FOSSEE, IIT Bombay
OpenFOAM Case Study Project
April 19, 2021

http://fossee.in

Report on

COMPUTATION OF AN AXISYMMETRIC JET USING OPENFOAM

Under the guidance of

Prof. Janani Srree Murallidharan

Mechanical Engineering,

Indian Institute of Technology, Bombay

Submitted by

Kunal Kailash Sharma

PhD, Chemical Engineering,

Indian Institute of Technology, Bombay

Email: skunal6996@gmail.com

ACKNOWLEDGEMENTS

I would like to express my sincere thanks, for the valuable guidance, constant support, advice, and encouragement by Prof. Janani Srree Murallidharan. Thorough understanding of the concepts which may look complex and their possible applications are made possible with the patient efforts of Prof. Janani, even in the scenarios of pandemic.

I am grateful to Mr. Divyesh Variya Sir and Mr. Ashley Melvin Sir for their directional and motivational support. I sincerely appreciate the efforts by my beloved family members and friends for their advice during my studies. A big thanks to all, who helped me knowingly or unknowingly contributing the present work.

Kunal Kailash Sharma
(Ph.D., Chemical Engineering
Indian Institute of Technology, Bombay)

ABSTRACT

The study is primarily centered in validating a paper as titled above. Further the consistency of the simulation results are compared with the experimental results obtained by **PL** (N. R.**P**ANCHAPAKESAN and J. L.**L**UMLEY). The simulation employed the **k-epsilon model** in **OpenFoam** Software for the simulation of the axisymmetric turbulent air jet using **RANS** (Reynolds Averaged Navier-Stokes equations) and the **blockMeshDict** utility for meshing of **3D wedge** geometry, the solver used was **simpleFoam**. Our forecast using the k-epsilon model and the experimental results by **PL** are in high similitude, even better than the simulation of the referred paper.

Keywords: CFD, OpenFoam, experimental results, simulation, axisymmetric.

TABLE OF CONTENTS

	Page
Abstract	ii
Acknowledgements	iii
Table of Contents	iv
1. Introduction	1
2. Problem Statement	3
2.1. Assumptions	3
2.2. Calculations:	3
2.2.1 Velocity Calculation	3
2.2.2 Wall Distance calculation	4
2.2.3 Turbulent Kinetic Energy Calculation (KE)	5
2.2.4 Dissipation Epsilon Calculation (ε)	5
2.2.5 Effective viscosity calculation (ν_t):	5
2.2.6. Mixing length Calculation	6
3. Equations:	7
3.1 Governing Equations:	7
3.2 Model Equations:	7
4 Simulation Procedure	8
4.1 Geometry and Mesh	8
4.2 Initial and Boundary Conditions	9
4.3 Solver	10
4.4 Post-Processing:	10
5 Results and Discussions	10
6 Conclusions	15
7 References	15

1. Introduction :

Science, in its initial stage of development has gone through many hurdles. The things has been difficult to prove or predict, there are theories which are originated from one person's perspective, have tremendous potential applications in practice today, but were discouraged by people at that point of time. To prove a concept one must conduct a series of experiments, amalgamate both the results and concept and hence the consistency of the concept is established. There are cases in which more than 95 percent of the experimental efforts are both resources and time consuming, which take years and years or may be generations for a discovery. We don't know, how many of the important theories or predictions have been lost, which could accelerate our progress even from now on.

Today, the 'efficiency' of the experiments are sky-rocketed by fuelling them 'simulations', even without conducting a particular experiment we can get the most reliable predictable result depicting the real scenarios which are at least a deciding factors whether to go for a particular run or not. Thanks, to the humongous potency of computations. Computational fluid dynamics is one of those revolutionary power-ups. It mainly focuses on predicting the dynamics of the fluid, computationally, to approach the real fluid behaviour. To get the access of this tool to everyone, openFoam is an open source software which is free for all to use and has a tremendous potential to simulate the near-to-real cases. With their numerous and wide applications like room space heating, **[1]** air conditioning, circular jets are demonstrating the utilities **[2].**

In jet system, the tiny cross-sectional inlet, which may be a pipe, a nozzle or a sharp-edged orifice (generalised as entrance) which is a primary component but heart and soul of the system. The incoming stream passes through the narrow cross-sectional area of the entrance and at the entrance downstream, the gas stream expands to occupy the whole volume due to the sudden expansion in the entertainment section and hence the reduction in the velocity exist in the axial and radial direction of the section, while the stream travels through the entrance the pressure energy is transformed into kinetic energy which after getting past is converted into pressure energy in the entrainment section, the outlet or jet stream exit.

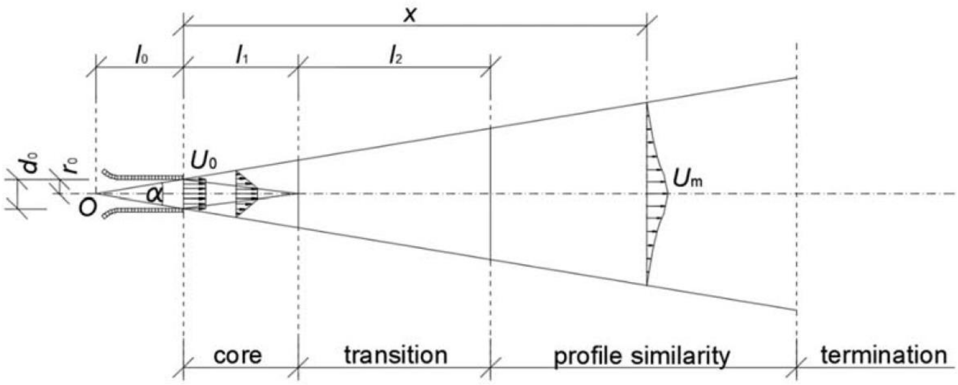


Figure 1: Schematic of the Jet system **[5] [6].**

The explanation of the figure is as follows:

- Core region ($4d_0$ to $6d_0$): The centreline velocity is equal to or greater than 95% of the supply velocity. At the exact centreline the supplied velocity continues **[4].** The velocity profile is conical **[5].**
- Transition zone ($6d_0$ to $20d_0$): There is an approximated centreline velocity decay in this region proportional to *x*-0.5 **[3]**.
- Profile similarity: Centreline velocity decay is approximately proportional to *x*-1, the transverse velocity profiles are similar. A fully developed turbulent 3D jet. **[1, 2].**
- Termination: Centreline velocity decay is approximately proportional to x^2 **[1].** The lateral mixing, results in the jet being decelerated due to its momentum transfer to the surrounding fluid **[2].**

U_0 - air velocity at the jet outlet;

Um - centreline velocity in any cross-section at x.

r_0 - radius of jet outlet;

d_0 - diameter of jet outlet=$2r_0$

l_0 – initial nozzle length;

l_1 - core zone length,

l_2 - transition zone length;

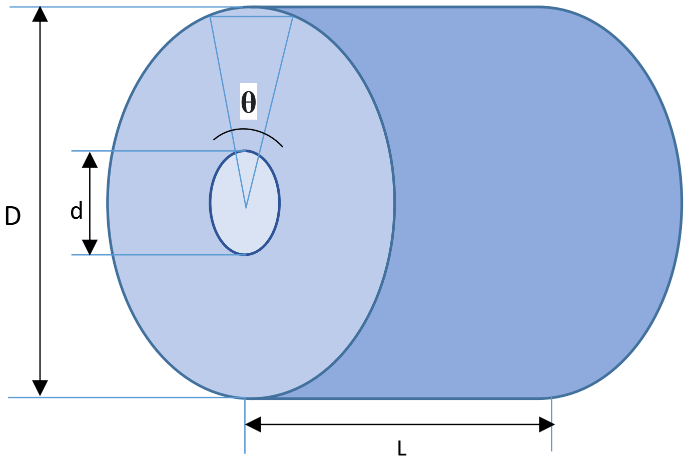
x - distance from jet entrance

α - stream angle

The paper presents the CFD analysis of the jet system using OpenFoam **V7**. Such analyses are essential in predetermining the function of the jet system for a particular application, and perform the corresponding modifications for the same, as per the need.

2. Problem Statement :

To compare the results of the practical experiment of an axisymmetric jet, done by **PL** (N.R.**P**ANCHAPAKESAN and J.L.**L**UMLEY), **[7]** with the predictions from CFD simulations and similarly validate the CFD simulation results by (Kannan B.T.) **[8]**



d=diameter of orifice (inlet)

D=diameter of the cylinder, entrainment section (wall).

D=300d

L=Length of the cylinder

L=250d

Figure 2: Problem Schematic.

2.1. Assumptions: [7] [8]

Following are the assumptions considered while performing the case study:

- Jet is axisymmetric- the geometry is symmetric across the axis. (here, consider x axis)
- Reynolds number = 11000 (representing turbulent conditions).
- Fluid: Air (to clarify all the fluid properties).
- Orifice diameter is 0.0061m.
- Turbulent Intensity (by PL) = 0.0001.
- Steady state and incompressible flow.

2.2. Calculations:

This sections encloses all the possible calculations involved.

2.2.1 Velocity Calculations:

Reynolds number is an essential dimensionless quantity to determine the nature of the flow either laminar or turbulent. For the cylindrical geometry,

- $Re<2100$, for laminar flow

- Re>4000, for turbulent flow
- 2100<Re<4000, transition flow regime.

Equation 1:

Reynolds number; Re = (Inertial Force)/ (Viscous force) = $\frac{du\rho}{\mu}$

Orifice diameter, d = 0.0061m

Reynolds number, Re = 11000

Dynamic viscosity, μ = 0.000018 Pa-s

Density, ρ = 1.2 kg/m^3

The jet inlet velocity at the orifice can be obtained

u=U_0= 27.04918033 (which agree with the PL experiment) **[7]**

2.2.2 Wall Distance calculation:

Wall functions are required to model the velocity near the walls. (The non-dimensional distance (y^+) and non- dimensional velocity (u^+) are used in the equation. The regimes are as follows:

- Viscous sub-layer 0 < y^+ < 5
- Buffer layer 5 < y^+ < 30
- Log-law layer 30 < y^+ < 300

For $k{-}\varepsilon$ model, this equation holds, keeping y_p within the log-law region of the wall to apply physics at the location **[9]**.

Equation 2:

$y^+ = y_p\,(0.5 C_f U_\infty{}^2)^{0.5} / \nu$

y_p is the distance from the wall,

U_∞ is the free stream velocity, and

ν is the kinematic viscosity.

The skin friction coefficient Cf is given by **[9]**,

Equation 3:

C_f=0.078 $Re^{-0.25}$ = 0.007616342

For the first cell centre y_p to be within the log-law region from the wall, the dimensionless wall distance y^+ should be 30<y^+<300

Here, we have taken y^+=100.

Hence, from equation 2**,**

y_p =0.000899

Therefore, the cell centre of the first cell should be 0.000899m away from the wall.

Cell width; $y_a=2*y_p$= 0.001797252

2.2.3 Turbulent Kinetic Energy Calculation (KE):

KE= 0.5* Average ($u' \cdot u'$) = 0.5*($u'^2+v'^2+w'^2$)

In the above formula:

u'=velocity vector

u'^2= x component of velocity vector

v'^2= y component of velocity vector

w'^2= z component of velocity vector

Assuming isentropic turbulence, for core region only away from the walls **[9]**: $u'^2=v'^2=w'^2$

Equation 4:

KE= 1.5* Average (u'^2) = 1.5(U_∞ I)^2 = 7.31658E-06

2.2.4 Dissipation Epsilon Calculation (ε):

The corresponding formula is:

Equation 5:

ε= [$(C_\mu)^{0.75}$ (KE)$^{1.5}$]/ (0.07*L)

Empirical constant **[9]**, C_μ= 0.09

ε= 3.04633E-08

With L as characteristic length = 250*d=250*0.0061= 1.525m

2.2.5 Effective viscosity calculation (ν_t): [9]

It is calculated by software itself **[9]** by the following formula:

$\nu_t=C_\mu$ KE$^2/\varepsilon$

ν_t = 0.000158155

Effective viscosity, $\nu_e=\nu_t+\nu$ =0.000173155 m^2/s

2.2.6. Mixing length Calculation

It is given by: **[9]**

l= C_{μ}*D =0.07*300*0.0061

l= 0.1281m

In the following table, the parameters are summarised:

Table 1: Summarising Calculations

Velocity Calculation		
Reynolds no.	Re	11000
orifice diameter (m)	d	0.0061
dynamic viscosity(Pa-s)	μ	0.000018
density(kg/m^3)	ρ	1.2
kinematic viscosity(m^2/s)	ν	0.000015
velocity (inlet) (m/s)	U	27.04918033
Wall Distance calculation		
yplus (constant)	y^+	100
kinematic viscosity(m^2/s)	ν	0.000015
Skin Friction Coefficient	C_f	0.007616342
distance of first cell away from the wall.(m)	y_p	0.000899
The cell width along y-axis.(m)	y_a	0.001797252
Turbulent Intensity (by PL)		
Turbulent Intensity	I	0.0001
Turbulent Kinetic Energy (KE)		
Turbulent Kinetic Energy (m^2/s^2)	KE	4.87772E-06
Dissipation Epsilon Calculation		
Empirical Constant	C_{μ}	0.09
Turbulent Kinetic Energy (m^2/s^2)	KE	4.87772E-06

Characteristic length (m)	L	1.525
Epsilon (m^2/s^3)	ε	1.65821E-08
Effective viscosity calculation (ν_e)		
Turbulent viscosity (m^2/s)	ν_t	0.000158155
Effective viscosity (m^2/s)	ν_e	0.000173155
Mixing length Calculation(m)	l	0.1281

3. Equations:

3.1 Governing Equations:

For steady state and incompressible flow the Reynolds averaged continuity equations and the momentum equation are the governing equations **[8].**

Equation 6:

$\nabla \cdot \mathbf{U}=0$

Equation 7:

$\nabla \cdot (\mathbf{UU}) + \nabla \cdot \tau = -\nabla p$

p is kinematic pressure

U is velocity vector

With viscous Stress; $\tau = \nu_e \nabla \mathbf{U}$

3.2 Model Equations:

The k-ϵ model equations are **[9]**:

Turbulent kinetic energy equation:

Equation 8

$(D/Dt)(\rho\ KE)=\nabla \cdot (\rho\ D_k \nabla\ KE)+P-\rho\epsilon$

KE=Turbulent kinetic energy [m^2s^{-2}]

D_k=Effective diffusivity for KE [-]

P=Turbulent kinetic energy production rate [m^2s^{-3}]

ϵ=Turbulent kinetic energy dissipation rate [m^2s^{-3}]

The turbulent kinetic energy dissipation rate equation

Equation 9

$(D/Dt)(\rho\epsilon)=\nabla.(\rho D\epsilon\nabla\epsilon) + (C_1\epsilon/KE)(P+0.6667*C_3\, KE\, \nabla.u) - C_2\rho\epsilon^2/KE$

$D\epsilon$ = Effective diffusivity for ϵ [-]

C_1 = Model coefficient [-]

C_2 = Model coefficient [-]

4. Simulation Procedure

As this case consists of three-dimensional turbulent simulation of an axisymmetric jet, before proceeding for the simulation one must have all the utility files ready. Here we are showing the steps in detail.

4.1 Geometry and Mesh

For the geometry and Mesh purpose the OpenFoam utility blockMesh is used, but the geometry is **'axisymmetric',** meaning, the **wedge** type geometry must be used, to satisfy the symmetry about the axis. Without wedge in normal cases the symmetry is missing from the simulation and will be shown in the later part of the report. **(In results and discussion section)**

The meshing is fine near the axis and get sparse away from the axis, the wedge geometry is formed by 2 blocks:

1. The tiny wedge (as d is too small) of the orifice inlet.
2. The wall build-up above it.

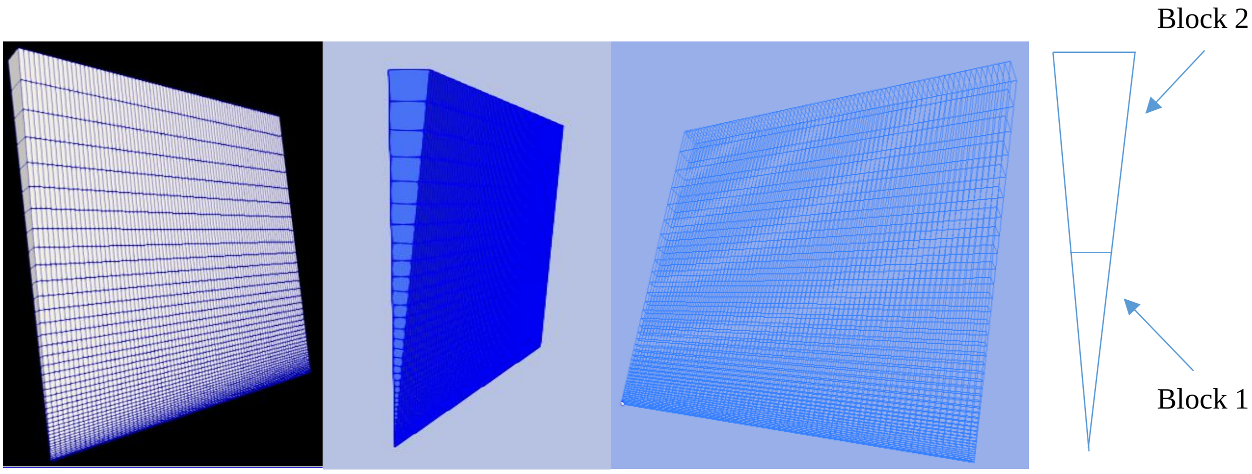


Figure 3 : Computational Geometry and Mesh of the Wedge (For Axisymmetry).

4.2 Initial and Boundary Conditions

The convergence condition for residuals is $10^{(-5)}$ .The following table summarizes the conditions:

Pressure:

Inlet	zeroGradient
Outlet	0
Wall	zeroGradient
Wedge1	wedge
Wedge2	wedge

Velocity:

Inlet	(0 0 26.55737705)
Outlet	zeroGradient
Wall	noSlip
Wedge1	wedge
Wedge2	wedge

4.3 Solver

The solver used was simpleFoam employing k-ε model, under incompressible turbulent conditions. The OpenFoam toolbox is shown here:

```
case
    |-------0
            |-------epsilon
            |-------k
            |-------nut
            |-------p
            |-------U
    |-------constant
            |-------ployMesh (Formed after running blockMesh)
            |-------transportProperties
            |-------turbulenceProperties
    |-------system
            |-------blockMeshDict
            |-------controlDict
            |-------fvSchemes
            |-------fvSolution
```

4.4 Post-Processing:

ParaFoam utility (Xming- Xserver) is used to post-process the simulation results, from OpenFoam V7, which is run using WSL (Windows Subsystem Linux Ubuntu 20.04 terminal).

5. Results and Discussions

The simulation results are as post-processed using paraview as shown below:

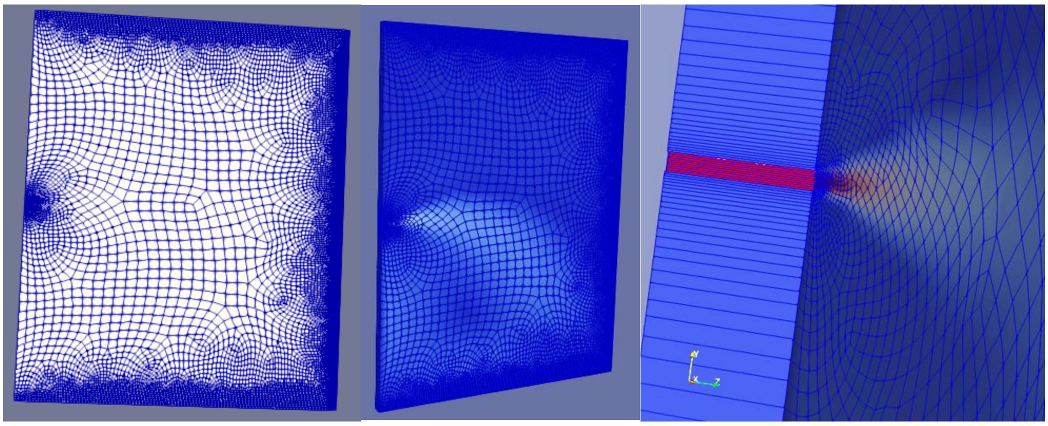


Figure 4: Steady state of jet 2D Geometry (Red shows max velocity, hence the jet inlet point)

- Initial simulations, were done on 2D, the mesh (from Ansys was imported to openFoam) was made fine near the edges of the plate and a bit coarse near the centre, but, the jet reached steady state pointing downwards which violates the 'axisymmetry'. Also, Mesh refining didn't help. Similar, thing is observed while combining the 3 block geometry (1st upper block=wall, 2nd tiny block=inlet, 3rd bottom block=wall). Here, the need of wedge geometry was felt.

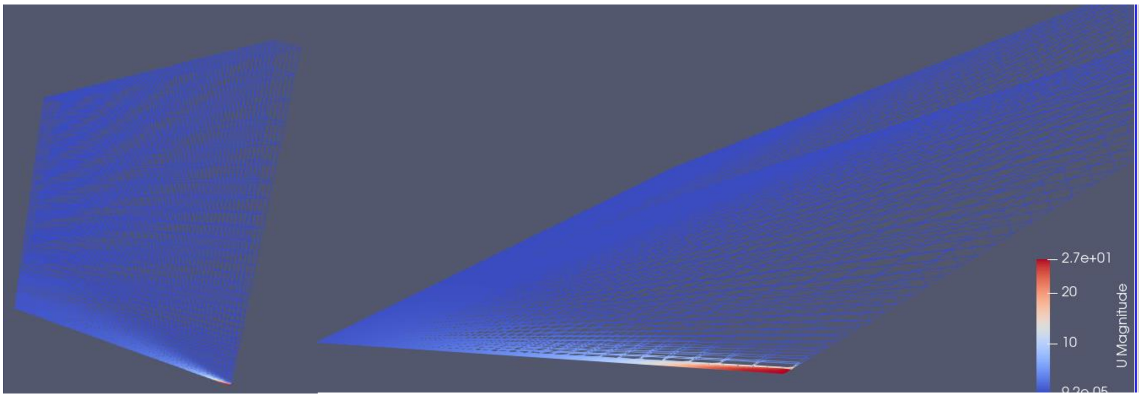


Figure 5: Steady state of jet and the bottom of the wedge (Red shows max velocity, hence the jet inlet point)

- In the figure 5, the wedge geometry is shown, at the bottom-most right of which the high-velocity jet flow is visible.
- The grid structure of the wedge is then revolved around the axis of the flow to create a half-cylinder geometry as shown in the figure 6.

- The centreline of the cylinder possesses the maximum jet velocity at any cross-section, which is verified by literature.
- The axisymmetric figure is the result of the wedge geometry, and hence they are used essentially for these applications, further the use of wedge geometry reduces the computational load on the system. It is observed by the fact, that the simulation is achieving the steady state within few time steps.

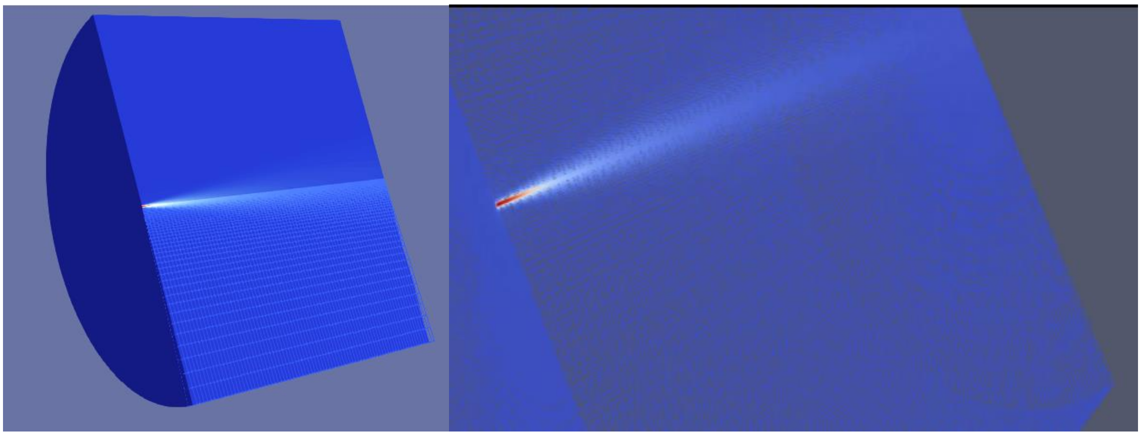


Figure 6: Steady state of jet and the centre of the half-cylindrical Section (Red shows max velocity, hence the jet inlet point)

- In the above figure the magnitude variation of the velocity vector overlaps with that of the axial velocity along the radial direction at any particular 'x' distance from the origin. Same curves are obtained at x=25, 50, 75, 100, 125, 150, 175, 200, 225.

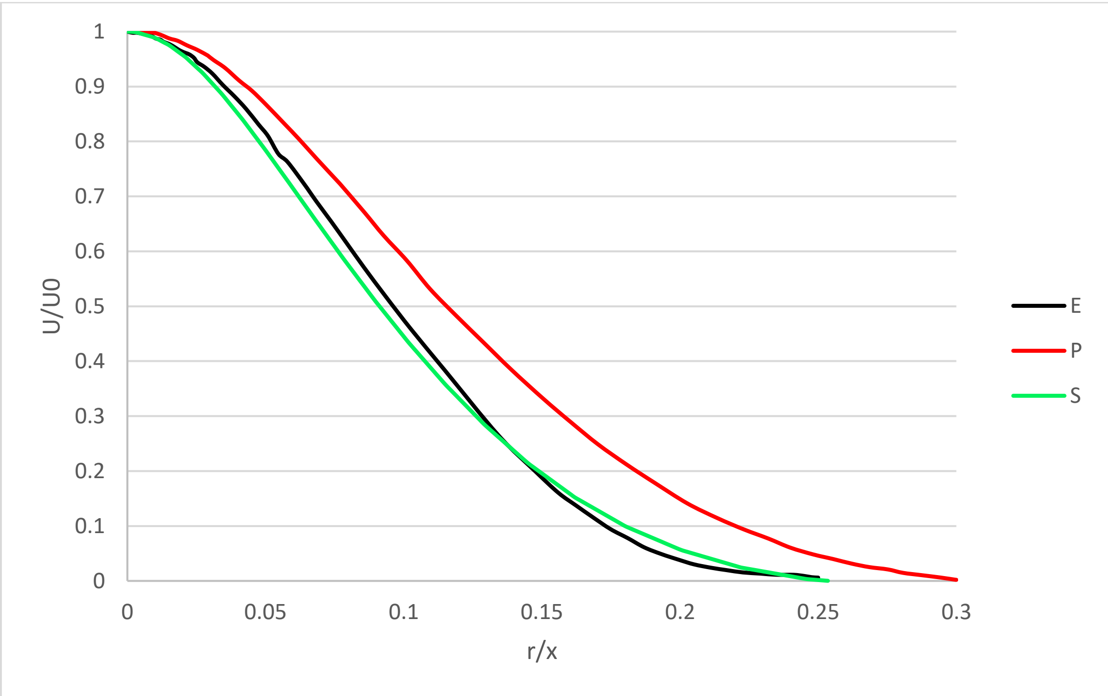
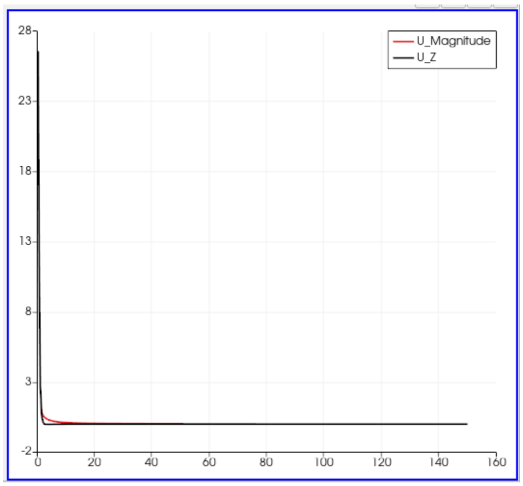
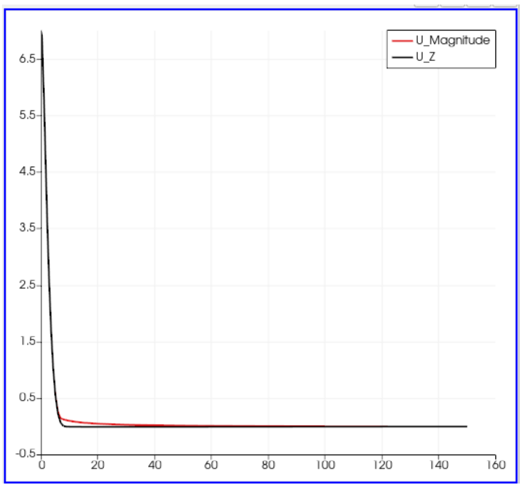


Figure 7: Side by side comparison of (a) Our simulation results (S), (b) experimental results by PL **[7]** (E) and (c) Simulation results of Kannan (P)**[8]** .Same curves are obtained at x=25, 50, 75, 100, 125, 150, 175, 200, 225. **Axial velocity distribution along radius.**

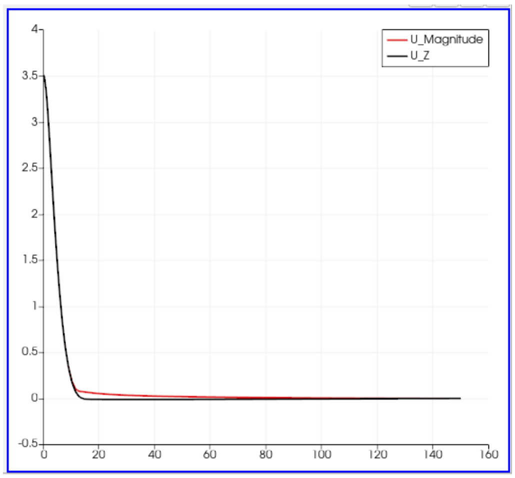
- Figure 7 shows the comparison, dimensionless velocity vs the dimensionless radial distance curves are plotted the in which it is easily evident that the current simulation results are in great agreement with the experimental results, even better than the simulation results in reference **[8].**



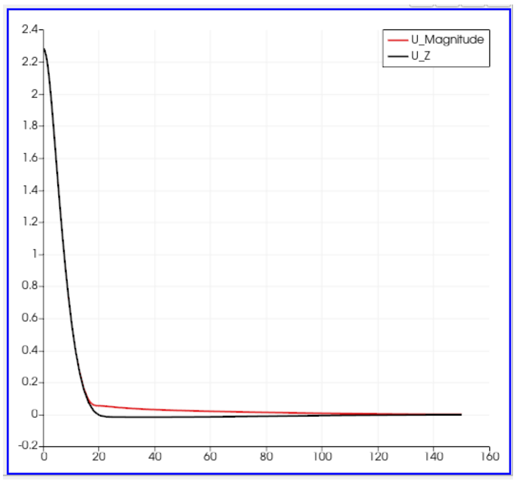
At x=0



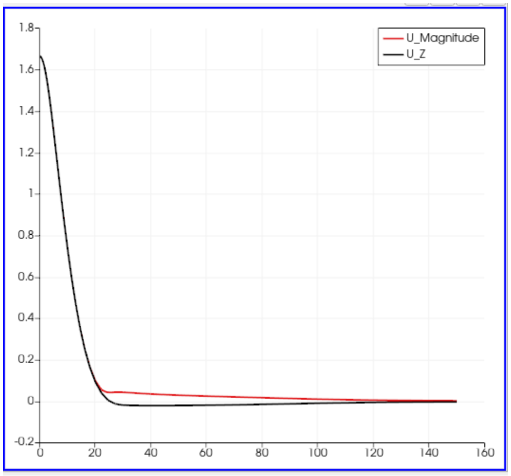
At x=25



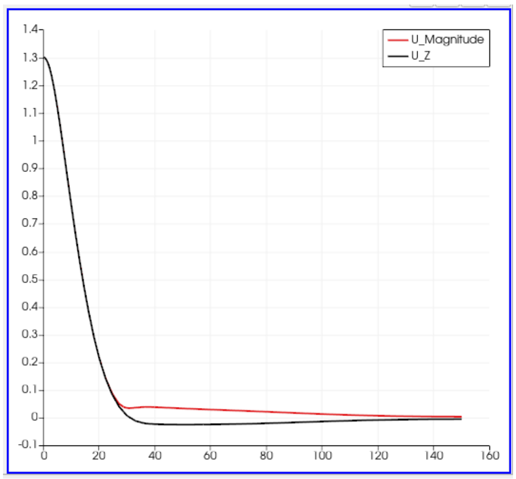
At x=50



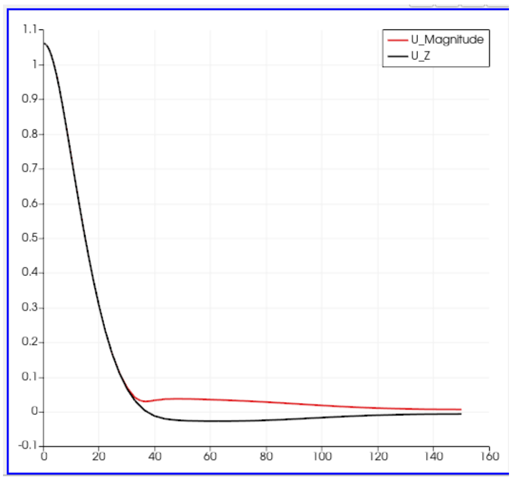
At x=75



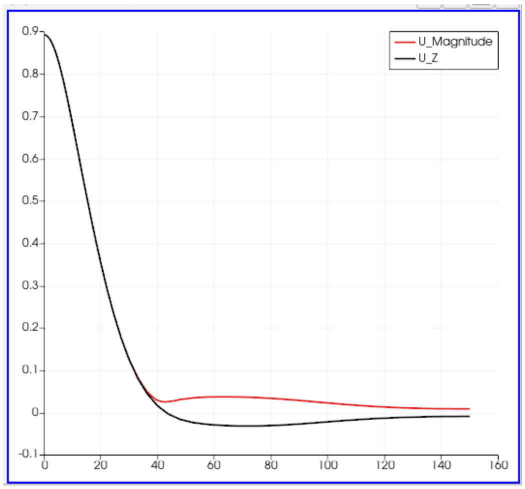
At x=100



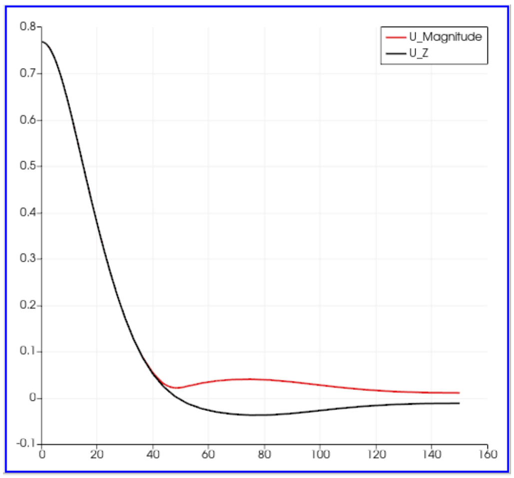
At x=125



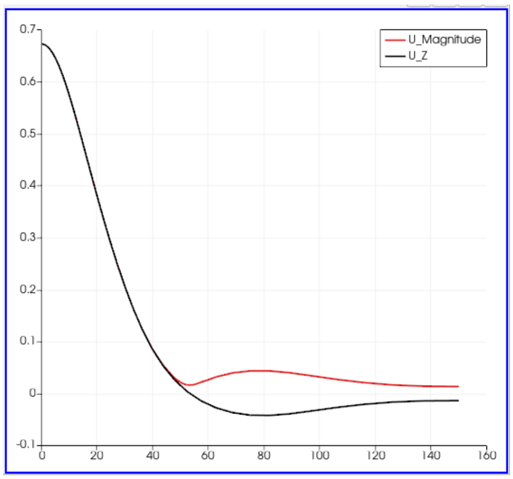
At x=150



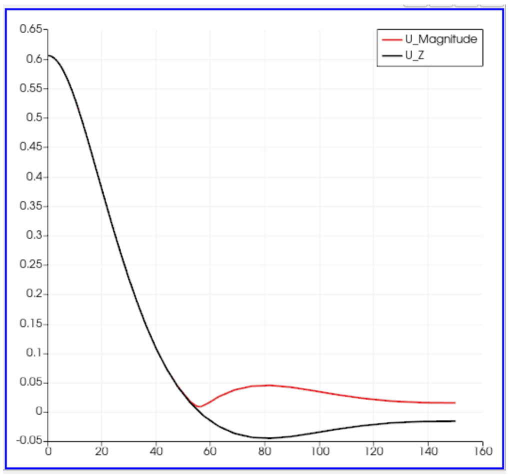
At x=175



At x=200



At x=225



At x=250

Figure 8: Series of the axial velocity profiles (black) and Velocity Vector magnitudes (red) at x=0, 25, 50, 75, 100, 125, 150, 175, 200, 225, 250 against dimensionless radial distance (r/x).

- Figure 8, compares the deviation between axial velocity components of velocity vector to the velocity vector magnitude itself. It is to be noted that the difference in velocities surges with the value of x. The change in the velocities starts happening at the higher radiuses with build-up of x. The velocities are more curved with x rising.

6. Conclusions:

- The simulated axisymmetric jet s made possible by the wedge geometry.
- The simulated orifice jet nozzle results are twinning with the experimental findings.
- Radial component of velocity is getting stronger away from the orifice.
- k-ε model is capable of speculating near-to-real scenarios.
- OpenFoam, an open-source software is robust in simulations. Deep learning and understanding of the same will surely help one to strengthen the basic CFD concepts.

7. References:

[1] H. Awbi, “Ventilation of Buildings”, (2003) 2nd ed. London: Spon Press, ISBN 0-415-27056-1.

[2] Theory of Jets [2017-10-05]. Available: http://shodhganga.inflibnet.ac.in/bitstream/10603/6989/8/08_chapter%202.pdf

[3] Miroslava Kmecova *et al.*, Circular Free Jets: CFD Simulations with Various Turbulence Models and Their Comparison with Theoretical Solutions, IOP Conference Series: Materials Science and Engineering (2019) **471.**

[4] F. Gori et al., Flow evolution of a turbulent submerged two-dimensional rectangular free jet of air. Average Particle Image Velocimetry (PIV) visualizations and measurements, International Journal of Heat and Fluid Flow, vol. 44, (2013) pp. 764-775, ISSN 0142-727X.

[5] Z. Yue, "Air jet in ventilation applications", Stockholm, ISSN 0284-141X, (2001).

[6] L. Zawadzki, Determination of the Air Velocity in the Free Stream Flowing out of a Cylindrical and Two-Gap Skewed Jet (Dual Slot Die), FIBRES & TEXTILES in Eastern Europe, vol. 18,(2010) pp. 39-43, ISSN 1230-3666.

[7] N.R.Panchapakesan, J. L.Lumley, Turbulence measurements in axisymmetric jets of air and helium. Part 1. Air jet. J Fluid Mech, 246 (1993) 197-223.

[8] Kannan B T, Computation of an Axisymmetric Jet using OpenFOAM, Procedia Engineering 127 (2015) 1292 – 1299.

[9] https://www.computationalfluiddynamics.com.au/turbulence-part-3-selection-of-wall-functions-and-y-to-best-capture-the-turbulent-boundary-layer/