TRIBHUVAN UNIVERSITY
INSTITUTE OF ENGINEERING
PULCHOWK CAMPUS

Thesis No: M-158-MSESP-2019-2023

COMPUTATIONAL ANALYSIS OF BIFURCATION OF RAGHUGANGA HYDROPOWER PROJECT

by
Rajendra Dhakal

A THESIS
SUBMITTED TO THE DEPARTMENT OF MECHANICAL AND AEROSPACE
ENGINEERING IN PARTIAL FULFILMENT OF THE REQUIREMENTS
FOR THE DEGREE OF MASTER OF SCIENCE IN
ENERGY SYSTEM PLANNING AND MANAGEMENT

DEPARTMENT OF MECHANICAL ENGINEERING
LALITPUR, NEPAL

APRIL, 2023

Copyright

The author has agreed that the library, Department of Mechanial and Aerospace Engineering, Pulchowk Campus, Institute of Engineering may make this thesis freely available for inspection. Moreover, the author has agreed that permission for extensive copying of this thesis for scholarly purpose may be granted by the professor(s) who supervised the work recorded herein or, in their absence, by the Head of the Department wherein the thesis was done. It is understood that the recognition will be given to the author of this thesis and to the Department of Mechanical and Aerospace Engineering, Pulchowk Campus, Institute of Engineering in any use of the material of this thesis. Copying or publication or the other use of this thesis for financial gain without approval of the Department of Mechanical and Aerospace Engineering, Pulchowk Campus, Institute of Engineering and author's written permission is prohibited.

Request for permission to copy or to make any other use of the material in this thesis in whole or in part should be addressed to:

Head

Department of Mechanical and Aerospace Engineering

Pulchowk Campus, Institute of Engineering

Lalitpur, Kathmandu

Nepal

TRIBHUVAN UNIVERSITY
INSTITUTE OF ENGINEERING
PULCHOWK CAMPUS
DEPARTMENT OF MECHANICAL AND AEROSPACE ENGINEERING

The undersigned certify that they have read, and recommended to the Institute of Engineering for acceptance, a thesis entitled **"Computational Analysis of Bifurcation of Raghuganga Hydropower Project"** submitted by Rajendra Dhakal in partial fulfilment of the requirements for the degree of Master of Science in Energy System Planning and Management.

Supervisor: Dr. Rajendra Shrestha
Professor
Department of Mechanical and Aerospace Engineering

External Examiner: Er. Sajan Shrestha
Deputy Manager
Nepal Electricity Authority

Committee Chairperson, Dr. Surya Prasad Adhikari
Head of Department
Department of Mechanical and Aerospace Engineering

Date: 11th April, 2023

Abstract

Computational analysis is the modern design optimization tool used widely for determining the effect of various loading parameters where operating conditions and geometry are not regular and complex for manual solutions to execute. This study has great significance in the design and optimization of penstock branches as the ancient technology used for the design has least possibility that it has both minimum head loss and good structural strength at the same time. Bifurcation of Raghuganga Hydropower Project of 40 MW installed capacity is chosen for this study in which, head loss, velocity and pressure distribution and stress distribution around the branching regions have been observed by varying the angle of cone of bifurcation. Upon varying the cone angles gradually starting from 3 degrees up to 15 degrees, the values of head loss have been reduced from 0.972 m to 0.086 m till 13-degree angle of cone and upon further increasing the angle to 15 degrees, the head loss increased sharply to 2.201 m. The structural analysis on the optimized cone angle profile, pipe thickness was varied from 25 mm till the values of stress was in the acceptable range. Upon simulations, it was found that optimum pipe thickness is 40 mm and sickle reinforcement of 75 mm with the value of maximum stress (Von-Mises) at the branching to be 167 MPa and minimum factor of safety of 1.49 for the material chosen i.e., E 250 corresponding to I S 2062.

Acknowledgements

Foremost I would like to express my sincere gratitude and appreciations to my supervisor Prof. Dr. Rajendra Shrestha of Department of Mechanical and Aerospace Engineering, Pulchowk Campus for guiding me through out the journey, providing me with immense support and supervision for my work with his patience, knowledge and enthusiasm to motivate me in this journey. His Guidance has helped me a lot in this work without whom this thesis would not have been possible.

I am grateful to Mr. Sanjay Pathak, Deputy Manager Nepal Electricity Authority and Mechanical Head at Raghuganga Hydropower Project, for helping me to provide the necessary details and drawings of the under-construction project and giving me the insight of it. Without his help, this study would not have been possible. I would like to thank him for his guidance in this study.

I would also like to thank my friends Er. Shankar Pandey, Er. Niroj Bhattrai and Er. Bikash Babu Shrestha, Prakriti Koju, Nepal Electricity Authority for the support and discussions of the results and findings and help me finish the works on schedule.

Finally, I would like to acknowledge the love and support of my family; my parents Kumar Prasad and Yashoda Devi, my Wife Kushum, My Brother Rajib. They all motivated as well as encouraged me for the journey and I kept going.

Table of Contents

List of Figures

List of Tables

List of Abbreviations

CFD	Computational Fluid Dynamics
FEA	Finite Element Analysis
HDPE	High Density Poly Ethelene
IS	Indian Standard
MW	Megawatt
GWh	Giga Watt Hour
K E	Kinetic Energy

CHAPTER-1: INTRODUCTION

1.1 Background

Hydropower stations have two or more generating units depending upon the availability of water and head. Penstock pipes are divided into two or more branches depending upon the flow requirements for each unit. Penstock pipe is the water transport system used to transfer water from higher head point to turbine. Penstock is pressurized conduit usually made up of steel plates of varying thickness for high head applications and HDPE pipes in low head applications.

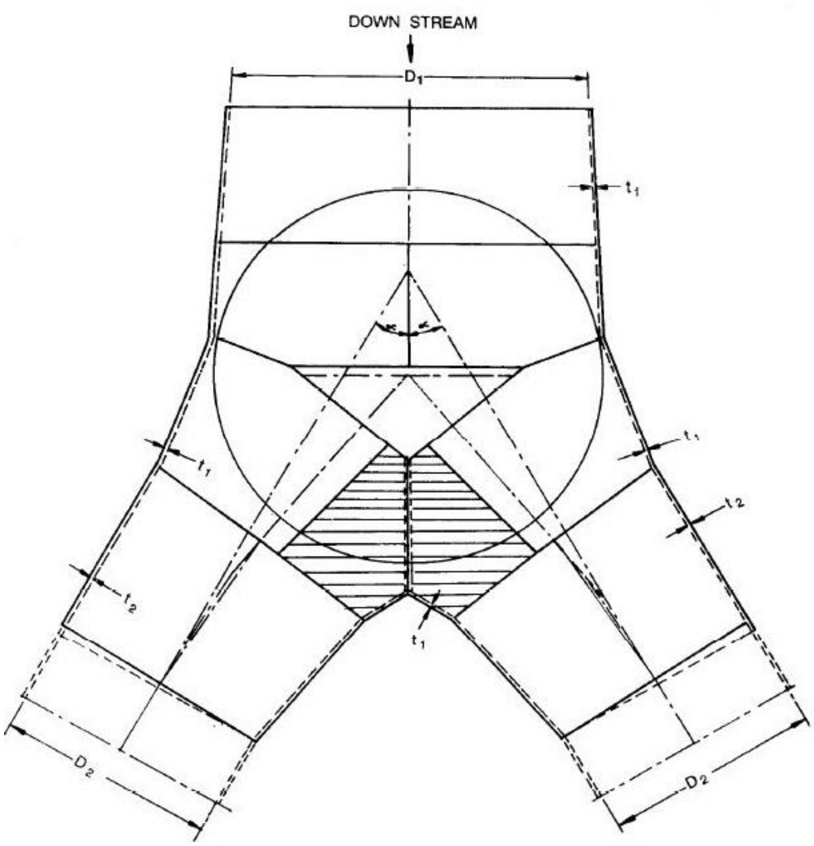

Figure 1: Typical Structure of Bifurcation *(Standards, 2001)*

Using individual penstock pipes for each unit in a power plant is not cost-effective and in most of the cases penstock convey the water near the power house and is then branched into different units. A penstock bifurcation is a point in a hydroelectric power system where a single water flow system splits into two or more separate channels to feed water to multiple turbines. This allows for the distribution of water flow and pressure to multiple turbines. This division may be symmetric or unsymmetric depending upon the requirements of the powerplant. In symmetric bifurcation, the flow is divided equally and in unsymmetric bifurcation, division of flow is unequal. Since bifurcation is installed near the power house and is located at the point of maximum pressure, it is a critical section and thus has to be designed with special care and considerations. The major factors to be considered are angle of bifurcation, losses in bifurcation and its structural stability as it affects the power plant in long run-in terms of vibration, power generations and repair maintenances.

1.2 Problem Statement

The design and analysis of a penstock bifurcation system for a hydroelectric power plant presents a complex engineering challenge. The goal is to design and analyze a penstock bifurcation system that can effectively distribute water flow and pressure to multiple turbines, while also ensuring structural integrity and minimizing energy loss. Bifurcation must also be designed to minimize vibrations and cavitations that might arise due to the change in pipe dimensions of the flow. If the wye branches are not designed carefully, the losses contribute to the reduction in power generation of the power plant and thus affect the revenue of the company.

The choice of bifurcation profiles can be made through either experimental examination using scaled-down model tests conducted in laboratories or by numerical simulations of fluid dynamics. Since the results from past experimental study don’t give empirical relations for all the hydropower stations as they are site specific, the analysis for minimum losses and best profile for the bifurcation should be done for penstock of each power plants.

In this study, the structure of bifurcation for given flow and angle of Raghuganga Hydro Power Project with installed capacity of 40 MW (2x20 MW Pelton turbine units) is modeled and studied to achieve minimum head loss in the branching by changing the taper angle of bifurcation and its structural analysis is done to obtain the economic thickness of the pipe for the wye branches analytically.

1.3 Objectives

1.3.1 Main Objective

The main objective of this study is to determine the profile for bifurcation of Raghuganga Hydropower Project with minimum head loss and determine the corresponding pipe thickness for the same analytically.

1.3.2 Specific Objectives

The general objectives of the study can be listed as follows.

- To model and perform CFD analysis of bifurcation by changing the taper angle of cone using the existing flow and other data available to obtain the profile with minimum loss.
- To perform structural analysis on the profile with minimum loss to calculate the pipe thickness at branches.

1.4 Scope

The main scope for this study is listed below.

- To model and perform CFD analysis of bifurcation by changing the taper angle of cone of bifurcated pipes by gradually increasing the angles starting from 3 degree.
- Structural analysis on the profile of bifurcation with minimum losses to obtain the optimum pipe thickness and details of reinforcements at branching.
- To recommend the new bifurcation with both structural requirements and minimum losses to be installed for the project.

CHAPTER-2: LITERATURE REVIEW

2.1 Design and Analysis of Pipe Branching

Numerical simulation using computational fluid dynamics (CFD) has been widely used now days in the analysis and optimization of penstock bifurcation systems for hydroelectric power plants. Several research studies have utilized CFD to examine the flow patterns and pressure distribution in penstock bifurcations with the aim of enhancing the system's efficiency.

The study by Sirajuddin Ahmed tested five symmetrical wye branches of conventional and spherical types for hydraulic losses under symmetrical and unsymmetrical flow conditions. The results showed a wide variation in loss factor depending on the type of wye and flow condition, and the minimum loss coefficient did not always occur under conditions of symmetrical flow. (Ahmed, 1965)Similarly, the study by Hua Wang conducted laboratory tests to determine head losses in conventional wyes and manifolds, with and without an internal tie-rod at the theoretical center of the wye. The wyes and manifolds had subtending angles of 45, 60, and 90 degrees and were symmetrical about the main pipe's longitudinal axis. The tests were conducted using a range of Reynolds numbers and the results were analyzed using the energy equation of Bernoulli for one-dimensional conditions. It was found that the coefficient of form loss is a function of the proportion of flow through the branches, the size of the tie-rod used and the subtending angle of the wye. (Wang, 1967)

The inquiry uncovered various issues that must be dealt with in the design, upgrading, and maintenance of hydropower facilities to guarantee secure operation. These issues encompass the following:

- The flow interruption rate and the consequent maximum pressure increase in the flow system.
- Increased stress concentration in geometrically irregular elements of the flow system, such as penstock bifurcations
- poor quality of welded or riveted joints.

According to the research, a penstock failure occurred due to water hammer causing an abrupt flow cut-off and resulting in an excessive pressure surge. The failure was also ascribed to the weak penstock construction, caused by inadequate reinforcement and poor welding quality at high-stress concentration areas. The incident serves as a warning that even small hydropower plants are at risk of breakdowns caused by water hammer, and it is important to check the quality of materials used and analyze various operating conditions using current computational methods and strain measurements to prevent such situations. (Adamkowski, 2001)

A study was conducted to compare different equations for the optimal design of penstocks in hydroelectric projects. It was found that these relations provide different values for the optimum penstock diameter, leading to different costs. Some of these relations only consider friction loss, but other losses also occur in practice and need to be considered. A novel approach was devised to optimize the design of penstocks by minimizing the annual project cost, while accounting for the total head loss (including friction and other losses) based on the Darcy-Weisbach formula. This new method was implemented on 21 hydroelectric projects ranging in capacity from 25 kW to 60 MW, and it led to a reduction in the annual cost of penstocks, ranging from 0.613% to 9.714% compared to previous designs. These findings validate the effectiveness of the new approach for the optimal design of penstocks in hydroelectric power projects. (Singhal M.K., 2015)

Study for optimization of the design of the penstock manifold and bifurcation in hydropower plants using modern techniques such as Computational Fluid Dynamics and Finite Element Method was done by Dipesh Thapa et. all. The primary objective of the research was to improve the theoretical understanding of the usage of these methodologies in Nepal's context. To achieve this, the optimization of the manifold arrangement in the Kulekhani-III Hydropower Project was carried out. The suggested design was simulated, examined, and improved until a satisfactory configuration was obtained. The bifurcation was subsequently fortified with thickness and reinforcements, and a three-dimensional model was created and scrutinized using Finite Element Analysis. The findings were verified using design codes, and a viable design was put forward for production and installation. (Dipesh Thapa, 2016)

The study by Ravi Koirala et all discusses the importance of optimizing the design of hydro power plants to ensure their reliability and efficiency. It notes that past design practices were based on experience and theoretical foundations, but modern technologies such as computational fluid dynamics (CFD) and finite element analysis (FEA) can now be used to improve the design process. The paper suggests that using these modern tools in conjunction with conventional methods can lead to more accurate and reliable designs for penstock bifurcation, which is a key component of hydro power plants. (Ravi Koirala, 2017)

The research focuses on the design and analysis of bifurcation in Upper Kallar Small Hydro electric project (2 Mw) using modern techniques such as Computational Fluid Dynamics (CFD) and Finite Element Method (FEM) to improve the reliability and safety of the design. The use of ASTM A 285 Grade C steel is found to be strong enough to withstand the stresses caused by the complex shape of the bifurcation. The result of the FEM analysis shows that a bifurcation with 20 mm in wall thickness and a sickle plate with 32 mm thickness meets the design requirements. Additionally, it is noted that it would be extra safe to have the bifurcation inside a reinforced concrete structure. (Project, 2017)

The main objective of this research is to quantify the head losses incurred in a system that utilizes bifurcation, trifurcation, and other arrangements to move water from surge tanks to powerhouses, supplying multiple turbines simultaneously. The study employs Computational Fluid Dynamics (CFD) to compute the coefficient of head losses and verify it against previous findings. Various mesh configurations are assessed, and the k-ω turbulence model is utilized with wall refinements to examine y+. Additionally, the SAS model is employed to investigate instability in the trifurcation. (Carlos Andres Aguirre, 2018)

The primary objective of this study is to measure the head losses in relation to the volumetric flow rate by employing Computational Fluid Dynamics (CFD) and to compare the outcomes with prior published data. Three fundamental branching designs were devised, and alterations in pressure, velocity, head loss, and mass flow were examined. The study advises further in-depth examination of one of the designs, and the conclusions are mostly relevant to the particular site. It is suggested that the branching section should be designed meticulously after analyzing the fundamental parameters carefully and considering the potential for flow irregularity within the branching structure. (Bipin Kandel, 2019)

Numerical analysis was utilized in this study to investigate the hydraulic and structural characteristics of the manifold in the Phukot Karnali Hydroelectric Project with a capacity of 480 MW. Computational simulations were carried out to observe the head loss, pressure distribution, velocity distribution, deformation, and stress in the manifold. The study focused on examining the effects of branch angle, cone length, and sickle plate for hydraulic analysis. The results indicated that head loss decreased with a reduction in branching angle and cone length, with the optimum branching angle calculated to be 30° and the ideal cone length at 9m. An optimized manifold profile was developed utilizing the best branch angle, cone length, and sickle plate. The head loss in the optimized profile at outlet-1, outlet-2, and outlet-3 was computed to be 0.13 m, 0.46 m, and 0.31 m, respectively. The optimized case was compared to the base case, which was designed by NEA Engineering Company, revealing a decrease in head loss of 37%, 15%, and 24% at outlet-1, outlet-2, and outlet-3, respectively. The study also performed a structural analysis, dividing the manifold into two parts, first bifurcation and second bifurcation. The initial pipe thickness of 60 mm at the first bifurcation and 50 mm at the second bifurcation were inadequate to meet allowable stress criteria, so the thickness of the pipe was increased to improve structural strength. The equivalent (von-Mises) stress at the first bifurcation with a 130 mm thick pipe and second bifurcation with a 70 mm thick pipe was 166 MPa and 161 MPa, respectively, where the allowable stress is 167 MPa. (Bardan Dangi, 2022)

2.2 Mathematical Equations and Models

2.2.1 Continuity Equation

Continuity equation and Navier Stokes equations are the governing equations used for flow calculations in steady incompressible, viscous and turbulent. So, the continuity equation and Navier Stokes in cylindrical coordinate system is as follows;

- Continuity Equation

$$\frac{\delta v_r}{\delta r} + \frac{\delta v_z}{\delta z} + \frac{v_r}{r} = 0$$

- Navier Stokes Equations

$$\delta V_r \frac{\delta V_\theta}{\delta r} + V_z \frac{\delta V_\theta}{\delta z} - \frac{V_r V_\theta}{r} = \vartheta\left(\frac{\delta^2 V_\theta}{\delta r^2} + \frac{\delta V_\theta}{r\delta r} - \frac{V_\theta}{r^2} + \frac{\delta^2 V_\theta}{\delta z^2}\right)$$

$$V_r \frac{\delta V_r}{\delta r} + V_z \frac{\delta V_r}{\delta z} - \frac{V_\theta^2}{r} + \frac{\delta \rho}{\rho \delta r} = \vartheta\left(\frac{\delta^2 V_r}{\delta r^2} + \frac{\delta V_r}{r\delta r} - \frac{V_r}{r^2} + \frac{\delta^2 V_r}{\delta z^2}\right)$$

$$\delta V_r \frac{\delta V_z}{\delta r} + V_z \frac{\delta V_z}{\delta z} + \frac{\delta \rho}{\rho \delta z} = \mathrm{g} + \vartheta\left(\frac{\delta^2 V_z}{\delta r^2} + \frac{\delta V_z}{r\delta r} + \frac{\delta^2 V_r}{\delta z^2}\right)$$

Where, V_r, V_θ and V_z are the radial, tangential and axial velocities respectively, g is acceleration due to gravity, ϑ is kinematic viscosity and ρ is the density of the fluid.

It is almost impossible to solve the equations analytically due to its complexity. Also, the presence of multiple domains also makes it more difficult to obtain the analytical solution. Thus, ANSYS CFX is used to obtain the solution of these equations.

2.2.2 Bernoulli's Equation

Bernoulli's theorem is a fundamental principle in fluid dynamics that relates the pressure, velocity, and elevation of an incompressible, inviscid fluid. Bernoulli's equation is a fundamental principle in fluid mechanics that describes the behavior of fluids as they move through pipes, nozzles, and other systems. It states that the sum of pressure, kinetic energy, and potential energy per unit volume of a fluid is constant along a streamline, assuming there is no work done by external forces. Bernoulli's equation has important applications in fields such as aviation, hydraulics, and meteorology, and has been used to develop models for fluid flow in many real-world situations. (Yunus A Cengel, 2013) Mathematically, it can be expressed as follows;

$$P + \frac{1}{2}\rho V^2 + \rho g h = constant$$

where:

P = pressure of the fluid

ρ = density of the fluid

V = velocity of the fluid

g = acceleration due to gravity

h = elevation of the fluid above a reference plane

2.2.3 Turbulence Model

Turbulence model in CFD is a method to include the effect of turbulence into the fluid flow simulation. Turbulent flows are prevalent in many engineering applications and thus majority of simulations require turbulence models (Simulating, 2020). These models allow calculation of mean flows. Shear Stress Transport (SST) model

The SST model is a formulation that combines the strengths of both k- ϵ and k- ω models, making it a versatile model for a variety of applications. The k- ω model is particularly effective at simulating flow in the viscous sublayer, while the k- ϵ model is better suited for predicting flow behavior in regions away from the wall. By combining these two models, the SST model is able to achieve optimal performance across a wide range of scenarios. (Rumsey, 2021). This model has following characteristics

- It accounts for the transport of turbulent shear stress and gives highly accurate predictions on the amount of flow separation under adverse pressure gradient.
- It exhibits less sensitivity to free stream conditions than many other turbulence models.
- It provides a platform for additional extensions such as SAS and laminar-turbulence transition.

The governing equations for SST model are as follows;

- Turbulence Kinetic Energy

$$\frac{\delta k}{\delta t} + U_j \frac{\delta k}{\delta_{x_j}} = P_k - \beta * k_w + \frac{\delta}{\delta_{x_j}} [(\vartheta + \sigma k \vartheta T) \frac{\delta k}{\delta x_j}$$

- Specific Dissipation Rate

$$\frac{\delta w}{\delta t} + U_j \frac{\delta w}{\delta x_j} = \ \alpha S^2 - \beta w^2 + \frac{\delta}{\delta x_j} \left[(\vartheta + \sigma_w \vartheta_T) \frac{\delta w}{\delta x_j} \right] + 2(1 - F_1) \sigma_{w2} \frac{1}{w} \frac{\delta k}{\delta x_i} \frac{\delta w}{\delta x_i}$$

The blending function F1 is given as;

$$F_1 = \tanh\{ \left\{ \min \left[\max \left(\frac{\sqrt{k}}{\beta * wy}, \frac{500\vartheta}{y^2 w} \right), \frac{4\sigma_{w2} k}{CD_{kw} y^2} \right] \right\}^4 \}$$

Kinematic eddy viscosity ϑ_T is given as;

$$\vartheta_T = \frac{a_1 k}{\max \, (a_1 w, SF_2)}$$

Second Blending Function F2 is given as;

$$F_2 = \tanh\{ \max \left(\frac{2\sqrt{k}}{\beta * wy}, \frac{500\vartheta}{y^2 w} \right)^2 \}$$

Production limiter P_k is given as;

$$P_k = min \left(\tau_{ij} \frac{\delta U_i}{\delta x_j}, 10\beta * kw \right)$$

2.2.4 Finite Element Analysis (FEA)

Finite Element Analysis (FEA) is a computational method used to solve engineering problems by breaking down complex systems into smaller, simpler parts. It is widely used in industries such as aerospace, automotive, and construction to predict the behavior of structures under different loading conditions. FEA is based on the concept of discretization, where the continuous system is divided into a finite number of smaller sub-domains or elements, each with a specific shape and size. The equations of motion or equilibrium for each element are then solved, and the results are combined to obtain a solution for the entire system.

FEA has numerous advantages over traditional analytical methods, such as the ability to handle complex geometries, material non-linearity, and boundary conditions. It is also highly adaptable to different types of analysis, including static, dynamic, thermal, and fluid analysis. FEA can help engineers optimize designs and reduce costs by predicting the behavior of a structure before it is built. It also allows for quick and easy evaluation of design modifications, reducing the time and cost of prototyping.

FEA has revolutionized the field of engineering by providing engineers with a powerful tool to simulate and analyze complex structures. It has helped to minimize the need for physical testing and experimentation, making the design process faster, cheaper, and more accurate. (Seshu, 2012)

CHAPTER-3: RESEARCH METHODOLOGY

3.1 Research Outline

This study mainly focuses on finding the correct bifurcation transition profile and its corresponding pipe thickness to withstand the stress under allowable working condition for the Raghuganga Hydropower Project. The angle of bifurcation and its layout has already been finalized and the transition of bifurcation from main pipe to branching with minimum head loss has been chosen for this study. The details of the project are shown in Table-1 and the layout of bifurcation is shown in figure-2 below.

Table 1: Salient Features of Hydropower Project

Salient Features of the Project		
1	Name of the Project	Raghuganga Hydroelectric Project
2	Location	Myagdi District of gandaki Province
3	Latitude	28°22’21”N to 28°25’45”N
4	Longitude	83°31’13”E to 83°34’35”E
5	Name of River	Raghuganga
6	Type of Plant	Peaking run-of-river
7	Gross Head (m)	292.83
8	Net rated Head (m)	281.56
9	Installed Capacity (MW)	2x20=40
10	Average Annual Energy	238.59 GWh
11	Design Flow (m3/s)	16.67
12	Diameter of Penstock (m)	2.15
13	Head race tunnel (m)	6270.106 m Concrete lined
14	Length of Penstock (m)	53.15
15	Penstock Diameter after Bifurcation (m)	1.52

16	Type of Turbine	Pelton, Vertical Axis
17	Rated flow for each unit (m3/s)	8.34
18	Transmission Line	220 kv 600m

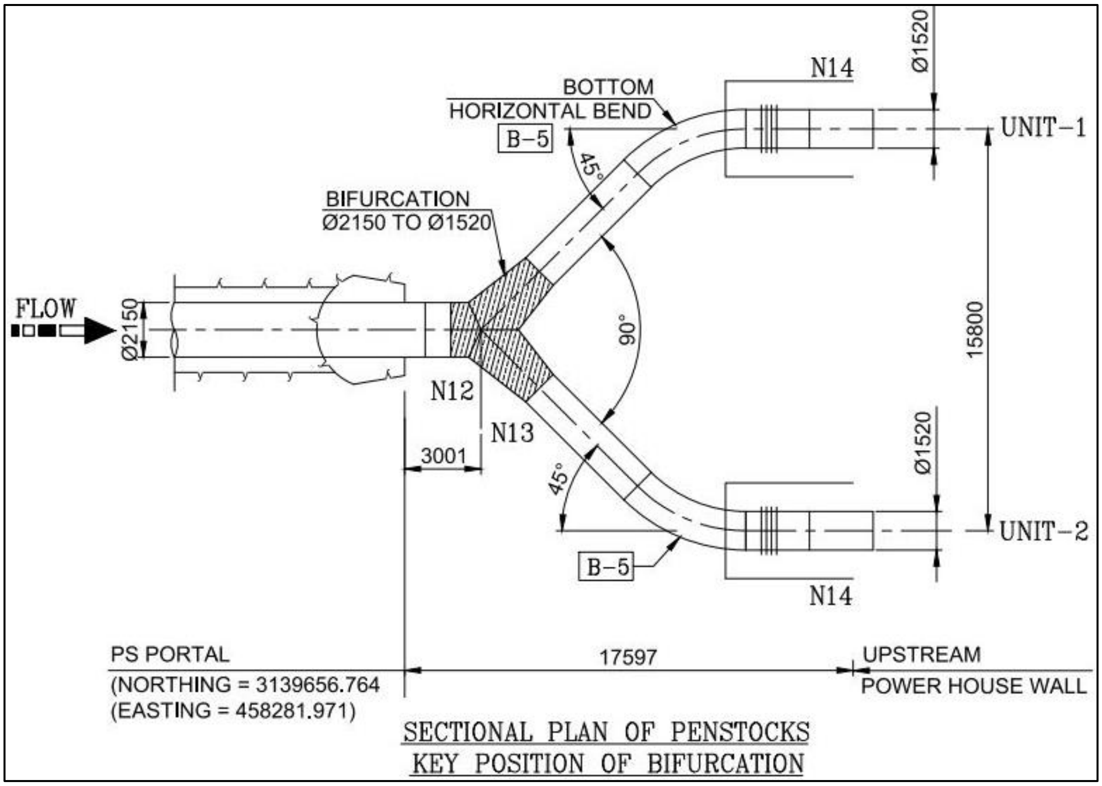

Figure 2: Plan of Bifurcation as provided by Project

The methodology adopted for this study is illustrated in the figure-3 below. The modeling shall be done in Solid works and then it is imported to ANSYS CFX to perform hydraulic and structural analysis to obtain the desired profile and plate thickness.

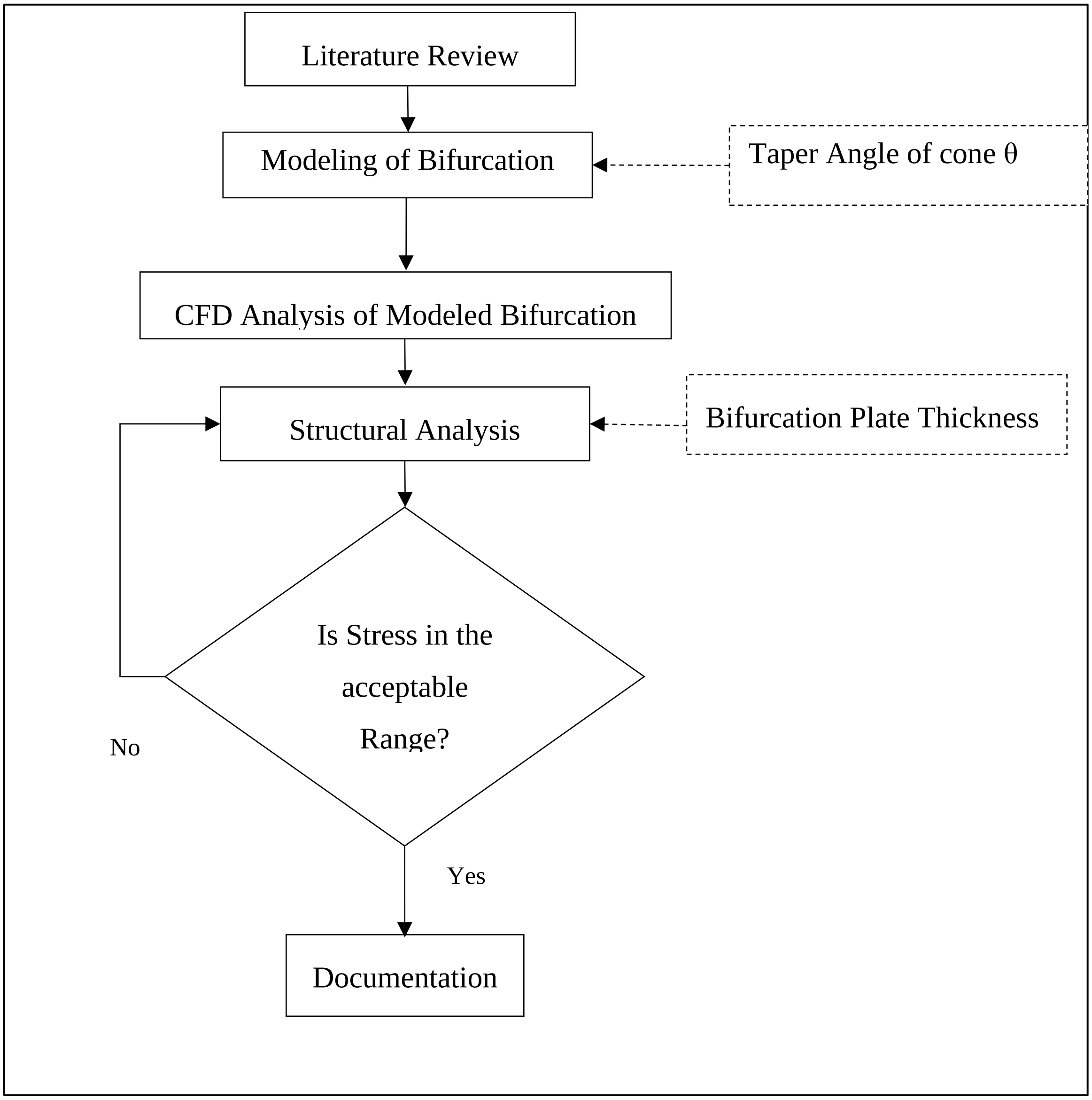

Figure 3: Research Methodology

3.2 Geometry Modeling

A penstock bifurcation is a point where a single penstock splits into two or more separate branches. In order to model this geometry, it is important to take into account factors such as the flow rate and pressure at the bifurcation point, as well as the angle and shape of the branches. Solid works is the software used to create a 3D model of the bifurcation in this study. The basic plan of the bifurcation and the dimension taken for modeling is shown in the figure-4.

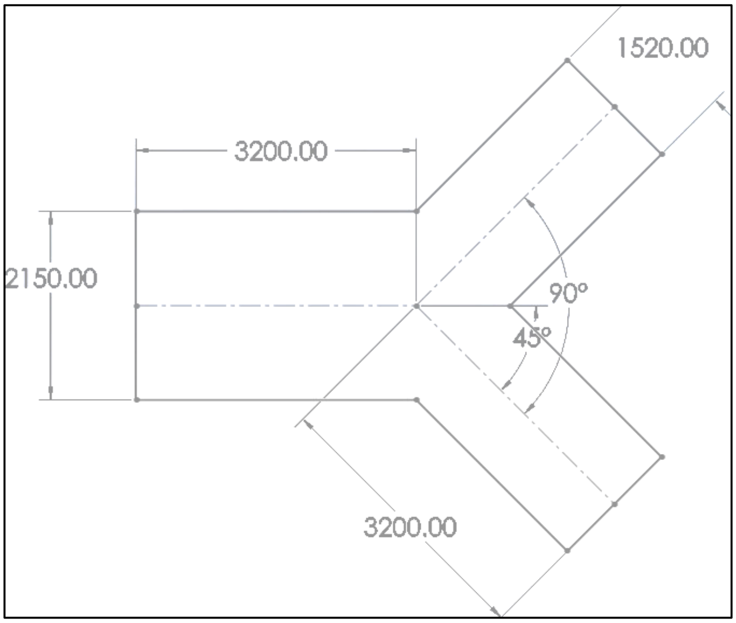

Figure 4: Basic Plan of Bifurcation for modeling

The plan for the first case to be modeled by changing the taper angle of bifurcation is shown in the figure-5 below.

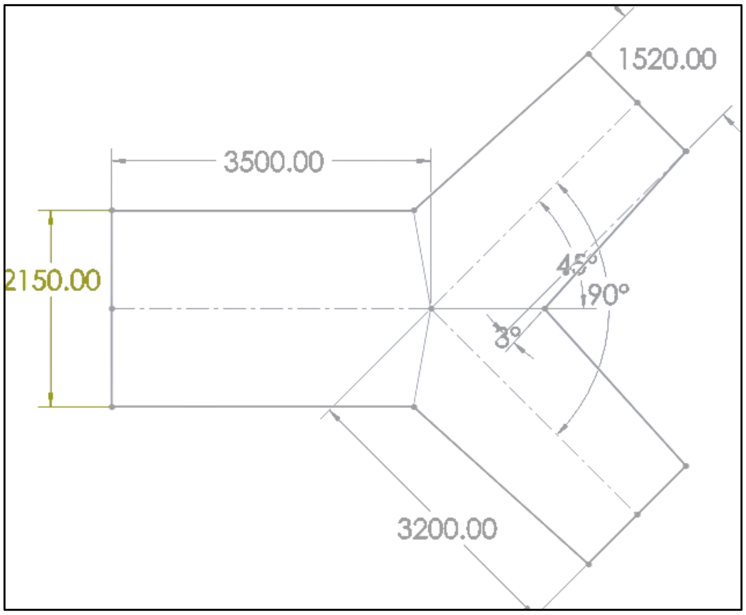

Figure 5: Plan for modeling of Case-1

The major dimensions used here are given by Civil designers i.e. the Pipe diameters for inlet and outlet of Bifurcation, the angle of bifurcation and the length segments of the branched pipes. For modeling of pipe branching, the taper angle of cone of the branch pipe is gradually increased from 3 degree up to 15 degrees by increasing by 2 degrees simultaneously and then its model is exported to ANSYS CFX in. STEP and then CFD analysis is carried out to get the profile with minimum hydraulic loss at the branching of the pipe. The isometric view of the 3D modeled bifurcation of case-1 is shown in the figure-6 below.

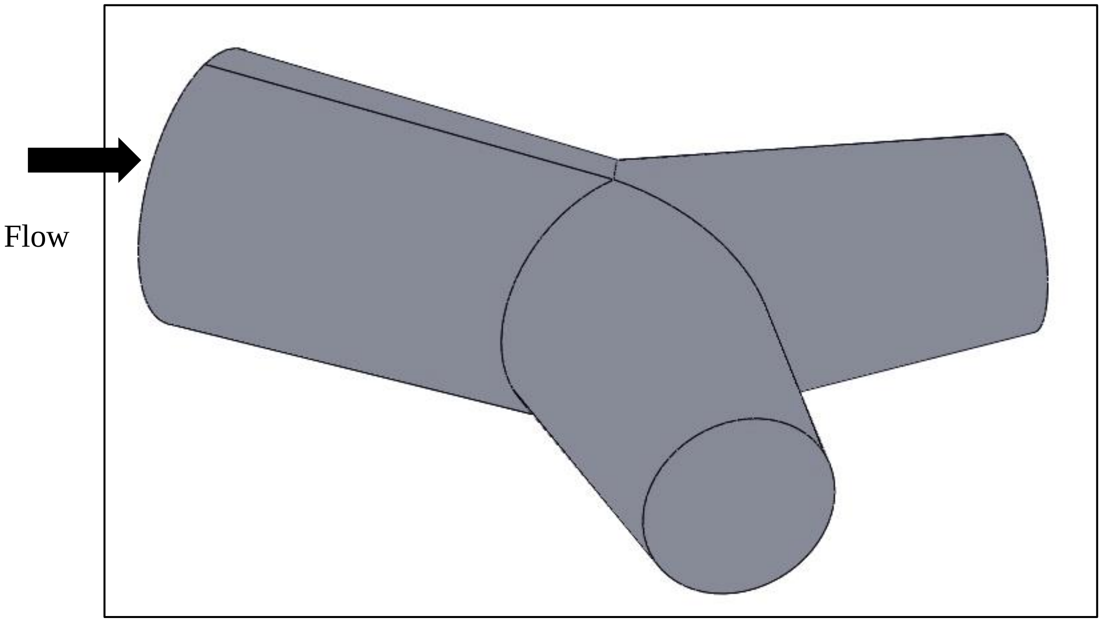

Figure 6: Isometric view of modeled bifurcation for case-1

The process explained above is carried out for 6 more times to get the 3D models of each case obtained by changing the taper angle of branched pipe from 3 degree to 15 degree.

3.3 Computational Fluid Modeling

3.3.1 Meshing of Geometry

The 3D modeled bifurcations mentioned above are then imported to ANSYS CFX for CFD analysis. Mesh is generated in CFX Mesh. Tetrahedral meshing is generated and mesh refinement of refinement order-3 has been included around the intersecting edges, face sizing has been used for the refinement of the mesh around the faces and inflation layer has been introduced around the boundary with 7 layers and growth rate of 1.2 to generate and refine the mesh to perform the subsequent simulation. The details of mesh is shown in the figure-7 below. In addition to this mesh independence test has been carried which is descried in the later section of the report to find the optimum mesh number for further simulations.

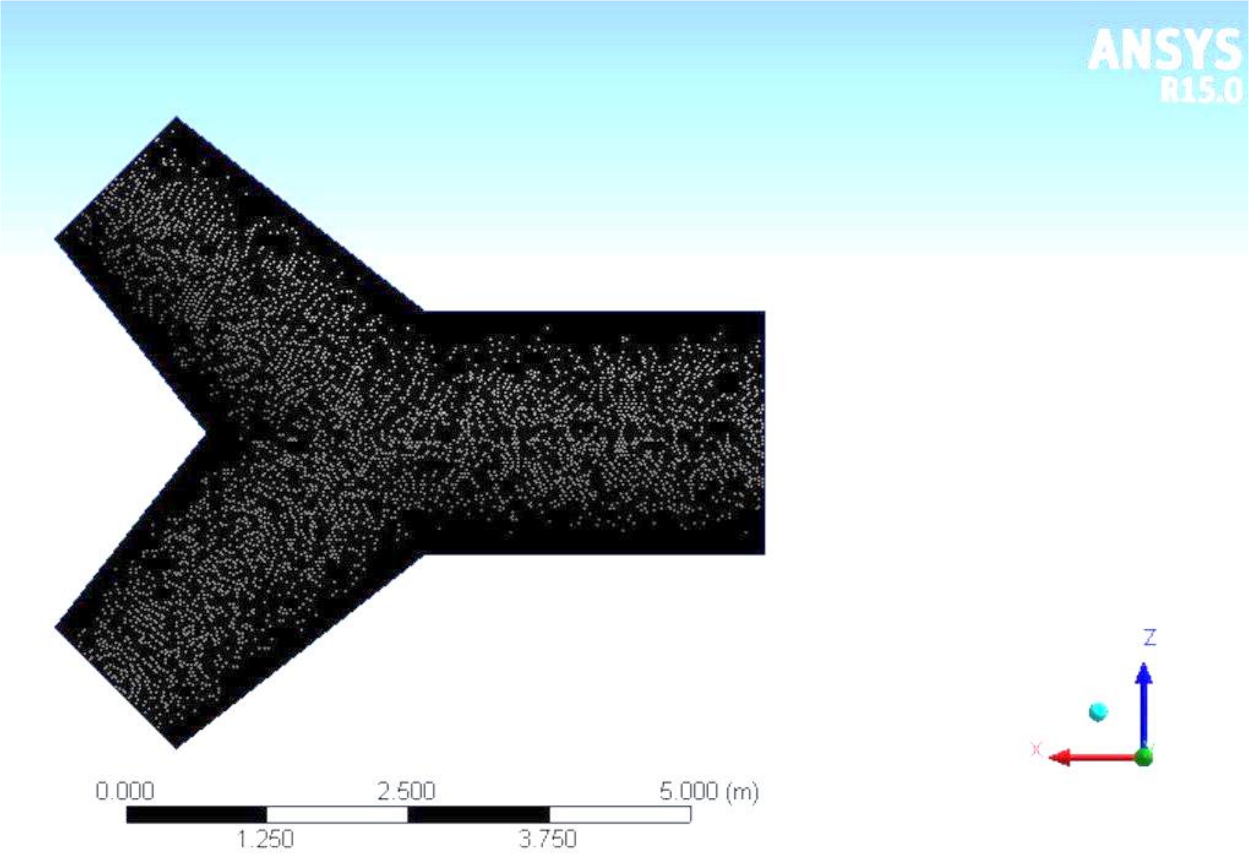

Figure 7: Tetrahedral Meshing for CFD analysis with necessary refinements

3.3.2 Set-up and Boundary Conditions

The set-up conditions and boundary conditions used for the analysis based on the available data are presented in the table-2 below.

The input values of parameters for setup conditions while performing computational analysis are obtained as below.

- For Total Pressure at inlet, the total design head is obtained by adding 15% of total net head i.e., net head is 281.56 m and addition of 42.23 m as surge head and thus the total head is 323.79 m and its corresponding value is 3.187 MPa.
- For Outlet mass flow rates, the design discharge of each unit is 8.34 m^3/s and its equivalent flow rate in kg/s is 8.34 x 10^3 kg/s.

Table 2: Boundary Conditions setup for CFD Analysis

Domain	**Boundaries**	
Default Domain	**Boundary - Inlet**	
	Type	INLET
	Location	Inlet
	Settings	
	Flow Direction	Normal to Boundary Condition
	Flow Regime	Subsonic
	Mass And Momentum	Total Pressure
	Relative Pressure	3.1870e+00 [MPa]
	Turbulence	High Intensity and Eddy Viscosity Ratio
	Boundary - Unit1	
	Type	OUTLET
	Location	Unit1
	Settings	
	Flow Regime	Subsonic
	Mass And Momentum	Mass Flow Rate
	Mass Flow Rate	8.3400e+03 [kg s^-1]
	Boundary - Unit2	
	Type	OUTLET
	Location	Unit2
	Settings	
	Flow Regime	Subsonic
	Mass And Momentum	Mass Flow Rate
	Mass Flow Rate	8.3400e+03 [kg s^-1]
	Boundary - Walls	
	Type	WALL
	Location	Walls
	Settings	
	Mass And Momentum	No Slip Wall
	Wall Roughness	Smooth Wall

3.3.3 Mesh Independence Test

Mesh independence testing is an important step in any CFD analysis to ensure that the results obtained from the simulation are not affected by the choice of mesh. The mesh is a discretization of the physical domain into small elements, and the quality and resolution of the mesh can greatly impact the accuracy and reliability of the simulation. By performing a mesh independence test, the user can determine the minimum mesh resolution required to accurately capture the physics of the problem, and also check that the results are not affected by the choice of mesh. Thus, inorder to carry out further simulations, mesh independence test is carried out by varying mesh number and noting the value of turbulent kinetic energy and its values are plotted to obtain the graph as shown in figure-8 below.

Table 3: Mesh Independence Test

S.N.	Numer of Mesh	Turbulent Kinetic Energy (J/kg)
1	92422	0.024
2	169736	0.028
3	285174	0.032
4	798239	0.034
5	1695639	0.036
6	2327152	0.037

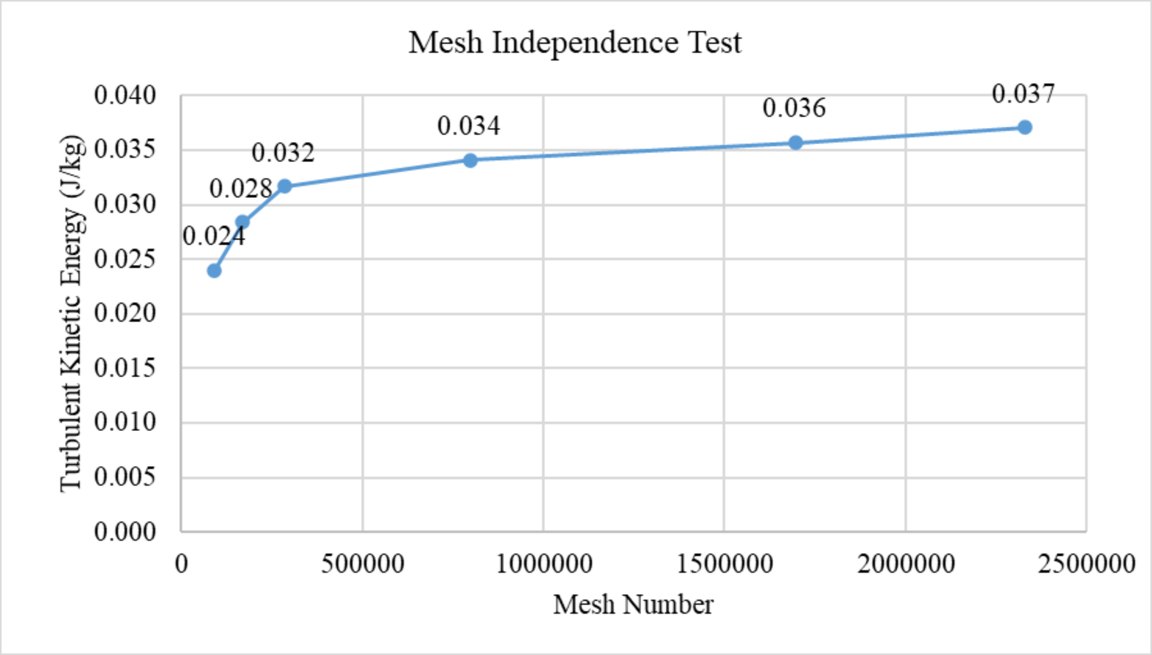

Figure 8: Mesh Independence Test

The results obtained from the above graph and table show that there is not more than 3.84% variation in output values when the mesh number is varied from 1695639 to 2327152. Thus, the mesh number of 1695639 is taken for subsequent simulations.

CHAPTER-4: RESULTS AND DISCUSSIONS

4.1 Hydraulic Analysis

The Bifurcations as described in methodology were modeled and the boundary conditions were set and its CFD analysis was carried out in ANSYS CFX and the results were noted. The different cases that have been analyzed and evaluated in this study and are named as cases are listed in the table-4 below.

Table 4: Total Cases Analyzed for Hydraulic Analysis

S. N.	Case No.	Taper Angle of Cone (Degree)
1	Case-1	3
2	Case-2	5
3	Case-3	7
4	Case-4	9
5	Case-5	11
6	Case-6	13
7	Case-7	15

4.1.1 Pressure and Velocity Distribution

The Distribution of Total Pressure and Velocity at the mid plane of each simulated case was evaluated and its corresponding values at inlet and outlets were noted. The figures 9-15, below are the results for the plot of velocity at mid plane for all seven simulated cases.

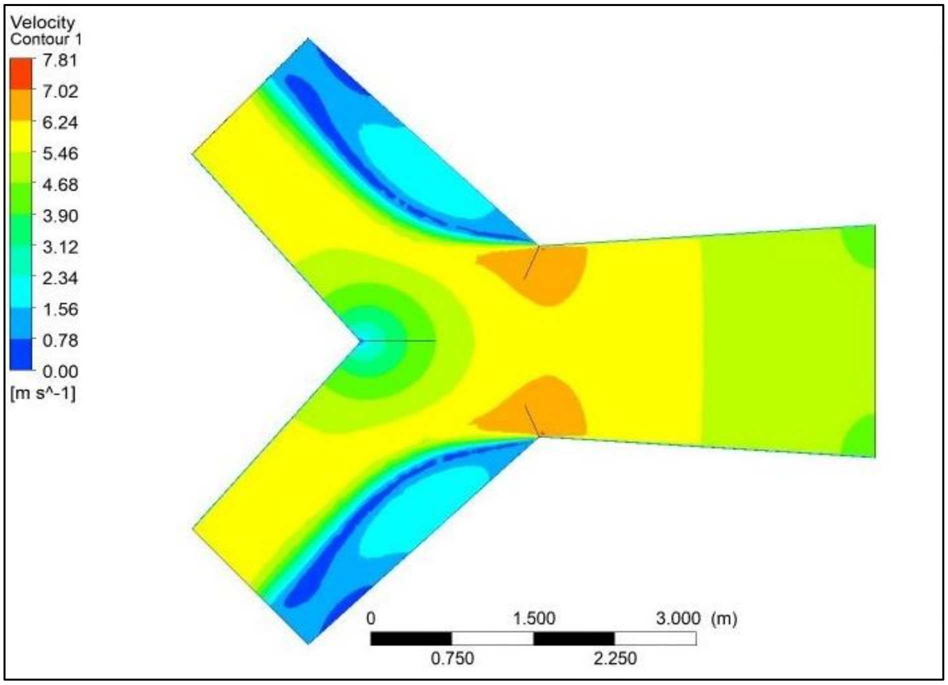

Figure 9: Velocity Plot at mid plane for case-1

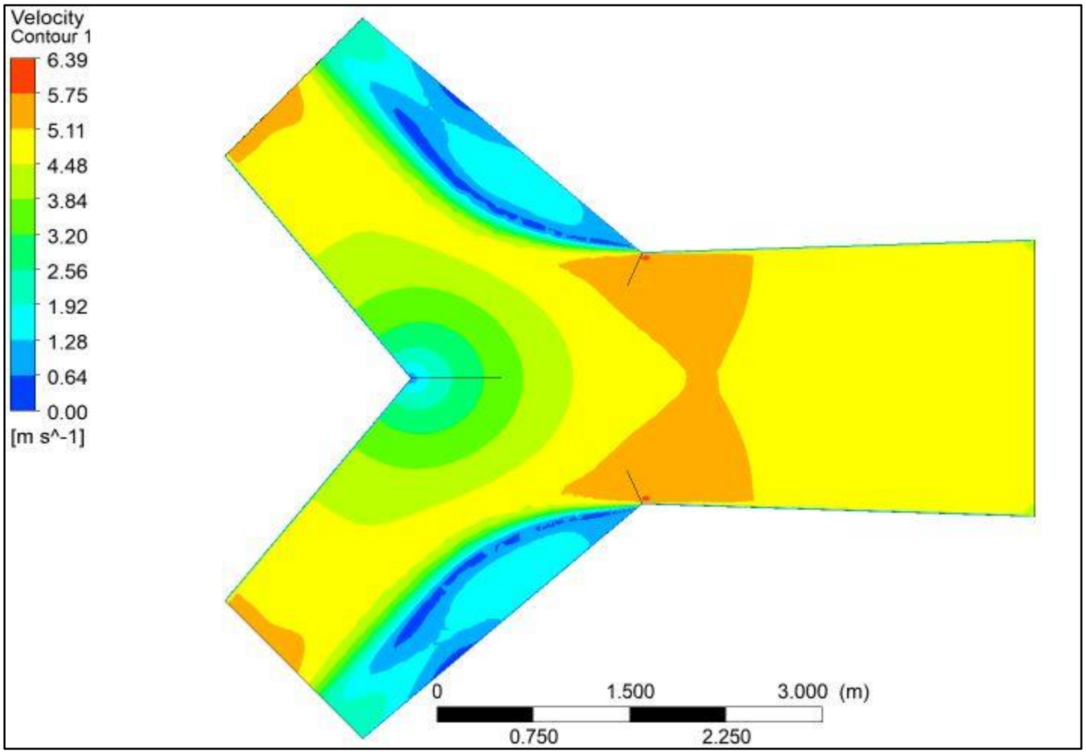

Figure 10: Velocity Plot at mid plane for case-2

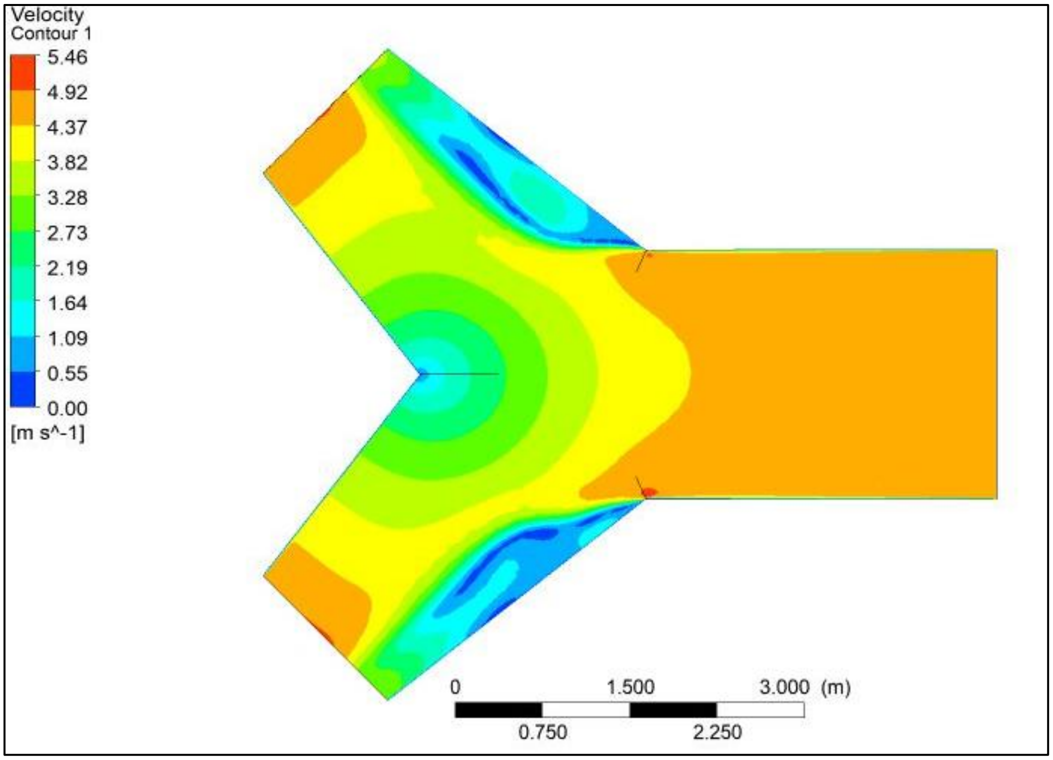

Figure 11: Velocity Plot at mid plane for case-3

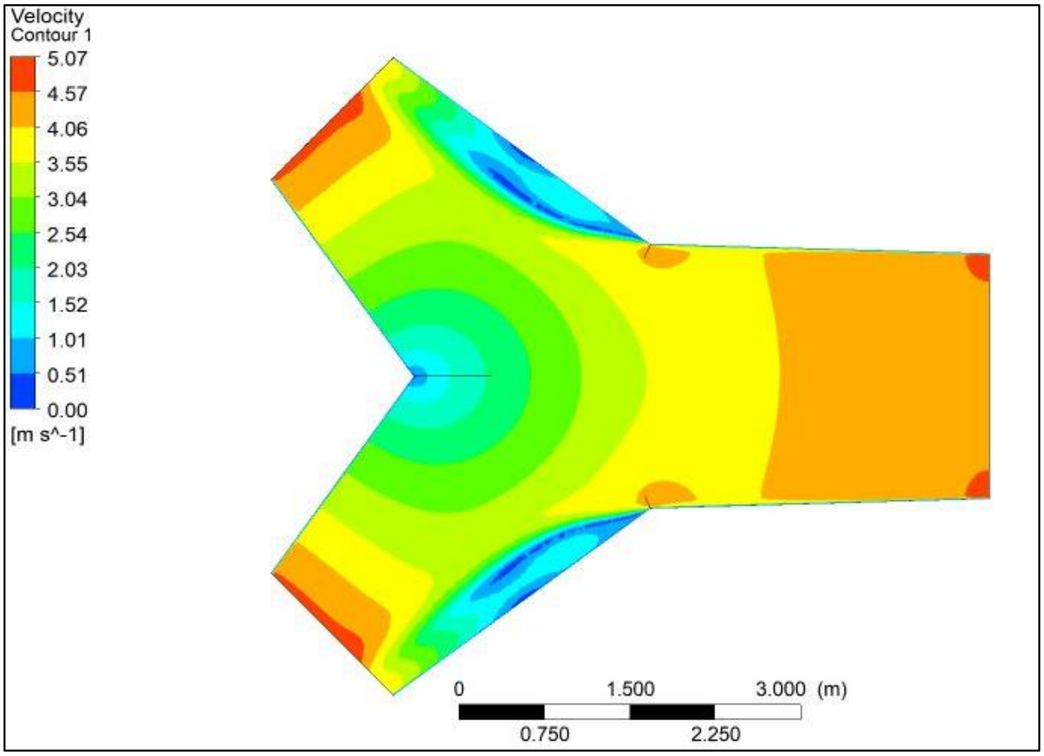

Figure 12: Velocity Plot at mid plane for case-4

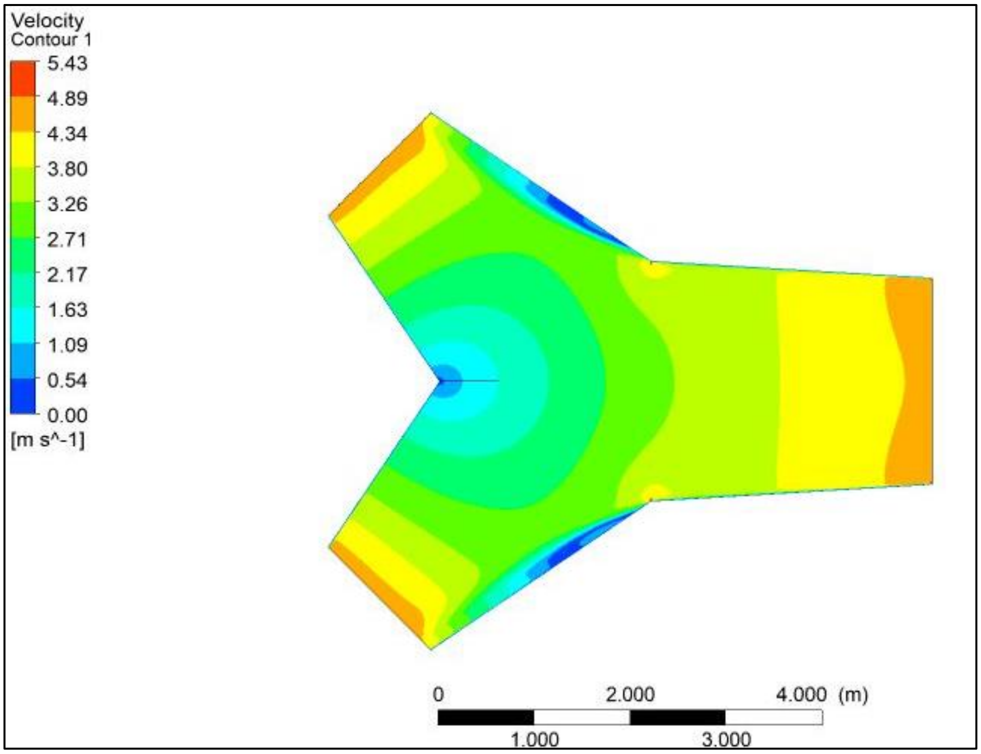

Figure 13: Velocity Plot at mid plane for case-5

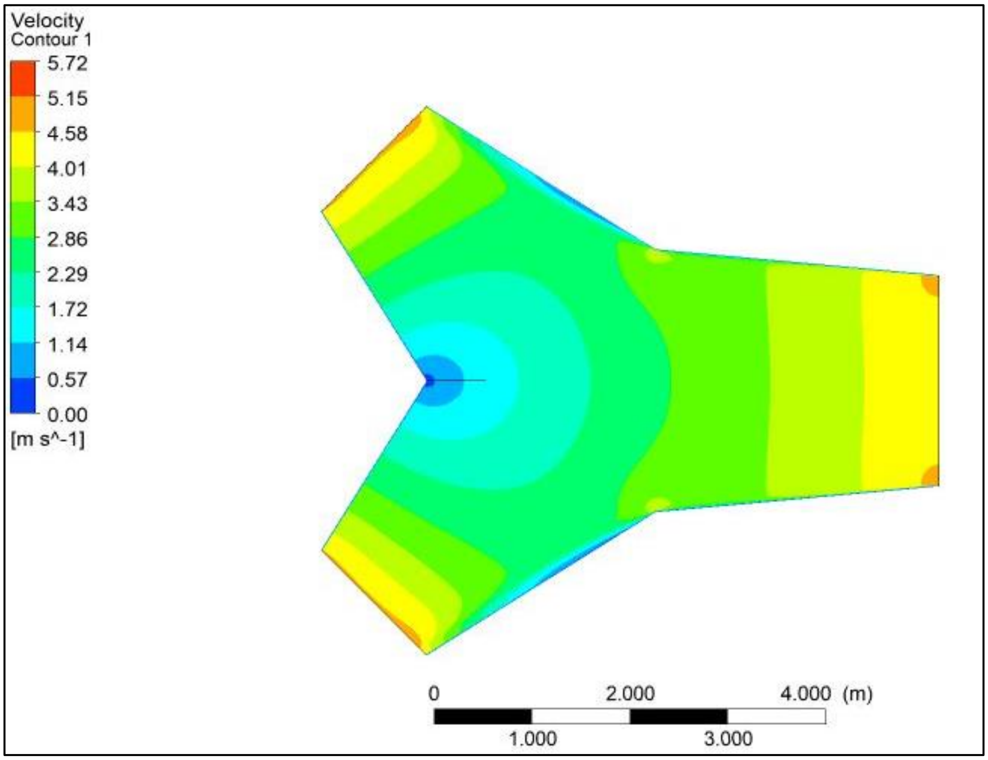

Figure 14: Velocity Plot at mid plane for case-6

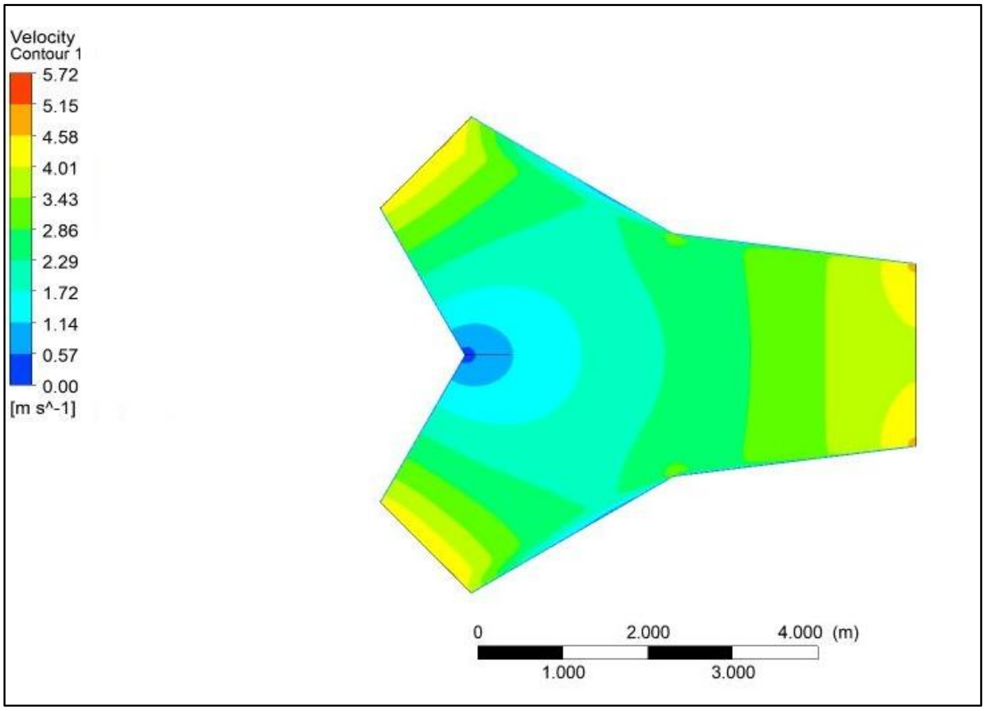

Figure 15: Velocity Plot at mid plane for case-7

The results of Total Pressure plot for all seven cases are shown in the figures 16-22 below.

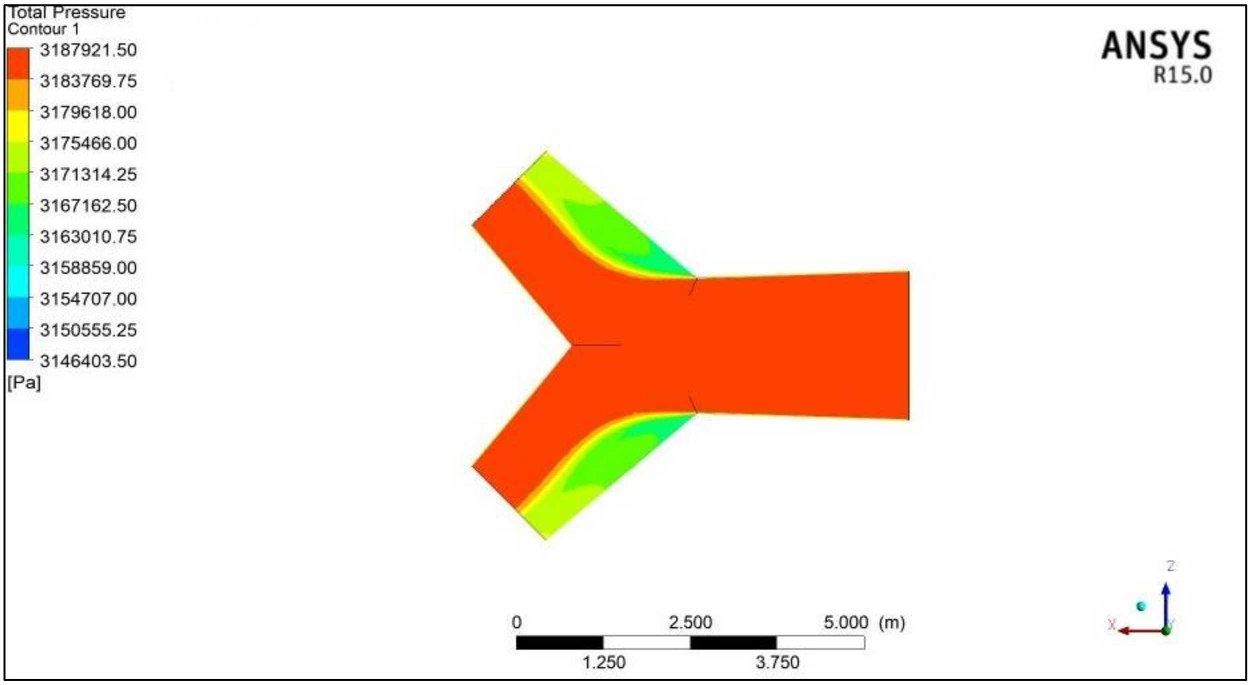

Figure 16: Pressure Plot at mid plane for case-1

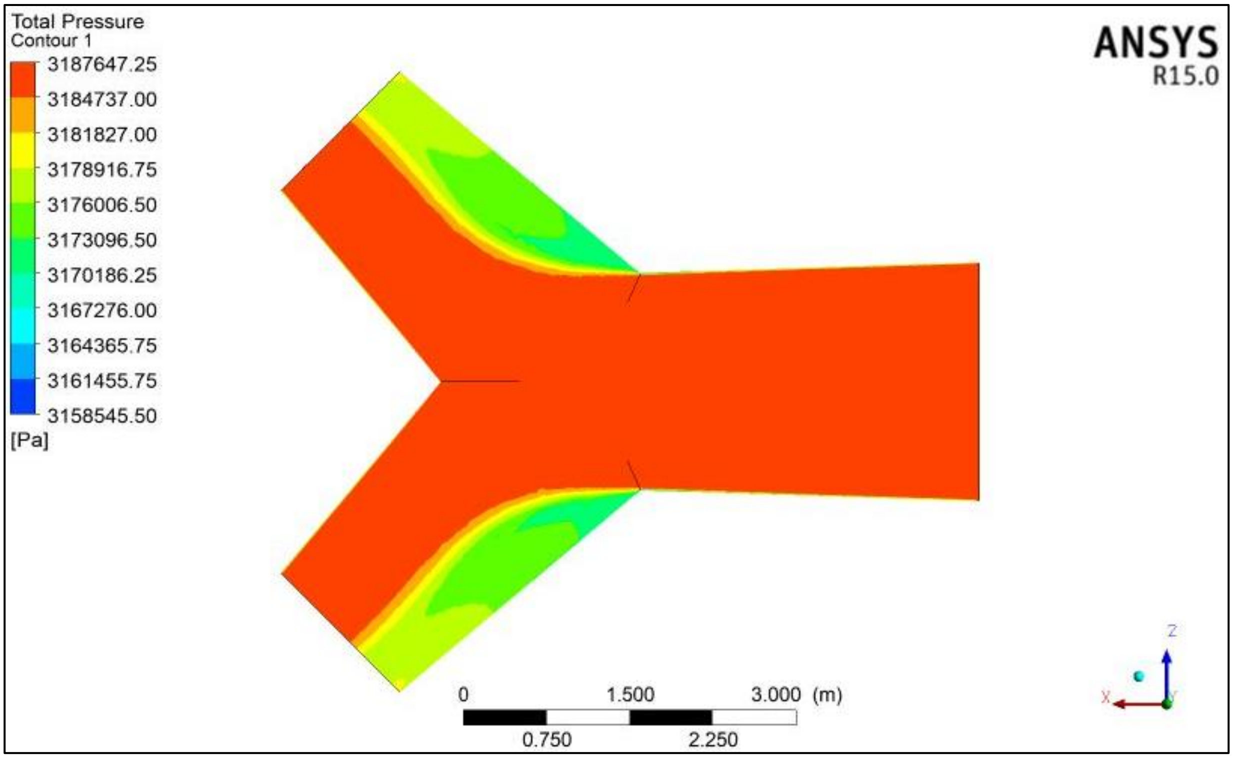

Figure 17: Pressure Plot at mid plane for case-2

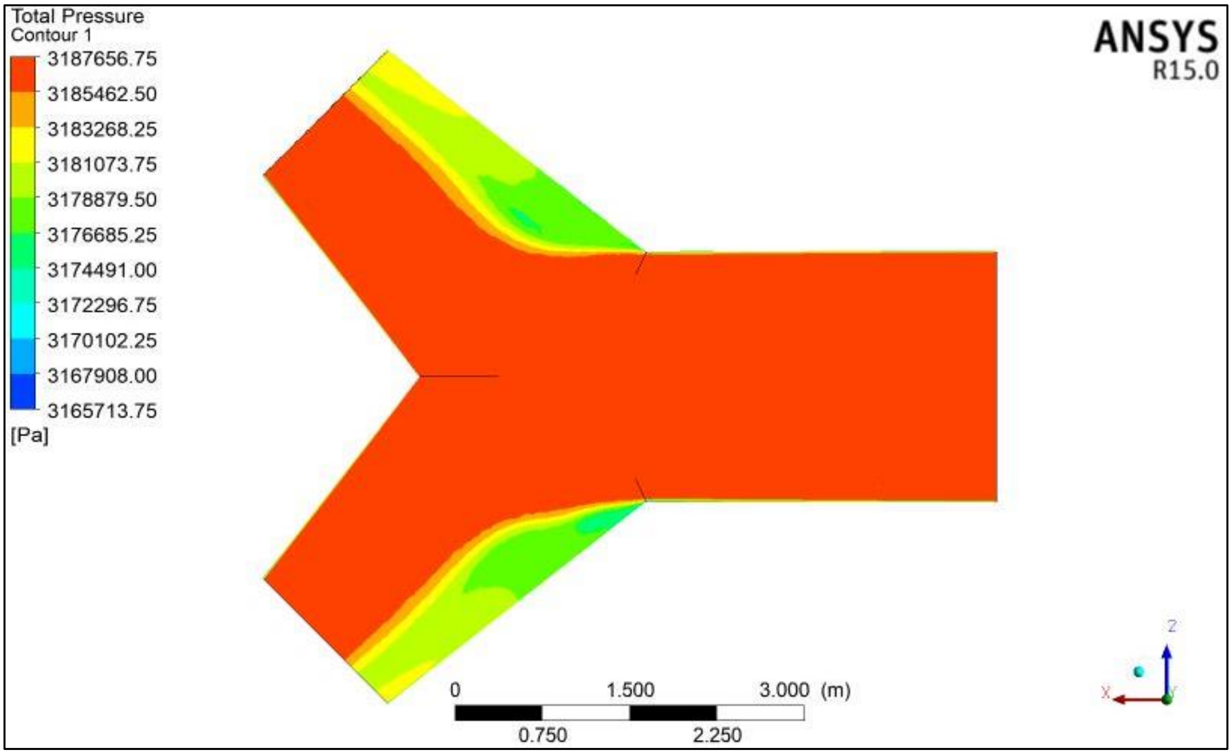

Figure 18: Pressure Plot at mid plane for case-3

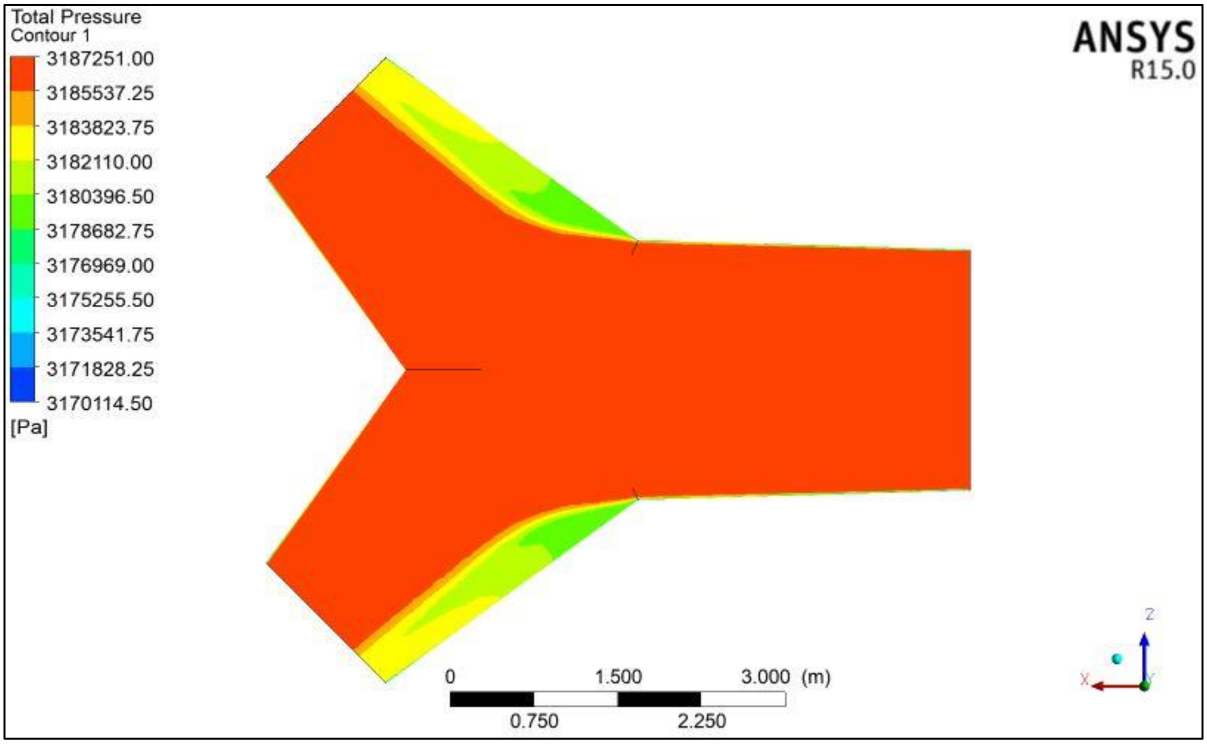

Figure 19: Pressure Plot at mid plane for case-4

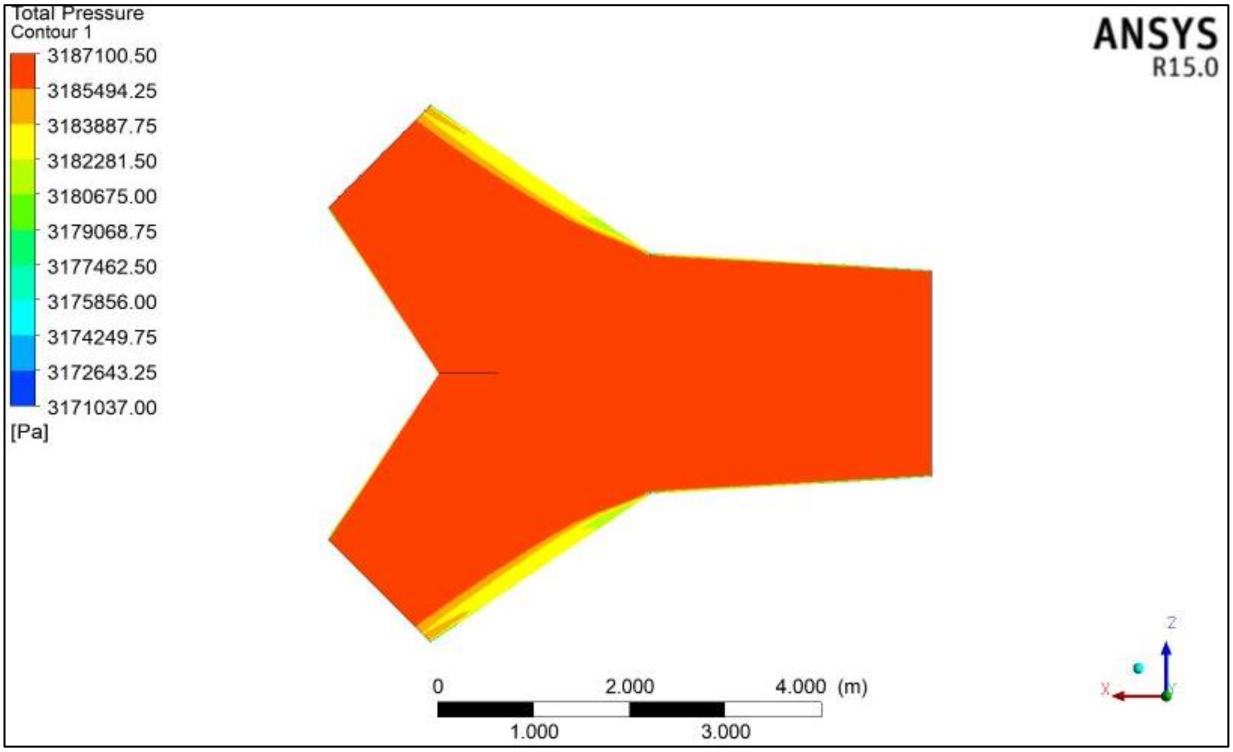

Figure 20: Pressure Plot at mid plane for case-5

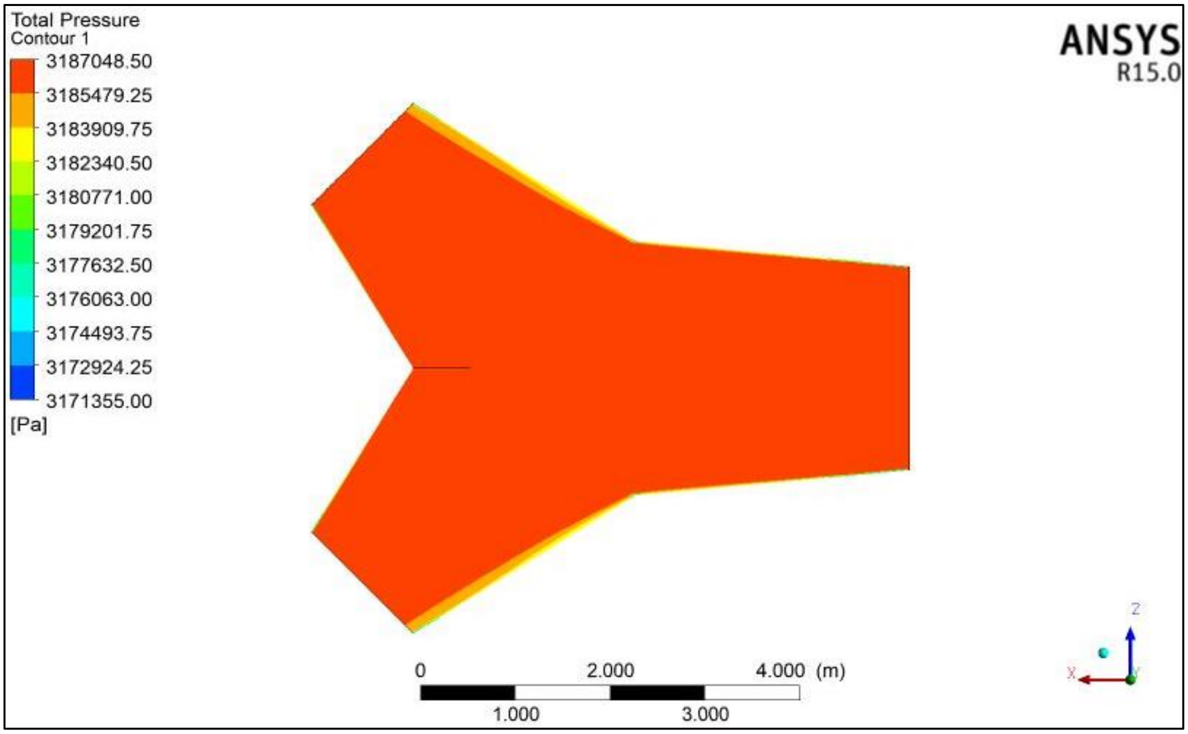

Figure 21: Pressure Plot at mid plane for case-6

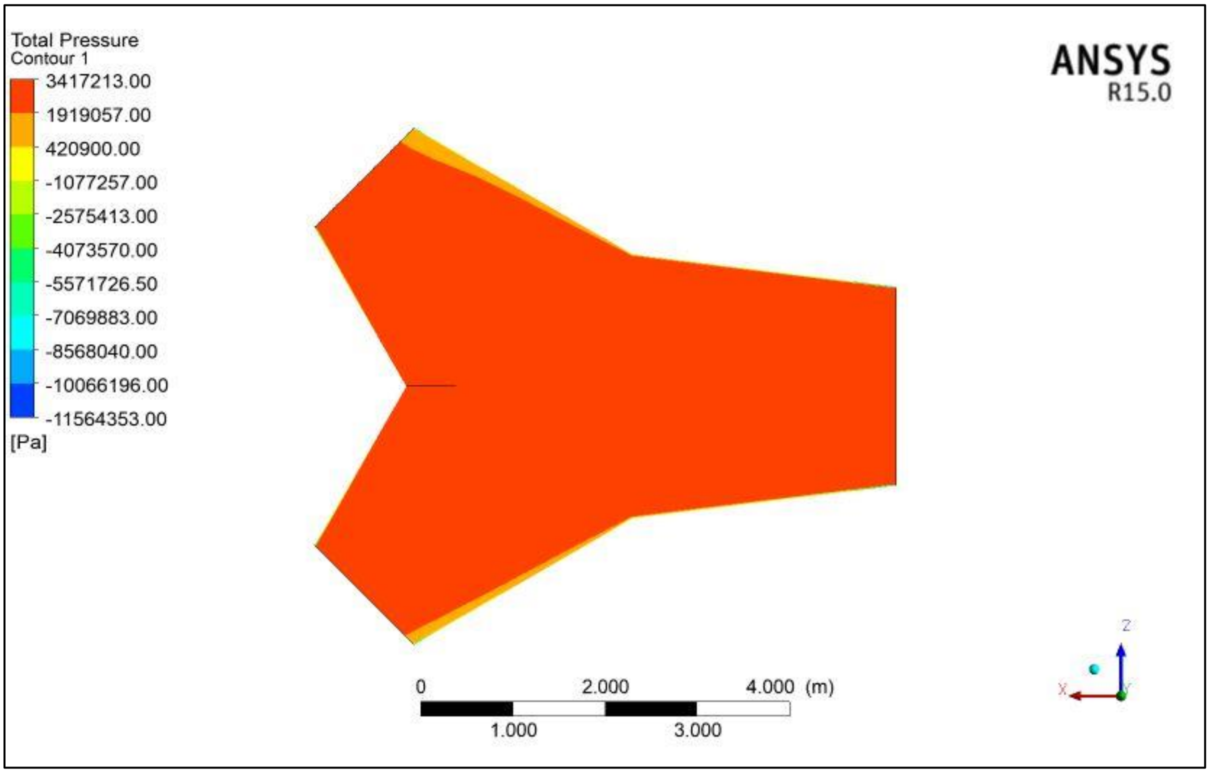

Figure 22: Pressure Plot at mid plane for case-7

4.1.2 Calculation of Head Loss

Head Loss for each branching unit Was Calculated by using equation-1 by using the values from the results of simulations. (C. A. AGUIRRE, n.d.)

$$H_{l_i} = \left(\frac{P_{in}}{\rho g} + \frac{V_{in}^2}{2g}\right) - \left(\frac{P_{Out_i}}{\rho g} + \frac{V_{Out_i}^2}{2g}\right) \qquad \text{Equation 1}$$

Where;

$H_{l_i} = Head\ loss\ at\ individual\ branching$ (m)

$P_{in} = Total\ Pressure\ in\ at\ Inlet$ (Pa)

$V_{in} = Total\ velocity\ at\ inlet\ of\ Bifurcation$ (m/s)

$P_{Out_i} = Total\ Pressure\ at\ oulet\ of\ Branching\ i$ (Pa)

$V_{Out_i} = Total\ Velocity\ at\ outlet\ of\ each\ branch$ (m/s)

$\rho = Density\ of\ water$ (kg/m^3)

$g = Acceleration\ due\ to\ gravity$ (m/s^2)

Total Head loss for each case was calculated by using the equation-2 as below.

$$Total\ Head\ Loss = \sum H_{l_i} \dots\dots\dots\dots\dots\dots\dots Equation\ 2$$

Here, the value of z-component in equation-1 is neglected and only the components of Pressure head and Velocity head has been taken into account for the calculation of head loss using Bernoulli's Equation. It is so because, the elevation of both branched pipes and bifurcation is the same and thus has no significance of its use.

Total head loss at the bifurcation was calculated using the above equation in all simulated cases and illustrated in the table-5 below.

Table 5: Head Loss Calculation and Data from Simulations

Cases	**Taper Angle (Degree)**	**Inlet Pressure (Pa)**	**Inlet Velocity (m/s)**	**Outlet Pressure-1 (Pa)**	**Outlet Velocity-1 (m/s)**	**Outlet Pressure-2 (Pa)**	**Outlet Velocity-2 (m/s)**	**Total Head Loss (m)**
1	3	3186990	4.532	3182140	4.681	3182150	4.684	0.972
2	5	3186990	4.573	3184810	4.594	3184860	4.608	0.436
3	7	3186990	4.615	3185840	4.568	3185840	4.573	0.239
4	9	3186970	4.654	3186290	4.580	3186280	4.581	0.147
5	11	3186940	4.694	3186510	4.598	3186510	4.600	0.097
6	13	3186920	4.726	3186550	4.612	3186560	4.612	0.086
7	15	3098660	4.226	3088000	4.114	3087840	4.107	2.201

A graph is plotted for the total head loss calculated and the taper angle of the cone. The figure-23 below shows the result.

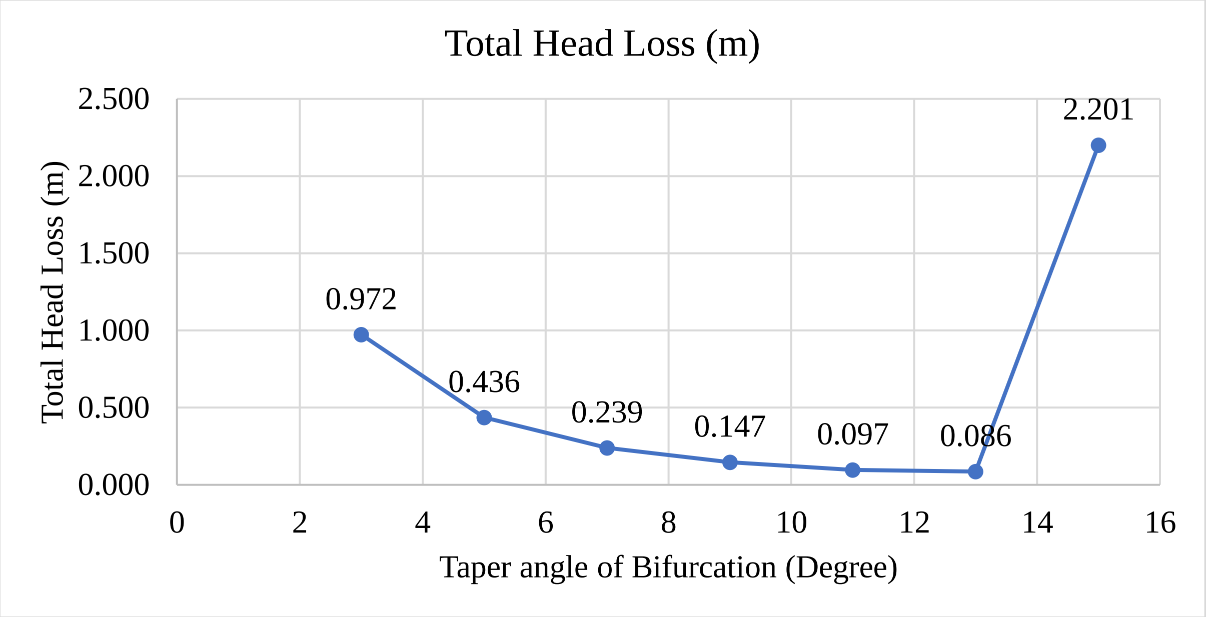

Figure 23: Head Loss Vs Taper Angle of Bifurcation

The result shows that, head loss is minimum for taper angle of 13 degrees for branching of pipes with total head loss of 0.086 m in branched section. The pressure and velocity distribution also show the same results as there is less velocity drop and pressure distribution in the junction.

As the cone angle of the bifurcation increases, the curvature of the cone also increases. Up to certain range, the values of head loss and turbulence kinetic energy decreases as the flow passage is not sufficient to attain the flow characters of minimum loss. But, increasing the cone angle beyond the limit causes the flow to separate more readily, creating more vortices and turbulence, and hence more energy loss due to flow separation as there is increase in curvature of cone and hence more area for fluid to interact. This additional energy loss due to flow separation results in an increase in the overall head loss of the bifurcation. The same has been illustrated by the plot of Total Turbulence Kinetic Energy for all simulated cases as shown in figure-24 below.

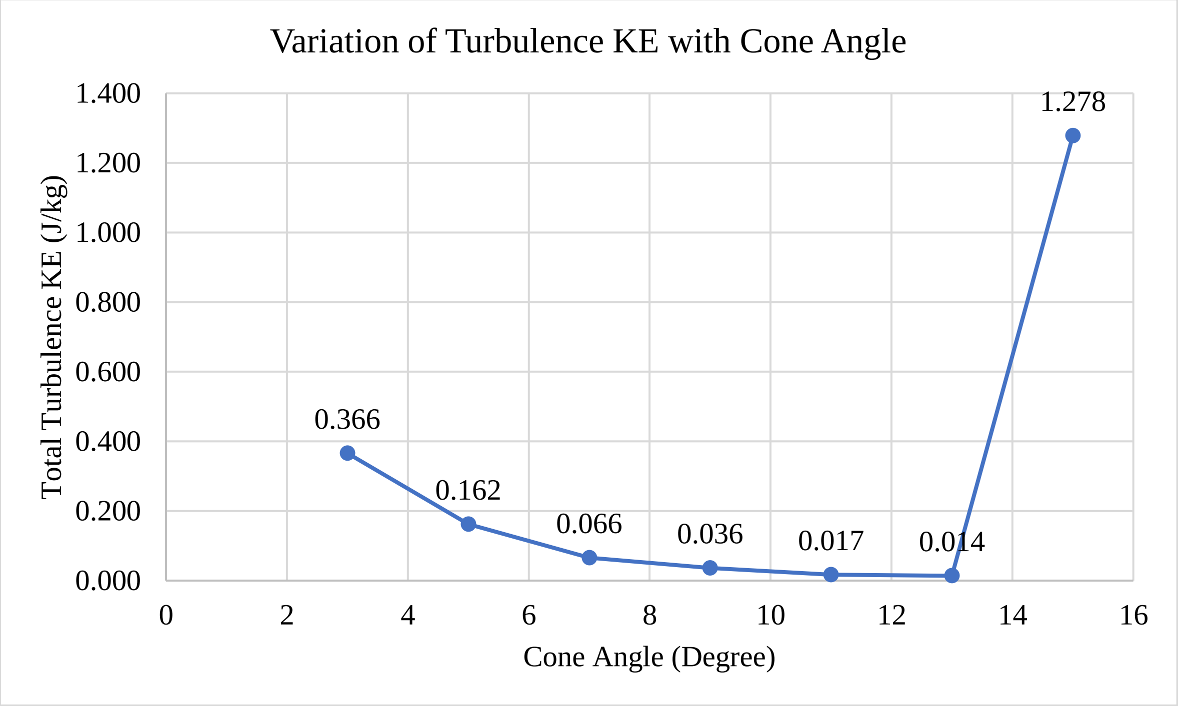

Figure 24: Plot of Total Turbulence K E vs Cone angle of bifurcation

4.2 Structural Analysis

The structural analysis of a penstock bifurcation is an important aspect of designing a safe and reliable hydraulic system. The bifurcation structure must be designed to withstand the fluid pressure, the weight of the fluid, and any external loads acting on it. So, the thickness of pipe at penstock bifurcation was varied starting from 25 mm gradually with the necessary sickle and other reinforcements and its structural analysis was done to obtain the values of equivalent stress under the acceptable range. The material of bifurcation is structural steel E 250 (Standard, 2011) having yield stress of 250 MPa corresponding to the Indian Standard Code of practice IS 2062: 2011. The properties of chosen material are as listed in the table-6 below.

Table 6: Properties of E 250 Material *(Standard, 2011)*

S. N.	Properties	Values
1	Ultimate Tensile Strength	410 MPa
2	Yield Stress (t<20 mm)	250 MPa
3	Yield Stress (t=20-40 mm)	240 MPa
4	Yield Stress (t>40 mm)	230 MPa

The 3D model of the bifurcation for structural analysis with initial pipe thickness of 25 mm and other necessary reinforcements like sickle and other ring stiffeners was modeled in SolidWorks and then imported in ANSYS STATIC STRUCTURAL for further simulation. Tetrahedral meshing was done for the imported model of bifurcation and simulation was run with the set-up conditions as follows.

- The Inlet, unit-1 and unit-2 branches were set as fixed support
- The total pressure of 3.187 MPa was applied as the internal pressure on exerted by the fluid on the interior all surfaces of the bifurcation and reinforcements

The analysis was run with above mentioned set-up conditions and the values of maximum stress and minimum factor of safety were noted for all the simulated cases and the results were compared with the values of material chosen being based on the Indian Standard code of practice and based on the results obtained from analysis, bifurcation able to withstand the operating condition was selected. The figures 25 and 26 below are the isometric view of modeled bifurcation for structural analysis and its meshing in ANSYS respectively.

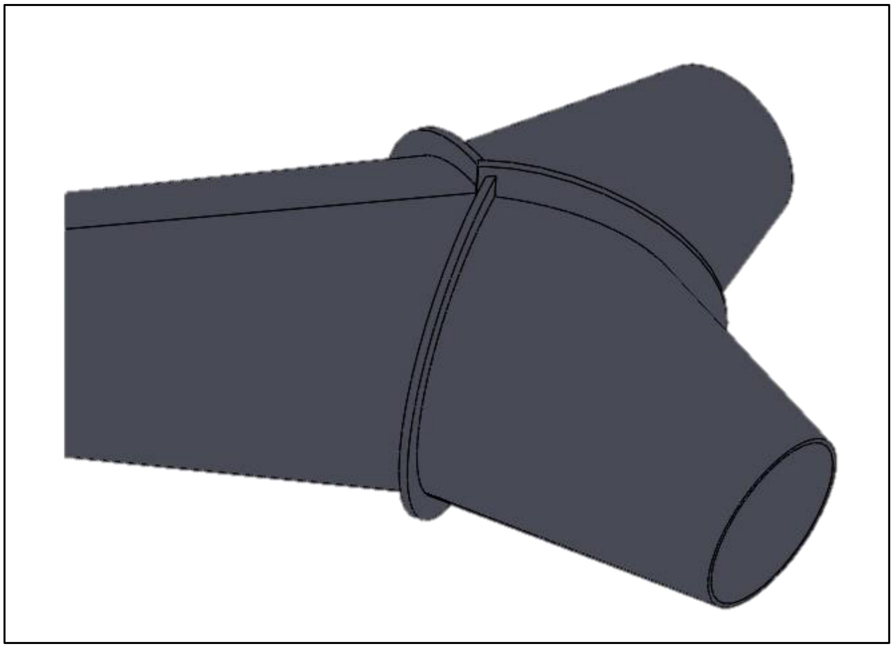

Figure 25: Model for Structural Analysis with pipe thickness of 25 mm

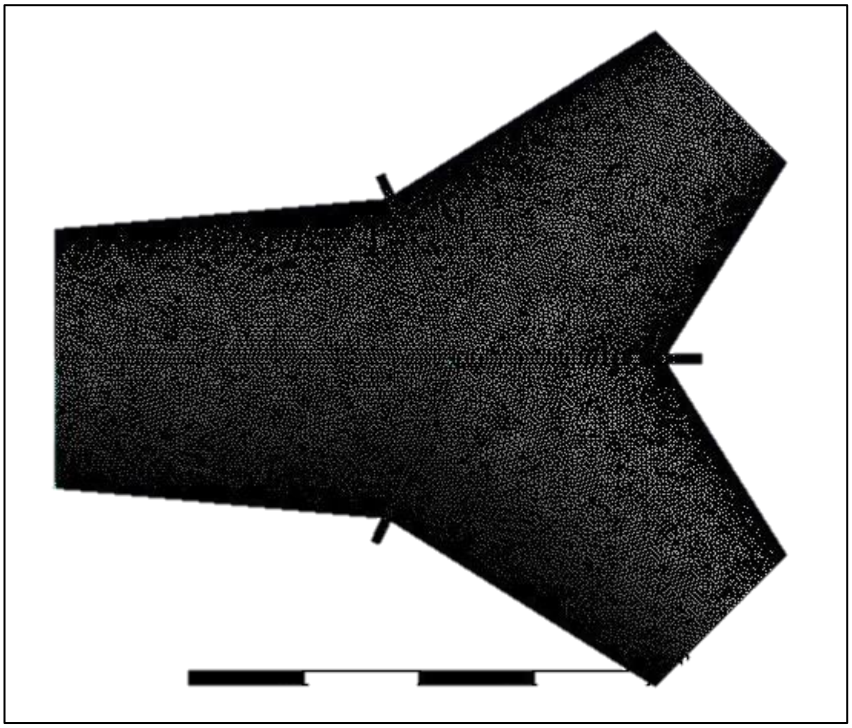

Figure 26: Meshing for Analysis in ANSYS STATIC STRUCTURAL

Upon simulation with the set-up conditions for structural analysis, the maximum stress of 290 MPa and factor of safety of 0.86 was obtained for pipe thickness of 25 mm and reinforcements of 50 mm; the results are illustrated in the figures 27 and 28 below.

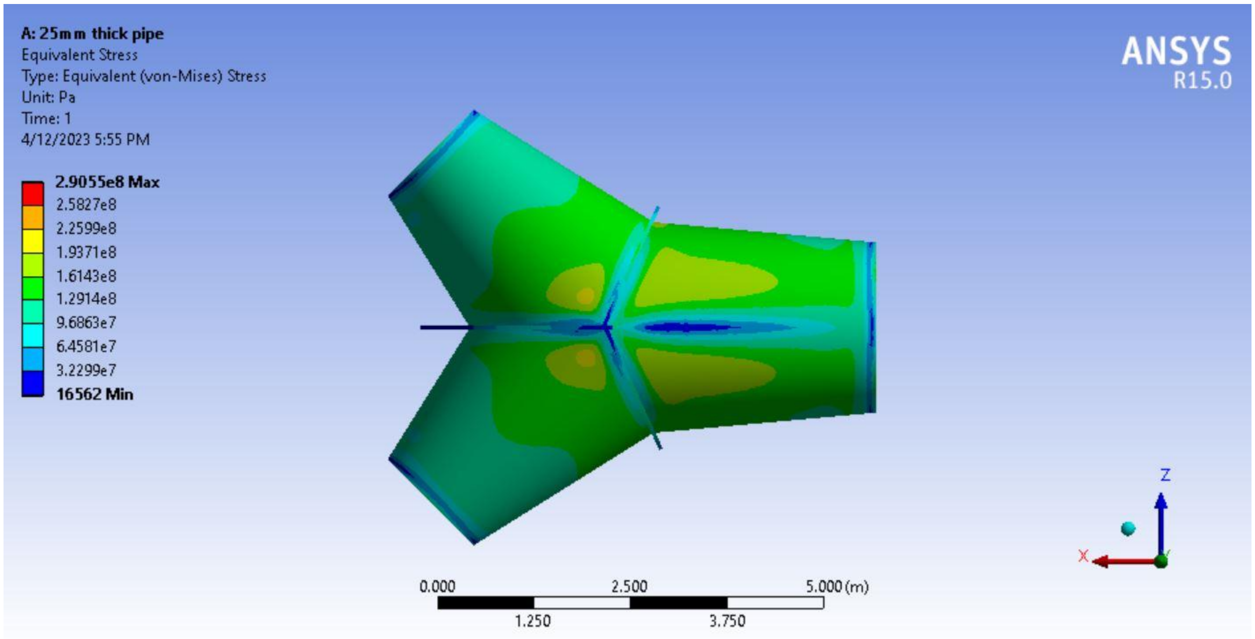

Figure 27: Maximum stress for 25 mm thick pipe and 50 mm reinforcements

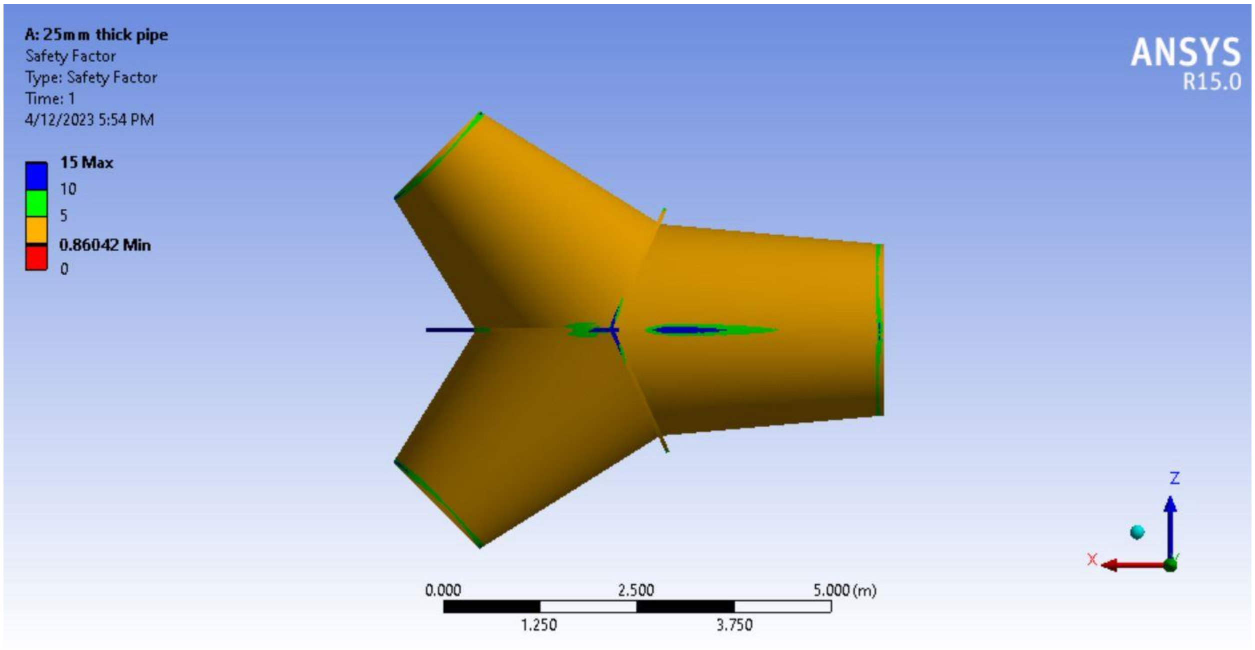

Figure 28: Minimum Factor of Safety for 25 mm thick pipe and 50 mm reinforcements

Similarly, with the same set-up conditions, 30 mm thick pipe for bifurcation and 50 mm thick reinforcements was modeled and analyzed and we got maximum stress in the branching to be 268 MPa and the minimum factor of safety to be 0.93 as illustrated in the figures 29 and 30 below.

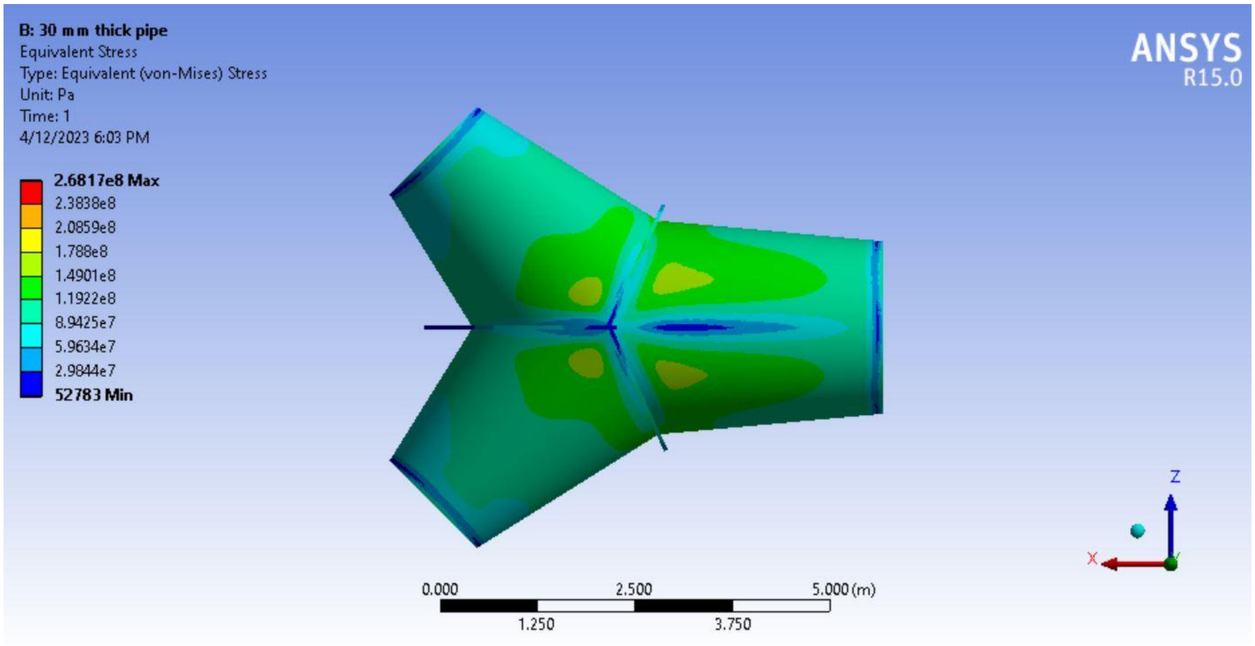

Figure 29: Maximum stress for 30 mm thick pipe and 50 mm reinforcements

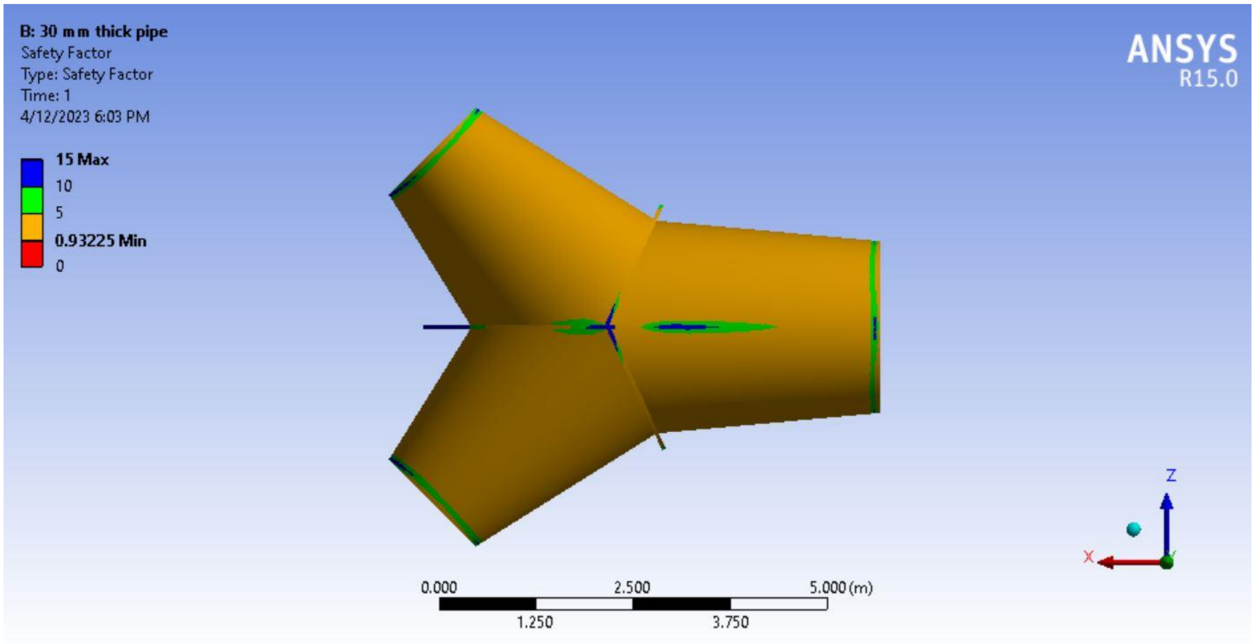

Figure 30: Minimum Factor of Safety for 30 mm thick pipe and 50 mm reinforcements

In the similar manner, the bifurcation was modeled with different pipe thickness and different reinforcements and its corresponding values of maximum stress and minimum factor of safety was noted till we got the stress in the acceptable range. the results from the simulation for different cases are as follows in table-7.

Table 7: Values of results obtained from structural analysis for various cases

S. N.	**Pipe Thickness (mm)**	**Reinforcements Thickness (mm)**	**Maximum Stress (MPa)**	**Minimum Factor of safety**
1	25	50	290	0.86
2	30	50	268	0.93
3	34	75	190	1.31
4	36	75	187	1.33
5	38	75	177	1.40
6	40	75	167	1.49

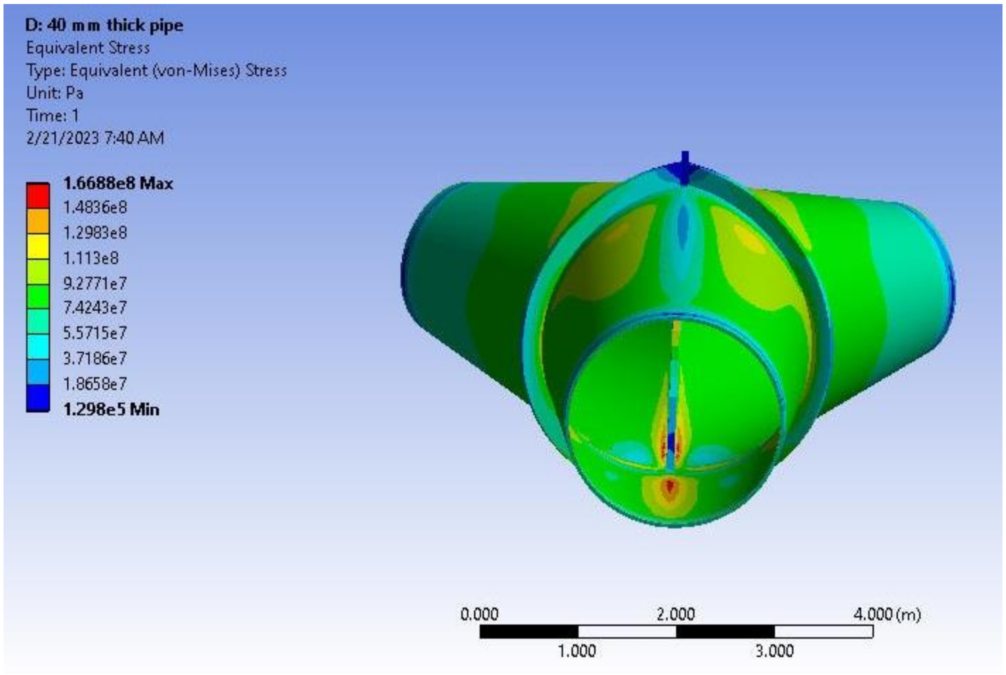

Figure 31: Maximum stress for 40 mm thick pipe with 75 mm reinforcements

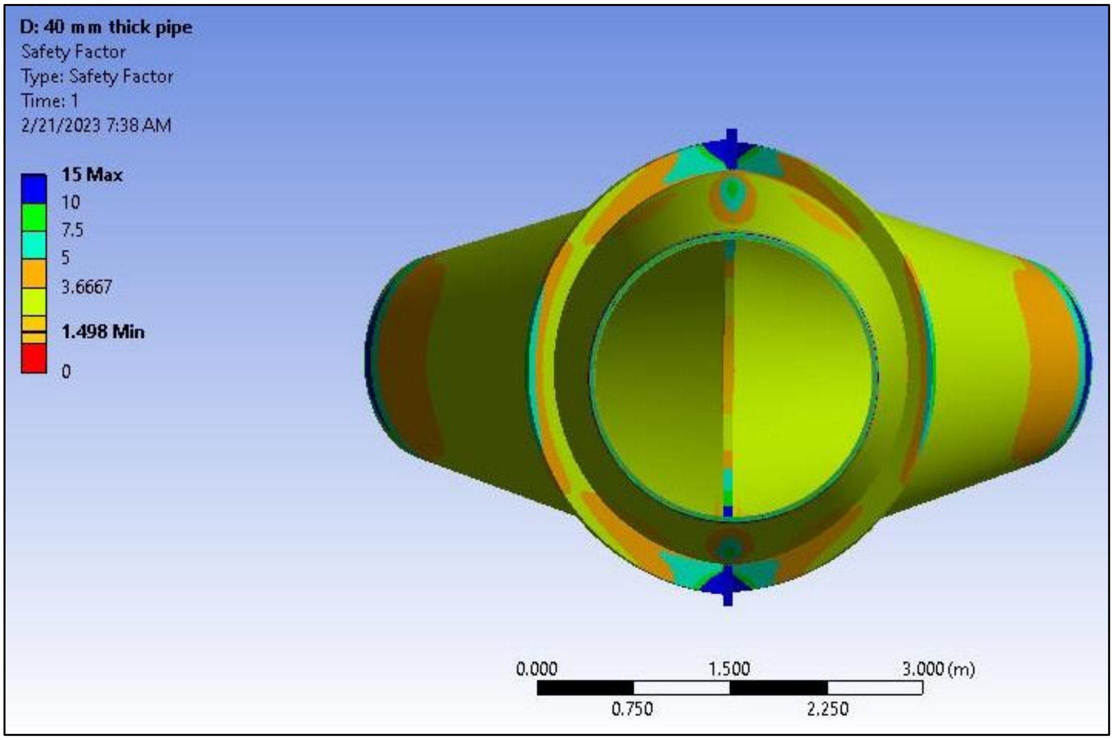

Figure 32: Minimum Factor of Safety for Bifurcation with pipe 40 mm thick

The results obtained from structural analysis show that, the maximum stress for the selected material is 167 MPa and the factor of safety is a minimum of 1.498 are shown in the figures 31 and 32 respectively, which is under acceptable range for the pipe thickness of 40 mm. Also, the thickness of reinforcements is 75 mm.

We can say that the values of maximum stress and minimum factor of safety is under the acceptable range because, as per the Indian Standard code of practice (IS 11639 part-2), when there is consideration of penstock pipe embedded inside the concrete structure, the values of stress in the steel pipes under normal operating condition without the consideration of concrete embedment should not exceed 90% of minimum yield stress or 2/3rd of the minimum ultimate tensile strength whichever is less. Also, for the chosen material E 250, the values of 90% of minimum yield stress is 273 MPa and 2/3rd of ultimate tensile strength is 207 MPa. So, the value of maximum allowable stress in the bifurcation under the acceptable range is 207 MPa (minimum of those two as discussed earlier) and the maximum stress obtained in the bifurcation region for the studied case for 40 mm thick pipe is 167 MPa which is less than the maximum allowable stress and thus it can be considered to be in the acceptable range. The datils of bifurcation with necessary reinforcements is shown in the figures 33, 34 and 35 below.

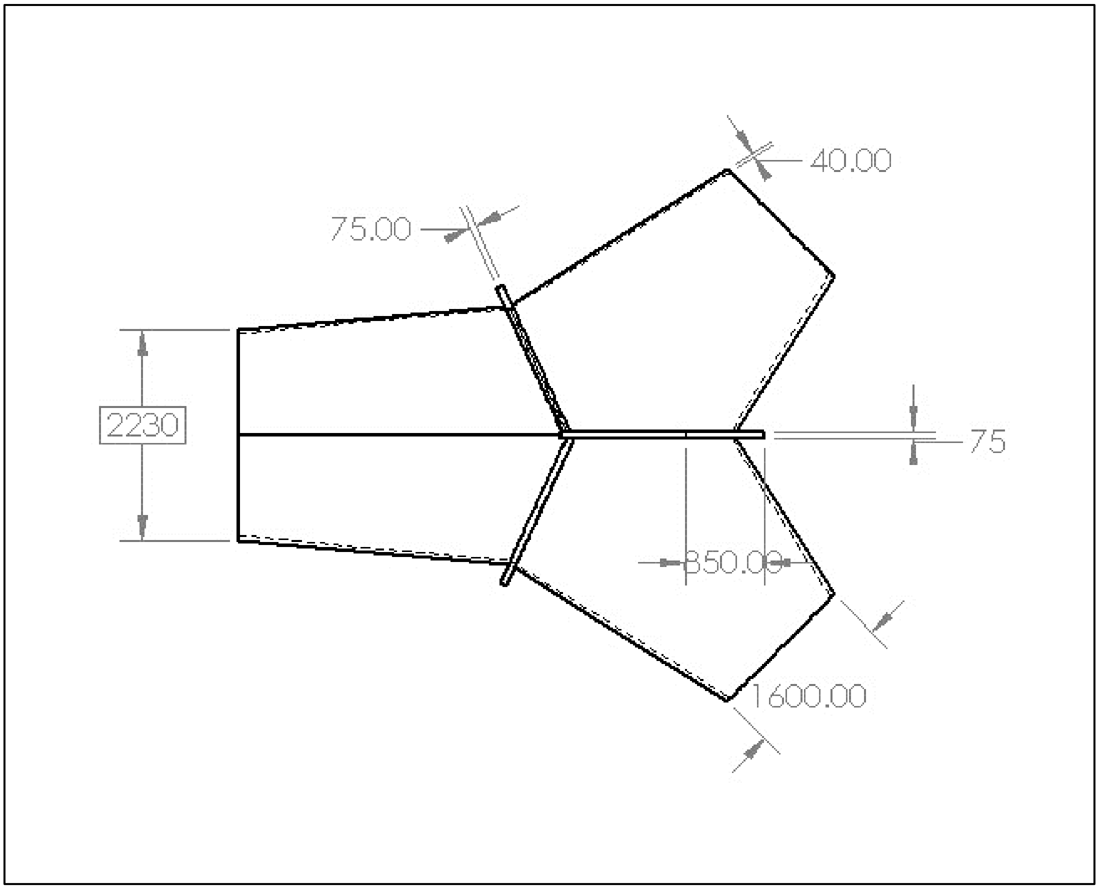

Figure 33: Details of bifurcation (All Dimension are in mm)

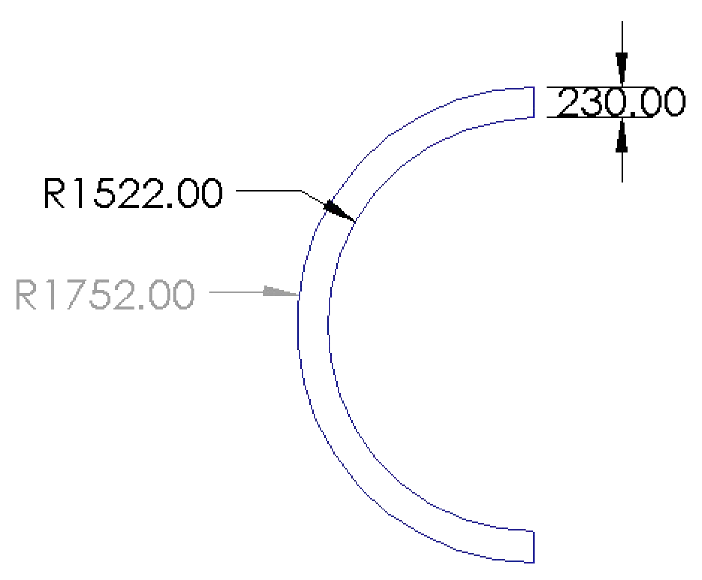

Figure 34: Details of Ring Girder (All Dimensions are in mm)

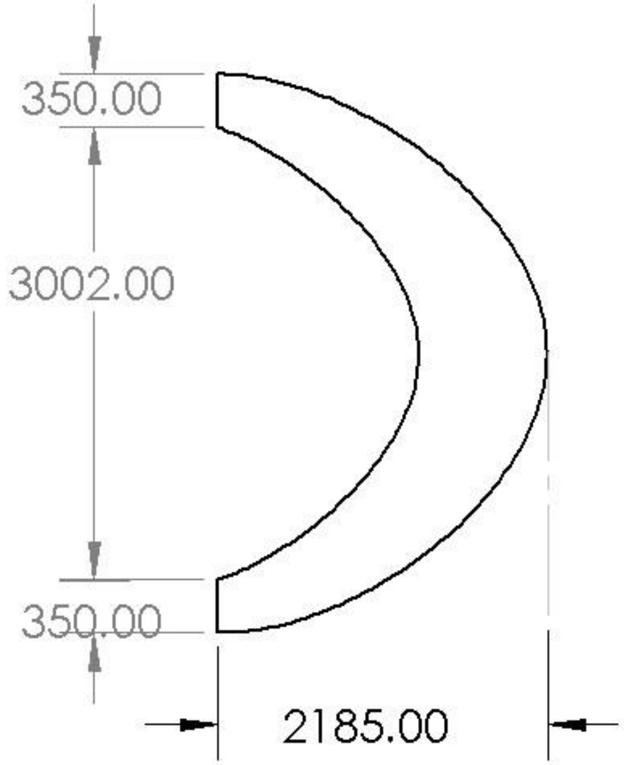

Figure 35: Details of Sickle (All Dimensions are in mm)

CHAPTER-5: CONCLUSION AND RECOMMENDATIONS

5.1 Conclusions

In this study, the taper angle of cone of bifurcation was increased starting from 3 degree to obtain the profile with minimum loss and then its structural analysis was carried out to know the necessary pipe thickness and reinforcements. The main findings are summarized as below.

a) Penstock bifurcation on given angle and cone length was modeled by varying the taper angle of cone by starting from 3 degree to 15 degrees increasing the angles by 2 degree.

b) CFD analysis was carried out for all the modeled cases in ANSYS CFX and the corresponding values for head loss was calculated and corresponding pressure and velocity distribution in the mid plane was observed.

c) The analysis of results showed that upon varying the taper angle of cone of bifurcation, the head loss decreases gradually, reaches minimum and then increase sharply. In this case, the value of head loss has decreased from 3 degree till 13-degree taper angle of cone, is minimum for 13 degrees (0.086 m) and has increased to 2.201 m for 15 degrees angle.

d) For the profile with minimum loss, branched pipe was modeled by varying the pipe thickness from 25 mm and its structural analysis was carried out to obtain the details of pipe thickness and necessary reinforcements so that the stress is within the acceptable limits. The material is chosen to be E 250 corresponding to IS 2062:2011.

e) The results from structural analysis shows that, the pipe with the material of E 250 with thickness of 40 mm and reinforcements of 75 mm thickness is suitable so that the maximum stress in the pipe with 167 MPa and minimum factor of safety to be 1.49.

5.2 Recommendations

The possible recommendations of the study can be summarized as follows

a) Results obtained from analytical study can be compared with the experimental results by performing the model tests.

b) Since only the portion of Bifurcation has been considered for the study, whole penstock including the manifold can be taken into consideration and head loss can be analyzed for the project.

REFERENCES

Adamkowski, A., 2001. Case STudy: Lapini Powerplant Penstock Failure. *Journal of Hydraulic Engineering*, 127(7), pp. 547-555.

Ahmed, S., 1965. *Head Loss in Symmetrical Bifurcations,* Vancouver: The University of British Columbia.

Bardan Dangi, T. R. B. A. B. T. B. C. S. B., 2022. *Numerical Analysis of Manifold: A case study of Phukot Karnali Hydroelectric Project.* Kathmandu, s.n.

Bipin Kandel, M. C. L., 2019. Computational Fluid Dynamics Analysis of Penstock Branching in Hydropower Project. *Journal of Advanced College of Engineering and Management,* Volume 5, pp. 37-43.

C. A. AGUIRRE, R. G. R. C., n.d. *HEAD LOSSES ANALYSIS IN SYMMETRICAL TRIFURCATIONS OF PENSTOCKS -HIGH PRESSURE PIPELINE SYSTEMS CFD,* s.l.: AGHEM.

Carlos Andres Aguirre, R. G. R. C. W. d. O. F. A., 2018. Numerical analysis for detecting head losses in trifurcations of high head in hydropower Plants. *Renewable Energy.*

Dipesh Thapa, M. C. L. T. R. B., 2016. *Flow Analysis and Structural Design of Penstock Bifurcation of Kulekhani III HEP.* Kathmandu, s.n.

Project, U. K. S. H. E., 2017. *Penstock Bifurcation Design,* Kerala: Kerala State Electricity Board Limited.

Ravi Koirala, S. C. H. P. N. B. C. B. T., 2017. Computational Design of Bifurcation: A Case Study of Darundi Khola Hydropower Project. *International Journal of Fluid Machinery and Systems,* 10(1).

Rumsey, C., 2021. *Menter Shear Stress Transport Model.* [Online] Available at: https://turbmodels.larc.nasa.gov/sst.html [Accessed 15 January 2023].

Seshu, P., 2012. *A Text Book of Finite Element Analysis.* New Delhi: PHI Learning Private Limited.

Simulating, i., 2020. *Turbulence models in CFD.* [Online] Available at: https://www.idealsimulations.com/resources/turbulence-models-in-cfd/ [Accessed 15 January 2023].

Singhal M.K., A. K., 2015. Optimum Design of Penstock for Hydro Projects. *International Journal of Energy and Power Engineering,* 4(4), pp. 216-226.

Standard, B. o. I., 1986. *Criteria for Hydraulic Design of Penstocks,* Delhi: Bureau of Indian Standard.

Standard, B. o. I., 2011. *Hot Rolled Medium and High Tensile Structural Steel- Specification (Seventh Revision),* New Delhi: Bureau of Indian Standard.

Standards, B. o. I., 2001. *Structural Design of Penstock-Criteria,* New Delhi: Bureau of Indian Standards.

Wang, H., 1967. *Head Losses Resulting from Flow Through Wyes and Manifolds,* Vancouver: The University of British Columbia.

Yunus A Cengel, C. M. C., 2013. *Fluid mechanics: fundamentals and applications.* 3rd ed. New York: McGraw-Hill.