FOSSEE, IIT Bombay
OpenFOAM Case Study Project
May 20, 2020

Isothermal incompressible fluid flow through a T-Junction

Submitted by

Vipinkumar c p

M.Tech ,Thermal and Fluids Engineering

Department of Mechanical Engineering

Lal Bahadur Shastri college of Engineering, Kasaragod, Kerala, India

13/3/2020

ABSTRACT

This case study demonstrates the simulation of isothermal incompressible water flow through a T-junction. In this study the T-junction with 38.1 mm NB is used. The geometric model is created using ANSYS Design Modular and the Meshing is done using ANSYS Meshing Academic R3. The simulations are performed using OpenFOAM-v6.The velocity profiles and pressure distributions are simulated for this case.

CONTENTS

1. Introduction

Pipe networks are widely employed in many "sector" to transport fluids and gases. These pipe networks consist of many components like valves, orifices, turbines, bends, etc. The interaction of these components with fluid flow results in pressure drop due to friction, which inventually cause in reduction in the flow rate, instability, and other undesirble effects. Hence, studying the nature of these interaction are essential in designing the pipe networks.

T-Junction is a commonly employed formation in the pipe networks. It is formed by the interjection of three pipes at right angles. The flow behaviour in T-Junction changes depending on whether it is an inflow or an outflow.

This study numerically investigates the effect of T-Junction on the laminar fluid flow in three different configurations shown in figure 1,2,3.

Case 1

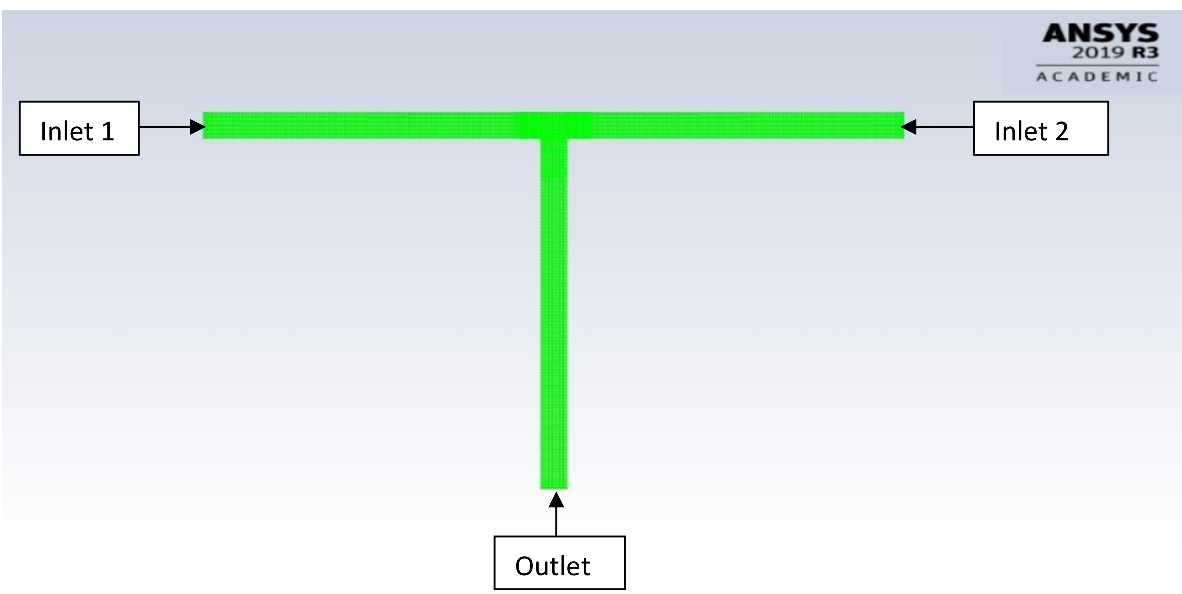


Figure 1: T-junction with two inlets and one outlet

Case 2

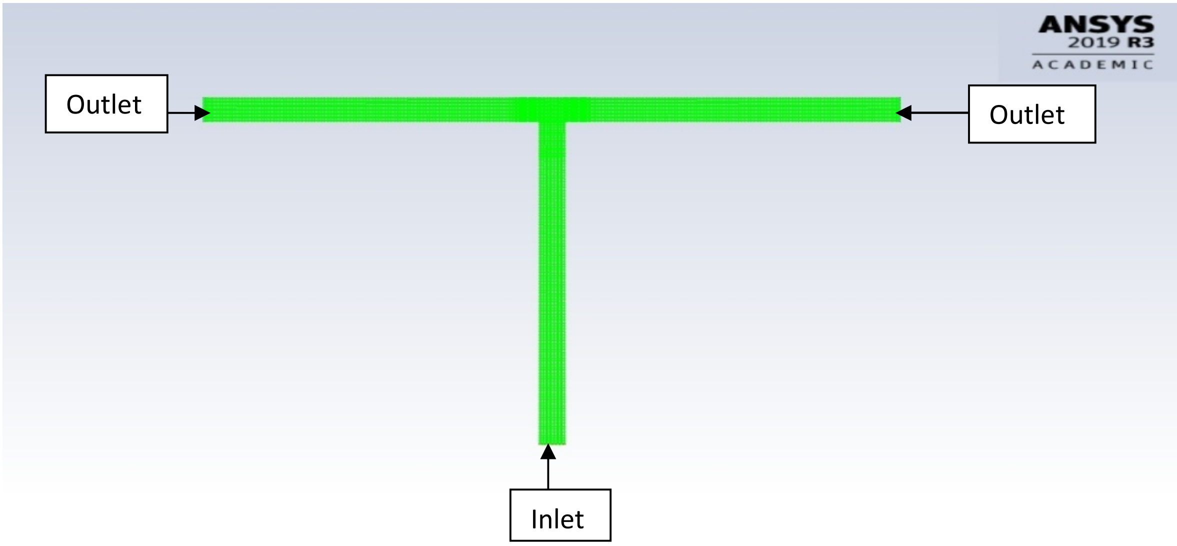


Figure 2: T-junction with one inlet and two outlets

Case 3

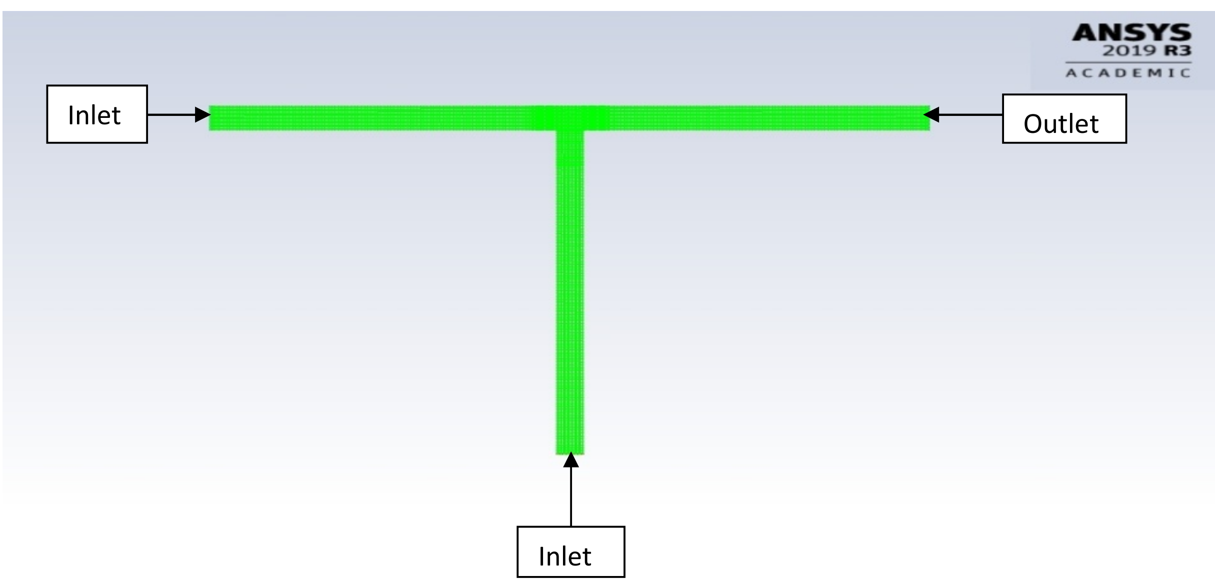


Figure 3: T-junction with two inlets and one outlet

2. Solution Methodology

2.1 Working Environment

The entire task of problem solving in Computational Fluid Dynamics can be divided into three stages, Pre-processing, simulation and Post-processing. Preprocessing is a process that must be performed before doing the actual simulation. During this process the geometry is made and the meshing is performed. Once this has been done, the next task is to perform computation on the model that we have just created. In order to visualize and analyze the results various post processing is required. It can be concluded that all three of them are interdependent and not performing any of the process correctly may lead to an incomplete solution. This section discusses about the various tools and software used throughout the course project.

2.2 Pre-processing

2.2.1 Geometry creation & Meshing

The geometry used in this study is the 2-D Geometry of T-junction. The main objective is to find out the pressure and velocity .The geometry consist of a one verical pipe with horizontal pipe by an angle of 90°. .The modeled flow distribution system considered for this purpose is of diameter 38.1 mm (1.5 inch).The length between the main pipe and branch pipe is 500 mm.

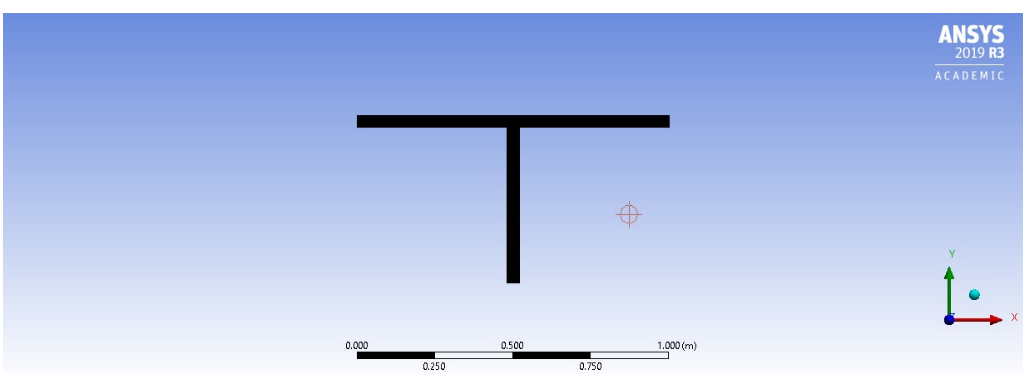


Fig 4: Meshed Geometry of T-junction

The geometry and mesh were built in Ansys academic and converted into openfoam mesh using openfoam utilities listed below

fluentMeshToFoam

fluentMeshToFoam is used to Convert a Fluent mesh (in ASCII format) to foam format including multiple region and region boundary
Synopsis : fluentMeshToFoam FLUENT_MESH , Convert a mesh file FLUENT_MESH from Fluent format to foam format. fluentMeshToFoam can only handle the following 2D cell types: tri, quad and 3D cell types: tet, hex, pyramid, prism.

checkMesh

checkMesh is a utility which analyzes and evaluates mesh statistics and quality parameters. It can be run anytime by a command checkMesh > log.checkMesh .The log contains information Statistics about number of points, faces, cells, etc.

2.3 Solver

OpenFOAM is an open source commercial CFD code written in C + +. It was chosen to perform simulation of the T-junction flow analysis. It consists of various embedded libraries which are accessible for review and modifications. The library consists of numerous mathematical models and CFD tools organized in directories.

2.3.1 IcoFoam Solver Specifications

The Navier-Stokes equations are analytical equations, hence it cannot be understood by the solver. In order to make the solver understand the equation, it has to be transferred in a discretized form. This process is known as discretization. The typical discretization methods are finite difference, finite element and finite volume methods. OpenFOAM is based on the Finite Volume Method approach.

IcoFoam solver

Governing Equations

The solver uses the PISO algorithm

Continuity equation: $\nabla \cdot u = 0$

Momentum equation: $\partial \partial t(u) + \nabla \cdot (u \otimes u) - \nabla \cdot (\nu \nabla u) = -\nabla p$

Where:

u = Velocity , p = Kinematic pressure

Inputs:

- p: kinematic pressure $[\frac{m^2}{s^2}]$
- U: velocity [m/s]

Solution controls

- Schemes
- Linear equation solvers
- controlDict

2.3.2 PISO algorithm

PISO algorithm (**Pressure-Implicit with Splitting of Operators**) was proposed by Issa in 1986 without iterations and with large time steps and a lesser computing effort. It is an extension of the SIMPLE algorithm used in computational fluid dynamics to solve the Navier-Stokes equations. PISO is a pressure-velocity calculation procedure for the Navier-Stokes equations developed originally for non-iterative computation of unsteady compressible flow, but it has been adapted successfully to steady-state problems.

PISO involves one predictor step and two corrector steps and is designed to satisfy mass conservation using predictor-corrector steps.

The algorithm can be summed up as follows:

1. Set the boundary conditions.
2. Solve the discretized momentum equation to compute an intermediate velocity field.
3. Compute the mass fluxes at the cells faces.
4. Solve the pressure equation.
5. Correct the mass fluxes at the cell faces.
6. Correct the velocities on the basis of the new pressure field.
7. Update the boundary conditions.
8. Repeat from 3 for the prescribed number of times.
9. Increase the time step and repeat from 1.

Advantages and disadvantages

1. Generally gives more stable results and takes less CPU time but not suitable for all processes.
2. Suitable numerical schemes for solving the pressure-velocity linked equation.
3. For laminar backward facing step PISO is faster than SIMPLE but it slower for concerning flow through heated fin.
4. If momentum and scalar equation have weak or no coupling then PISO is better than SIMPLE

2.4 Post-processors

ParaView is an open-source, multi-platform data analysis and visualization application. ParaView users can quickly build visualizations to analyze their data using qualitative and quantitative techniques. The data exploration can be done interactively in 3D or programmatically using ParaView's batch processing capabilities. The pressure and velocity distribution on T-junction was visualized using Parafoam

3. Boundary Conditions

3.1 Transport properties

Kinematic viscosity of water at 30°C is $0.801*10^{-6}\ \frac{m^2}{s}$ and fluid density is taken as 996.26 kg/m^3

3.2 Inlet conditions

Inlet condition for flow through T-junction with different flow configurations described below as three different cases.

Case 1

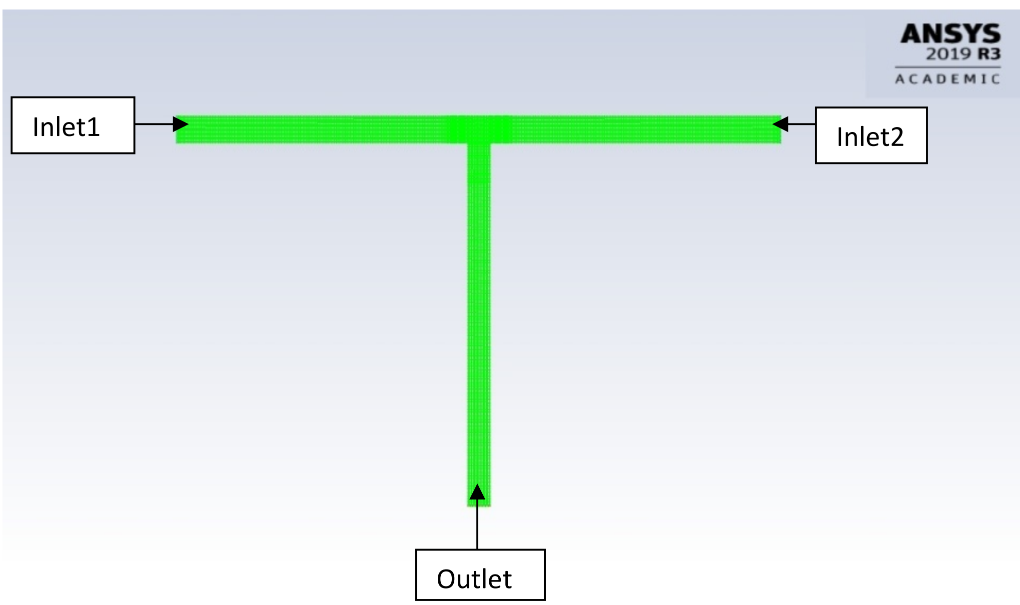


Figure 5 : T-junction with two inlets and one outlet

Velocity conditions

The velocity conditions for T-junction with two inlets and one outlet described below.

Boundary	Inlet Conditions
Inlet 1	0.01m/s
Inlet 2	-0.02 m/s
Outlet	zeroGradient
Wall	noSlip
frontAndBackPlanes	empty

Pressure conditions

The pressure conditions for T-junction with two inlets and one outlet described below.

Boundary	Inlet conditions
Inlet 1	zeroGradient
Inlet 2	zeroGradient
Outlet	uniform 0
wall	zeroGradient
frontAndBackPlanes	empty

Case 2

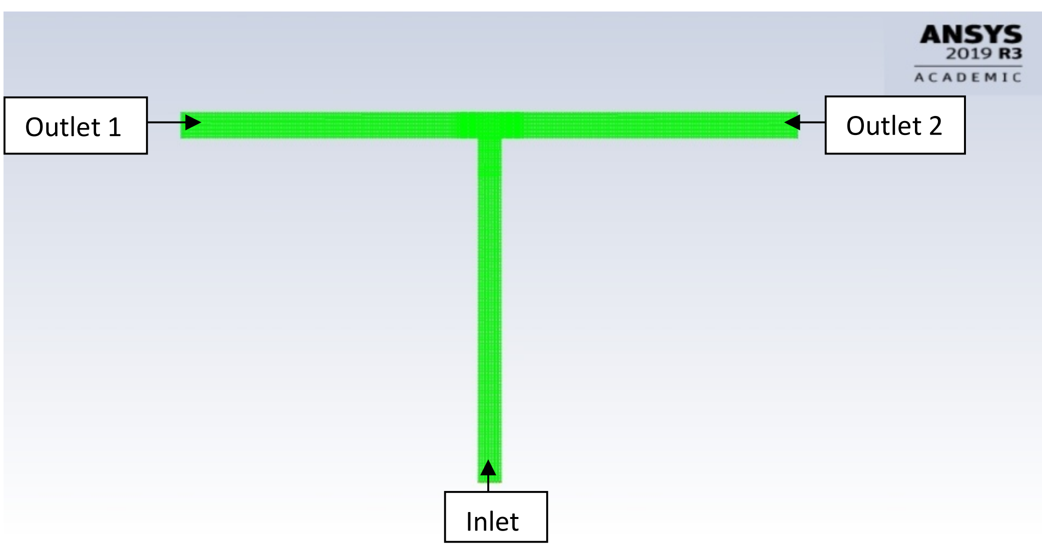


Figure 6: T-junction with one inlet and two outlets

Velocity conditions

The velocity conditions for T-junction with one inlet and two outlets described below.

Boundary	Inlet Conditions
Inlet	0.02m/s
Outlet 1	ZeroGradient
Outlet 2	ZeroGradient
Wall	NoSlip
frontAndBackPlanes	Empty

Pressure conditions

The pressure conditions for T-junction with one inlet and two outlets described below.

Boundary	Inlet conditions
Inlet 1	ZeroGradient
Outlet 1	uniform 0
Outlet 2	uniform 0
wall	ZeroGradient
frontAndBackPlanes	empty

Case 3

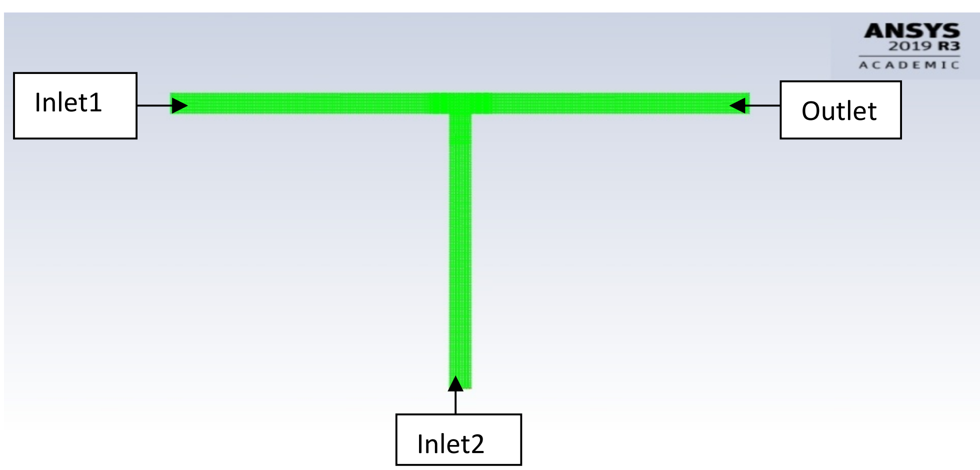


Figure 7: T-junction with two inlets and one outlet

Velocity conditions

The velocity conditions for T-junction with two inlets and one outlet described below.

Boundary	Inlet Conditions
Inlet 1	0.02m/s
Inlet 2	0.01 m/s
Outlet	zeroGradient
wall	noSlip
frontAndBackPlanes	empty

Pressure conditions

The pressure conditions for T-junction with two inlets and one outlet described below.

Boundary	Inlet conditions
Inlet 1	zeroGradient
Inlet 2	zeroGradient
Outlet 2	uniform 0
wall	zeroGradient
frontAndBackPlanes	empty

3.3 Time and data input/output control

The OpenFOAM solvers begin all runs by setting up a database. The database controls Input/Output and, since output of data is usually requested at intervals of time during the run, time is an inextricable part of the database. The controlDict dictionary sets input parameters essential for the creation of the database. The keyword entries in controlDict are listed in the following sections.

application	icoFoam
startFrom	startTime
startTime	0
stopAt	endTime
endTime	180
deltaT	0.05
writeControl	timeStep
writeInterval	20

RESULTS

The velocity and pressure gradient plots for flow through a T-junction with 3 different cases are shown in figures below the fluid is taken as water with density 996.25 $\frac{kg}{m^3}$ and kinematic viscosity is taken as 0.801* $10^{-6}\frac{m^2}{s}$ and The Reynolds number of Flow in 3 cases ranges between 450-2000 (laminar flow) . The geometry of a simplified flow model was employed in this study to represent the pipe flow situation as the flow in a straight pipe i.e. main pipe with one vertical pipe i.e. branch pipe which is connected with the main pipe by an angle of 90°.The modeled flow distribution system considered for this purpose is of diameter 38.1 mm (1.5 inch).The length between the main pipe and branch pipe considered 500 mm. The inlet and outlet varied in 3 different cases. The figure shows the various Contours of Pressure and velocity of pipe having t-junction with different configurations of inlet and outlet of three different cases.

Case 1

The below plots describes pressure and velocity gradients for the T-junction with two inlets and one outlet. In this case the inlet velocity for two different inlets given as 0.01 m/s and 0.02 m/s and a constant pressure outlet. In The color contours shows how the pressure gets decreased at the connecting branch of pipe due to which pressure loss occurred in pipe and the velocity gets increased at the outlet of the pipe .The maximum flow velocity which is attained at the center region of the outlet is 0.052m/s which is likely greater than the combination velocity of both inlets and the velocity of out near the wall region which closes to value zero. Towards the junction the location where the velocity get the value zero due to the mixing of flow.As the flow get mixed and it flow through outlet with higher velocity than velocity of two inlets.

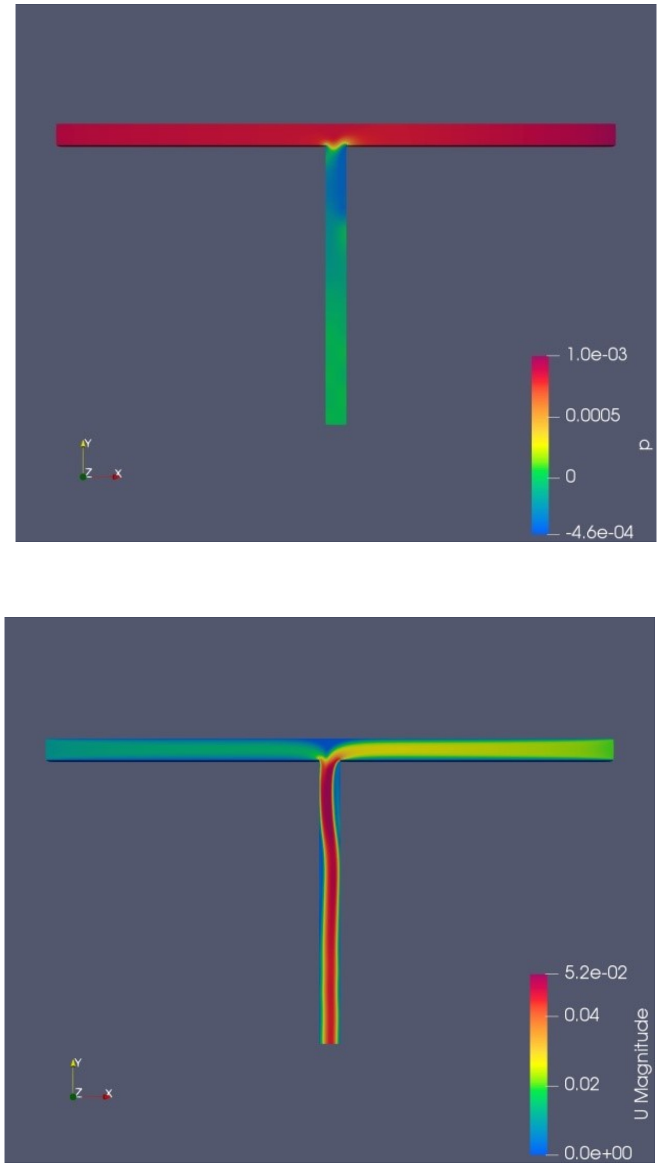
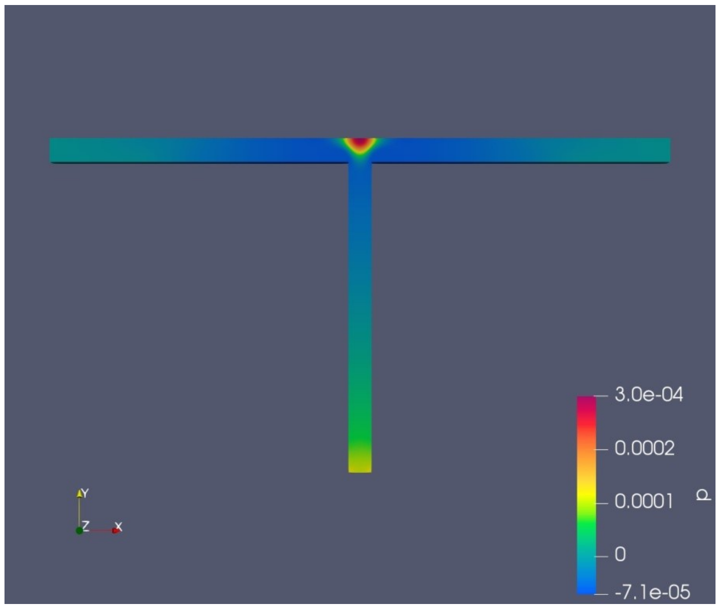


Figure 8: pressure and velocity distribution plots for T-junction with two inlets and one outlet

Case 2

The below plots describes pressure and velocity gradients for the T-junction with one inlets and two outlet. In this case the inlet velocity given as 0.02 m/s and a constant pressure outlets . In The color contours shows how the pressure gets increased at the junction of Diverging connecting branch of pipe due to which pressure occurred in pipe and the velocity gets decreased to zero at that region. Towards the junction the location where the velocity get the value zero due to the diverging of the flow. The maximum flow velocity which is attained at the center region of the outlets ranges between 0.02 m/s to 0.01 m/s and the velocity of out near the wall region which closes to value zero.



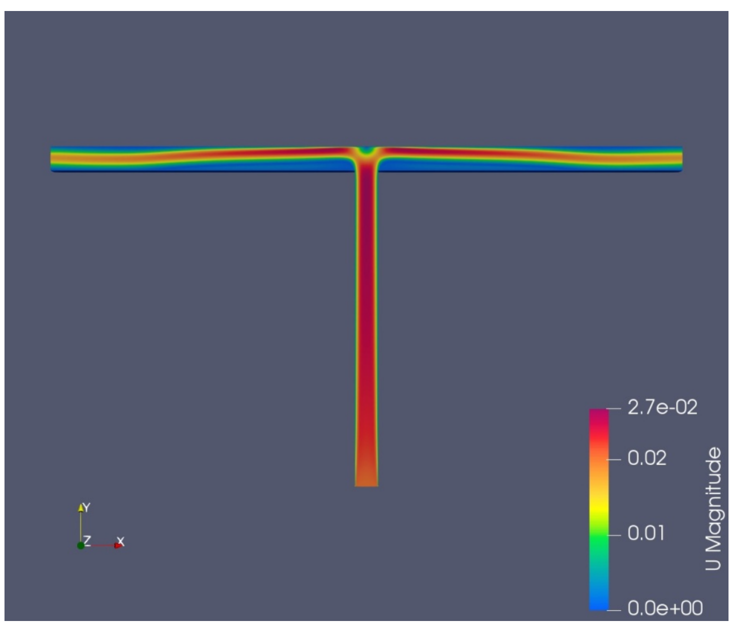
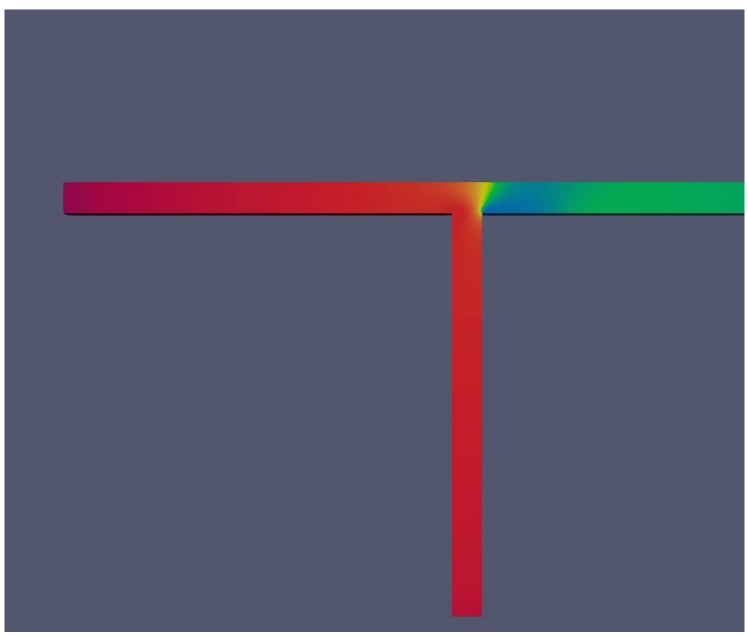


Figure 9: pressure and velocity distribution plots for T-junction with one inlet and two outlets.

Case 3

The below plots describes pressure and velocity gradients for the T-junction with two inlets and one outlet. In this case the inlet velocity for two different inlets given as 0.01 m/s and 0.02 m/s and a constant pressure outlet. In The color contours shows how the pressure gets decreased at the connecting branch of pipe due to which pressure loss occurred in pipe and the velocity gets increased at the outlet of the pipe .The maximum flow velocity which is attained at the center region of the outlet is 0.046 m/s which is likely greater than the combination velocity of both inlets and the velocity of out near the wall region which closes to value zero. Towards the junction the location where the velocity get the value zero due to the mixing of flow .As the flow get mixed and it flow through outlet with higher velocity than the velocity of two inlets. The inertia at the curve causes bend in the flow velocity at junction where mixing occurs is slightly more and flow becomes uniform gradually towards outlet.



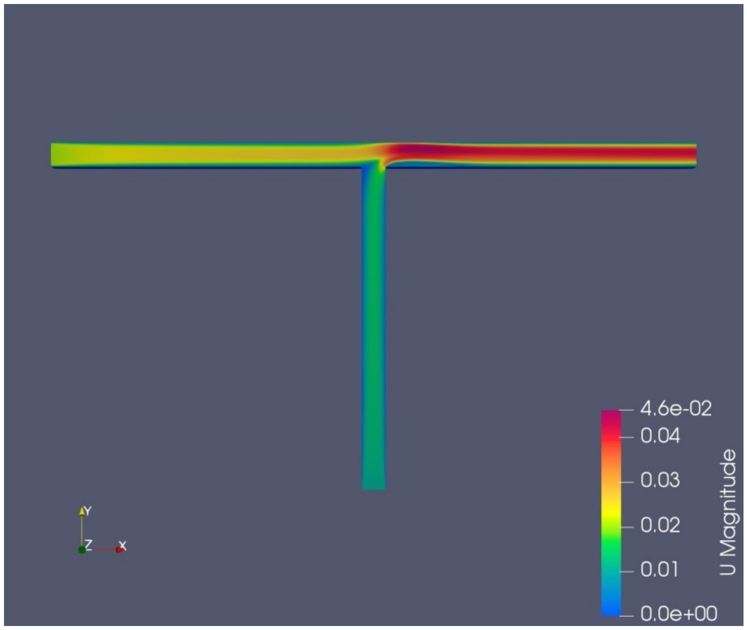


Figure 10: pressure and velocity distribution plots for T-junction with two inlets and one outlet

References

[1]. "Numerical Prediction Of Pressure Loss Of Fluid In A T-Junction," International Journal Of Energy And Environment *Volume 4, Issue 2, 2013 Pp.253-264* Mohammed Abdulwahhab, Niranjan Kumar Injeti, Sadoun Fahad Dakhil

[2]. “Fluid Flow In T-Junction Of Pipes” Master’s Thesis, Paritosh R Vasava, Lappeenranta University Of Technology

[3] “Cfd Analysis Of Flow Through T-Junction Of Pipe “ International Research Journal Of Engineering And Technology (Irjet). Volume:4 Issue:2 | Feb-2017 P-Issn: 2395-0072 .Mr.G.B.Nimadge1, Mr.S.V.Chopade2

[4] https://www.openfoam.com/documentation/guides/latest/doc/guide-applications-solvers-incompressible-icoFoam.html

[5] https://cfd.fossee.in/case-study-project/completed-case-studies