I, Raj Narayan Gopalakrishnan, hereby submit this original work as part of the requirements for the degree of Master of Science in Aerospace Engineering.

It is entitled:
CFD Analysis of Turbulent Twin Impinging Axisymmetric Jets at Low Reynolds Number

Student's name: Raj Narayan Gopalakrishnan

This work and its defense approved by:

Committee chair: Peter Disimile, Ph.D.

Committee member: Shaaban Abdallah, Ph.D.

Committee member: Milind Jog, Ph.D.
CFD Analysis of Turbulent Twin Impinging Axisymmetric Jets at Low Reynolds Number

A thesis submitted to the
Graduate School
of the University of Cincinnati
in partial fulfillment of the
requirements for the degree of

Master of Science

in the Department of Aerospace Engineering and Engineering Mechanics
of the College of Engineering
November 2, 2017

by

Raj Narayan Gopalakrishnan
B.Tech, University of Kerala, 2010

Committee Chair: Dr. Peter J. Disimile
Dr. Shaaban Abdallah
Dr. Milind Jog
Abstract

CFD analysis of turbulent twin-impinging round jets was performed to establish the growth profile and velocity characteristics of resultant jet. To ensure that a high degree of confidence can be assigned to the planned multi-jet impingement simulations, a single axisymmetric jet was first numerically resolved and compared to published jet experiments results. This required the definition of inlet boundary condition of the jet to be accurately documented. To that end, a turbulent flow exiting a long circular pipe was first modelled and analyzed to ensure that the inlet boundary condition to a single axisymmetric jet was in good agreement with the experiments.

Once, the development length based on mean velocity was calculated, the pipe flow at a Reynolds number of 7,500 was analyzed and compared with in house and published experimental data. It was observed that the solution using the SST turbulence model performs better than the solution obtained using the Realizable k-ε model in the pipe domain. Once the analysis was completed, velocity and turbulence components at the outlet of the pipe were extracted and used as input for the single jet flow simulations.

Single axi-symmetric round jet flow was analyzed using computational techniques and validated with experimental results to establish the suitable turbulence model for simulation of low Reynolds number jets exiting from fully developed pipe. It was observed that although all the turbulence models studied could closely predict the mean velocity field, they were not able to accurately predict the turbulence intensity distributions. From the models studied, it was concluded that SST k-ω model was the best turbulence model for simulating low Reynolds number jet flow exiting from fully developed pipe.
Based on the insight gained from single jet analysis, CFD analysis on turbulent impinging jets was performed. Multiple Reynolds numbers and impingement angles were considered for the study to gain better understanding of the parameters affecting resultant jet growth and distribution. Based on the mesh obtained from grid sensitivity study, jets impinging at 30°, 45° and 60° at constant Reynolds number of 7500, and jets impinging at 30° angle at Reynolds number of 5000, 7500 and 10000 were numerically analyzed. It was observed that the profile of the resultant jet closely matched with the prediction of elliptical profile predicted by past researchers. It was also seen that higher jet growth was predicted in case of jets impinging at higher angle and higher momentum of jets were predicted in case of jets impinging with higher Reynolds number. Hence, it was concluded that based on the application required; ideal impinging strategy can be applied for obtaining optimal results.
Acknowledgements

All the work presented in this Master’s thesis is made possible due to the guidance and help of my research advisor - Dr. Peter Disimile – without whose patient guidance, I would have lost track and pursued too many activities, down many a rabbit holes. Thank you for all your help and guidance over the last 3 years.

I would like to thank Dr. Shaaban Abdallah and Dr. Milind Jog for taking the time to review this work and for being members of my thesis defense committee. Prof. Abdallah has always lent a patient ear to my doubts and clarified my queries regarding CFD methods.

To Brian Landers and the team at ESI – who provided the experimental data for the work – the effort you guys put into obtaining experimental results is deeply appreciated and heartfully acknowledged.

To Akshay, Dhaval, Karthik, Sanjay, Viswanathan – the questions during weekly meetings were quite invigorating and the insight you provided were very helpful. I will always be in your debt for the kind support and honest feedbacks that you always provided.

To my family and my friends – Your support, kindness and encouragement made me who I am today. I will forever be indebted to you.
Publications


Contents

Abstract ......................................................................................................................................................... i
Acknowledgements ......................................................................................................................................... iv
Publications ..................................................................................................................................................... v
Contents ........................................................................................................................................................ vi
List of Figures ............................................................................................................................................... ix
List of Tables .................................................................................................................................................. xiii
List of Equations .......................................................................................................................................... xiv
Nomenclature ................................................................................................................................................ xvi
   Greek Symbols ............................................................................................................................................. xvi

1 Introduction ............................................................................................................................................. 1
   1.1 Pipe flow ............................................................................................................................................. 1
   1.2 Round Jet Study ............................................................................................................................... 3
   1.3 Impinging Jet Study ......................................................................................................................... 5

2 Computational Methodology ............................................................................................................. 12
   2.1 Geometry Generation ....................................................................................................................... 12
   2.2 Mesh Generation ............................................................................................................................. 13
   2.3 Pre-Processing .................................................................................................................................. 16
      2.3.1 Quality Checks and Solver conditions ...................................................................................... 17
      2.3.2 Choice of Turbulence Models ................................................................................................. 17
      2.3.3 Material properties used .......................................................................................................... 19
      2.3.4 Boundary Conditions ................................................................................................................ 19
      2.3.5 Solution Methods ...................................................................................................................... 20
      2.3.6 Underrelaxation Parameters ................................................................................................... 20
      2.3.7 Convergence Criteria ............................................................................................................... 21
   2.4 Simulation and Post Processing ...................................................................................................... 21

3 Pipe Flow Analysis .............................................................................................................................. 23
   3.1 Geometry and Mesh Generation ....................................................................................................... 23
   3.2 Boundary conditions and Solver setup used ..................................................................................... 26
   3.3 Grid Sensitivity Study ...................................................................................................................... 27
3.4 Pipe Length Determination ................................................................. 31
3.5 Validation of Velocity Profile ............................................................ 39
  3.5.1 Normalized Velocity plot ............................................................... 39
  3.5.2 Law of the wall plot ................................................................... 41
  3.5.3 Velocity Defect plot .................................................................. 43
4  Single Jet Analysis .............................................................................. 46
  4.1 Geometry and Mesh Generation ....................................................... 46
  4.2 Turbulence Models used ................................................................. 50
  4.3 Boundary Conditions used ............................................................... 51
  4.4 Grid Sensitivity Study ...................................................................... 52
  4.5 Effect of Turbulence Models ........................................................... 56
    4.5.1 Velocity Profiles ..................................................................... 57
    4.5.2 Turbulence Profiles ................................................................. 63
5  Impinging Jet Analysis ....................................................................... 71
  5.1 Geometry and Mesh Generation ....................................................... 72
  5.2 Turbulence Models used ................................................................. 74
  5.3 Boundary Conditions and Solver used ............................................ 74
  5.4 False Diffusion study .................................................................... 77
  5.5 Full model vs Symmetry model ....................................................... 86
  5.6 Grid Sensitivity Study ..................................................................... 90
  5.7 Effect of Impingement Angle .......................................................... 92
    5.7.1 Velocity Profile Validation ......................................................... 92
    5.7.2 Velocity Contours ................................................................... 99
    5.7.3 TKE Contours ......................................................................... 102
    5.7.4 Center-line Velocity Profile .................................................... 105
    5.7.5 Center-line TKE Profile ............................................................ 108
    5.7.6 Spread profile of the jet .......................................................... 112
  5.8 Effect of Reynolds Number ............................................................ 121
    5.8.1 Velocity Contours ................................................................. 122
    5.8.2 TKE Contours ....................................................................... 125
    5.8.3 Center-line Velocity Profile .................................................... 129
    5.8.4 Center-line TKE Profile ............................................................ 131
    5.8.5 Spread profile of the jet .......................................................... 132
6 Conclusion ................................................................................................................................. 137
6.1 Pipe flow study .................................................................................................................. 137
6.2 Round jet study ............................................................................................................... 138
6.3 Impinging jet study ........................................................................................................ 139
7 Recommendations .......................................................................................................... 142
8 References .......................................................................................................................... 144
9 Appendix .................................................................................................................................. 148
  9.1 Comparison of impinging jet exiting from fully developed pipe against impinging jet exiting from short pipe........................................................................................................ 148
List of Figures

Figure 1: Injector plate used in F-1 rocket engine and modes of jet impingement used .......................... 7
Figure 2: Notation of flow field used by Elangovan [1996] ........................................................................... 10
Figure 3: Geometry of the Pipe ..................................................................................................................... 24
Figure 4: Inlet and Diameter .......................................................................................................................... 24
Figure 5: Mesh along the centerline of the pipe ............................................................................................. 25
Figure 6: Mesh at Inlet and at Walls .............................................................................................................. 25
Figure 7: Close view of O-Grid mesh at Inlet ................................................................................................. 26
Figure 8: Velocity development along flow direction .................................................................................... 28
Figure 9: Turbulence kinetic energy development along flow direction ...................................................... 29
Figure 10: Grid sensitivity study based on Velocity value ........................................................................... 30
Figure 11: Grid sensitivity study based on Turbulence Kinetic energy value ............................................. 30
Figure 12: Development length [39] ........................................................................................................... 32
Figure 13: y’ plot for the pipe ....................................................................................................................... 33
Figure 14: Centerline velocity profile for 1m long pipe with different Turbulence Models ..................... 34
Figure 15: Centerline turbulent kinetic energy profile for 1m pipe with different Turbulence Models .... 35
Figure 16: Centerline velocity profile for different pipes lengths with Realizable k-ε model .................. 36
Figure 17: Centerline velocity profile for different pipes lengths with SST model .................................. 36
Figure 18: Centerline turbulence kinetic energy profile for different pipes lengths with Realizable k-ε model .......................................................................................................................... 37
Figure 19: Centerline turbulence kinetic energy profile for different pipes lengths with SST-model ...... 38
Figure 20: Normalized velocity plot - Real k-ε model compared with Landers’ and Eggels’ data .......... 40
Figure 21: Normalized velocity plot - SST model compared with Landers’ and Eggels’ data ............... 40
Figure 22: Law of wall plot - SST data vs Eggels ....................................................................................... 42
Figure 23: Law of wall plot - Real KEp data vs Eggels ............................................................................... 43
Figure 24: Velocity defect data comparison – CFD SST model vs Eggels ............................................... 45
Figure 25: Velocity defect data comparison – CFD Real KEp model vs Eggels .................................... 45
Figure 26: Geometry for single axi-symmetric jet simulation ................................................................. 47
Figure 27: Side view of geometry for single axi-symmetric jet simulation ............................................. 48
Figure 28: Front view and close-ups showing the mesh with 4.3 million nodes in detail ......................... 49
Figure 29: Side view of mesh for single axi-symmetric round jet ........................................................... 50
Figure 30: a) Velocity and b) Turbulence Kinetic energy used as inlet condition for the round jet ....... 52
Figure 31: Front view of grid with a) 0.25 million nodes and b) 4.3 million nodes ............................... 53
Figure 32: Side view of central plane for grid a) 0.25 million nodes and b) 4.3 million nodes ......... 54
Figure 33: Grid sensitivity study based on Mean Velocity value ............................................................... 55
Figure 34: Grid sensitivity study based on Turbulence Kinetic energy value ........................................ 55
Figure 35: Velocity contour along central plane for single axi-symmetric jet obtained using various turbulence models .............................................................................................................. 58
Figure 36: Zoomed view of velocity contour along central plane for single axi-symmetric jet obtained using various turbulence models .................................................................................. 59
Figure 37: Profile of normalized velocity along centreline ...................................................................... 60
Figure 38: Radial profile of normalized velocity over radius normalized by half width at 3D from pipe exit .................................................................................................................................................. 61
Figure 78: Non-dimensional velocity profile along 14D in horizontal plane for 30-degree impingement case

Figure 79: Non-dimensional velocity profile along 16D in horizontal plane for 30-degree impingement case

Figure 80: Non-dimensional velocity profile along 18D in horizontal plane for 30-degree impingement case

Figure 81: Non-dimensional velocity profile in horizontal plane for 30-degree impingement case

Figure 82: Non-dimensional velocity profile in horizontal plane for 45-degree impingement case

Figure 83: Non-dimensional velocity profile in horizontal plane for 60-degree impingement case

Figure 84: Non-dimensional velocity profile in vertical plane for 30-degree impingement case

Figure 85: Non-dimensional velocity profile in vertical plane for 45-degree impingement case

Figure 86: Non-dimensional velocity profile in vertical plane for 60-degree impingement case

Figure 87: Velocity profile along horizontal plane for jet impingement angle of 30 degrees

Figure 88: Velocity profile along horizontal plane for jet impingement angle of 45 degrees

Figure 89: Velocity profile along horizontal plane for jet impingement angle of 60 degrees

Figure 90: Velocity profile along vertical plane for jet impingement angle of 30 degrees

Figure 91: Velocity profile along vertical plane for jet impingement angle of 45 degrees

Figure 92: Velocity profile along vertical plane for jet impingement angle of 60 degrees

Figure 93: TKE profile along horizontal plane for jet impingement angle of 30 degrees

Figure 94: TKE profile along horizontal plane for jet impingement angle of 45 degrees

Figure 95: TKE profile along horizontal plane for jet impingement angle of 60 degrees

Figure 96: TKE profile along vertical plane for jet impingement angle of 30 degrees

Figure 97: TKE profile along vertical plane for jet impingement angle of 45 degrees

Figure 98: TKE profile along vertical plane for jet impingement angle of 60 degrees

Figure 99: Velocity profile along the center line for various impingement angles

Figure 100: Normalized velocity profile along the center line for various impingement angles

Figure 101: Normalized velocity profile along the center line compared with 1/x profile

Figure 102: Normalized TKE profile along the center line for 30-degree jet impingement

Figure 103: Normalized TKE profile along the center line for 45-degree jet impingement

Figure 104: Normalized TKE profile along the center line for 60-degree jet impingement

Figure 105: Normalized TKE profile along the center line for various angles of impingement

Figure 106: Turbulence intensity profile along center line for various angles of impingement

Figure 107: Profiles of jet at various locations after impingement for 30-degree case

Figure 108: Profiles of jet at various locations after impingement for 45-degree case

Figure 109: Profiles of jet at various locations after impingement for 60-degree case

Figure 110: Profiles of jet at 12D for various impingement angles

Figure 111: Profiles of jet at 16D for various impingement angles

Figure 112: Profiles of jet at 20D for various impingement angles

Figure 113: Comparison of velocity profile with elliptical profile at 12D for 30-degree case

Figure 114: Comparison of velocity profile with elliptical profile at 16D for 30-degree case

Figure 115: Comparison of velocity profile with elliptical profile at 20D for 30-degree case

Figure 116: Comparison of velocity profile with elliptical profile for 30-degree case

Figure 117: Comparison of velocity profile with elliptical profile for 45-degree case

Figure 118: Comparison of velocity profile with elliptical profile for 60-degree case
Figure 119: Velocity profile along horizontal plane for jets impinging at Re=5,000 ..........123
Figure 120: Velocity profile along horizontal plane for jets impinging at Re=7,500 ..........123
Figure 121: Velocity profile along horizontal plane for jets impinging at Re=10,000 .......123
Figure 122: Velocity profile along vertical plane for jets impinging at Re=5,000 ..........124
Figure 123: Velocity profile along vertical plane for jets impinging at Re=7,500 ..........124
Figure 124: Velocity profile along vertical plane for jets impinging at Re=10,000 .......125
Figure 125: TKE profile along horizontal plane for jets impinging at Re=5,000 ..........126
Figure 126: TKE profile along horizontal plane for jets impinging at Re=7,500 ..........126
Figure 127: TKE profile along horizontal plane for jets impinging at Re=10,000 .......126
Figure 128: TKE profile along vertical plane for jets impinging at Re=5,000 ..........127
Figure 129: TKE profile along vertical plane for jets impinging at Re=7,500 ..........128
Figure 130: TKE profile along vertical plane for jets impinging at Re=10,000 .......128
Figure 131: Velocity profile along the center line for various Reynolds number .......129
Figure 132: Normalized velocity profile along the center line for various Reynolds number ....130
Figure 133: Normalized TKE profile along the center line for various Reynolds number ....131
Figure 134: Turbulence intensity profile along center line for various Reynolds number ....132
Figure 135: Profiles of jet at various locations after impingement for Re=7,500 .......133
Figure 136: Profiles of jet at various locations after impingement for Re=5,000 .......133
Figure 137: Profiles of jet at various locations after impingement for Re=10,000 ......134
Figure 138: Profiles of jet at 12D for various Reynolds number ..........134
Figure 139: Profiles of jet at 16D for various Reynolds number ..........135
Figure 140: Profiles of jet at 20D for various Reynolds number ..........135
Figure 141: Velocity profile along the center line for cases with different pipe development length ......149
Figure 142: TKE profile along the center line for cases with different pipe development length ......149
List of Tables

Table 1: Mesh quality check parameters............................................................................................................. 15
Table 2: Mesh quality parameters used..................................................................................................................16
Table 3: Value of material properties used ..............................................................................................................19
Table 4: Under Relaxation parameters used ...........................................................................................................21
Table 5: y' Calculation table.....................................................................................................................................24
Table 6: Aspect ratio across various mesh configuration ..........................................................................................27
Table 7: Comparison of values predicted by Turbulence model .............................................................................39
Table 8: Data taken from Eggels..........................................................................................................................44
Table 9: Data calculated from Eggels....................................................................................................................44
Table 10: Grid Independence data ........................................................................................................................56
Table 11: Explanation of parameters used in false diffusion equation .................................................................79
Table 12: Grid independence data ........................................................................................................................92
Table 13: Growth profile of resultant jet in y and z direction..................................................................................117
Table 14: Growth profile of resultant jet in y and z direction for various Reynolds number .........................136
List of Equations

Equation 1: Equation for $y'$ calculation ................................................................. 14
Equation 2: Reynolds Number .............................................................................. 26
Equation 3: Equation for calculation of entrance length for turbulent flow ............... 32
Equation 4: Expression for false diffusion .................................................................. 78
Nomenclature

Acronyms

CFD – Computational Fluid Dynamics

HWHM- Half width at the half Maximum

HIA – Half Impingement Angle

FIA – Full Impingement Angle

TKE – Turbulence Kinetic Energy

TI – Turbulence Intensity

SIMPLE - Semi-Implicit Method for Pressure Linked Equations

FD – False Diffusion

RANS – Reynolds Averaged Navier-Stokes

LES – Large Eddy Simulation

Symbols

$U_c$ – Centerline Velocity (m/s)

$U$ – Local Velocity (m/s)

$U_{avg}$ – Average Velocity (m/s)

$D$ – Diameter of pipe (mm)

$k$ – Turbulence kinetic energy (m$^2$/s$^2$)

$r_{1/2}$ – Jet half width (m)
\( y^+ \) – Non-dimensional wall distance

\( y \) – First node height (mm)

\( u_t \) – Wall Shear Velocity (m/s)

\( \text{Re} \) – Reynolds number

\( L_e \) – Entrance length (mm)

\( b_1 \) – half width of jet in plane of pipe

\( b_2 \) – half width of jet in plane perpendicular to the plane of pipe

**Greek Symbols**

\( \varepsilon \) – Dissipation rate (m²/s³)

\( \omega \) – Specific Dissipation Rate (s⁻¹)

\( \nu \) – Kinematic Viscosity (m²/s)

\( \theta \) - Half impingement angle (degrees)
1 Introduction

For the impingement jet study, it was decided that the impinging jet flow exiting into ambient atmospheric condition must replicate the experimental conditions used by Disimile et al. [1] and followed by Brian Landers [2]. To ensure that confidence can be allocated to the results obtained for impinging jet study, it was decided that a single jet at the same Reynolds number will be initially studied (both experimentally and computationally) which will act as a baseline model. Prior to the single jet study, it was planned that the a single pipe flow at the same Reynolds number will be numerically studied, the outlet condition of which will act as the jet inlet discharge condition.

The literature review that has been undertaken to gain technical insight before conducting computational study on impinging jets can be classified into 3 sections based on the component under consideration – Pipe flow, Single round jet study and Impinging jet study. From the literature review, the previously established works that has provided deep insight into the flow physics for each section is mentioned below.

1.1 Pipe flow

Flow of the fluid through circular pipes is one of the most fundamental flows with large industrial and research applications. The flow of water through the pipe system in a building, the flow of blood through the arteries and veins, and the flow of oil through long distance pipes are simple examples of pipe flow that shows its ubiquitous nature. The pipe flow can either be laminar, turbulent or transitional based on the Reynolds number. Though simple enough to be explained in an undergraduate fluid mechanics course, turbulent pipe flow holds significant research potential and is still a topic of research.
One of the seminal works in turbulent pipe flow was performed by Nikuradse [3] in early 1930’s, who studied the effect of roughness on the pipe flow since numerous earlier studies mainly concentrated on smooth pipes. He suggested that an entrance length of 40 diameters was sufficient for obtaining a fully developed flow in a rough pipe while it takes more (approximately 50 diameters) to achieve full development in case of smooth pipe. From these studies, he concluded that the friction factor only depended on Reynolds number in some conditions. It was his observation that for both smooth and rough pipes at small Reynolds numbers, the friction factor was almost same and increased slightly with increase in Reynolds number. A further increase in Reynolds number did not effectively increase the friction factor, and the friction factor became a function of relative roughness of the pipe.

Laufer [4], meanwhile recommends a lower value of development length for turbulent pipe flow. His suggested value of 30 diameters for development length has been questioned by Lien et al. [5] who also points that the Laufer’s prediction of 55 heights for fully developed channel flow is inaccurate. Lien’s views may be correct since he was able to corroborate his results with multiple research studies done after Laufer’s study. Dean [6] had tabulated the entrance length required for fully developed flow in channels and shown that Laufer’s prediction of 55 heights is one of the smallest, if not the smallest value of development length for channel flow. His own prediction for development length in channel flow was 108 heights (significantly larger than Laufer’s value).

Laufer [4] had also shown the effect of Reynolds number in the mean and instantaneous velocity profile. This observation is noted by other researchers of which Toonder [7] and Wagner et al. [8] has the Reynolds number close to the value of interest to the current study. Toonder compared his experimental values with Eggels et al. [9] Direct Numerical Simulations (DNS) data and showed good agreement. This bolsters Toonder’s results and adds credibility to his view that the flow
profie is Reynolds number dependent. The detailed view of logarithmic layer for various Reynolds numbers plotted by Toonder also shows the variation of the velocity profile over Reynolds number range.

Wagner et al. [8] on the other hand performed a DNS analysis for bulk Reynolds number ranging from 5,300 to 10,300 and compared his solution with Laser Doppler Anemometry (LDA) data obtained by Westerweel et al. [10]. His work also showed the dependence of flow profile in the logarithmic region on Reynolds number. Eggels et al. [9] meanwhile performed both DNS and experimental LDA, Particle Image Velocimetry (PIV), and Hot Wire Anemometry (HWA) analyses on turbulent pipe for a Reynolds number of approximately 7,000. Their study showed an excellent match between the experimental data and computational data thereby establishing a good reference dataset to which other researchers could compare.

A Reynolds Averaged Navier Stokes (RANS) and Large Eddy Simulations (LES) based analysis of turbulent pipe flow was performed by Vijiapurapu and Cui [11], who evaluated the performance of various turbulence models in the pipe domain. They employed k-ε model, k-ω model, RSM model, and LES in their analysis and observed that for the Reynolds number used in their study (Re = 100,000), the various turbulence models and LES were in close agreement. However, it be noted that their analysis considered a Reynolds number more than an order of magnitude above the Reynolds number used in the present study.

1.2 Round Jet Study

Round jet flow is a form of free shear flow that has multitude of industrial applications, from water jet exiting the nozzles used by firemen for fire suppression to fuel jet in aircraft combustion chambers. Jet flow observed in nature, such as thermal plumes and volcanic exhausts also exhibit a circular profile and has been source of investigation for Bejan et al. [12]. The wide range of
applications has ensured that a large number of research work was done to establish some of its underlying physics.

Round jet flows can exist in either a laminar or turbulent state; and for our current study we are studying turbulent axisymmetric round jets. An exhaustive review on turbulent round jets has been conducted by Ball et al. [13] in 2012, who examined the work of Tollmein (done in 1926) to round jet analysis done as recent as 2010. This work provided immense and valuable insight into the experimental and computational work done in the field of round jets. Their work briefly discusses the effect of initial conditions of the jet on similarity profile; which has been established by other authors. They also discuss the coherent structures observed in round jets and the length scales associated with the flow. At the end of their work, they have pointed out the parameters and physics of the round jet that still remain unknown; mainly the nature of interaction of the coherent structures and the mixing transition. It is indeed interesting to notice that even in 2012, the complete physics of such a ubiquitous fluid dynamic phenomenon was not completely understood.

Meanwhile, Kaushik et al. [14] and Dewan et al. [15] had performed a detailed survey on CFD (Computational Fluid Dynamics) treatment of round jet, which provided excellent insight into the recent computational works done in the field of research on round jets. While Kaushik et al. [14] discuss both laminar and turbulent jets, Dewan et al. [15] has concentrated on turbulent plane and round jets thereby providing a generalized view on jets and the recent research done.

One of the important aspect of round jets is the similarity nature of its mean velocity profile. It was believed from data obtained from experimental analysis that the round jet velocity profile exhibited self-similar nature, and that the profile was similar for all jets irrespective of the inlet flow conditions. It was George W. K. [16], who in his pivotal work established the dependence of turbulent scalar properties on inlet flow conditions using analytical methods. This work proved
instrumental in initiating new generation of research where the initial conditions of jet flow and its impact were analyzed. Mi et al. [17] examined round jets with two different initial conditions (a top hat profile and a fully developed pipe flow) and were able to establish the dependence of turbulence scalar properties on initial conditions similar to that presented by George W. K. [16]. Similar study was conducted by Ferdman et al. [18] who studied the effect of initial conditions on velocity and turbulence characteristics. Their study displayed a dependence of decay rates of jets on initial flow condition. From these studies, it was concluded that the round jet velocity and turbulence profile has direct association with initial conditions.

Fellouah et al. [19] established that there was direct correlation between the mean velocity condition at inlet to the turbulence and Reynolds stress profiles downstream. Their observation was similar to those made by Dimotakis [20] who established a Reynolds number range of 1-2 x 10^4 based on outer scale for the flow to become sustainably turbulent. These studies helped to establish the dependence of flow physics on the Reynolds number. From these studies, it was concluded that when validating computational results of axisymmetric round jet, unique data set data was acquired from specific experimental setup, where both initial conditions and Reynolds number matched. Any mismatch in boundary or flow conditions would affect data set quality and lead to a possible mismatch. Hence, work done by Landers [2] was used for validation since the experimental conditions including the initial flow profile generation and Reynolds number matched with the current study.

1.3 Impinging Jet Study

Impinging jets has large number of industrial applications since it is capable of enhancing fluid mixing and increasing the spread rate of the resultant jet. The application of impinging jets has been found in space propulsion rocket fuel-oxidizer mixing chamber, in internal combustion
engines and in spray generation. The team at ESI (Engineering & Scientific Innovations, Inc) is currently looking at establishing the use of impinging jets as an option for fire suppressant discharge.

The fuel-oxidizer injector plate used in F-1 rocket engine of Saturn V rocket, along with the cross section of injector plate and mode of impingements is shown in Figure 1. Other research activities [21], looking at the F-1 injector which consisted of 15 rings of oxidizer holes and 14 rings for fuel holes, with total of 2,832 orifices.

“The orifices are arranged in pairs such that the propellant being expelled through the holes intersect or impinge in a doublet, like-on-like pattern (i.e., two streams of oxidizer impinge, and two streams of fuel impinge). The impinging jets atomize, the fuel and oxidizer vapors intermix, the mixed vapors react (combust), and the resulting hot gases flow out of the combustion chamber to produce thrust. Other impingement patterns tested (and rejected) include a triplet impingement pattern (where three streams intersect) and like-on-unlike (where fuel and oxidizer impinge).”

-Quote from [21]
The work done by Disimile et al. [1] has provided insight into the spread characteristics of the resultant jet after impingement at constant Reynolds number of 7500. They studied the round impinging jets at 2 angles; 30 and 45 degrees. From their study, they observed that growth characteristics of the resultant jet along plane normal to impinging jet plane after 45-degree impingement was approximately 50% more than the growth after 30-degree impingement. They also observed elliptical profile for the resultant jet, similar to the observation made by Rho et al. [22] with major axis being 2.5 times the minor axis in case of 45-degree impingement and 1.6 times the minor axis in case of 30-degree impingement. The significant difference in growth along the major axis when compared with minor axis for different impingement angles is an indicator of its effect in resultant jet characteristics.

Rho et al. [22] performed an experimental study on cross jet mixing flows exiting nozzle condition. They considered circular nozzles and an impingement angle of 45-degrees for their study. The Reynolds number considered for the case under study was 52,000 and 65,000. They observed an
elliptical profile for the resultant jet after impingement, which has consistently been noted by other researchers. The shape of the resultant jet was observed to shift from elliptical profile to circular profile further downstream. It was also concluded that, beyond impingement zone; the mean velocity profile can be correlated to semi-empirical equations based on jet half-width.

Landers and Disimile [23] [24] [25] [26] has done significant amount of work on the near field of single jets, which were used as the baseline case for their work on impinging jets. Landers [2] congegrated the results from the above-mentioned studies into his thesis based on the experimental analysis performed on impinging jets at various angles. Hence, for the current study; experimental data from Landers [2] is used for the validation of results obtained from CFD analysis to ensure reasonable agreement.

Prof. N. Rajaratnam of University of Alberta has performed significant research on topics related to turbulent jets in general and impinging jets in particular. His book on turbulent jets [27] has been an ideal technical source for the single jet study. Rajaratnam and Khan [28] studied impinging jets at 4 different angles (30, 60, 90 and 120-degrees) at Reynolds number of 30,000. They established the physics of the flow based on 2 regions; a zone from nozzle exit to impingement point, and a zone beyond impingement point. Detailed observations related to pressure and velocity characteristics were made in these regions for analysis. They observed that beyond the impingement point, the growth of the jet in the plane normal to the plane of nozzle was thrice the growth observed along the plane of nozzle. Also, as they moved further downstream, the flow tended to become axisymmetric with growth the rate persisting at 1.5 times the growth rate for single round jet.

Rajaratnam and Wu [29] performed experimental study on impinging jets at 60-degree angle with unequal momentum. They considered incompressible flows with different velocities exiting each
of the nozzle. The value of velocities studied ranged from 0 to 42 m/s and the ratio of velocities considered were 0.39, 0.47, 0.59, 0.68 and 0.79. Hence, none of their cases considered symmetric profile. Similar to [28], the physics of the flow was established based on the location with respect to impingement point. Beyond impingement point, they observed that the resultant jet axis can be predicted using momentum considerations. They observed that the growth of the jet in plane of the nozzle did not significantly depend on the velocity ratio. But along the plane normal to the plane of nozzle, they were able to establish direct correlation between the velocity ratio and growth profile. Similar to the observation made in [28], the growth of the combined jet in the plane normal to the plane of nozzles was significantly more than the growth in the plane of the nozzles.

Hall et al. [30] worked on submerged axisymmetric turbulent intersecting jets to establish the properties of the flow field. Previous work proved that the vector addition of velocity field or momentum flux densities may not be the ideal approach to combine the properties of intersecting flows. Hence, they considered two normal planes to study the flow characteristics – one in-line with the nozzles and one plane perpendicular to nozzle plane. For submerged jets, they observed that along with a forward flow; a significant reverse flow exists in the plane along the nozzle. From their study, they concluded that the spread of the reverse flow was significant along the plane perpendicular to the nozzle when compared to the spread along the nozzle plane. The spread along the forward direction was found to be symmetric in both the planes and followed Gaussian profile.

Elangovan et al. [31] presented their work on interaction of twin intersecting axisymmetric turbulent jets at low Mach numbers (0.2). The exit diameter of the nozzles was 10mm each and the spacing between the centerlines of the nozzles was 31mm with impingement angles considered being 0, 10, 20 and 30-degrees. They established 3 zones based on jet interactions as shown in Figure 2 –merging region, combining region and combined region.
As seen in Figure 2, significant recirculation occurs in the merging region with ambient flow entrainment into the free shear layers of jets. At the Merging Point (MP), the free shear layers of individual jet interact with one another. Combining region was defined as the zone beyond MP extending up to the location where the centerline velocity becomes maximum. This location was defined as Combining Point (CP). Beyond CP, the region was characterized by the resultant jet resembling a single jet flow. Hence, this region was suitably named combined region.

From their study [31], they observed that the near field flow physics was strongly dependent on the impingement angle, and that the resultant flow field downstream of combining point resembled elliptic profile. They also observed the axis-switching characteristics which is considered a phenomenon closely associated with non-circular jets. Like previous researchers, they also observed that the growth of the resultant jets in the plane normal to the nozzle plane was significantly higher than growth in the nozzle plane. Regarding the entrainment of the ambient fluid into the jet shear layer, they observed that the entrainment was higher at lower angle, with
maximum being at 0 degrees and consistently reduced with increasing impingement angle. Similarly, the team [32] studied the effect of impingement angle and distance between nozzles centers for sonic and supersonic conditions.

An observation made by Landers [2] that seems to be very astute is that only limited amount of published research work has been observed for impinging jets, either computational or experimental. Most of the published results available are very application-specific and mainly dealing with supersonic flows. Hence, obtaining well established data set for validation has proven quite difficult for the case under consideration. Considering the ubiquitous nature of impinging jet in industrial application, it can only be concluded that the results, if any, has not been tabulated or published either because of sensitive nature of application (such as space craft propulsion fuel mixing, defense-related study) or because of proprietary nature of application (such as impinging jets in combustion chambers of engines, spray formation systems). Hence, it was considered very significant to establish one of the initial works on impinging jets at low Reynolds number involving both computational and experimental side.
2 Computational Methodology

Any CFD analysis performed to gain understanding of the underlying flow physics involves the following steps:

2. Mesh Generation
3. Pre-Processing
4. Solving
5. Post-Processing.

The detailed explanation of each steps for the corresponding analysis is mentioned below and in later sections.

2.1 Geometry Generation

The starting point of every CFD (Computational Fluid Dynamics) analysis is the creation of the geometry, which ideally defines the flow path under consideration. Based on the complexity of the geometry, in some cases model simplifications can be performed. Model simplification would imply reducing the complexity of the model without altering the flow physics, thereby leading to reduced mesh count and computational power requirement. This has to be done by a person with insight on flow path and with the understanding that any simplification will affect the results at some level.

In the current study, fluid flow through a pipe that exits into ambient domain requires only a simple cylindrical geometry ending into a large domain and does not require any geometry modification. For geometry generation, initial calculations based on the literature review were performed to estimate the entrance length required for the flow to become fully developed just upstream of the
pipe exit. The length of pipe section longer than the calculated value was used for the creation of computational geometry.

For single jet and impinging jet study, a domain representing ambient conditions of standard atmospheric temperature and pressure was modelled. For the single jet study, it was modelled as a truncated frustum of a cone and for impinging jet study, it was modelled as a cuboidal domain based on meshing philosophy applied. The size of the ambient domain was selected to be large enough so as to ensure no significant numerical interaction with the jet flow under consideration, and small enough to reduce computational resource usage. The details regarding the dimensions for the domains are mentioned in respective sections.

The commercial computer-aided design (CAD) software by Dassault Systèmes, SolidWorks 2015 and 2016 has been used for generation of the geometries for the current study. The geometry files are saved as *.sldprt files which is the native file format in SolidWorks. These file types are later converted into *.step files, which can be easily imported into the mesh generation software.

2.2 Mesh Generation

In any computational simulation, the overall volume of the flow path has to be discretized into finite number of smaller volumes called meshes. The quality and quantity of the mesh is generally defined by the geometry under investigation. The quality determines the degree of accuracy while the node count affects computational time and power required. Hence, using an appropriate mesh count with high quality mesh elements is critical to every computational study.

One of the major factors that define the mesh for any geometry is the turbulence model to be used in the analysis. Each turbulence model has its own specific requirements for the height of the first node away from the bounding wall. Usually the displacement of the first node in the mesh is
defined by inner wall variables utilized in turbulent boundary layer analysis and is given by the variable \( y^+ \). This quantity \( y^+ \) represents the non-dimensional wall distance which is determined as the ratio of product of first node height and friction velocity to kinematic viscosity.

\[
y^+ = \frac{y^* u_\tau}{\nu}
\]

Equation 1: Equation for \( y^+ \) calculation

An accurate value of \( y^+ \) ensures that the mesh near the wall is properly positioned and the flow physics near the wall can be adequately captured. In the current study, both epsilon (\( \varepsilon \)) and omega (\( \omega \)) based eddy viscosity turbulence models are utilized. The general agreed-upon view is that the epsilon based turbulence models require a minimum \( y^+ \) of 30 since it uses wall function to resolve viscous sublayer and buffer layer. Meanwhile, the omega based models have more accurate near-wall capturing capability and use meshes with \( y^+ \) of approximately 1. Hence, different models required the generation of different meshes.

But, based on the literature review for an axisymmetric jet, it was believed that the Realizable k-\( \varepsilon \) model was the model of choice with high level of accuracy. Mesh requirements for that model was investigated and it was indicated that the Realizable k-\( \varepsilon \) model had the capability to resolve down to viscous sublayer while analyzing wall bounded flows. Hence, for the pipe flow analysis; it was evident that the same mesh configuration could be used for both the turbulence models.

Mesh at the outlet of the pipe domain defines the mesh at the inlet into the ambient domain, since they are inter-dependent. Hence, while generating corresponding mesh for the single jet and impinging jet domain, care was taken to ensure that there exists one-to-one mesh connectivity between the mesh at the exit of pipe domain and at the inlet of jet domains.
Commercial simulation software package, Ansys 16.2 has been used for the generation of native geometry, meshing of flow path, pre-processing, solving and post-processing for all the cases in the current study. The bundled mesh creation software, ICEM-CFD has been used for mesh generation for pipe flow, single jet flow and impinging jet flow study.

While setting up the mesh, various quality checks were performed on the grid in order to ensure that the grid did not contain any mesh elements of low quality. Mesh elements with lower quality can generate inaccurate results and hence should be avoided. For the current study, the following ICEM-CFD quality check variables were used and defined in Table 1: Determinant, Angle, Aspect ratio and Quality as found in [33].

<table>
<thead>
<tr>
<th>Variable</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Determinant</td>
<td>The value ranges from 0-1 with 1 representing perfectly consistent prism mesh while 0 represents bad mesh with degenerate edges. Negative value of determinant represents inverted cells with negative (non-physical) volume.</td>
</tr>
<tr>
<td>Aspect Ratio</td>
<td>Ratio of minimum element edge length to maximum edge length. An aspect ratio close to 1 is ideal for simulation but increases the mesh count. New generation CFD solvers can support meshes with aspect ratio up to 100.</td>
</tr>
<tr>
<td>Angle</td>
<td>Minimum value of internal angle for each element. If the mesh is skewed, the internal angle will be low, and the solution obtained from this mesh may not be as accurate.</td>
</tr>
<tr>
<td>Quality</td>
<td>Weighted value of Determinant and Orthogonality.</td>
</tr>
</tbody>
</table>

Table 1: Mesh quality check parameters

All the meshes generated and used for the grid independent study were rigorously checked to ensure that they didn’t have any low-quality mesh cells. The minimum value of quality parameters used in the meshes is shown in Table 2: -
<table>
<thead>
<tr>
<th>Variable</th>
<th>Minimum Value used</th>
</tr>
</thead>
<tbody>
<tr>
<td>Determinant</td>
<td>0.4</td>
</tr>
<tr>
<td>Aspect Ratio</td>
<td>&lt;100</td>
</tr>
<tr>
<td>Angle</td>
<td>30</td>
</tr>
<tr>
<td>Quality</td>
<td>0.4</td>
</tr>
</tbody>
</table>

Table 2: Mesh quality parameters used

Ansys Fluent has the capability to use skewed meshes with the aspect ratio of up to 100. In this study, it was always confirmed that the aspect ratio is held below the maximum supported value. After achieving the required mesh qualities, the mesh file was saved as a *.blk file (block file). On completion of meshing, the mesh file suitable for input into Ansys Fluent was generated and exported.

2.3 Pre-Processing

The commercial CFD solver, Ansys Fluent 16.2 was used to computationally solve the flow field in the pipe and jet domain. The mesh generated from ICEM CFD was imported into Fluent where a preliminary quality check was again conducted. Since the flow is turbulent, one of the primary factors to choose was an appropriate turbulence model. The turbulence models are usually selected based on the application; some models are said to behave better in internal flows while others in external flows. Since the main aspect of this analysis is to extract velocity profiles from jet flows, the turbulence model capable of handling free shear flows was initially chosen. The details regarding turbulence models, boundary conditions, materials used, and solution parameters considered are described in sections below.
2.3.1 Quality Checks and Solver conditions

Once the mesh is imported into Ansys Fluent, a preliminary quality check is performed to ensure that basic mesh quality parameters like Orthogonal quality, Orthogonal Skew and Aspect ratio is maintained. Once completed; pressure based solver (ideal for incompressible flows) and steady state simulation setup was chosen as the solver type.

2.3.2 Choice of Turbulence Models

Based on the literature review on axisymmetric jets, it was noted that the Realizable k-ε model performs well for free shear flows since it has been modified to accurately account for the spread rates of planar and round jets. T.-H. Shih et al. [34] who modeled the aforementioned model notes that the effect of rotation on both turbulent kinetic energy (k) and eddy dissipation rate (ε) has been well captured in Realizable k-ε model and hence it outperforms standard k-ε model for free shear flows. However, the SST model was also considered for the current analysis since claims of SST model performing better than standard k-ε model were made in case of plane jet flows by Menter [35]. Since a comparison between Realizable k-ε model and SST model for axisymmetric jet was not found by the author, a study to compare the performance of each of these eddy viscosity models on the pipe flow, which serves as the inlet boundary condition for jet flow was undertaken. As such, a brief summary of these models is provided below.

Realizable k-ε model

The Realizable k-ε model was generated as an improvement over Standard k-ε model at NASA by T.-H. Shih et al. [34]. The “realizability” condition implies that:

1. Schwarz’ inequality for turbulent shear stress is satisfied
2. Normal stresses are not allowed to have negative values.
Realizable k-\(\varepsilon\) model uses a variable value for the eddy viscosity coefficient \(C_\mu\) in the turbulence kinetic energy and dissipation rate equation whereas the standard k-\(\varepsilon\) model uses a constant value of 0.09. Though Realizable k-\(\varepsilon\) model is based on standard k-\(\varepsilon\) model, it outperforms its predecessor significantly by accurately capturing the spread rate of planar and round jets.

**SST Model**

The k-\(\omega\) SST turbulence model, or commonly called SST model [35] is one of the most popular RANS turbulence model in use today. It is a combination of k-\(\omega\) model in the boundary layer flow and standard k-\(\varepsilon\) model for outer layer. The near-wall capability of k-\(\omega\) model to capture the viscous sublayer without using wall functions makes the SST model ideal for internal flows and for flows where boundary layer capture is crucial. It is also designed to blend into standard k-\(\varepsilon\) model in case of free shear flows. It is one of the most touted turbulence model with the capability to correctly simulate flows with adverse pressure gradients and separation zone.

**Standard k-\(\varepsilon\) model**

Proposed in 1974 by Launder and Sharma [36], it is one of the most common and oldest turbulence model in use. It’s a semi-empirical model with the turbulent kinetic energy (k) and dissipation rate (\(\varepsilon\)) based on model transport equations. It has been observed that the Standard k-\(\varepsilon\) model performs poorly in case of adverse pressure gradients. As further research provided conclusive evidence on the strength and weakness of the Standard k-\(\varepsilon\) model; modifications were made to it to improve its performance. Such modifications have led to the creation of few variations, of which two models are RNG k-\(\varepsilon\) model and Realizable k-\(\varepsilon\) model.
**Standard k-ω model**

Standard k-ω model by Wilcox [37] is another popular model among CFD users. It is an empirical model based on model transport equations for the turbulence kinetic energy (k) and the specific dissipation rate (ω). The Standard k-ω model has better wall physics capturing capability than Standard k-ε model and can handle meshes with y+ closer to 1. Similar to Standard k-ε model, it has many weaknesses due to which modifications were made, which eventually lead to the creation of SST k-ω model.

### 2.3.3 Material properties used

Material properties for air is chosen from the Fluent material database and used for the calculation of Reynolds number and y+ parameters. The value of various material properties used is as shown in Table 3 below.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>1.225</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Specific Heat</td>
<td>1006.43</td>
<td>J/kg-K</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>0.0242</td>
<td>w/m.K</td>
</tr>
<tr>
<td>Dynamic Viscosity</td>
<td>1.79E-05</td>
<td>kg/m.s</td>
</tr>
</tbody>
</table>

Table 3: Value of material properties used

### 2.3.4 Boundary Conditions

Once the turbulence model to be used in the analysis is selected, the next step is to choose the boundary setup which will provide a good approximation of the real-world physics. The choice of boundary condition was based on the parameters available. For a single pipe simulation, velocity was provided as the inlet boundary condition with outlet boundary condition being pressure outlet...
of 1 bar (standard atmospheric condition) to replicate the flow exhausting into ambient conditions. The wall of the pipe was modelled as no-slip smooth walls. For single and impinging jet study, the velocity and TKE data from the exit of the pipe was used as the inlet condition. More details regarding the boundary conditions for each stage of analysis is mentioned in the corresponding stage.

2.3.5 Solution Methods

For pressure-velocity coupling, SIMPLE scheme by Patankar [38] is applied. The SIMPLE algorithm uses iterative steps to calculate velocity and pressure corrections thereby ensuring mass conservation. 2\textsuperscript{nd} order Upwind scheme was used for spacial discretization of convection terms in the solution equations.

2.3.6 Underrelaxation Parameters

The default value of under-relaxation parameters used for the analysis are as shown in Table 4. Modifications of under-relaxation factors can be done for pressure-based solver to control the variation of computed variables at each iteration. By modifying these values, we can ideally speed-up and slow down solution convergence.
2.3.7 Convergence Criteria

Convergence criteria selection is critical to ensure that the solution obtained is devoid of any numerical errors. As the iterative solver solves the equations, the residual should ideally decrease indicating smooth convergence. For the current study, a convergence criterion of $10^{-6}$ was set for continuity, velocity components, $k$, $\varepsilon$ and $\omega$.

2.4 Simulation and Post Processing

Ansys Fluent solver was programmed to run the required number of iterations in order to achieve the convergence criteria for the mesh under consideration with the applied boundary conditions. Once the residuals have come below the set value, the solver automatically stopped the analysis and the solution file was then saved manually to be post processed.

Ansys CFX Post was used for post processing the solution file because of its versatility and stability. A state file storing the details of section planes and curves was generated in CFX Post in

<table>
<thead>
<tr>
<th>Under-relaxation Factor</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>0.3</td>
</tr>
<tr>
<td>Momentum</td>
<td>0.7</td>
</tr>
<tr>
<td>Density</td>
<td>1</td>
</tr>
<tr>
<td>Body Forces</td>
<td>1</td>
</tr>
<tr>
<td>Turbulent Kinetic Energy</td>
<td>0.8</td>
</tr>
<tr>
<td>Specific Dissipation Rate</td>
<td>0.8</td>
</tr>
<tr>
<td>Turbulent Viscosity</td>
<td>1</td>
</tr>
<tr>
<td>Energy</td>
<td>1</td>
</tr>
</tbody>
</table>

Table 4: Under Relaxation parameters used
order to reduce the post processing time. Once generated, the state file could be used for other cases of analysis of same geometry. The velocity and turbulence kinetic energy along the different locations was generated using CFX Post and exported for analysis.
3 Pipe Flow Analysis

In order to expedite the computational analysis and minimize the overall mesh count; the pipe flow preceding the development of a single axisymmetric jet was numerically resolved separately. This allowed for an autonomous grid independence study for the pipe and jet along with potentially providing for a higher mesh resolution in the jet’s free shear layer. The velocity profile within the pipe section was validated against published and in-house experimental results and the computational mesh modified accordingly in order to obtain a solution-independent grid. Once a grid independent solution was obtained, the same mesh was used to generate inlet flow parameters to be used in the axisymmetric jet study.

3.1 Geometry and Mesh Generation

Since the results of this study is to be applied to impinging jets, the work of Disimile et al. [1] was used for initial test conditions and validation. They had used pipe exit Reynolds number of 7,500 based on the mean centerline velocity and pipe diameter. Hence the Reynolds number for the current study was also taken as 7,500. A pipe diameter of 0.02 m (20mm), the same value used in Disimile’s experimental study was utilized.

Ansys Fluent, a commercial CFD solver was employed in the present investigation. Hence, the default values for dynamic viscosity and density of air were taken from the Fluent’s properties table and used for the calculation of mean inlet velocity. Once the geometrical features (diameter and length of the pipe) were determined, the geometry was modelled using commercial mesh generation software, Ansys ICEM CFD as shown in Figure 3 and Figure 4. Once the geometry was created in ICEM CFD, it was extracted as a *.tin file and saved separately.
Based on the variables at hand, the required height of the first node in the mesh was calculated as shown in Table 5 based on an ability to obtain a $y^+$ of approximately 1.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reynolds Number</td>
<td>7,500</td>
<td></td>
</tr>
<tr>
<td>Diameter</td>
<td>20</td>
<td>mm</td>
</tr>
<tr>
<td>Velocity</td>
<td>5.478</td>
<td>m/s</td>
</tr>
<tr>
<td>Kinematic Viscosity</td>
<td>1.46e-5</td>
<td>m²/s</td>
</tr>
<tr>
<td>Required $y^+$ Value</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>First Node height required</td>
<td>36</td>
<td>μm</td>
</tr>
</tbody>
</table>

Table 5: $y^+$ Calculation table
Once the first node height was obtained, ICEM CFD was used for mesh generation. Based on the complexity of the geometry, multiple meshing strategies could be used. For the current simulation, simple hexahedral mesh was generated employing the O-Grid method as shown in Figure 5 and Figure 6.

![Figure 5: Mesh along the centerline of the pipe](image)

![Figure 6: Mesh at Inlet and at Walls](image)

The O-Grid method used in the current simulation is simple combination of a cartesian grid and polar grid type element as shown in Figure 7. O-Grids are generally used to model geometries where an O-shaped mesh is required, either at a confined location or totally encompassing a geometric feature. In some occasions, proper capture of the geometry may not be feasible without the use of O-Grids.
3.2 Boundary conditions and Solver setup used

Since the Reynolds number of flow was 7,500, the corresponding velocity was calculated using Equation 2 where $U_{avg}$ is the average velocity, $D$ is the diameter of pipe and $\nu$ is the kinematic viscosity.

$$Re = \frac{U_{avg} \times D}{\nu}$$

Equation 2: Reynolds Number

An average velocity value normal to inlet along with a turbulence intensity of 5% (moderate value of turbulence) was given as inlet boundary condition. Since the solution will serve as the input to the jet flow, atmospheric pressure was given as the outlet boundary condition to simulate pipe exit into ambient air. No-slip wall condition was used to simulate the wall boundary at the pipe surface. The SIMPLE algorithm was used for pressure-velocity coupling with 2\textsuperscript{nd} order upwind scheme used for spatial discretization.
### 3.3 Grid Sensitivity Study

For any CFD analysis, it has to be ascertained that the solution is independent of the node count used in the analysis. This is achieved by running the same simulation over numerous grid configurations with varying node count and closely monitoring the important flow parameters. Once the flow parameter is observed to remain unchanged with the increase in node count, the solution is said to have attained grid independence.

For the current study, 5 different mesh configurations were generated for the grid independence study. A pipe length of 1 meter was used for the grid independence analysis based on the value obtained from literature survey and the pipe length used in experimental setup. The grid independence study was performed with Realizable k-ε model. Node count modification feature of ICEM CFD was used in successive operations to vary the number of mesh nodes along each edge by a factor of 0.9. This helped in maintaining uniformity among different mesh configurations, which would be difficult to preserve if new block meshes were generated for each case. Along with this, it was also carefully ascertained that under no circumstance did the quality of the mesh go below the minimum values discussed earlier.

<table>
<thead>
<tr>
<th>Node Count (million)</th>
<th>Aspect Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.27</td>
<td>51</td>
</tr>
<tr>
<td>0.44</td>
<td>40.5</td>
</tr>
<tr>
<td>0.78</td>
<td>34.2</td>
</tr>
<tr>
<td>0.99</td>
<td>36.4</td>
</tr>
<tr>
<td>1.34</td>
<td>40</td>
</tr>
</tbody>
</table>

Table 6: Aspect ratio across various mesh configuration
To study the development of velocity and turbulence kinetic energy along the flow length, profiles were extracted at 25 cm, 50 cm and at 100 cm (outlet) and results plotted. It can be observed from Figure 8 that velocity profile is initially flatter (at 25 cm) and becomes sharper (at 50 cm) before reaching a developed profile. This observation is consistent with flow development and is discussed in detail in the next section. It is to be noted that the centerline velocity value initially increases and then decreases. Hence the maximum value of velocity along the centerline will not be at the 100 cm but at value close to 50 cm.

![Velocity development profile](image)

**Figure 8: Velocity development along flow direction**

Similarly, Figure 9 displays the variation of turbulence kinetic energy (TKE) along the radial direction at locations 25 cm, 50 cm and 100 cm downstream from the inlet. TKE along the centerline seems to be constantly increasing with the increase in distance from inlet. In radial direction, TKE profile reaches a maximum closer to the wall and then decreases towards the flow center. It is noted that the maximum value of turbulence kinetic energy occurs at approximately
85% of radius and not along centerline. Hence, maximum value of turbulence kinetic energy along centerline is much less than the maximum value of turbulence kinetic energy along the locations where the profiles were extracted.

Figure 9: Tubulence kinetic energy development along flow direction

Once the analyses were completed, the maximum value of velocity and turbulence kinetic energy along the centerline and at outlet were calculated and plotted across various mesh configurations. Figure 10 shows the variation of maximum value of velocity at the centerline of the pipe and at outlet of the pipe with increasing mesh count.
Similarly, variation of maximum value of turbulence kinetic energy at the centerline and at outlet of the pipe for various mesh configurations is plotted in Figure 11.

Figure 10: Grid sensitivity study based on Velocity value

Figure 11: Grid sensitivity study based on Turbulence Kinetic energy value
From Figure 10 and Figure 11, it becomes evident that the solution becomes grid independent at the mesh count of 0.99 million nodes; for both velocity profile and turbulence kinetic energy profiles. It can be noted that the velocity parameter becomes independent of grid at lower node count (as low as 0.43 million nodes) while the turbulence kinetic energy parameter requires higher node count to attain grid independence.

Hence for future analyses, 0.99 million node mesh will be considered as the baseline mesh. In the case with different Reynolds numbers, the first node height will be suitably manipulated to maintain a \(y^+\) value of 1, while maintaining a total mesh count of 0.99 million nodes.

### 3.4 Pipe Length Determination

Before simulating the jet flow field and any subsequent jet interactions, confidence that the jet is initiated from a fully developed pipe flow condition must be attained. If the flow exiting the pipe is not fully developed, the resulting jet may not produce reliable universal results which would nullify the validity of entire simulation.

Development length, or entrance length as it is usually called, is the length of pipe or duct required through which the flow should pass before it attains fully developed or self-similar velocity profile. As the flow enters the pipe, an annular shear layer grows from the wall until it reaches the pipe center where it comes together. This point of intersection defines the length of entrance region. Beyond this point, the velocity profile is self-similar (Figure 8) and does not change and the flow is said to be hydrodynamically fully developed (Figure 10). It is to be noted that the pressure drop and wall shear stress also becomes constant once the flow is fully developed.
The entrance length can vary based on if the flow is laminar, turbulent, or transitional. In the current study, the Reynolds number is 7,500 which lies in turbulent region. Based on a commonly accepted correlation (Equation 3) from [40], the entrance length was estimated to be 19.47 diameters.

\[
\frac{L_e}{D} \sim 4.4 \times Re_d^{1/6}
\]

Equation 3: Equation for calculation of entrance length for turbulent flow

Significant research has been conducted on the turbulent pipe flow, of which the seminal ones were done by Nikuradse [3] and Laufer [4]. In his work, Laufer states that based on the “mean-velocity” distribution, an entrance length of 30 diameters can yield a fully developed flow. Likewise, Nikuradse has stated that an “approach length” of 50 diameters for smooth pipes, and 40 diameters for rough pipes were sufficient to yield a fully developed flow in case of turbulent flow.

To validate the results with these studies, the effect of length in the development of mean velocity and turbulent kinetic energy was examined. Three pipe lengths were selected for analysis starting with the lowest length of 50 diameters (1 meter) and then incrementing 25 diameters twice. The
geometries were modeled in ICEM CFD and 0.99 million mesh was used for all the cases. The first node height of 36 microns was used for all the cases in order to obtain a $y^+$ of 1. Both Realizable $k$-$\varepsilon$ model and SST models were considered and their results were analyzed.

Once the converged solution was obtained, $y^+$ value at the wall was plotted for the cases to ensure that the $y^+$ has not exceeded 1 during the simulation. From Figure 13, it becomes evident that the $y^+$ was maintained below 1 everywhere in the domain, except at the inlet. This is because the initial value of wall shear is slightly larger, thereby increasing the value of computed $y^+$.

![Figure 13: $y^+$ plot for the pipe](image)

Based on the solution obtained from the pipe with a length of 50 diameters, the centerline velocity profile was extracted for both SST and Realizable $k$-$\varepsilon$ model (marked as RealKEp) case and plotted.
It can be noted from Figure 14 that the flow approached an asymptote at 20-25 diameters (0.4-0.5 meter) for the Realizable k-ε model, while it took 40-45 diameters (0.8-0.9 meter) for the SST model. Beyond 40 diameters (0.8 meters), velocity value has not changed significantly for either cases representing a fully developed flow. This is consistent with the work of Nikuradse [3].

However, it should be noted that the turbulence model does in fact affect the mean velocity solution. The Realizable k-ε model seems to have reached their asymptotic state much faster than the SST model. Also, the difference between the mean velocity values predicted by both the models at the pipe outlet (ie at 1 meter) was observed to be approximately 7%.

While the mean velocity has attained a fully developed state with an entrance length of 40 diameters, it is observed that the turbulent kinetic energy requires more length to reach the hydrodynamically developed state as seen in Figure 15.
It becomes evident from Figure 15 that the turbulent kinetic energy has not reached an asymptotic state for SST turbulence model even at a length of 50 diameters (1 meter) and requires more pipe length, whereas Realizable k-ε model has reached an asymptotic state around 30 diameters (0.6 meter). The purpose of the length determination study was to obtain fully developed flow at pipe exit, which is typically based on the development of mean velocity. Even then, the ongoing variation of turbulent kinetic energy profile needs to be studied to understand the effect of length in its development.

Therefore, additional analysis was performed on the pipes of length 75 diameters and 100 diameters. The results of all three cases were then examined in order to understand the effect of length in establishing the development of flow variables.
Figure 16: Centerline velocity profile for different pipes lengths with Realizable k-ε model

Figure 17: Centerline velocity profile for different pipes lengths with SST model

From Figure 16 and Figure 17, it becomes evident that the flow has reached the asymptotic state by 40 diameters (0.8 meter) with respect to mean velocity using SST model. This is in good
agreement with the available literature and acts as one of the validation method for the pipe flow. However, the Realizable k-ε model reaches an asymptotic state at around 25 diameters (0.5 meters) which is earlier than the value predicted by Nikuradse.

The turbulent kinetic energy profile along the centerline for both the turbulence models were also plotted for various pipe lengths (Figure 18 and Figure 19). It can be seen that the turbulence kinetic energy parameter predicted by Realizable k-ε model reaches an asymptote state at around 30-35 diameters (0.6-0.7 meters) when compared to 55 -60 diameters (1.1-1.2 meters) taken by SST model.

![Figure 18: Centerline turbulence kinetic energy profile for different pipes lengths with Realizable k-ε model](image)

Figure 18: Centerline turbulence kinetic energy profile for different pipes lengths with Realizable k-ε model
Figure 19: Centerline turbulence kinetic energy profile for different pipes lengths with SST-model

The various values predicted by both Realizable k-ε model and SST turbulence models are tabulated in Table 7 so that a better understanding of the performance of these models can be achieved. It is to be noted that, at 50 diameters (1 meter); the value of turbulence kinetic energy predicted by Realizable k-ε model is almost 66% higher than that predicted by the SST model. It is to be noted that this difference may play a significant role in the turbulence properties in the axisymmetric jet.
Based on these analyses, it was concluded that a pipe length of 40 diameters (0.8 meters) or greater is optimum for attaining a fully developed mean flow within the pipe domain. This provided support for using pipe with length equaling 45 diameters (0.91 m or approximately 3 ft.) for the experimental arrangement. Although it is understood that the turbulence kinetic energy may not have quite reached a fully developed state (depending on the turbulence model), a pipe length of 1 m (50 diameters) was selected for further pipe simulations.

### 3.5 Validation of Velocity Profile

#### 3.5.1 Normalized Velocity plot

For the validation of velocity profile obtained from the CFD analysis, the local velocity normalized against maximum centerline velocity was plotted for both the turbulence models and was compared with the Eggels et al. [9] and Landers and Disimile [23] experimental data. Eggels’ DNS and LDA data showed remarkable similarity and hence only DNS data was extracted and used for the validation of current study. It is noted that Eggels’ DNS data and Landers and Disimile LDA data are in good agreement. However, the result from current analysis using Realizable k-ε model over-predicts the value of velocity outside the core region and close to the wall as seen in Figure

<table>
<thead>
<tr>
<th>Turbulence Model</th>
<th>Realizable k-ε Model</th>
<th>SST Model</th>
<th>% Difference (approximate)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Velocity profile development length (m)</td>
<td>0.4-0.5</td>
<td>0.8-0.9</td>
<td>47</td>
</tr>
<tr>
<td>TKE profile development length (m)</td>
<td>0.6-0.7</td>
<td>1.1-1.2</td>
<td>43</td>
</tr>
<tr>
<td>Velocity value after development (m/s)</td>
<td>~6.5</td>
<td>~7</td>
<td>7</td>
</tr>
<tr>
<td>TKE value after development (J/kg)</td>
<td>~0.25</td>
<td>~0.15</td>
<td>66</td>
</tr>
</tbody>
</table>

Table 7: Comparison of values predicted by Turbulence model
Also, as observed in Figure 21; the SST model is in good agreement with the data from Eggels’ and Landers’. The SST model appears to model the flow physics more accurately inside the pipe.

Figure 20: Normalized velocity plot - Real k-ε model compared with Landers’ and Eggels’ data

Figure 21: Normalized velocity plot - SST model compared with Landers’ and Eggels’ data
3.5.2 Law of the wall plot

Law of the wall represents the velocity at any point beyond the buffer layer and shows its dependence on the wall roughness and distance from wall for all turbulent cases. This law was first established by Theodore von Kármán in 1930 [41]. He proposed the value of von-Karman constant which is a non-dimensional constant in the Law of the wall equation. His predicted value of 0.4 for the constant still holds today.

The major advantage of the correct validation against law of the wall is that it guarantees that the flow physics near wall is well captured. If the viscous sublayer results obtained from the analysis matches with the law of the wall plot, it is inferred that the viscous sublayer has been properly meshed and captured in the analysis.

For validation against Law of the wall, data from Eggels et al. [9] was used since the Reynolds number used by Eggels was approximately 7,000. Hence, any Reynolds number dependence could be considered negligible. In their study, Eggels et al. had performed velocity measurements using both experimental (LDA) and computational (DNS) analysis of turbulent pipe flow and validated his results against each other. Validating the results of the current study against Eggels’ provides confidence on the accuracy of the current analysis. It is to be noted that Eggels claimed that their velocity profile had failed to match with the law of the wall, and a similar condition is observed in our current study. This bolsters the validity of results obtained from the current analysis.
Figure 22 shows the result from current SST analysis when compared to Eggels’ data. The von-Karman constant of 0.4 and a roughness constant of 5.5 (same as used by Eggels) was used for plotting the law of the wall. The SST model appears to have captured the viscous sublayer with enough accuracy thereby ascertaining the capture of wall flow physics. In the log law region, there seems to be no conformity to the logarithmic law as noted by Eggels. Similar development was observed by Wagner [8] also, who claimed that a well-defined log law region did not exist until a Reynolds number of 10,300 for his case. The absence of well-defined log law region appears to be due to the low value of Reynolds number.

Similarly, plotting the law of the wall profile for the solution obtained from Realizable k-ε model; we observed that the Realizable k-ε model showed further deviation from the DNS data as shown in Figure 23.
In order to gain additional confidence on the reliability of the velocity profile obtained from the different turbulence models, the data from each model was extracted to plot velocity defect profile.

### 3.5.3 Velocity Defect plot

Although the Log law plot can provide confidence in the capture of the near wall flow physics, the low Reynolds number limits the reliability of such a plot in the buffer and log law region. Also deviation from Eggels’ data in the buffer region and log-law region needs to be further analyzed. This in part may have arisen since Eggels’ used honed pipe with very low surface roughness, thereby reducing overall turbulence generating capability of the wall flow. Hence, to better validate the central core flow away from the wall obtained from the current study, a velocity defect plot was generated from the SST model’s and Eggels’ [9] data. Again, Eggels’ DNS data was used since the DNS data and LDA data showed very good similarity. Since the value of kinematic
viscosity used in Eggels’ analysis was not mentioned in his article, the value of kinematic viscosity of air (1.46e-5 m²/s) obtained from Ansys Fluent was utilized. Although this may produce some numerical differences, no major differences in results is expected.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter of Pipe(D)</td>
<td>95.4mm</td>
</tr>
<tr>
<td>Length of test section</td>
<td>77D</td>
</tr>
<tr>
<td>Reₐ (Re based on centerline velocity)</td>
<td>7,200</td>
</tr>
<tr>
<td>Reₜ (Re based on wall shear velocity)</td>
<td>371</td>
</tr>
</tbody>
</table>

Table 8: Data taken from Eggels

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Uₜ</td>
<td>1.102</td>
</tr>
<tr>
<td>uₜ</td>
<td>0.057</td>
</tr>
</tbody>
</table>

Table 9: Data calculated from Eggels

From Figure 24, it becomes clear that the data obtained using SST model compares very well with Eggels’ study and reinforces the proper capturing of all the aspects of flow, from viscous sublayer to the flow core. But Figure 25 shows that the deviation of results obtained from Realizable k-ε model when compared to experimental data is significant. From these validation studies, it can be concluded that SST k-ω model is the ideal turbulence model capable of accurately simulating the flow physics for pipe domain at low Reynolds number. It was decided that the ideal turbulence model for the impinging jet will not be chosen based only on the study of flow physics in the pipe domain, and an indepth comparison in the single jet domain will be performed to ensure the usage of the best available model.
Figure 24: Velocity defect data comparison – CFD SST model vs Eggels

Figure 25: Velocity defect data comparison – CFD Real KEp model vs Eggels
4 Single Jet Analysis

In the present section, the flow exiting a fully developed round pipe was used as the inlet condition for the single axi-symmetric round jet. We have previously noted that the purpose for an autonomous pipe and jet simulation was to control the grid points in both the studies so that the simulations could be performed independently with a high level of mesh refinement and accuracy. The velocity profile obtained from the pipe flow simulation was validated with published results, whereby a considerable agreement was observed. This provided high confidence in the pipe flow solution, which was then used as the inlet boundary condition of the single jet simulation.

In the current study, a single round axi-symmetric jet was analyzed using SST k-ω model initially at Reynolds number = 7,500. Multiple meshes were generated to identify the smallest mesh setup by which the solution becomes independent of grid resolution. Once a grid independent solution was obtained, the result from the simulation was compared with in-house experimental results which indicated discrepancies in turbulence intensity data obtained between computational and experimental approach. This was further analyzed by performing the single jet analysis using other turbulence models and comparing the results.

4.1 Geometry and Mesh Generation

The reason for performing single jet CFD analysis was to gain a better understanding of the computational capabilities of current turbulence models in capturing the jet’s free shear layer physics. Once adequate understanding was obtained with single jet, the most suitable turbulence model was selected to perform the analysis of two jets impinging at an angle. Work performed by Disimile et al. [1] on impinging jets has been taken as the validation source for impinging jet analysis, wherein a Reynolds number of 7500 was used. Hence for the single jet study, and the pipe flow study which preceded this section; Reynolds number used was 7500. Disimile et al. [1]
had used pipe internal diameter of 20 mm in their study, which was maintained in the present work and in Landers [2].

Since in the experimental conditions, the jet exiting the pipe mixes with a still ambient fluid (air); the geometry designed for the current study had to represent a similar setup. This was attained by keeping the boundaries very far from the jet path, so that the presence of any form of computational boundary would not manipulate the jet. With this consideration, the computational geometry was generated in form of frustum, with the pipe flow exit (and jet inlet) in the smaller section and the domain exit at the larger section. All the surfaces, including the curved periphery was modelled to allow flow of mass in either direction and was set as pressure outlet boundary conditions where reverse flow was permitted.

A pictorial representation of the geometry is shown in Figure 26. All surfaces other than inlet was modelled as pressure outlet. In order to maintain solution reliability, the domain was constructed with the smaller side of frustum with radius of 15D while the larger side had 25D as shown in Figure 26 and Figure 27.

Figure 26: Geometry for single axi-symmetric jet simulation
The axial flow length of 25D was maintained for the flow to develop inside the geometry. Further downstream was not modelled or analyzed since the primary interest of this study was to simulate the near field physics of round jets.

Once the geometry was designed and initial boundary conditions established, the next step involved in the analysis was grid generation. Two separate O-grids were used to capture the outer and inner periphery of the frustum as shown in Figure 28. It shows the mesh with node count of 4.3 million nodes.
Figure 28: Front view and close-ups showing the mesh with 4.3 million nodes in detail

It can be observed in Figure 28 that the mesh close to the center (which is the pipe exit domain) is very fine when compared to the mesh at the periphery. This has been maintained for the accurate transfer of resulting flow data from the pipe simulation, since using a coarse mesh leads to data averaging.

The dark ring seen in the right most snapshot of the mesh indicates the boundary layer mesh within the pipe flow. In order to accurately capture of boundary layer physics, the first node height had to be reduced so that the distance from the wall in wall coordinates, $y^+$ is closer to 1. This condition yielded a first node height of 0.036 mm in the pipe domain. Since the pipe domain exits into the jet domain, the same first node height was initially maintained into the jet domain. Figure 29 shows the side view of the mesh. As can be clearly observed, the central core region was captured with very fine mesh while mesh was allowed to grow coarser as it reached the periphery of the domain.

Various mesh quality checks using different parameters were performed to ensure that the mesh obtained was of adequate quality. Detailed description of the parameters and allowable limits for the same have been presented in section 2.2. For the current study, it was established that the aspect
ratio never exceeded 100 (the maximum aspect ratio allowed by Ansys Fluent). Once a mesh with the required quality was generated, it was saved as *.msh file from ICEM-CFD which was then input into the commercial finite volume solver, Ansys Fluent.

Figure 29: Side view of mesh for single axi-symmetric round jet

4.2 Turbulence Models used

One of the primary concerns in this analysis was the ability of the turbulence models to reliably capture the physics of free shear flows. It was noted during literature review that certain models performed better for round jet flow when compared to others. Also, there were articles that implied the dependence of the flow field on initial conditions [18] and proposed that round jets cannot be accurately simulated easily with the turbulence models and parameters currently in use [42]. Yoder et al. [43] has studied the modeling of turbulent free shear flows and have concluded that the general turbulence models available may not be able to accurately predict even the basic flow
characteristics. For this study, four turbulence models were selected for examination; Realizable $k$-$\varepsilon$ model, SST $k$-$\omega$ model, Standard $k$-$\varepsilon$ model and Standard $k$-$\omega$ model. Studies comparing the efficacy of Realizable $k$-$\varepsilon$ model with Standard $k$-$\varepsilon$ model and Standard $k$-$\omega$ model were found and so was studies comparing SST $k$-$\omega$ model with Standard $k$-$\varepsilon$ model and Standard $k$-$\omega$ model. But, no study comparing all these four models for turbulent round jets was found by the authors. Hence, this study provides insight into the area of choice of best turbulence model when simulating round jet flow exiting fully developed pipe at low Reynolds number. The turbulence models used in the current study is described in section 2.3.2.

4.3 Boundary Conditions used

After choosing a turbulence model, the simulation requires the proper boundary conditions that will impart the real-world environment to the simulation. The flow from fully developed pipe was to be used as the inlet boundary condition to the jet flow. Velocity and turbulence kinetic energy profiles as shown in Figure 30; were extracted from the outlet of pipe flow simulation and were provided as the input boundary condition. In order to simulate that the jet escaping into ambient fluid at atmospheric conditions, the outlet condition for the round jet simulation was set as pressure outlet at ambient pressure. The periphery of the domain was also set as pressure outlet condition at ambient pressure in order to replicate natural conditions. This allowed for entrainment of the ambient fluid from all the directions. Air at ambient temperature and pressure was used as the working fluid. Solver settings used can be found in section 2.3. SIMPLE algorithm was utilized for pressure-velocity coupling with the 2$\text{nd}$ order Upwind scheme used for spatial discretization. A convergence criterion of $10^{-6}$ was used to confirm that the solution had fully converged with minimal possible error. Once the residuals for the convergence criterions were satisfied, the run was stopped, and the results file extracted to be post processed.
Figure 30: a) Velocity and b) Turbulence Kinetic energy used as inlet condition for the round jet

4.4 Grid Sensitivity Study

As shown in the earlier pipe study; it is imperative that the solutions obtained from any CFD analysis be resolved over multiple meshes in order to ascertain that the solution achieved is not dependent on the grid size used. Assessing grid independence is carried out by running simulations using multiple mesh configurations for the same geometry and boundary conditions and observing the change in target parameters as the mesh is altered. Ideally, a very small mesh is generated initially which satisfies the bare minimum mesh quality requirements. Based on this mesh, further meshes are typically generated by refining the mesh size over every iteration. The results from all these meshes are compared, and the mesh size beyond which no significant change in target parameter occurs for any variation in mesh size is considered as the point of grid independence. This mesh at the point of grid independence is then taken as the minimal mesh required to obtain a node independent solution for that particular geometry and simulation setup.

For the current study, six (6) different mesh geometries were generated starting from 0.25 million nodes. All the analysis for the grid independence study was performed using Realizable k-\( \varepsilon \) turbulence model. The mesh at the inlet of the jet domain was matched with the mesh from the outlet of the pipe used and the initial mesh height of 0.036 mm into the jet domain was initially
maintained. This was required for a smooth continuation of mesh from the pipe outlet and for the accurate transfer of flow variables without any interpolation or averaging. However, starting with a very low mesh node count and maintaining such small grid spacing at center yielded regions with coarse mesh and high aspect ratio. In the current study those regions were successfully maintained at the periphery of the domain, ie away from the area of interest. This can be clearly observed in Figure 31 where the mesh close to the center of the domain is maintained at a height of 0.036mm whereas the peripheral mesh was on the order of millimeters. This meshing philosophy was applied in order to keep the starting node count low. As meshing iterations proceeded with the generation of finer meshes, the mesh along the periphery was refined while adding nodes in the central region as shown in Figure 31.

Figure 31: Front view of grid with a) 0.25 million nodes and b) 4.3 million nodes

A similar mesh refinement tactic was applied along the axial flow direction thereby reducing the aspect ratio with every iteration of mesh enhancement. Impact of this refinement strategy can be clearly observed in Figure 32.
Figure 32: Side view of central plane for grid a) 0.25 million nodes and b) 4.3 million nodes

Once all the test cases were successfully converged, the results from the analyses were extracted for post processing. Mean velocity and turbulence kinetic energy (TKE) were obtained from all the mesh configurations and plotted systematically to identify the point of grid independence as shown in Figure 33 and Figure 34.
Figure 33: Grid sensitivity study based on Mean Velocity value

Figure 34: Grid sensitivity study based on Turbulence Kinetic energy value
It is evident from Figure 33 and Figure 34 that the solution becomes independent of the grid size at around 1.38 million nodes since no significant change was observed in the velocity or turbulence kinetic energy value beyond 1.38 million nodes, even with tripling the mesh size. The maximum variation from 1.38 million case is of the order of 1.7% in terms of mean velocity and 5% in terms of turbulence kinetic energy for mesh size above 1.38 million nodes as seen in Table 10. These values are negligible when compared to the variation observed in experimental data [2] which indicated a range of 10-18% fluctuation in the calculation of mean velocity values and 22-41% fluctuation in the calculation of turbulence kinetic energy values for different experimental setup in the case of single jet analysis.

<table>
<thead>
<tr>
<th>Million nodes</th>
<th>Vel (m/s)</th>
<th>TKE (m^2/s^2)</th>
<th>Vel (m/s)</th>
<th>TKE (m^2/s^2)</th>
<th>% Difference from 1.38 million node case</th>
<th>% Difference from 1.38 million node case</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.25</td>
<td>0.1554</td>
<td>0.0436</td>
<td>0.0189</td>
<td>0.0125</td>
<td>13.8836</td>
<td>39.9865</td>
</tr>
<tr>
<td>0.44</td>
<td>0.1480</td>
<td>0.0391</td>
<td>0.0116</td>
<td>0.0079</td>
<td>8.4796</td>
<td>25.3214</td>
</tr>
<tr>
<td>0.78</td>
<td>0.1404</td>
<td>0.0338</td>
<td>0.0040</td>
<td>0.0026</td>
<td>2.9040</td>
<td>8.3026</td>
</tr>
<tr>
<td>1.38</td>
<td>0.1364</td>
<td>0.0312</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2.44</td>
<td>0.1341</td>
<td>0.0296</td>
<td>-0.0023</td>
<td>-0.0016</td>
<td>-1.7191</td>
<td>-5.0892</td>
</tr>
<tr>
<td>4.3</td>
<td>0.1351</td>
<td>0.0303</td>
<td>-0.0013</td>
<td>-0.0009</td>
<td>-0.9762</td>
<td>-2.8290</td>
</tr>
</tbody>
</table>

Table 10: Grid Independence data

Hence for further analyses, 1.38 million node mesh was considered as the baseline mesh. Using this mesh, a single axisymmetric turbulent jet was simulated as exiting a long constant diameter round pipe with fully developed turbulent velocity profile.

4.5 Effect of Turbulence Models

The single jet analysis was initially performed using Realizable k-ε model and was further simulated using SST k-ω model, Standard k-ε model and Standard k-ω model. It has to be noted that the choice of Realizable k-ε model as the first model used for analysis was based on the literature review, which claimed that the performance of Realizable k-ε model was adapted
specifically for accurate prediction of the spread rate of round jets. During literature review, it was also noted that the SST k-ω model’s performance was claimed superior to that of Standard k-ε model and Standard k-ω model [Menter [44]]. But a research work comparing the performance of SST k-ω model and Realizable k-ε model for single round jets was not found during literature survey.

4.5.1 Velocity Profiles

From Figure 35, it can be observed that there exists significant difference in the distribution of area of high velocity (as shown within the demarcated region) when calculated by four turbulence models. It was imperative to look closely into this zone to establish the accuracy of each turbulence model. Figure 36 gives the zoomed in look at demarcated zone. A general trend observed was that the epsilon based turbulence models generated smaller regions of higher velocity when compared to omega based models. Of the four turbulence models under study; Standard k-ω model seems to have the largest zone with maximum velocity, denoting lower turbulence dissipation.
Figure 35: Velocity contour along central plane for single axi-symmetric jet obtained using various turbulence models
Figure 36: Zoomed view of velocity contour along central plane for single axi-symmetric jet obtained using various turbulence models

In order to obtain a holistic understanding of the general trend of velocity at central zone, velocity profiles were extracted from central line and different locations downstream of pipe exit. These profiles were compared with data from Landers [2] in order to establish the turbulence model which provided solution closest to the experimental result.

Velocity data extracted from axial central line was normalized with maximum velocity along the center line (which occurs at the exit of the pipe flow) and was plotted over axial distance measured...
in pipe diameters. From Figure 37; we notice that the Standard k-ω model follows a more deviated path when compared to the three other remaining models. This consistent deviation of results obtained for Standard k-ω model demands further investigation on the performance of the model. Another key-point to be noted from the Figure 37 is that the performance of SST k-ω model, Realizable k-ε model and Standard k-ε model are very similar except in the region between 5D to 10D from the pipe exit. This may be due to the breakdown of potential core in that region suggesting different models simulates this region differently.

![Profile of normalized velocity along centreline](image)

Figure 37: Profile of normalized velocity along centreline

Initially, velocity data extracted from different location downstream of the pipe exit were plotted against data from Landers [2] as shown in Figure 38 to Figure 41. It is to be noted that Landers observed a difference of up to 4.5% in the velocity data based on the experimental setup used. He also noted that there exists an uncertainty factor of 4% on the experimental calculation of velocity.
components based on the instrumentation used. The general trend observed was that the performance of Realizable k-ε model and SST k-ω model closely matched each other with Standard k-ε model being another close contender. It has to be noted that, beyond 6D; data from experimental study also showed a wider spread. Meanwhile, the Standard k-ω model showed the maximum deviation from the experimental data at 9D and 10.33D. As seen in Figure 40 and Figure 41, the Standard k-ω model starts to deviate from the general trend while the 3 other turbulence models illustrate similar results.

Figure 38: Radial profile of normalized velocity over radius normalized by half width at 3D from pipe exit
Figure 39: Radial profile of normalized velocity over radius normalized by half width at 6D from pipe exit

Figure 40: Radial profile of normalized velocity over radius normalized by half width at 9D from pipe exit
It was concluded that the performance of the four turbulence models was comparable in accuracy when considering the mean velocity. However, different parameters need to be examined, in order to distinguish the most suitable model for the current single simulation and future impinging jet study. Turbulence intensity was identified as a suitable parameter in previously established works and hence it was taken for consideration in the present study also.

As done previously, central plane was considered for extraction of data for plotting contours and profiles. Turbulence intensity data was extracted along the center line and at various locations downstream of pipe exit and compared with experimental data obtained from [2]. The turbulence kinetic energy (TKE) contours along the central plane is shown in Figure 42. It is observed that there exists significant difference in the simulation of turbulence by the four turbulence models.
While the SST k-ω model and the Standard k-ε model displayed similar performance for TKE parameter, Realizable k-ε model and Standard k-ω model predicted lower values of TKE, with Standard k-ω model predicting the lowest.

Figure 42: Turbulence kinetic energy contour along central plane for single axi-symmetric jet obtained using various turbulence models
Figure 43 gives the zoomed in look at demarcated zone shown in Figure 42, so that a better judgement of the profile can be obtained. The presence of low turbulence region in the middle of the jet for Standard k-ω model indicates that the model predicts lower turbulence in the high velocity region.

Figure 43: Zoomed view of turbulence kinetic energy contour along central plane for single axi-symmetric jet obtained using various turbulence models.
For obtaining clear understanding of the nature of turbulence simulated by the different models, it was imperative to look at the turbulence intensity (TI) data at different locations and compare it with experimental results from [2]. Initially, TI for different models were plotted along the center line and compared as shown in Figure 44. Again, the Standard k-ω model stands as the odd-one-out, predicting very low values of TI until 18D downstream of pipe exit.

![Profile of turbulence intensity along centre line](image)

Figure 44 : Profile of turbulence intensity along center line

An interesting observation can be made regarding Realizable k-ε model was that beyond 18D, the TI dips the most for that model. This can be observed in Figure 43 also, as the contour for Realizable k-ε model alone indicates the presence of lower TKE zone near the right end of the zone. This needed to be further analyzed to ensure that confidence can be assigned to that model.
Warda et al. [45] had used relative turbulence intensity (RTI) instead of normal turbulence intensity in their study. This motivated the calculation of RTI profile for various turbulence models. It can be observed in Figure 45 that the RTI values obtained for all 3 models except Realizable k-ε model showed similar trend of initially peaking to a maximum value and then become constant (similar to the trend shown by Warda et al. [45]) while Realizable k-ε model predicted an initial cresting and then slowly decreasing. The trend of reduction in RTI after reaching a maximum value seems to indicate underlying inaccuracy in capturing the turbulence physics by the Realizable k-ε model.

![Profile of relative turbulence intensity along centreline](image)

Figure 45: Profile of relative turbulence intensity along center line

The need to further ascertain the performance of Realizable k-ε model in predicting TI parameter was recognized and TI data profiles were extracted at locations 3D, 6D, 9D and 10.33D from pipe exit for comparison and validation. Again, the performance of Realizable k-ε model and SST k-ω
model were close to one another, with Standard k-ε model being the next close model. Standard k-ω model, as observed in other scenarios, performed differently.

Figure 46 to Figure 49 shows the radial profile of TI at different locations downstream of pipe exit. It is noted that the Realizable k-ε model and SST k-ω model were able to predict the TI value at 3D and 6D with some success. Standard k-ε model was able to closely predict the TI value only at 3D, while Standard k-ω model performed poorly in all locations. Beyond 6D, no model was able to accurately predict the TI for single round jet even though they were able to predict the general trend (except Standard k-ω model). All models except Standard k-ω model either correctly or over-predicts TI while Standard k-ω model under-predicts it at all the locations.

![Radial profile of turbulence intensity over normalized radius](image)

Figure 46: Radial profile of turbulence intensity at 3D from pipe exit
Figure 47: Radial profile of turbulence intensity at 6D from pipe exit

Figure 48: Radial profile of turbulence intensity at 9D from pipe exit
From the current results, it was concluded that the performance of Standard k-ω model was not adequate enough to properly model the physics of single axi-symmetric round jet. Hence, it was decided to forego the Standard k-ω model from future consideration. Since the performance of Realizable k-ε model and SST k-ω model were comparable (except for the inaccuracy of Realizable k-ε model in calculating relative turbulence intensity near the end of the domain), it was deemed that SST k-ω model was the better of the two. Also, performance of Standard k-ε model and SST k-ω model were quite comparable (except for Standard k-ε model over-predicting turbulence intensity at 6D from pipe exit). From these observations, it was determined to use SST model alone to analyze impinging round jets.

Figure 49: Radial profile of turbulence intensity at 10.33D from pipe exit
5 Impinging Jet Analysis

The current study is performed based on the insight gained from the analysis of single turbulent round jet as discussed in section 4 at low Reynolds number of 7500. In the case of single jet study, the inlet boundary condition was modelled as exiting from a fully developed pipe into ambient conditions. In order to obtain the velocity and TKE profile of fully developed pipe, it was decided that the pipe flow be modelled separately and the profile from the pipe flow simulation used as the inlet condition for jet. The results from the study of single round jets were closely validated with experimental data which provided confidence that the strategy used was accurate. Hence, the same strategy was used in the case of impinging jet study. For the study of impinging jets, the full model generated is shown in Figure 50. The pipe sections were analyzed first, followed by impinging jet domain. This methodology has successfully helped in reducing the computational requirements since the cases were treated separately.

Figure 50: Schematic of full geometry
5.1 Geometry and Mesh Generation

As discussed in section earlier, the geometry was initially modelled fully with pipe and ambient domain. It was designed as a pipe with length 50D exiting into ambient air (at standard pressure and temperature) domain with length of 30D and height and width parameter being 20D each. The pipes are inclined at an angle of $\theta$ to the central plane, which is termed as the Half Impingement Angle (HIA). In the current study, the HIA considered are $15^\circ$, $22.5^\circ$ and $30^\circ$. The mesh for the inclined pipe domain was maintained same as the mesh generated for single pipe simulation used for jet; which has been established to be grid independent and validated with experimental results. This gives us the confidence that the profile extracted from the pipe exit is accurately predicting the flow physics. The mesh in the outlet of pipe domain is as shown in Figure 51. As with the previous study, the geometry was generated using SolidWorks, while the structured mesh was created using proprietary mesh generation code ICEM-CFD.

![Figure 51: Mesh at Pipe domain outlet](image)

In order to ensure that accurate transfer of velocity and TKE data occurs from the pipe domain to jet domain, it was necessary to maintain 1 to 1 grid connectivity between the pipe domain and the
domain downstream. Hence, the inlet of the jet domain (which acts as the exit of the pipe domain) was very finely meshed, to maintain 1 to 1 grid connectivity. The grid generated near the pipe exit/flow inlet section of jet domain is as shown in Figure 52. It can be seen that the area corresponding to the pipe exit is meshed in correspondence to the pipe domain mesh and one-to-one connectivity between pipe mesh and jet domain mesh has been maintained. The band of very fine mesh seen in the diagram corresponds to the wall surface mesh generated in the pipe domain. It is to be also noted that the area between the jets are meshed with fine mesh to ensure that the flow physics due to jet impingement is accurately captured.

The midplane showing the cut section of the mesh generated for the impingement domain is shown in Figure 53. Care has been taken to ensure that the mesh extruding from the inlet section is gradually growing towards the center to ideally capture the growth of the jets with sufficient numerical accuracy.
5.2 Turbulence Models used

The choice of turbulence model for the simulation of impinging jets is based on the work done earlier on single jets which was established in earlier section on single jet. Various turbulence models were analyzed for single jet flow conditions, and it was found that SST k-ω model performed the best for the given range of Re under consideration. Hence, for the current analysis, SST k-ω model was deemed as the best option in terms of turbulence models. The details regarding SST k-ω model were discussed in sections earlier.

5.3 Boundary Conditions and Solver used

The velocity and TKE profile extracted from the pipe exit is shown in Figure 54, Figure 55, Figure 56 and Figure 57. It is to be noticed that three separate velocity components were used in this study (velocity u, v and w) instead of single velocity profile (as used in single jet study) to ensure that all the appropriate velocity vectors are transferred from the pipe domain into impinging jet domain.
Velocity $v$ is shown in negative components since one of the pipe is pointed in negative $y$ direction, while the other is pointed in positive $y$ direction.
The schematic of boundary conditions used in the study is shown in the Figure 58. The green region denotes the zone where pipe exit profile conditions are applied. The surface bounding the inlets (shown in blue) are modelled as pressure outlet condition. This denotes a condition of standard atmospheric pressure and temperature. This allows the flow to enter that surface in all direction since pressure outlet condition in Ansys Fluent allows for backward flow. The side surfaces of the domain (shown in white) are also modelled as pressure outlet conditions. This is necessary to ensure that flow from all the direction is permitted to mix with the center jet flow, thereby ensure accurate physics of ambient air entrainment. The exit zone of the domain is also modelled as pressure outlet, thereby replicating a fully open surface for the jet to grow.
Figure 58: Schematic of boundary conditions used in the analysis

The solver settings involved in this analysis is similar to that used in the case of single round jet study and pipe study as explained in earlier sections. SIMPLE algorithm was utilized for pressure-velocity coupling with the 2nd order Upwind scheme used for spatial discretization. A convergence criterion of $10^{-6}$ was used to confirm that the solution had fully converged with minimal possible error. Once the residuals for the convergence criterions were satisfied, the run was stopped, and the results file extracted to be post processed.

5.4 False Diffusion study

One of the major concerns involved in the meshing of impinging jet domain was that the same meshing philosophy used for the case of single jet domain could not be used. In case of single jet, a single O-grid exiting the pipe domain was extruded into the jet domain, which radially expanded outwards; thereby capturing the relevant flow physics. In the case of impinging jets, there would be two such O-grids which need to interact with each other (at the point of impingement). But it is
not possible to generate 2 intersecting O-grids, as the mesh cannot overlap with each other. This lead us to the conclusion that the only possible solution is to keep the O-grid mesh straight out of the pipe domain parallel to the central plane, without any alignment towards the flow direction. The region between the jets were to be finely meshed to capture the all flow details with necessary precision.

One of the major drawbacks of this approach is known to be False diffusion (FD). FD is defined as the artificial diffusion introduced by the numerical scheme when the flow has predefined obliquity to the grid lines and when there exists a non-zero gradient of flow variables in the direction normal to flow. It is a multi-dimensional phenomenon usually observed in cases with large Peclet number. FD was extensively studied by de Vahl Davis and Mallinson [46] who proposed an approximation expression to represent it in two-dimensional state as shown in Equation 4.

\[
\Gamma_{\text{False}} = \frac{\rho \ast U \ast \Delta x \ast \Delta y \ast \sin 2\theta}{4(\Delta y \ast \sin^3 \theta + \Delta x \ast \cos^3 \theta)}
\]

Equation 4 : Expression for false diffusion

The parameters used in Equation 4 is defined in Table 11. Suhas Patankar [38] has provided a very clear discussion of the issues related with FD in his book. He helps draw attention to the fact that:

1. FD is present only when a mesh obliquity exists with respect to velocity profile.
2. FD becomes maximum when \(\sin 2\theta\) is maximum, which is attained when \(\theta\) becomes 45 degrees i.e. flow direction is at 45° to the grid alignment.
3. FD can be reduced by mesh refinement, i.e. having smaller values for $\Delta x$ and $\Delta y$ respectively.

4. FD cannot be removed by using Central difference scheme, since Central difference scheme are prone to produce unrealistic results at large grid Peclet number.

<table>
<thead>
<tr>
<th>Symbols</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>$I_{False}$</td>
<td>False Diffusion Coefficient</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density of the fluid</td>
</tr>
<tr>
<td>$U$</td>
<td>Velocity of flow</td>
</tr>
<tr>
<td>$\Delta x$</td>
<td>Mesh size in x-direction</td>
</tr>
<tr>
<td>$\Delta y$</td>
<td>Mesh size in y-direction</td>
</tr>
<tr>
<td>$\theta$</td>
<td>Angle made by the velocity vector with x-direction</td>
</tr>
</tbody>
</table>

Table 11: Explanation of parameters used in false diffusion equation

This has helped us in gaining significant understanding on the importance of choosing the appropriate numerical scheme for the study. The work done by Raithby [47] and [48] provides insight into the research that has gone into gaining an understanding of this numerical phenomenon. Along the same lines, the work done by Patel et al. [49] and [50] has suggested an updated numerical scheme which suffers less from FD issues. Recent works (as of 2012) by Karadimou [51] show that there is still significant interest in this aspect of numerical schemes.

This brings our attention to the problem of jet impingement study, where we have mesh with predetermined obliquity with the grid lines. It was considered important to establish the effect of FD due mesh obliquity in accurate prediction of velocity profiles. A qualitative and quantitative representation of this numerical issue was necessary before proceeding with impingement jet
study. Hence, it was decided that a single jet case will be studied where the mesh is not aligned with the inlet velocity vector and the results were to be compared with straight jet (where the flow and velocity vectors are aligned) result. The validation of straight round jet results with experimental data has provided us with confidence in the accuracy of those results, and hence they can be used as baseline data for future validation.

For the study with misaligned flow, a domain similar to single jet study was designed as shown in Figure 59.

![Figure 59: Schematic of flow setup used for false diffusion study](image)

For the study, a mesh obliquity (θ) of 15° angle was considered. The center line of the flow is shown in red color while the center plane to which the mesh is parallel is shown in black. The mesh generated for the study is shown in Figure 60. The mesh used is exactly same as the mesh generated for straight jet flow. The velocity and TKE contour from study was generated and overlaid with the mesh as shown in Figure 61 and Figure 62.
Figure 60: Mesh for flow setup used for false diffusion study

Figure 61: Velocity profile for case with misaligned mesh
For obtaining a clear understanding of the effect of mesh obliquity, velocity profile along the center line of the flow was taken for consideration. From the study of single jet established in section 4, it was established that none of the turbulence models used were capable of predicting the TKE parameter with much accuracy beyond a distance of 6D from the pipe exit for the current range of Re used. Hence, for the impingement jet study, TKE parameter was not considered as a factor of special interest and only velocity data was closely analyzed. Still, in the spirit of technical growth and research interest; TKE parameter was also plotted for the misaligned jet study and compared with straight jet data. The velocity profile along the center line is shown in Figure 63 and the TKE profile along the center line is shown in Figure 64. It is observed that there exists a difference in value for velocity and TKE along the center line for misaligned jet when compared to straight jet. In case of velocity profile, this difference corresponds to 4.8% where as in TKE data, the difference is close to 63.5%.

Figure 62: TKE profile for case with misaligned mesh
Even with using the same mesh that was used for straight jet, a considerable difference in the value of velocity data was observed. This indicates that numerical scheme in the commercial CFD solver used also suffers from FD. It was decided that a refined mesh was to be tested to see the impact of mesh size on the FD parameter.
Instead of generating a mesh with higher node count for the same domain size, the domain size was brought down to half the initial size and the mesh count maintained the same; thereby ideally reducing the Δx and Δy to half of the initial value. The new generated mesh overlaid with previous mesh is shown in Figure 65. The yellow line represents the 10.33 D location from the pipe exit which based on the centerline of the pipe identifies the impingement point.

![Figure 65: Refined mesh vs initial mesh generated for false diffusion study](image)

Based on the refined mesh, CFD analysis was performed on the inclined jet flow and the results compared to the results from previous mesh and straight mesh study. Again, the velocity and TKE parameters were plotted along the center line and data compared with each other as shown in Figure 66 and Figure 67. The TKE value difference has come down from 63.5% to 33% whereas the velocity difference has reduced from 4.8% to 3%. A mesh refinement by a factor of 2 has only reduced the difference in velocity values by a margin of 1.8%; whereas significantly affected the TKE parameter.
This implied that, if the domain size similar to single jet domain was to be used; a significantly large mesh size will be required to generate accurate values for velocity data. Hence, it was deemed necessary that the domain size used for the jet impingement study be reduced by a considerable margin so that high quality mesh can be generated at similar node counts. The dimensions hence used for the domain in case of impingement jet study as (shown in Figure 50) can be seen to be
significantly smaller than the dimensions used for single jet study. This reduction in domain size has helped us to efficiently allocate the nodes to the areas of higher interest without any significant increase in node count. Care has been taken to ensure that the reduction in domain dimensions did not affect any simulation physics.

5.5 Full model vs Symmetry model

Another major step considered to ensure that accurate flow results were obtained at minimal usage of computational resources was to study the flow using full model (as shown in Figure 53) and compare it with the results obtained from the symmetry based model (as shown in Figure 68). Since the boundary conditions are symmetric about the central plane, it can be assumed that the flow physics beyond interaction point is also symmetric in a steady state solution. If the velocity data obtained from full model and symmetric model is comparable, then it can be assumed with confidence that the symmetric model can produce accurate results which captures the flow physics completely.

![Figure 68: Mesh along center plane for symmetry model jet domain](image)

The region corresponding the zone of impingement is very finely meshed as can be seen from the picture above. The node count generated for the symmetric mesh is close to 1.35 million. Even though, this value is very close to the mesh used for single jet mesh, it is to be considered that the
The fluid domain used in the case of jet impingement study is significantly smaller, thereby ensuring higher mesh quality.

The planes where data were considered for the comparison is shown in the Figure 69. The naming convention used by Landers [2] is followed in the current study. In the horizontal plane, the jet impingement occurs at angle $2\theta$ to one another. The impingement point lies on the intersection of horizontal and vertical plane, at a distance of 10.33D from the pipe exit. The line formed at the intersection of horizontal and vertical plane is termed as the center line.

![Figure 69: Location of planes used for the jet impingement study](image_url)

The velocity and TKE contour along the horizontal and vertical planes are plotted as shown in Figure 70 to Figure 73. The symmetry model seems indicate that the flow physics observed in the full model is ideally captured in the symmetry model also.
Figure 70: Velocity profile along horizontal plane for full and symmetry model

Figure 71: TKE profile along horizontal plane for full and symmetry model

Figure 72: Velocity profile along vertical plane for full and symmetry model
To quantitatively compare the effect of symmetry model vs the full model for jet impingement study, the center line velocity and TKE data was plotted and difference between them taken into consideration. It was observed that the maximum difference in velocity value was close to 2.8% whereas the maximum difference in TKE value was close to 2.3%. The graph showing the profile of velocity and TKE data is as shown in Figure 74 and Figure 75. Since the difference in values are negligible, it was concluded that using symmetry model for the jet impingement study was an effective method to reduce the computational resource usage without any major omission of flow physics.
5.6 Grid Sensitivity Study

The philosophy behind grid sensitivity study has been explained in previous sections. For every independent CFD analysis performed, it was carefully verified that the solution was independent of the mesh count used by running the same simulation over various meshes and finding the
smallest mesh where the target parameter does not change significantly with the change in mesh size.

For the case of jet impingement study, the target parameter considered was velocity data along the center line at and after the impingement point. The velocity value at 10.33D and 12D along the center line was carefully noted for multiple meshes, and the general trend in variation was observed. The data was plotted systematically to identify the point of grid independence as shown in Figure 76.

![Grid Sensitivity study](image)

**Figure 76: Grid sensitivity study performed for impinging jet domain**

From the above figure, it can be seen that the change in value for velocity is not significant when the mesh size is increased beyond 1.35 million nodes. Table 12 shows the magnitude of variation of velocity value for other meshes with respect to 1.35 million nodes. Since the relative change in value is seen to be small, the mesh with node count of 1.35 million nodes was considered as the point of grid independence.
Table 12: Grid independence data

<table>
<thead>
<tr>
<th>Node Count (millions)</th>
<th>At 10.33D along centerline</th>
<th>At 12D along centerline</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Diff. wrt to 1.35 million case</td>
<td>% Diff. wrt to 1.35 million case</td>
</tr>
<tr>
<td>0.69</td>
<td>3.139</td>
<td>0.015</td>
</tr>
<tr>
<td>0.96</td>
<td>3.151</td>
<td>0.003</td>
</tr>
<tr>
<td></td>
<td>3.154</td>
<td>0</td>
</tr>
<tr>
<td>1.86</td>
<td>3.156</td>
<td>-0.002</td>
</tr>
<tr>
<td>2.59</td>
<td>3.158</td>
<td>-0.004</td>
</tr>
</tbody>
</table>

For all the further studies, a grid size of 1.35 million nodes was used for the symmetric model of jet impingement domain. It is to be noted that with the study with different impingement angle, using the same grid size and meshing philosophy; changes in θ, Δx and Δy will occur; there by changing the values of FD per case basis. But, it has also been noted that the effect of FD is significantly less on velocity parameter when compared to TKE parameter. Hence, no significant difference between the simulated data and real physics is expected.

5.7 Effect of Impingement Angle

After systematically attaining a grid size where the solution was found to be independent to further changes in the number of grid elements; 3 different cases with impingement angles of 30°, 45° and 60° and at a Reynolds number of 7,500 were analyzed. Contours and data values were taken along multiple planes to quantitatively and qualitatively understand the flow physics. The experimental data from Landers [2] and Disimile et al. [1] has been used for validation of the CFD data. This helps in providing confidence in the accuracy of computational results, and acts as the stepping stone for further studies done on impinging jets.

5.7.1 Velocity Profile Validation

Velocity data from Landers [2] was used to validate the simulation results to ensure that the data obtained from simulation is of highest quality. Landers has provided non-dimensional velocity...
profiles along horizontal and vertical planes, taken at different distances downstream of jet impingement for all the jet impingement angle considered. Hence, the data extracted from [2] is plotted against the data obtained from simulation. It is to be noted that Landers observed a difference of up to 4.2% in the velocity data at horizontal plane and up to 4.4% in vertical plane based on the experimental setup used. He also noted that there exists an uncertainty factor of 4% on the experimental calculation of velocity components based on the instrumentation used.

For the 30-degree case, at 4 different location (12D, 14D, 16D and 18D) downstream of pipe exit, the data was taken along horizontal plane and compared with the experimental data as shown from Figure 77 to Figure 80. It can be observed that there exists a good match between the computational results and the experimentally obtained data. This provides confidence in the accuracy of the computational results.

Figure 77: Non-dimensional velocity profile along 12D in horizontal plane for 30-degree impingement case
Figure 78: Non-dimensional velocity profile along 14D in horizontal plane for 30-degree impingement case

Figure 79: Non-dimensional velocity profile along 16D in horizontal plane for 30-degree impingement case
Figure 80: Non-dimensional velocity profile along 18D in horizontal plane for 30-degree impingement case

The data from all the locations are plotted together in one figure as shown in Figure 81. From this figure, it can be observed that the experimental data falls on a narrow band of values. But as the impingement angle is increased, the spread in experimental data has increased as shown in Figure 82 and Figure 83. This may be due to the difficulties in accurate measuring of velocity parameters when the flow field experiences high mixing.
Figure 81: Non-dimensional velocity profile in horizontal plane for 30-degree impingement case

Figure 82: Non-dimensional velocity profile in horizontal plane for 45-degree impingement case
Figure 83: Non-dimensional velocity profile in horizontal plane for 60-degree impingement case

Similarly, data was extracted from vertical plane and plotted along multiple locations downstream of pipe exit and compared with Landers’s experimental data. In the vertical plane, the data spread observed in the experimental results has increased as seen in Figure 84 to Figure 86. This can possibly be attributed to problems during the experiment, since the vertical plane witnesses high amount of interaction. Based on these validations, it was concluded that the results obtained from simulations are of high quality, and further studies were conducted on the impinging jets.
Figure 84: Non-dimensional velocity profile in vertical plane for 30-degree impingement case

Figure 85: Non-dimensional velocity profile in vertical plane for 45-degree impingement case
Figure 86: Non-dimensional velocity profile in vertical plane for 60-degree impingement case

5.7.2 Velocity Contours

Once the simulation results were validated with experimental data, velocity contours were taken along the horizontal and vertical plane to understand the flow characteristics of the resultant jet formed from impingement. The velocity contours along the horizontal plane is shown below from Figure 87 to Figure 89 and along the vertical plane is shown from Figure 90 to Figure 92. The black (+) sign denotes the geometrical impingement point, 10.33D from the exit of the pipe domain and along the centerline of the jet domain.
Figure 87: Velocity profile along horizontal plane for jet impingement angle of 30 degrees

Figure 88: Velocity profile along horizontal plane for jet impingement angle of 45 degrees

Figure 89: Velocity profile along horizontal plane for jet impingement angle of 60 degrees
Figure 90: Velocity profile along vertical plane for jet impingement angle of 30 degrees

Figure 91: Velocity profile along vertical plane for jet impingement angle of 45 degrees

Figure 92: Velocity profile along vertical plane for jet impingement angle of 60 degrees
From the above contours, the following observations could be made:

1. Higher the impingement angle, lower the velocity of jet near the zone of impingement.

2. The velocity falls from 3.1 m/s at impingement point for 30-degree case to 2.3 m/s for the case with 60-degree impingement. This represents a reduction of 26% in velocity.

3. Hence, in applications where higher momentum of jet is required; it may be beneficial to use lower impingement angle.

4. In Figure 92, a region of lower velocity region can be seen growing towards the impingement zone from the left. This region corresponds to the region of interaction of free shear layer of jets. Even though the velocity is low, the region represents zone of high interaction between the jets.

5.7.3 TKE Contours

TKE contours were determined along the horizontal and vertical plane to understand the turbulence characteristics of the resultant jet formed from jet impingement. The TKE contours along the horizontal plane is shown below from Figure 93 to Figure 95 and along the vertical plane is shown from Figure 96 to Figure 98. For all the cases, a common scale was used for plotting TKE data for easier understanding of turbulence values obtained.

From the TKE contours, the following observations could be made:

1. Higher the impingement angle, larger the zone of high turbulence zone formed near the impingement region. This is visually clear from Figure 94 and Figure 95.

2. From Figure 97 and Figure 98, we can observe that the region corresponding to lower velocity (as seen in Figure 91 and Figure 92) denotes regions of higher TKE values. This is due to the interaction of free shear regions of the jets.
Figure 93: TKE profile along horizontal plane for jet impingement angle of 30 degrees

Figure 94: TKE profile along horizontal plane for jet impingement angle of 45 degrees

Figure 95: TKE profile along horizontal plane for jet impingement angle of 60 degrees
Figure 96: TKE profile along vertical plane for jet impingement angle of 30 degrees

Figure 97: TKE profile along vertical plane for jet impingement angle of 45 degrees

Figure 98: TKE profile along vertical plane for jet impingement angle of 60 degrees
From the velocity and TKE contours, it can be concluded that for cases requiring higher momentum near the zone of impingement; it would be beneficial to use lower impingement angle. But if a higher mixing is considered requisite at the impingement zone, it would be advantageous to invest in higher impingement angle; as higher TKE generally denotes better mixing.

5.7.4 Center-line Velocity Profile

To quantitatively understand the difference in the flow characteristics of the resultant jet formed from impingement at various angles, the velocity data along the centerline is taken. The center line forms as the intersection of horizontal and vertical plane. Hence, it carries the characteristics of both the planes.

The velocity profile along the center line for the 3 cases of impingement angle is plotted in Figure 99 and compared with the single jet data. Beyond impingement point; the decay rate of velocity along the center line of the resultant jet resembles the decay rate for the single jet. Also, it is to be noted that the maximum velocity in the case of 30-degree impingement angle is higher than the value observed in case of 60-degree impingement (an observation made already from the contour plots).
To further understand the decay physics, normalized velocity profile along the center line was plotted as shown in Figure 100.
From Figure 100, it can be seen that the maximum velocity seen along the center-line shifts to the right, and moves beyond impingement point as the jet impingement angle is increased. This effect can be explained to the fact that at higher impingement angle, the individual jets have already spread significantly before reaching the point of interaction; and hence carry lesser energy with them. The area of interaction between the jets has also increased significantly due to the higher spread of the jets; and hence the zone of higher velocity gets shifted beyond the impingement zone. The center-line decay of velocity for an axi-symmetric jet can be modelled with a 1/x decay profile, where x represents the distance from the pipe exit. In Figure 101, a 1/x profile was generated and plotted against single jet results showing that the single jet center line velocity decay obeys the 1/x decay profile. This decay rule is observed to be obeyed beyond the potential core of the round jet, the extent of which has been a topic of significant research as shown in [13], [52], [53] and [54]. For the current study, 10D was considered as the length of potential core region for round jet.

![Normalized velocity profile along center line](image)

Figure 101: Normalized velocity profile along the center line compared with 1/x profile

107
Similarly, when $1/x$ profile was plotted for the resultant jet formed from jet impingement, it can be seen that beyond 20D (almost 10D downstream of the jet impingement point); the resultant jet also obeys a $1/x$ decay profile. This implies that at 10D from jet impingement point; the resultant jet formed has fully developed characteristics of a single jet and does not behave as 2 independent jets.

5.7.5 Center-line TKE Profile

The TKE profile along the center line for the 3 cases of impingement angle is plotted as shown in Figure 102 to Figure 104 and qualitatively compared with experimental data obtained by Landers [2]. From the single jet results, it was established that beyond 6D, the simulation overpredicted the TKE parameters quantitatively. In a qualitative sense, it was observed that the simulation followed the similar trend observed in experimental data. Hence, only qualitative comparison with Landers’s experimental results are performed by using scaled parameters. It is to be noted that Landers observed a difference of up to 14.5% in the $u_{rms}$ data based on the experimental setup used. Hence, slight mismatch between experimental data and simulation results is not considered significant.

From Figure 102 it can be observed that the CFD results predict the peak in TKE parameter 1.5D before the peak observed in experimental data. Also, the experimental data shows a significantly sharper fall in TKE parameter compared to the results obtained from the simulation.
As the impingement angle is increased to 45 degrees, the difference in trend between the experimental data and computational results is seen to have reduced. This can be observed in Figure 103 where the peak in TKE obtained from experimental results and computational results are off by a margin of 0.6D. Still, a significant difference in the post impingement TKE profile exists between experimental and computational data.

With further increase in impingement angle to 60 degrees, the difference in trend between the experimental data and computational results is seen to become negligible before impingement as seen in Figure 104. Both experimental and computational data predicts the zone of highest TKE at the same location (almost 1D before impingement point). Beyond impingement point, the difference in TKE value predicted by the experimental methodology significantly differs from the CFD results.
Figure 103: Normalized TKE profile along the center line for 45-degree jet impingement

Figure 104: Normalized TKE profile along the center line for 60-degree jet impingement
It is to be considered that the jet impingement leads to highly turbulent mixing regions where significant interaction of turbulent scales occur. The turbulence model used in this steady state analysis may not be able to account of all the interaction and hence the significant difference in TKE value predicted by experimental and CFD.

The data from all the different impingement angles is plotted together as shown in Figure 105 for easier visualization. It can be seen the experimental data follows a very narrow band of values before and after impingement (except for the spread in experimental data for the 30-degree case). Similarly, the data from 45-degree and 60-degree case also follows a close band of values. At 5D beyond impingement point, it can be observed that all the simulation data follows a similar trend, with 30-degree case having higher normalized TKE when compared to 45 and 60-degree impingement cases.

Figure 105: Normalized TKE profile along the center line for various angles of impingement
A better understanding of the turbulent properties can be attained by plotting the turbulence intensity along the center-line as shown in Figure 106. It can be observed that the maximum turbulence intensity observed in the single round jet is 11.7%. In the case of 30-degree impinging jets, this value goes up to 28.7% which denotes an increase in turbulence intensity of 145%.

Between 30 and 45-degree case, the turbulence intensity increases from 28.7% to 34.7% which represents an increase of 21%. And between 45 and 60-degree case, the turbulence intensity increases from 34.7% to 40.5% signifying an increase of 17%. This implies that at higher impingement angle, we can expect higher turbulence intensity which will aide better mixing.

![Figure 106: Turbulence intensity profile along center line for various angles of impingement](image)

Figure 106: Turbulence intensity profile along center line for various angles of impingement

### 5.7.6 Spread profile of the jet

To estimate the spread profile of the jet in horizontal and vertical planes, velocity values were considered along radial locations for every 10 degrees at 3 locations downstream of jet
impingement point; at 12D, 16D and 20D beyond pipe exit. Along these radial locations, the velocity values were calculated to estimate the half-width. Half-width is defined as the location along the line where the velocity becomes half of the center-line velocity. Using the half-width half-max (HWHM) as the measuring parameter, the profile of the jet at that plane was plotted. It is interesting to observe that the jet attains an elliptical profile after impingement as shown in Figure 107. The half width along the radial lines was normalized using pipe diameter. A similar observation was made by Disimile et al. [1] and Rho et al. [22]. This provides more confidence in the legitimacy of the results and act as a validation criterion.

Similarly, jet profile was analyzed for various locations for the 45-degree and 60-degree case. The results from the analysis is shown in Figure 108 and Figure 109. From these 3 figures, it can be observed that the growth of jet in z direction (in vertical plane) is significantly higher when compared to the growth in y direction (in horizontal plane).
Figure 108: Profiles of jet at various locations after impingement for 45-degree case

Figure 109: Profiles of jet at various locations after impingement for 60-degree case
Interestingly, it was observed that for all the impingement angle; the half distance in y direction reaches a maximum value for any given location beyond 12D and does not change with impingement angle. For example, the b1 parameter (half distance in y direction normalized with diameter of pipe) is approximately 1.3 for all the cases at 16D from pipe exit and 1.7 for all cases at 20D. This is clearly represented in Figure 110 to Figure 112.

Even though the spread in y direction is not affected by the impingement angle, the growth of the jet in z-direction shows promising results. With the increase in in impingement angle, the growth in z-direction increases as we move further downstream.

![Profile of resultant jet after impingement](image)

Figure 110: Profiles of jet at 12D for various impingement angles
Figure 111: Profiles of jet at 16D for various impingement angles

Figure 112: Profiles of jet at 20D for various impingement angles
The data regarding the growth of the jet in y and z direction is represented using parameters $b_1$ and $b_2$; where $b_1$ is the half width along the y direction measured in pipe diameters and $b_2$ is the half width along the z direction. The data is shown in Table 13, where the growth along y and z directions are normalized to show the effect of location and angle on growth along the planes. The parameter $b_2/b_1$ represents the ratio of growth in vertical plane to growth in horizontal plane. For a round jet, the value of $b_2/b_1$ is 1 at every location.

From the below table we can conclude that the growth at 20D in vertical plane achieved by jets impinging at 30-degree angle can be achieved by 16D by 45-degree impinging jets and approximately at 14D by 60-degree impinging jets.

<table>
<thead>
<tr>
<th>Case</th>
<th>Location</th>
<th>$b_1$</th>
<th>$b_2$</th>
<th>$b_2/b_1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>30-degree Impingement</td>
<td>12D</td>
<td>1.17</td>
<td>2.26</td>
<td>1.94</td>
</tr>
<tr>
<td></td>
<td>16D</td>
<td>1.32</td>
<td>3.23</td>
<td>2.45</td>
</tr>
<tr>
<td></td>
<td>20D</td>
<td>1.69</td>
<td>4.13</td>
<td>2.44</td>
</tr>
<tr>
<td>45-degree Impingement</td>
<td>12D</td>
<td>1.23</td>
<td>2.76</td>
<td>2.24</td>
</tr>
<tr>
<td></td>
<td>16D</td>
<td>1.30</td>
<td>4.17</td>
<td>3.21</td>
</tr>
<tr>
<td></td>
<td>20D</td>
<td>1.66</td>
<td>5.56</td>
<td>3.35</td>
</tr>
<tr>
<td>60-degree Impingement</td>
<td>12D</td>
<td>1.32</td>
<td>3.26</td>
<td>2.47</td>
</tr>
<tr>
<td></td>
<td>16D</td>
<td>1.32</td>
<td>5.07</td>
<td>3.86</td>
</tr>
<tr>
<td></td>
<td>20D</td>
<td>1.68</td>
<td>7.09</td>
<td>4.23</td>
</tr>
</tbody>
</table>

Table 13: Growth profile of resultant jet in y and z direction

To identify the profile of the resultant jet, an elliptical profile was created using $b_1$ as the semi-minor axis and $b_2$ as the semi-major axis and overlaid with the computational profile obtained. As seen in Figure 113 the simulation results closely follow the elliptical profile. This helps in concluding that the resultant jet formed by the interaction of 2 impinging round axi-symmetric jets at low Re is an elliptical jet.
Profiles taken at locations further downstream also indicate the presence of an elliptical profile as shown in Figure 114 and Figure 115. As we move downstream, it is observed that the match between the elliptical profile and computational data has decreased for the 30-degree impingement case.

Figure 114: Comparison of velocity profile with elliptical profile at 16D for 30-degree case
Figure 115: Comparison of velocity profile with elliptical profile at 20D for 30-degree case

Growth profile along all the location for the 3 impingement angle cases are plotted as shown in Figure 116 to Figure 118. It is observed that at impingement angles of 30-degree, the profile at 12D closely matches with the elliptical profile; but doesn’t tend to match at 20D. But at case with higher impingement angle of 60-degree, the further downstream locations have better match than the upstream location. In all the cases of impingement angles, the computed profile closely follows an elliptical shape at 16D location.

For the case with 60-degree impingement angle, we can observe a strong deviation from the elliptical profile at 12D. This may be caused by the intense interaction between fluid streams which accounts for further developing profile.
Figure 116: Comparison of velocity profile with elliptical profile for 30-degree case

Figure 117: Comparison of velocity profile with elliptical profile for 45-degree case
Figure 118: Comparison of velocity profile with elliptical profile for 60-degree case

5.8 Effect of Reynolds Number

Once the study based on different impingement angle was completed, different Reynolds number impinging at 30-degree angle were analyzed to understand the effect of Reynolds number on the flow and spread characteristics of the resultant jet. The previous work on same Reynolds number jets has provided confidence in the efficacy of computational tools in accurate prediction of velocity and spread characteristics of the jets. Based on the insight gained from that work, it was considered beneficial to understand the impact of Reynolds number of jets in prediction of spread characteristics.

For the current study, three different Reynolds number were considered; starting with the Reynolds number of 7,500 as used in previous case (which also acts as a baseline) and Reynolds number of 5,000 and 10,000 by modifying the velocity parameter in the pipe. Similar to the single jet study, pipe flow simulation was performed for generating the inlet velocity profiles for the jets at
Reynolds number of 5,000 and 10,000 respectively; and the profiles were added as the inlet condition in jet domain analysis.

5.8.1 Velocity Contours

Velocity contours were extracted along the horizontal and vertical plane to visually represent the flow characteristics of the resultant jet formed due to impingement. The velocity profile along the horizontal plane is shown from Figure 119 to Figure 121 while the profile along the vertical plane is shown in Figure 122 to Figure 124.

Along the horizontal plane, it is observed that at lower Reynolds number, the centerline velocity approaching the impingement point (denoted by + sign) is lower when compared to cases with higher Reynolds number. This impacts the post-impingement velocity data also linearly. After impingement, the case with lower Reynolds number exhibit lower velocity magnitude when compared to cases with higher Reynolds number.
When the data from vertical plane is considered, it is observable that the maximum velocity region is very concentrated near the impingement zone. In cases with different impingement angles, it was observed that this shifted towards right of impingement point with the increase in
impingement angle. An observation can be made here that the qualitative characteristics of the jets does not seem to depend heavily on the Reynolds number, in the sense that irrespective of the Reynolds number; the zone of maximum velocity in the vertical plane lies on the same region. This hypothesis must be tested considering the TKE parameter also and quantitatively evaluated by considering the values of velocity and TKE along the center-line.

Figure 122: Velocity profile along vertical plane for jets impinging at Re=5,000

Figure 123: Velocity profile along vertical plane for jets impinging at Re=7,500
5.8.2 TKE Contours

To further establish the characteristics of the interaction of jets impinging at different Reynolds numbers, Turbulence Kinetic Energy parameter along the horizontal and vertical planes were considered. The profiles along the horizontal planes are shown from Figure 125 to Figure 127 and along vertical planes are shown from Figure 128 to Figure 130.

Considering the TKE parameter, it can be observed that in the case with Reynolds number 5,000; the maximum value of TKE observed in the free shear region is 4-6 times lesser than the values of TKE observed in the case with Reynolds number 10,000. This implies that by just doubling the velocity, the effective TKE has increase more than 4-fold. Also, it can be observed that the magnitude of TKE observed in the case of jets impinging at 30 degrees at Re=10,000 is 70% more than the TKE observed in the case of jets impinging at 60 degrees at Re=7,500.
Figure 125: TKE profile along horizontal plane for jets impinging at Re=5,000

Figure 126: TKE profile along horizontal plane for jets impinging at Re=7,500

Figure 127: TKE profile along horizontal plane for jets impinging at Re=10,000
Hence, the optimal scenario for extraction of higher TKE from resultant jets will be by utilizing higher velocity impinging jets rather than using higher impingement angle; when it is observed that 33% increase in Reynolds number (velocity) can yield 70% increase in TKE.

The characteristics of TKE along the vertical plane is consistent with the behavior of velocity parameter along the vertical plane. It is observed that the zone of maximum TKE for all the different Reynolds number cases occur at the same location in the vertical plane. This helps us further conclude that the Reynolds number does not affect the characteristics of the jet qualitatively.

Figure 128: TKE profile along vertical plane for jets impinging at Re=5,000
Figure 129: TKE profile along vertical plane for jets impinging at Re=7,500

Figure 130: TKE profile along vertical plane for jets impinging at Re=10,000
5.8.3 Center-line Velocity Profile

To obtain a quantitative understanding of the difference (or similarities) in flow characteristics of the resultant jet formed from impingement at various Reynolds number, the velocity parameter was initially studied along the center-line of the jets. Similar to the cases with different impingement angle, the data from single jet study was added to act as a baseline.

It can be seen from Figure 131 that the point of maximum velocity along the center-line lies close to 10D from the pipe exit for all the Reynolds number cases. As observed from the velocity contour plots, the area of maximum velocity region coincides for all the cases. Also, similar to observation made from velocity contour plot, the maximum value of velocity was observed for the case with maximum inlet Reynolds number.

![Velocity profile along center line for various Reynolds number](image)

Figure 131: Velocity profile along the center line for various Reynolds number
Normalized velocity profile was generated by dividing the local velocity along the center-line with the maximum velocity along the center-line for the corresponding case and plotted against distance axial distance from pipe exit. This helped in categorically isolating the zone of maximum velocity for all the cases, which was observed to occur very close to the impingement point (10.33D from pipe exit) as seen in Figure 132.

Similar to the cases with different impingement angle, 1/x profile was used to compare the decay profile of the resultant jet with the standard decay profile of round jet. It can be observed that the decay profile of resultant jet matches very closely with 1/x profile from 20D onwards (close to 10D from impingement point). This could be an indicator of the distance required for the resultant jet to attain a fully developed state beyond the impingement point.

Figure 132: Normalized velocity profile along the center line for various Reynolds number
5.8.4 Center-line TKE Profile

Considering the center-line normalized TKE, we observed that the trend is similar to that of the velocity profile. For all the cases, the normalized TKE parameter peaked at the same location; approximately 2D before the impingement point as shown in Figure 133. This indicates that the Reynolds number in the range analyzed in this study does not influence the qualitative aspect of the flow, and that it affects the quantitative aspects of the flow considerably.

![Normalized TKE profile along center line for various Reynolds number](image)

Figure 133: Normalized TKE profile along the center line for various Reynolds number

Considering the turbulence intensity along the center-line produces the same trend observed in the case of normalized TKE as seen in Figure 134. For all the cases, the maximum value of TI corresponds to approximately 28.6%, and the region of maximum TI corresponds to 2D from impingement point. Hence, it can be concluded that for the increase in TKE parameter, there is an almost linear increase in velocity which maintains the TI values approximately equal across the Reynolds number studied.
Figure 134: Turbulence intensity profile along center line for various Reynolds number

5.8.5 Spread profile of the jet

From the previous plots, we were able to conclude that the behavior of the resultant jet formed due to impingement did not exhibit any Reynolds number dependence qualitatively and they performed ideally similar to one another when normalized values were considered. This observation gains further credence when the growth profile of the resultant jet is considered. As observed from Figure 135 to Figure 137, it is clear that the growth profile predicted for the resultant jet is identical for all the values of Reynolds number considered.

This implies that the jet growth characteristics is strongly dependent on the impingement angle and not significantly on the velocity of impingement. This understanding of the flow physics will help in optimal allocation of resources when higher spread rate is required over higher jet momentum.
Figure 135: Profiles of jet at various locations after impingement for Re=7,500

Figure 136: Profiles of jet at various locations after impingement for Re=5,000
Isolating the profiles along one specific location for all the Reynolds number considered helps to generate the plots shown in Figure 138 to Figure 140. From all three figures, it can be observed that the profile at any location matches irrespective of the Reynolds number. With this we conclude that increasing the Reynolds number may not ideally increase the spread rate of the jet, even though the energy carried by the jet is increased.
Based on the growth data of jet in y and z directions, a table is generated as shown in Table 14. It is observed that growth parameters along similar location is the same for different value of Reynolds number. Only 0.5% growth in $b_2/b_1$ parameter observed when the velocity is double from Re of 5,000 to Re of 10,000. Since the value of increase is negligible, it can be concluded
with confidence that the Reynolds number does not influence the flow characteristics of the resultant jet qualitatively.

<table>
<thead>
<tr>
<th>Case</th>
<th>Location</th>
<th>b1</th>
<th>b2</th>
<th>b2/b1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Re 7,500</td>
<td>12D</td>
<td>1.17</td>
<td>2.26</td>
<td>1.94</td>
</tr>
<tr>
<td></td>
<td>16D</td>
<td>1.32</td>
<td>3.23</td>
<td>2.45</td>
</tr>
<tr>
<td></td>
<td>20D</td>
<td>1.69</td>
<td>4.13</td>
<td>2.44</td>
</tr>
<tr>
<td>Re 5,000</td>
<td>12D</td>
<td>1.18</td>
<td>2.27</td>
<td>1.93</td>
</tr>
<tr>
<td></td>
<td>16D</td>
<td>1.33</td>
<td>3.23</td>
<td>2.43</td>
</tr>
<tr>
<td></td>
<td>20D</td>
<td>1.70</td>
<td>4.13</td>
<td>2.43</td>
</tr>
<tr>
<td>Re 10,000</td>
<td>12D</td>
<td>1.16</td>
<td>2.25</td>
<td>1.95</td>
</tr>
<tr>
<td></td>
<td>16D</td>
<td>1.31</td>
<td>3.22</td>
<td>2.46</td>
</tr>
<tr>
<td></td>
<td>20D</td>
<td>1.68</td>
<td>4.13</td>
<td>2.45</td>
</tr>
</tbody>
</table>

Table 14: Growth profile of resultant jet in y and z direction for various Reynolds number.
6 Conclusion

6.1 Pipe flow study

Pipe flow analysis was performed for the Reynolds number of 7,500 and the outlet data was extracted to serve as the inlet boundary condition for the next phase of the study wherein a single axisymmetric jet will be examined over a range of Reynolds numbers and eventually used for a jet impingement study. Grid independence was initially performed to establish a mesh configuration so that the solution obtained from the analysis did not depend on the mesh used. Afterwards, the effect of flow length on the development of velocity and turbulent flow variables were studied in order to establish the minimum distance required to obtain a fully developed flow with respect to mean velocity. It was observed that Realizable k-ε model reached an asymptotic state showing full development at flow length of 20-25 diameters while SST model predicted 40-45 diameters as the development length, showing a difference of approximately 47%. It was also observed that the value of centerline velocity predicted by both the models showed a difference of approximately 7%.

While comparing the distance taken by both the models to attain full development in terms of turbulence kinetic energy, it was noted that Realizable k-ε model again predicted lower value for full development. While the SST model predicted 55-60 diameters to attain full development, the Realizable k-ε model required only 30-35 diameters to reach asymptotic state for turbulence kinetic energy, showing a difference of approximately 43%. Based on these results, it was concluded that a pipe length greater than 40 diameters (0.8 meters) was optimum for attaining a fully developed mean flow within the pipe domain, and a longer pipe was needed for the turbulent kinetic energy to reach an asymptotic state.
To gain confidence on the accuracy of the results obtained from the analysis, the results were validated against experimental results; both published (Eggels et al. [9]) and in-house (Landers and Disimile [23]). Normalized velocity plot was generated for both the turbulence models and compared with the experimental results. While the in-house experimental results and SST results compared well with the published results, the computational data obtained from Realizable k-ε model showed mismatch.

Law of the wall plot was generated from result obtained from SST model and compared with Eggels’ data. This provided confidence in the mesh since good agreement was observed between both the profiles in viscous sublayer. But, disparity was observed beyond the viscous sublayer, which may have arisen due to the fact that Eggels’ utilized a very smooth honed pipe with lower surface roughness. The mismatch in buffer layer and log law region was further analyzed using velocity defect plot. A close agreement was observed between both the profiles there by ensuring the proper capturing of all the aspects of flow, from viscous sublayer to the flow core for SST model based analysis. From all these validations, it can be concluded that SST model under this Reynolds number condition performs better than the Realizable k-ε model in capturing the pipe flow physics.

6.2 Round jet study

Single jet simulations and analyses were performed at a Reynolds number of 7500 using various turbulence models to evaluate the performance of each model. This was done in order to determine the best possible turbulence model for a low Reynolds number round jet exiting from fully developed pipe flow. Based on the results from this study, jets impinging at an angle will be modeled in the next phase of analysis using the turbulence model that was found superior. Initially, grid independence was performed using Realizable k-ε model to ensure that the solution obtained
was independent of the grid size used. This was attained by performing the simulation using a very small mesh (0.25 million nodes) and gradually increasing the node count. It was found that beyond 1.38 million nodes, the solution did not show much dependence on the grid count, with only 2% difference in velocity and 5% difference in turbulence kinetic energy (TKE) with almost threefold increase in node count. Hence, for the purpose of single axi-symmetric round jets; 1.38 million node mesh was taken as the grid independent mesh for the current Reynolds number and simulation setup.

It was observed that all the turbulence models performed well in case of mean velocity profile (with Standard k-ω model showing slight fluctuation at 10.33D). But, while plotting TI; it was observed that none of the turbulence models were able to accurately predict the TI values beyond 6D. Standard k-ω model was found to under predict TI values at all the locations, while Realizable k-ε model and SST k-ω model were able to predict TI values till 6D. The performance of Standard k-ε model was comparable with that of Realizable k-ε model and SST k-ω model. It was concluded that for the simulation of round axi-symmetric jets originating from a fully developed pipe flow at low Reynolds numbers, the SST k-ω model was the best model to be used and Standard k-ω model may be the least appropriate model.

6.3 Impinging jet study

From the current study based on jet impingement at constant angle and constant Reynolds number, the following conclusions were made based on the validated computational results.

1. SST k-ω model can predict velocity characteristics of impinging jets with high level of accuracy, but may not be suitable for predicting TKE parameters at low Reynolds number.
2. The mesh obliquity with the velocity vector has an adverse effect on the numerical solution obtained and is referred to as False Diffusion (FD).
3. False diffusion increases with the increase in velocity, mesh oblique angle and the mesh size in all directions.

4. The only possible method to reduce false diffusion for any given set of boundary conditions is to minimize mesh size by maximizing node count; which will be computationally expensive.

5. Symmetry models, when compared to full domain models are equally capable of predicting the flow physics arising from jet impingement.

6. For the study with different jet impingement angles, the velocity data obtained from computational analysis was validated with experimental data from [2].

7. The Gaussian non-dimensionalized velocity profile matches with experimental data for all the impingement angle in both horizontal and vertical planes.

8. Region of higher velocity along the center-line was observed to shift beyond impingement point at higher values of impingement angles.

9. The resultant jet formed from the jet interaction obeys 1/x profile of jet center-line velocity decay.

10. Normalized TKE profile matches with experimental data at higher impingement angle and does not seem to agree well with experimental result at lower impingement angle.

11. The resultant jet follows an elliptical profile beyond impingement point.

12. The growth of the jet along the plane perpendicular to the pipe plane is significantly higher than the growth along the pipe plane.

13. The growth profile in both the planes is dependent on the impingement angle.
14. For cases with different Reynolds number at constant impingement angle, the region of maximum velocity was observed to be at the same location irrespective of Reynolds number. Similar was the case with the zone of maximum TKE.

15. The turbulence intensity magnitude predicted by the jets at Re=10,000 impinging at 30-degree angle was lower than the turbulence intensity for case with Re=7,500 impinging at 60-degree angle.

16. The resultant jet formed from the jet interaction at different Reynolds number also obeys 1/x profile of jet center-line velocity decay.

17. Significant effect of Reynolds number was not observed in the growth characteristics of the resultant jet in either planes.

18. Based on the requirement of the application (maximum spread or maximum momentum), an optimal jet impingement strategy can be achieved.
7 Recommendations

As part of future research options, the following methodologies should be considered to ensure that high quality computational results can be obtained so that better understanding of the jet impingement flow physics can be achieved.

1. Adaptive Mesh Refinement (AMR)

In the current study, static mesh based on insight gained from single jet study was generated to mesh the flow domain. But this may not be the ideal or even optimal solution, since jet impingement study involves major flow field interaction. Hence, it may be ideal to study the impingement flow physics using numerical methods with mesh capable of adaptively refining in zones of high interaction in an iterative process. Ideally, a uniform coarse mesh with no significant biasing is to be generated and AMR methodology used to perform local grid adaptation (refinement and coarsening) to obtain high quality results. Such as mesh generated may ideally suffer less False Diffusion.

2. Asymmetric Jets

Current analysis only considered symmetric jets. Hence, symmetric domain models were used. If one were to completely understand the full extent of flow physics involved with jet impingement study, it is considered significant to consider cases with asymmetric jet impingement, i.e., jets with different Reynolds number. This may help us gain insight into the jet growth characteristics and help isolate the impact of velocity in growth physics.

3. Impingement zone variation

The impingement zone considered in the current study was located 10.33D from the pipe exit. This was taken as a common parameter for the cases with different impingement angles and different Reynolds number cases. This distance was maintained same as the
impingement distance used by Disimile et al. [1], which served as the validation data for the current study. Since confidence on the computational methodology has been gained from the current study, the next parameter to be changed must be the distance of impingement point from pipe exit. This will help establish the effect of the impingement zone on jet growth profile.

4. **Thermal Jets**

Only isothermal conditions were considered in the current analysis. The effect of temperature must be considered to replicate the real-life applications, since no realistic condition will exist in isothermal state. Also, modification of thermal profile will impact the density of the fluid in consideration, thereby ideally affecting the flow properties. Hence, it is deemed as an interesting venture to consider thermal jets as an extension of the current study. It will be insightful to gain knowledge on the diffusion of thermal energy from the jet to the ambient fluid via mixing with entraining medium.

5. **Different exit conditions**

The flow considered in this study exited from a fully developed pipe for all the cases. It will be technically interesting to understand the flow physics due to jet impingement for cases with jet exiting from non-fully developed pipes, nozzles and bend pipes. Also, all the cases in the current study were in incompressible domain. To gain better understanding of the flow physics, compressible flow domain must be considered.

6. **Unsteady analysis**

Finally, unsteady analysis needs to be performed to understand the effect of temporal variations in the growth characteristics of impinging jets. This can be performed as an unsteady RANS based analysis or higher order LES simulations.
8 References


[33] Ansys, "Ansys Help".


9 Appendix

9.1 Comparison of impinging jet exiting from fully developed pipe against impinging jet exiting from short pipe

As part of scientific curiosity, a comparative jet impingement study was performed between a case with jet discharging from fully developed pipe flow profile and a case with jet discharging from a shorter pipe. The length of the pipe in the short pipe case is maintained low enough so as to ensure that the flow has not attained full development. The Reynolds number of the flow exiting from the pipes is maintained at a constant value of 7500 and the impingement angle was 30 degrees. This helped in isolating the effect of velocity profile of the jet discharge condition in the flow characteristics downstream.

The velocity and TKE parameters along the centerline are plotted and compared with one another. From the velocity profile comparison (seen in Figure 141), it was observed that the jet discharge inlet condition did not make significant difference in the centerline velocity profile. The normalized velocity matched quite perfectly with very minimal variation in data. Hence, it was concluded that the jet discharge condition did not have prominent effect on velocity parameter. But from Figure 142, it can be noted that the turbulence parameter downstream the impingement zone reflects a change in jet discharge condition. A slightly higher value of TKE parameter is observed for the case with the shorter pipe beyond jet impingement.
Figure 141: Velocity profile along the center line for cases with different pipe development length

Figure 142: TKE profile along the center line for cases with different pipe development length