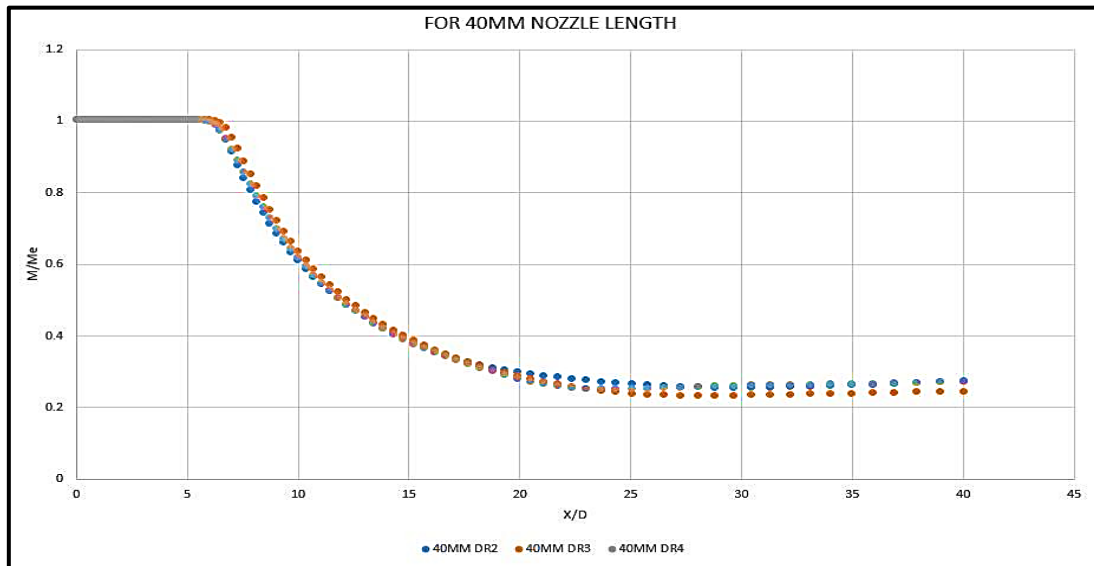


Fig:5.33 : The centerline velocity decay for 40 mm length nozzle

Also, From Fig 5.34, the velocity decay rate is much faster for DR=4 nozzle than other two diameter ratio for 50 mm length. Whereas for 60 mm length the DR=4 performs better than other two DR's (from Fig 5.35)

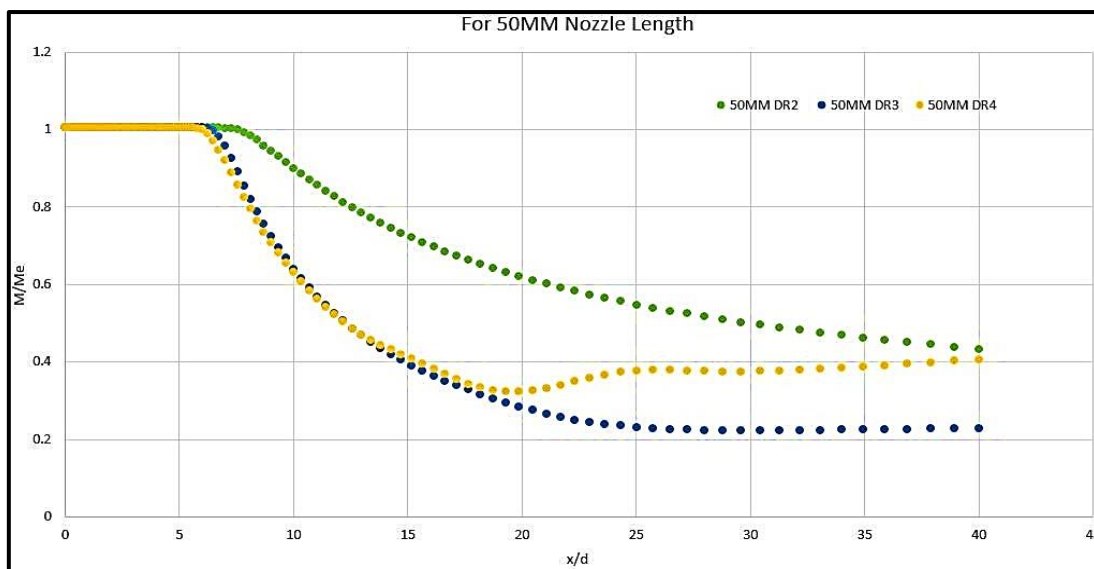


Fig:5.34 : The centerline velocity decay for 50 mm length nozzle

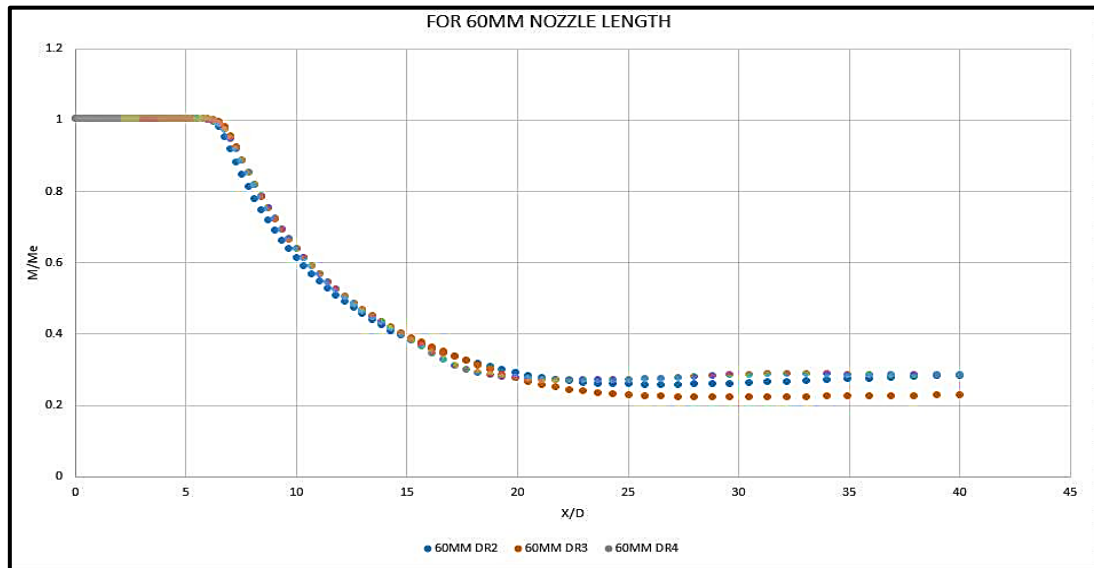


Fig:5.35 : The centerline velocity decay for 60 mm length nozzle

The fastest velocity decays occur in nozzles whose diameter ratio 4. However, they accelerate in the span-wise direction (the major axis plane) near outer boundaries of the jet. As a result of this behavior, they show double-jet formation. It should be noted that the mixing efficiency may not be concluded from the centerline velocity decay rates due to unique formation (the double-jet) for the second contraction profile nozzles. For the effects of nozzle lengths on the centerline velocity decay, demonstrates that as nozzle length becomes shorter, the centerline velocity decays faster. Nozzle of length 10mm that has the shortest nozzle length decays faster compared to nozzles of other lengths. For all jets, the vena-contracta effect is observed in very near field where the jet issues from outlet. The vena-contracta effect is the initial acceleration of fluid after emerging from an outlet.

The effect of nozzle geometry on the velocity decay was also investigated by Lovaraju [14]. The findings of Lovaraju et al. experiment are shown in Fig 5.36 to compare with the results of this project.

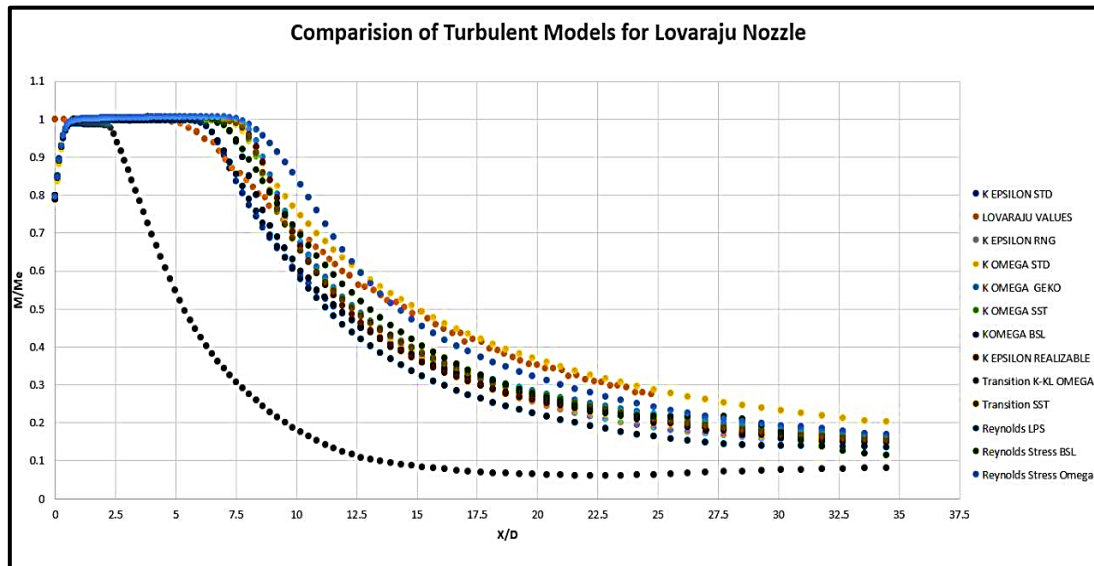


Fig:5.36 : The centerline velocity decay in the different turbulent models

5.12 EXPERIMENTAL RESULTS AND DISCUSSION

The experiments of a convergent subsonic jet were conducted at the Apollo Engineering College Chennai, India 'Compressed air was provided to the primary settling chamber from a storage tank via a control valve to achieve the desired Mach number. The isentropic relations were used to set the back pressure for the primary nozzle, while the settling chamber pressure (P_0) was regulated to attain the desired Mach number of 0.6. The experiment was conducted on the nozzle of dimension 10 mm length and diameter ratio 3. Firstly, the compressor was switched on and the air was sent into the pressure tank and was stored in it. Then the nozzle was fixed to the high-speed jet facility by using an adapter. WIKA sensor was used to measure the variation of Pitot pressure in the axial directions of the jet at different locations. The sensor has a range of 0–10 bar and an accuracy of $\pm 0.15\%$ full scale after re-zero calibration.

The measurement of total pressure was done using a Pitot tube with inner and outer diameters of 0.4 and 0.6mm respectively.

The probe was mounted on a traverse mechanism with a linear translation resolution of 0.1 mm. Then the experiment was started by releasing the air into the settling chamber by rotating the control valve.

The pressure values coming out of the jet were calculated by the WIKA sensor using the pitot tube which was mounted on the traverse mechanism. The pressure values are calculated for every quarter position up to the length of 30 cm by rotating the traverse mechanism in the axial direction of the jet.

The experiment was carried out for a while of 16 min and the data that was obtained from the sensor was used for the post-processing work to get accurate results of the total pressure.

The mixing characteristic are associated with experimental values of jet based on the far field decay, potential core length and characteristics decay. Due to the presence of compression waves and expansion waves of different strength, the static pressure is assumed to be constant throughout the study. Furthermore, when mixing of jet is fast, the potential core length will be small and it leads to rapid decrease in characteristic decay. The pressure oscillation in jet gives the shock wave structure.

The nozzles are analyzed based on the below parameters like

- Center line decay characteristics
- X/D vs P_{total}/P_{static}

5.12.1 Center line pressure decay

Centerline pressure decay is used to measure the efficiency of jet mixing and it also shows the length of jet core. Jet cone can be defined as the axial distance up to which the supersonic region presents. This jet cone is constant for all type of profile like circular and non-circular jet. More ever, it also independent of the jet spread and axis switching. The ratio of pitot

pressure P_t and Chamber pressure P_o is plotted against the axial distance X/D .

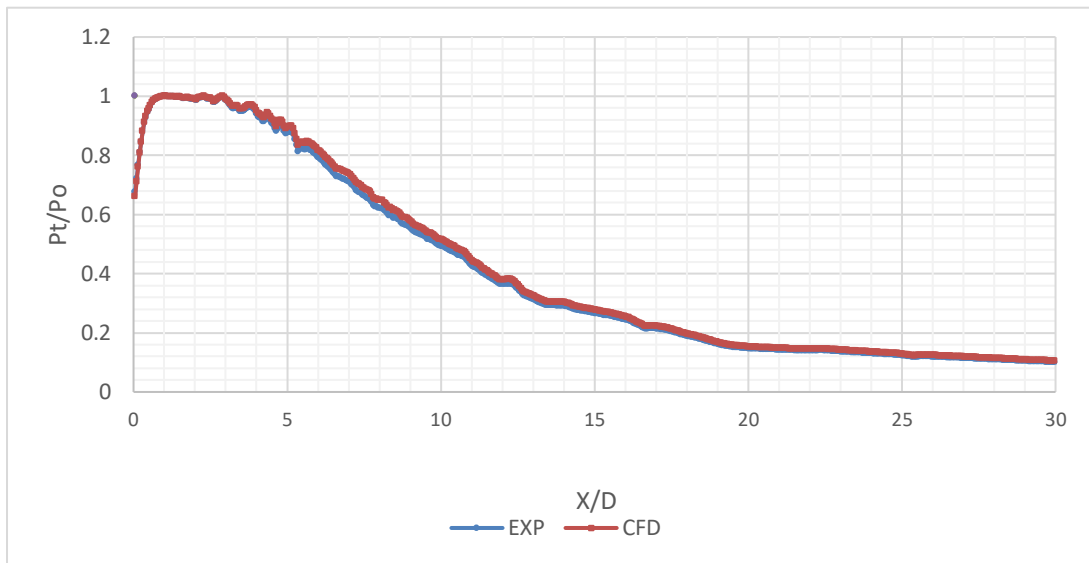


Fig:5.37 : The centerline velocity decay of Length= 10mm nozzle (DR= 3)

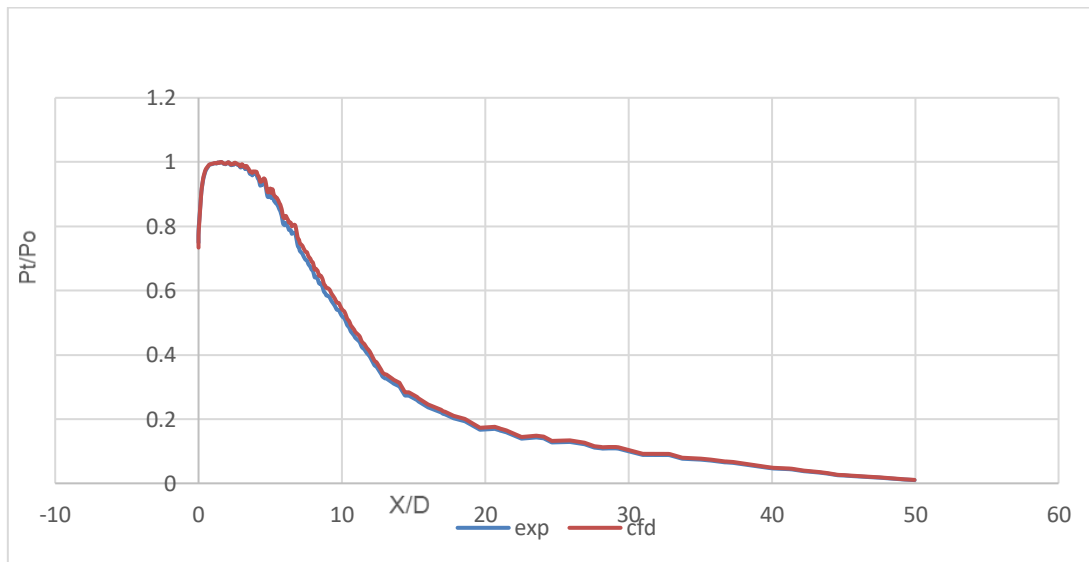


Fig:5.38 : The centerline velocity decay of Length= 20mm nozzle (DR= 3)

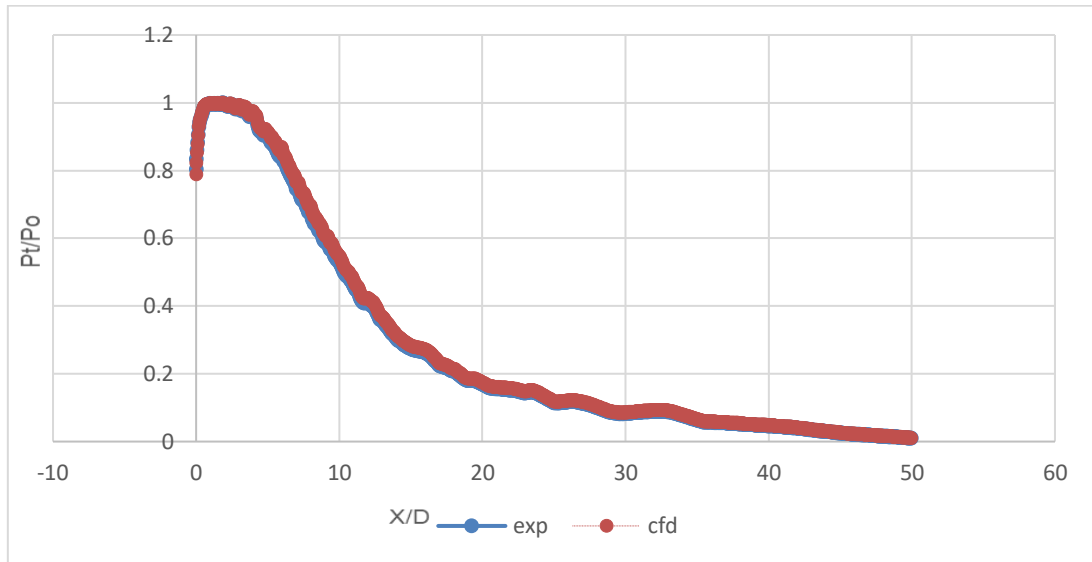


Fig:5.39 : The centerline velocity decay of Length= 30mm nozzle (DR= 3)

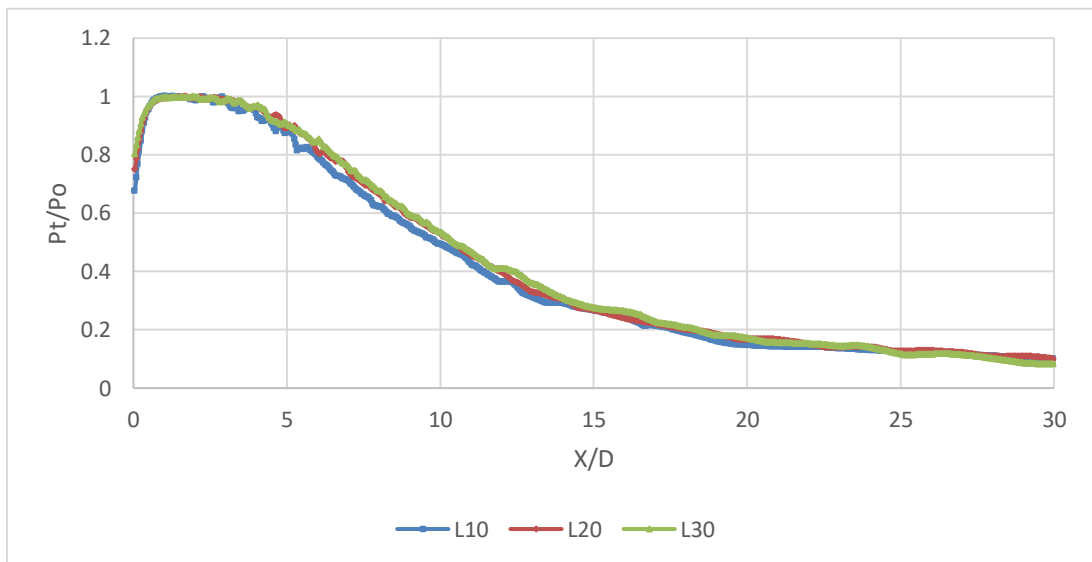


Fig:5.40 : Decay of various length at NPR 1.1- Experimental Results

- **For 10mm Length Nozzle**

The velocity ratio starts at a relatively high value at X/D and decreases gradually as X/D increases. The potential core length in this case would be the region where the velocity ratio remains relatively high and doesn't decrease significantly. This region seems to be concentrated near the nozzle exit ($X/D \approx 0.4$).

- **20mm Length Nozzle**

The velocity ratio starts at a relatively low value at $X/D = 0$ and increases gradually as X/D increases. The potential core length in this case would be the region where the velocity ratio remains relatively low and doesn't increase significantly. This region seems to be concentrated near the nozzle exit ($X/D \approx 40$).

- **30mm Length Nozzle**

The velocity ratio starts at a relatively high value at $X/D = 0$ and decreases gradually as X/D increases. Similar to the 10mm length nozzle, the potential core length in this case would be the region where the velocity ratio remains relatively high and doesn't decrease significantly. This region also seems to be concentrated near the nozzle exit ($X/D \approx 40$).

Overall, the potential core length appears to be more concentrated near the nozzle exit for all three nozzle lengths. This observation suggests that the flow undergoes significant changes near the nozzle exit, which is typical in fluid dynamics, as the flow adjusts to the ambient conditions.

Table 5.3 : Comparison of potential core length

Nozzles	Experimental values of potential core length
DR=3 L=10mm	4.1
DR=3 L=20mm	3.8
DR=3 L=30mm	3.7

Comparing the experimental values of potential core length for different nozzle ($DR=3$, lengths of 10mm, 20mm, and 30mm) with the qualitative analysis from the provided data, we can observe the following:

1. $DR=3$, $L=10mm$ Nozzle

- Experimental Potential Core Length: 4.1
- Qualitative Analysis: The potential core length was observed to be concentrated near the nozzle exit, which aligns with the experimental value. The qualitative analysis indicated that the potential core length would be near the nozzle exit, which corresponds to the experimental result.

2. $DR=3$, $L=20mm$ Nozzle

- Experimental Potential Core Length: 3.8
- Qualitative Analysis: Similarly, the potential core length was observed to be near the nozzle exit for the 20mm length nozzle. The qualitative analysis indicated that the potential core length would be concentrated near the nozzle exit, which aligns well with the experimental result.

3. $DR=3$, $L=30mm$ Nozzle

- Experimental Potential Core Length: 3.7
- Qualitative Analysis: Again, the potential core length was observed to be concentrated near the nozzle exit for the 30mm length nozzle. The qualitative analysis suggested a similar trend, where the potential core length would be near the nozzle exit, supporting the experimental result.

Overall, the qualitative analysis based on the provided data aligns with the experimental values of potential core length for all three nozzle configurations. This consistency indicates that the qualitative analysis is in

good agreement with the experimental findings, confirming the behaviour of the potential core length near the nozzle exit for different lengths of the nozzle with a Diameter Ratio (DR) of 3.

5.12.2 Shadow graph Images

For 10mm diameter ratio 3 shadow graph images were taken. It is well known that under expanded jet forms expansion fans and compression waves in the jet by periodic reflection of waves from the jet boundary. The length of the shock cell is significant and the numbers of shock cells are more for the plain jet, as shown in the Fig 5.41 below. vortices pass through different Mach number zones and become weak and lose their identity. The pressure differential upstream and downstream gets strengthened as there is a favourable pressure gradient “pressure valley” at the rear side of the tab in the under expanded jet flow.



Fig:5.41 : NPR of shadow graph images

CHAPTER 6

CONCLUSION

A computational analysis of non-circular nozzles was presented in this project. A CFD model was developed to predict mass entrainment ratios, the turbulence intensities, jet half-width variations, and mean velocity profiles in non- circular nozzles. Three different geometrical nozzle modifications were applied to non- circular jets to condition and control the flow properties and consequently enhance the entrainment process. These modifications were the

- Nozzle lengths.
- Diameter Ratio

A detailed analysis of the modified nozzle geometries was performed. The results were compared to circular nozzle as well as reference. This thesis addressed the flow properties of geometrically modified circular nozzles utilizing ANSYS Fluent tools. The general conclusions drawn from studying the results based on implemented CFD model were as follow:

1. For nozzle length considerations, non-circular nozzle with the smallest nozzle length ($L=2D_e$) has higher mass entrainment ratios than nozzles of other length. These results are also consistent with the previous experimental study [20].
2. Mass entrainment ratios are much higher in all circular nozzles in high diameter ratio than low valued diameter ratio.
3. Centerline velocity decay, which is an indicator of a better mixing, is faster in low Diameter ratio and less length circular nozzles.
4. Among all turbulent models K-epsilon model results proper mixing than other models.
5. In conclusion for this study, nozzle with $DR=3$ and Length 10mm is the best nozzle geometry for mixing.

REFERENCES

- [1] Akin, J. E., Bass, J., "Asymmetric turbulent jet flows", *Computer Methods in Applied Mechanics and Engineering*, 191:515-524, 2001.
- [2] Azad, M., Quinn, W. R., Groulx, D., "Mixing in turbulent free jets issuing from isosceles triangular orifices with different apex angles", *Experimental Thermal and Fluid Science*, 39:237–251, 2012.
- [3] ANSYS, FLUENT, *Theory Guide*, 2011.
- [4] Berg, J. R., Ormiston, S. J., and Soliman, H. M., "Prediction of the flow structure in a turbulent rectangular free jet", *International Communications in Heat and Mass Transfer*, 33:552-563, 2006.
- [5] CD-Adapco, Star-CCM+, *User Guide Version 8.04*, 2013.
- [6] Falcone, A., M., Cataldo, J. C., "Entrainment velocity in an axisymmetric turbulent jet", *Journal of Fluids Engineering*, 125:620-627, 2003.
- [7] Grinstein, F. F., Gutmark, E., Parr, T., "Near field dynamics of subsonic free square jets. A computational and experimental study", *Physics of Fluids*, 7:1483- 1497, 1995.
- [8] Gutmark, E., Schadow, K. C., "Flow characteristics of orifice and tapered jets", *Physics of Fluids*, 30:3448-3454, 1987.
- [9] Gutmark, E. J., Grinstein F. F., "Flow control with non-circular Jets", *Annual Review of Fluid Mechanics*, 31:239-272, 1999.
- [10] Gutmark, E., Schadow, K. C., Parr, T. P., Parr, D. M., and Wilson. K. J., "Combustion Enhancement by Axial Vortices", *Journal of Propulsion and Power*, 5:555-560, 1989.
- [11] Hammond, A. D., "Thrust losses in hovering for jet VTOL aircraft", NASA, Report No. SP-116, 1966.
- [12] Heyt, J. W., Diaz, M. J., "Pressure drop in flat-oval spiral air duct", *ASHRAE Transactions*, 81:221-232, 1975.
- [13] Ho, C., Gutmark, E., "Vortex induction and mass entrainment in a small aspect ratio elliptic jet", *Journal of Fluid Mechanics*, 179: 383-405, 1987.

- [14] Hussain, F., Husain, H. S., "Elliptic jets. Part 1. Characteristics of unexcited and excited Jets", *Journal of Fluid Mechanics*, 208:257-320, 1989.
- [15] Jadhav, T., "Multiple round jets. An experimental and computational study", 37th. National and 4th. International conference on fluid mechanics and fluid power, *FMFP*, 2010.
- [16] Lovaraju P, Rathakrishnan E. Subsonic and Transonic Jet Control with Cross-Wire. American Institute of Aeronautics and Astronautics 2006; 44(11): 2700-2705.
- [17] Mi, J., Nathan, G. J., "Statistical Properties of Turbulent free jets issuing from nine differently-shaped nozzles", *Flow Turbulence Combust*, 84:583–606, 2010.
- [18] Miller, R. S., Madnia, C. K., and Givi, P., "Numerical simulation of non-circular jets", *Computers and Fluids*, 24:1–25, 1995.
- [19] New, T. H., "Effects of notch geometry and sharpness on turbulent jets issuing from indeterminate-origin notched nozzles", *European Journal of Mechanics B/Fluids*, 29:309-320, 2010.
- [20] Patankar, S. V., Spalding, D. B., "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows", *International Journal of Heat and Mass Transfer*, 15:1787-1806, 1972.
- [21] Quinn, W. R., "Near-field measurements in an equilateral triangular turbulent free jet", *AIAA Journal*, 43:2574-2585, 2005.
- [22] Quinn, W. R., "On Mixing in an Elliptic Turbulent Free Jet", *Physics of Fluids*, 1: 1716-1722, 1989.
- [23] Ricou, F. P., Spalding, D. B., "Measurements of entrainment by axisymmetrical turbulent jets", *Journal of Fluid Mechanics*, 11:21-32, 1961.
- [24] Shamami, K. K., Birouk, M., "Three-dimensional numerical simulation of an equilateral triangular turbulent free jet", *International Journal of Numerical Methods for Heat and Fluid Flow*, 19:894-912, 2009.
- [25] Shih, T-H., Liou, W. W., Shabbir, A., Yang, Z. and Zhu, J., "A new k- ϵ eddy viscosity model for high Reynolds number turbulent flows", *Computers and Fluids*, 24:227-238, 1995.
- [26] Trabold, T.A., Esen, E. B., and Obot, N. T., "Entrainment by turbulent jets issuing from sharp-edged inlet round nozzles", *Journal of Fluids Engineering*, 109:248-254, 1987.

- [27] Trentacoste, N., Sforza, P. M., "Some remarks on three-dimensional wakes and jets", *Air Force Office of Scientific Research*, Report No. 68-9, 1968.
- [28] Versteeg, H. K., Malalasekera, W., "An Introduction to Computational Fluid Dynamics", *Prentice Hall*, 2007.
- [29] Wilcox, D. C., "Turbulence Modeling for CFD", *DCW Industries Inc*, 1993.

ENHANCING MIXING CHARACTERISTICS OF CONVERGENT NOZZLE

E.Rambabu¹ S.Balaji² Dr.P.Booma Devi³

1&2 UG Scholar, Department of Aeronautical Engineering,
Sathyabama Institute of Science and Technology, Chennai- India

3. Associate professor, Department of Aeronautical Engineering, Sathyabama
Institute of Science and Technology, Chennai- India

rambabuelumalai17@gmail.com, balajiaero20@gmail.com,
boomajeev@gmail.com

ABSTRACT

This thesis presents a computational analysis for determining the flow properties of novel circular nozzles. In last few decades, circular nozzles have been investigated intensively due to their remarkably superior capabilities in enhancing mass entrainment over non circular nozzles. In this thesis, to increase the amount of ambient fluid entrained in the jet flow, three different geometrical modifications are applied to circular nozzles. In this study, the mixing characteristics of a circular free jet were investigated computationally for different turbulence models. Found that K-epsilon turbulent model deemed to be fit for $M=0.6$ The study found that the variation in length of the nozzle leading to improved mixing characteristics. The flow properties of air emanating from geometrically modified circular nozzles are studied by using computational fluid dynamics (CFD). This computational analysis shows that suitably modified circular nozzles are very effective passive flow conditioning devices and can modify the flow field. Particularly, nozzle with length variation entrains the ambient fluid at a higher rate than the diameter ratio modified nozzles.

Keywords: CFD, Circular Nozzle, K-epsilon turbulent model

1. INTRODUCTION

Jet flows are widely used in many engineering applications such as gas turbine combustors, engine injectors, water jets, and hydraulic drilling. In most of these engineering applications, main purposes are mixing and mass entrainment enhancement in the flow domain. The efficiency of all those applications is highly dependent on flow properties of the jet flow. In this thesis, novel non-circular nozzles are studied to improve mass entrainment rates of non-circular nozzles.

Axisymmetric jets are the most common jet applications because they are easy and convenient to produce, and they provide stable flows due to their smooth and straightforward geometries. In the past few decades, researchers have begun to look for alternative jet designs that are more suitable than a circular jet for particular applications including combustion, mixing, and cleaning. The search for alternative jet designs

in hopes of finding better jet designs eventually led them to investigate non-circular jet flows. Many researchers reported that non-circular jet flows showed better mixing and mass entrainment over circular jets. Experimental studies suggested that enhancement of mixing and mass entrainment is related to the azimuthal distortion caused by the sharp edges of non-circular jets it is highly important to understand the flow properties of a jet flow in order to find new ways of control and improvement over common jet configurations. The objective of this paper is to study the flow properties of different circular nozzles to find the best nozzle design for better mixing and higher mass entrainment rates. In this study, circular nozzles are modified geometrically and the effects of those modifications on improving the flow properties are studied.

2. METHODOLOGY OF COMPUTATIONAL STUDIES

2.1 Finite Volume Method

The Star CCM+ solver uses the finite volume method, which is a numerical method that divides the problem domain into finite number of volumes and evaluates conservation laws (conservation of mass and momentum) for every finite volume. These volumes are called cells of a computational grid. The next step is to find discrete versions of conservation laws in integral form for every control volume in the domain. The main objective is to obtain a set of linear equations corresponding to the number of cells in the grid

2.2 Segregated Flow Model

There are two unsteady flow solvers available in Star CCM+, namely coupled flow solver and segregated flow solver. Coupled flow model solves conservation equations for mass and momentum simultaneously using a time-marching approach. The coupled flow solver is mainly used in compressible flows, natural convection problems, and the flows with large body forces.

The segregated flow model solves the flow equations in a segregated manner. A predictor-corrector approach is used to link momentum and continuity equations, which are solved separately. The segregated flow model offers a good compromise between accuracy and computation time for constant density flows. It is also suitable to use segregated flow model for mildly compressible flows. It is highly recommended to use segregated flow where the computational resources are issue and the physics of problem is suitable to use segregated flow

There are several algorithms to couple pressure-velocity and derive equations for the pressure from continuity and momentum equations. The default algorithm in Star-CCM+ is "Semi Implicit Method for Pressure Linked Equations (SIMPLE)" algorithm. SIMPLE is not only default algorithm for the solver, but also the most widely used one to solve segregated flow model. SIMPLE algorithm was introduced by Patankar and Spalding. In SIMPLE algorithm, initial pressure field is defined by using boundary conditions. The gradients of velocity and pressure are

computed according to the initial pressure field. The discretized momentum equations are solved to compute intermediate velocity field. The face mass fluxes are checked in every cell to see if continuity equation is satisfied for every cell. To satisfy continuity equation, the pressure correction is applied and the pressure field is updated by using new corrected pressure values. The face mass fluxes and cell velocities are corrected with respect to updated pressure field. These fundamental steps can be repeated as long as the convergence criteria are satisfied. After carrying out several benchmark computations to test the present model, they reported well-known spreading rate anomaly was completely removed. In addition, this new dissipation rate equation is expected to enhance numerical stability in turbulent flow conditions. In conclusion, after investigating the previous computational studies carefully, and because of the reasons listed in this section, the realizable k- ϵ turbulence model is selected for this study.

2.3 Nozzle Geometry

Detailed information regarding nozzle properties is given in Table 2.1 for all nozzle configurations. Table 2.1: Nozzle configurations.

	Inlet Diameter(mm)	Outlet Diameter(mm)	Length(mm)	Convergence Angle(in deg)
NOZZLE - 1	20	10	10	26.56
	30	10	10	45.968
	40	10	10	56.81
NOZZLE - 2	20	10	20	14.036
	30	10	20	26.565
	40	10	20	36.87
NOZZLE - 3	20	10	30	9.462
	30	10	30	18.435
	40	10	30	26.565
NOZZLE - 4	20	10	40	7.125
	30	10	40	14.036
	40	10	40	20.556
NOZZLE	20	10	50	5.711
	30	10	50	11.31

-5	40	10	50	16.699
NOZZLE - 6	20	10	60	4.754
	30	10	60	9.452
	40	10	60	14.036

3. ANALYSIS

3.1 Grid Independence Study:

Three sets of grid cells, a coarse grid of about 24,000, a medium grid of about 43,300, and a fine grid of 1, 71,800, were used for the grid independence study.

Table 3.1 Turbulence Model Comparison for Computational Analysis

K Epsilon	Standard
	RNG
	Realizable
K Omega	Standard
	GEKO
	BSL
	SST
Transition K-KL Omega (3eqn)	-
Transition SST (4 eqn)	-
Reynolds Stress(eqn)	Linear Pressure Strain
	Stress - Omega
	Stress - BSL

The SST model employs a $k-\epsilon$ formulation in the inner region of the wall boundary layers and switches to a transformed $k-\epsilon$ formulation in the outer region of the boundary layers and in the free shear layer/mixing regions.

It should be noted that the intention here is not so much as to make a detailed evaluation of the different models and their different variants in computing the present flow field. Rather, it is to select one that can represent flow among these models.

The primary jet potential core length (PCL) is defined as the axial extension until the primary exit nozzle velocity prevails for subsonic jets (Lovaraju et al. 2011). Mach 0.6, for the seven different turbulence models and for the experiment conducted by lovaraju et al. It is evident that the $k-\epsilon$ standard model produces data that agrees well with the experimental data, consistent with its advantage in predicting the flow field characteristics, and the $k-\epsilon$ standard model decays slower than the experimental variation in the characteristic decay region. Other turbulence models decay at a slower rate when compared with the experimental decay.

3.1 Analysis of Diameter Ratio= 2 with Turbulent Model – K EPSILON STD

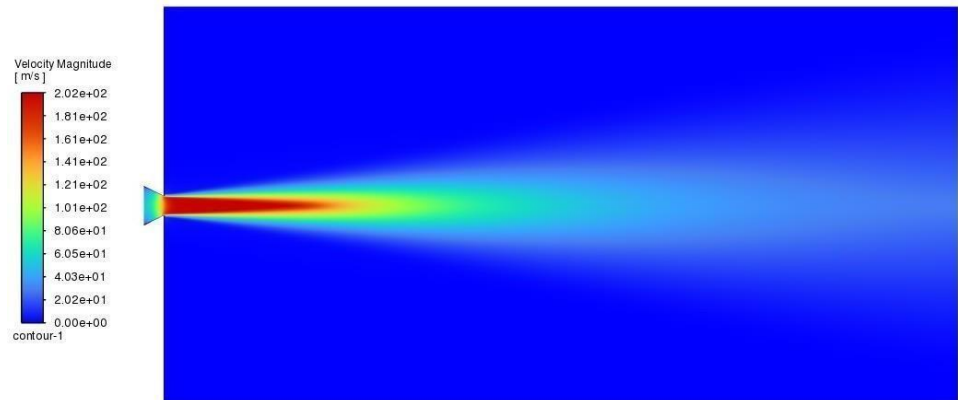
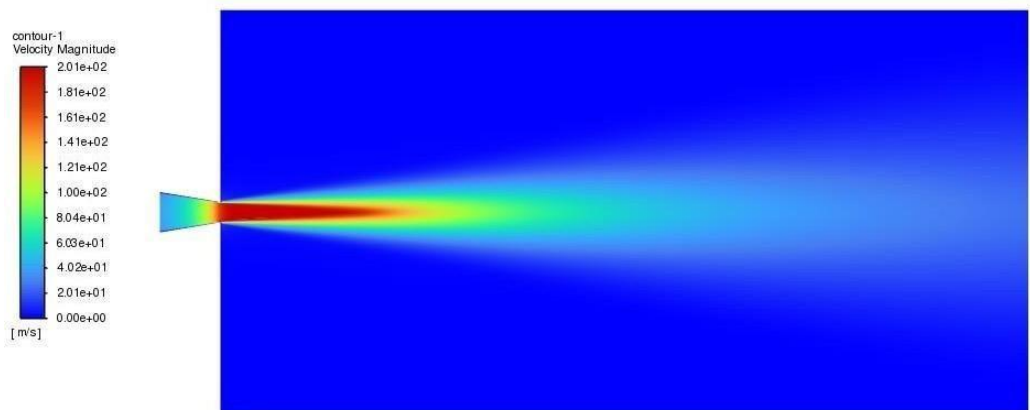


Figure 3.2: Velocity Contour of 10 mm length nozzle with Diameter Ratio of 2



3.2 Centerline Velocity Decay

The Centerline Velocity Decay The centerline velocity decay rate is a practical indicator to estimate mass entrainment ratios in the jet flows because as the velocity decay rate gets higher, the potential core diminishes earlier; therefore, the jet entrains more fluid from the surroundings. The higher mean stream-wise velocity decay rate of jets issuing from sharp-edged slots implies higher entrainment ratios and consequently better mixing in the jet flows. Relatively shorter potential core in non- circular jets also indicates the superior mixing capabilities of those jets over their circular counterparts.

Table 3.2 Potential Core Length for all models

Length (in mm)	Diameter Ratio	Mach No.	Turbulent Model	Core Length (in mm)
10	2	0.6	K Epsilon Standard	6.233
	3			5.0
	4			5.298
20	2	0.6	K Epsilon Standard	6.233
	3			6.233
	4			5.99
30	2	0.6	K Epsilon Standard	5.75
	3			5.99
	4			5.75
40	2	0.6	K Epsilon Standard	5.99
	3			6.48
	4			5.99
50	2	0.6	K Epsilon Standard	5.99
	3			5.99
	4			5.99
60	2	0.6	K Epsilon Standard	6.233
	3			6.481
	4			6.5

The centerline velocity decays are plotted as a function of normalized

axial distances (X/De) for all nozzles in Figure 3.2 This figure is also important to see how the different contraction profiles, the twist angles, and nozzle lengths influence the centerline velocity decay.

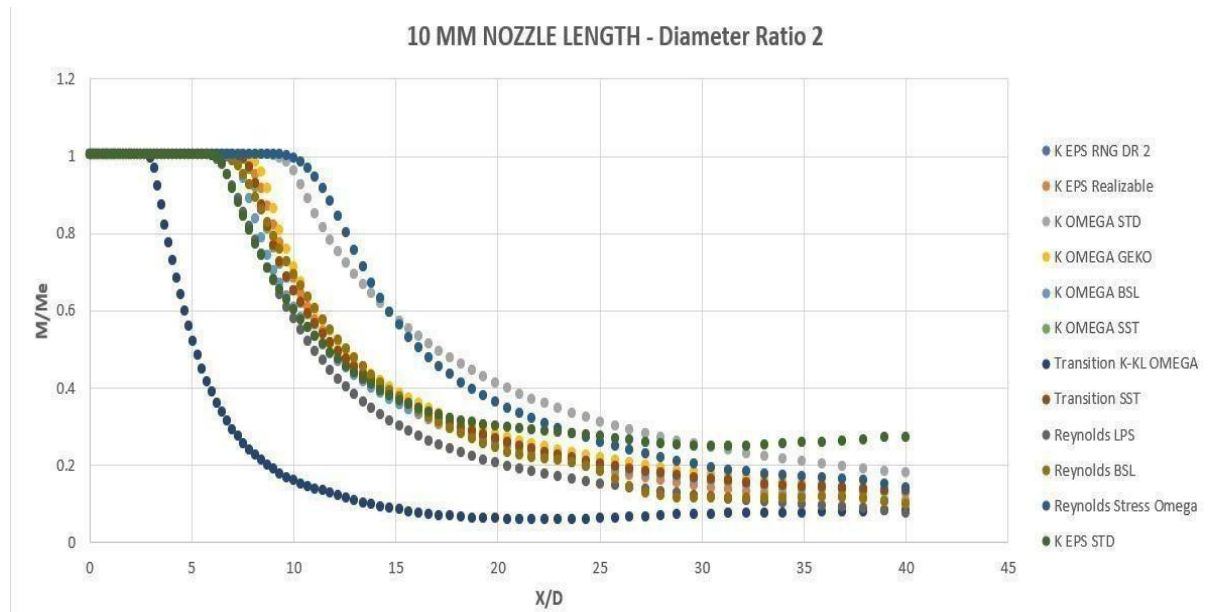


Figure 5.25: The centerline velocity decay in the jets for all turbulent models

As can be seen in Figure 3.2 the velocity decay rates in circular nozzles are much faster in K-omega model than others. Also, From Figure 3.2, the velocity decay rate is much faster for DR=3 nozzle than other two diameter ratio for 10 mm length. Whereas for 20 mm length the DR=2 performs better than other two DR's (from Figure)

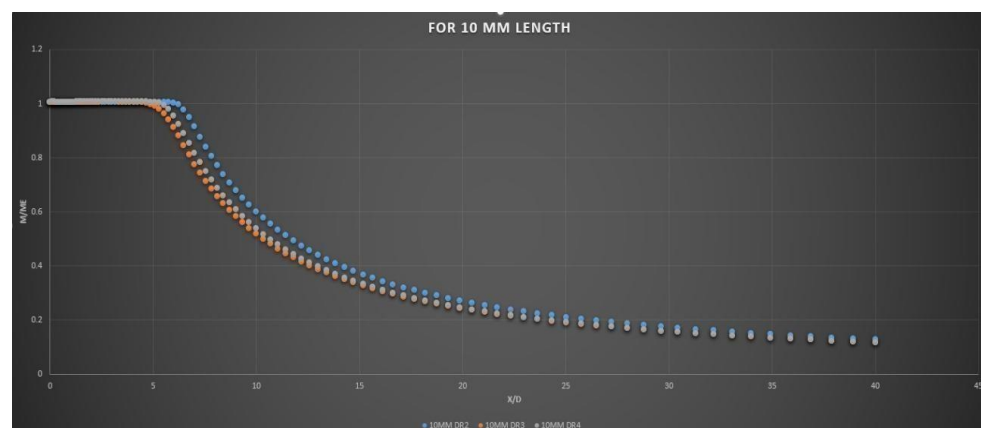


Figure 3.3: The centerline velocity decay for 10mm

length nozzle

CONCLUSIONS

A computational analysis of non-circular nozzles was presented in this thesis. A CFD model was developed to predict mass entrainment ratios, the turbulence intensities, jet half-width variations, and mean velocity profiles in non-circular nozzles. The CFD model was used to perform the numerical analysis for all nozzles. Three different geometrical nozzle modifications were applied to non-circular jets to condition and control the flow properties and consequently enhance the entrainment process. These modifications were the □ Nozzle lengths.

□ Diameter Ratio

A detailed analysis of the modified nozzle geometries was performed. The results were compared to circular nozzle as well as reference. This thesis addressed the flow properties of geometrically modified circular nozzles utilizing ANSYS Fluent tools. The general conclusions drawn from studying the results based on implemented CFD model were as follow:

1. For nozzle length considerations, non-circular nozzle with the smallest nozzle length ($L=2De$) has higher mass entrainment ratios than nozzles of other length. These results are also consistent with the previous experimental study
2. Mass entrainment ratios are much higher in all circular nozzles in high diameter ratio than low valued diameter ratio.
3. Centerline velocity decay, which is an indicator of a better mixing, is faster in low Diameter ratio and less length circular nozzles.
4. Among all turbulent models K-epsilon model results proper mixing than other models.
5. In conclusion for this study, nozzle with $DR=2$ and Length 30mm is the best nozzle geometry for mixing.

REFERENCES

- [1] Akin, J. E., Bass, J., "Asymmetric turbulent jet flows", Computer Methods in Applied Mechanics and Engineering, 191:515-524, 2001.
- [2] Trentacoste, N., Sforza, P. M., "Some remarks on three-dimensional wakes and jets", Air Force Office of Scientific Research, Report No. 68-9, 1968.
- [3] Grinstein, F. F., Gutmark, E., Parr, T., "Near field dynamics of subsonic free square jets. A computational and experimental study", Physics of Fluids, 7:1483-1497, 1995.
- [4] Berg, J. R., Ormiston, S. J., and Soliman, H. M., "Prediction of the flow structure in a turbulent rectangular free jet", International Communications in Heat and Mass Transfer, 33:552-563, 2006.
- [5] Ho, C., Gutmark, E., "Vortex induction and mass entrainment in a small aspect ratio elliptic jet", Journal of Fluid Mechanics, 179:383-405, 1987.
- [6] Miller, R. S., Madnia, C. K., and Givi, P., "Numerical simulation of non- circular jets", Computers and Fluids, 24:1-25, 1995.
- [7] Hussain, F., Husain, H. S., "Elliptic jets. Part 1. Characteristics of unexcited and excited Jets", Journal of Fluid Mechanics, 208:257-320, 1989.
- [8] Shih, T-H., Liou, W. W., Shabbir, A., Yang, Z. and Zhu, J., "A new k- ϵ eddy viscosity model for high Reynolds number turbulent flows", Computers and Fluids, 24:227-238, 1995.
- [9] Gutmark, E., Schadow, K. C., "Flow characteristics of orifice and tapered jets", Physics of Fluids, 30:3448-3454, 1987.

- [10] Lovaraju P, Rathakrishnan E. Subsonic and Transonic Jet Control with Cross-Wire. American Institute of Aeronautics and Astronautics 2006; 44(11): 2700-2705.
- [11] Ricou, F. P., Spalding, D. B., "Measurements of entrainment by axisymmetrical turbulent jets", Journal of Fluid Mechanics, 11:21-3
- [12] Falcone, A., M., Cataldo, J. C., "Entrainment velocity in an axisymmetric turbulent jet", Journal of Fluids Engineering, 125:620-627, 2003.