Proceedings of 12th IOE Graduate Conference
Peer Reviewed
ISSN: 2350-8914 (Online), 2350-8906 (Print)
Year: 2022 Month: October Volume: 12

Flow Analysis in Asymmetric and Symmetric Bifurcation with Varied Layout: A Case Study of Daram Khola HEP

Prashant Neupane [a], Mahesh Chandra Luintel [b]

[a, b] *Department of Mechanical & Aerospace Engineering, Pulchowk Campus, IOE, Tribhuvan University, Nepal*
[a] 076msree012.prashant@ioe.edu.np, [b] mcluintel@ioe.edu.np

Abstract
A bifurcation is used whenever it is needed to divide the fluid flow into more than one turbines for power generation. Its design is conventionally done by using analytical techniques, design codes and guidelines. Nowadays with the advancement of computing devices, computational methods can be used for the design process for more accurate results. In this study, a case of Daram Khola HEP has been considered where the layout of bifurcation is mainly constrained by the geological arrangement of penstock and powerhouse orientation. Asymmetric bifurcation layouts with conventional design approach are developed, modeled and analyzed in ANSYS platform to determine head loss and flow distribution pattern in the branch pipes. These layouts are revised by incorporating symmetric bifurcation layout with angle of bifurcation 60° and adding a bend pipe just upstream of the bifurcation. Multiple layouts are proposed with change in upstream bend angle by 1° in each revision, ranging the bend angle from 24° to 32°. Flow simulation, analysis and head loss calculation is done for each layout and the results are compared. The difference in mass flow rate at the two outlets has decreased from 892.83 kg/s in the asymmetrical layout to 140.82 kg/s in the symmetric layout with bend angle 31°. The head loss in outlet 1 and outlet 2 of the asymmetrical layout are 154.52 mm and 571.51 mm respectively, while for the symmetric layout, the head loss is minimum for outlet 1 at a bend angle of 32° i.e. 223.30 mm and for outlet 2 at 24° i.e. 171.08 mm. Since the mass flow rate difference in the two outlets is minimum for bend angle 31° and head loss in the two outlets are also close to the lowest head loss for each outlets in the considered range, it is concluded to be the optimum layout.

Keywords
Bifurcation, Head loss, CFD, Turbulence

1. Introduction

Bifurcation is one of the critical parts of a hydropower project which contributes to head loss in the penstock manifold. The water flow pattern inside a bifurcation is complex due to geometries with varied cross sections and sudden change in flow direction. The behavior of water in such complexities cannot be easily predicted. Hence, special care and design considerations are required both hydraulically and structurally while working with a bifurcation [1]. The design and layout of a bifurcation are determined by the available head of water, flow rate, geological constraints and fabrication and economic constraints.

The head loss in the bifurcation decreases the net head available at the turbine inlet. Determination of such head loss by analytical methods is a tedious task. Although some empirical relations are available in the design codes in the form of head loss coefficients, they are applicable only on selected geometries having pre-specified values of angle of bifurcation. Hence, for analyzing the flow and determining the head loss along a bifurcation layout, which needs to be repeatedly revised, Computational Fluid Dynamics (CFD) approach of problem solution is a suitable method and is the current industrial practice.

The fundamental equations that govern the flow of incompressible fluids are equations 1 and 2.
Mass conservation equation (continuity equation)

$$\frac{\partial \rho}{\partial t} = \rho \nabla V \tag{1}$$

Momentum Conservation equation

$$\rho\left(\frac{DV}{Dt}\right) = \rho g + \nabla \cdot \tau'_{ij} - \nabla p \tag{2}$$

Empirical relations from experiments are available to determine the head loss in bifurcation for general cases. The Bureau of Indian standards (BIS) [2] has formulated one such equation to determine head loss which is given by equation 3

$$\Delta H = \alpha \frac{v_o^2}{2g} \tag{3}$$

Where,

$$\alpha = Head\,Loss\,Coefficient$$
$$v_o = mean\,velocity\,of\,flow\,in\,the\,main\,pipe$$

Value of α is influenced by branch angle, change in sectional area, flow distribution ratio and Reynolds number. Different experimental curves are available for different values of branch angle, flare angle, flow distribution ratio and Reynolds number[2]. Graphical representation from the Miller experiments and Munich test also provide an estimation of the head loss coefficient for a specified bend angle and flow ratio[3] . For multiple varied cases of the bifurcation layout, mathematical modeling and empirical relations are not much useful. Hence, a computational method can be utilized to determine the flow behavior in such complex cases[4]. Computational Fluid Dynamics (CFD) can model the flow conditions and best determine the flow behavior when provided with appropriate boundary conditions[5]. The flow at the bifurcation area around the junction of the branches is of complex pattern and very difficult to describe by general mathematical equations of differential or integral forms. In such case, the better alternative could be to conduct model analysis or the Computational fluid dynamics (CFD)analysis that will capture the flow pattern can give the best estimation of the flow parameters[6]. Finite element method of discretization divides the fluid domain into a number of discrete subdomains of tetrahedral or hexahedral elements, each of which is represented by a discrete set of points. The nodal parameters known at the boundary are known as a boundary conditions. The governing differential equations are converted into a system of algebraic equations valid at each of these discrete points. The coefficient matrix of the linear equations of each element is formed which is known as element stiffness matrix. The elemental stiffness matrixes of all elements are assembled to form global stiffness matrix. The matrix is solved to obtain the nodal parameter at each node. All of these tasks can be done with the help of the available CFD tools. ANSYS CFX and FLUENT are the strong CFD tools for modeling of the flow in any boundary conditions and flow load.[7].

Laminar and turbulent flow is characterized by the study of Reynolds' Number, which is defined as the ratio of inertial force to the viscous force. In real field, most of the fluid flow in an engineering design or research problem are turbulent. Among several techniques for solving the turbulence model, Direct Numerical Simulation (DNS), Scale Resolving Simulation (SRS), Large Eddy Simulation (LES) Reynolds Averaged Navier-Stokes Simulations (RANS) are some major techniques, each having their associated accuracy and computational time and cost as a tradeoff. DNS and SRS are more demanding and complex for the problem considered.

The K-ε Turbulence Model can be used to model the turbulent flow in pipe with bifurcation. The realizable k-ε model provided results closest to the experimental method while performing comparative similarity study on hydraulic losses of a Y-bifurcation[8]. With this model, the turbulent flow is characterized by 3 mean fields: the mean velocity V, the turbulence kinetic energy K and the dissipation rate ε. This model is valid in turbulent areas[9]. Two partial differential equations (transport equations) are solved in this type of turbulence model: the turbulent kinetic energy k and the turbulence eddy dissipation ε (i.e., the rate at which the turbulent kinetic energy dissipates)[10]. It can also be stated as the simplest turbulence model for which only initial and/or boundary conditions needs to be supplied. Hence, it is less computationally expensive and properly model the flow in free stream region.

Thapa[7] has applied CFD and FEM for the design and structural analysis of penstock bifurcation for the manifold of Kulekhani-III Hydropower Project. The flow pattern and head loss were reviewed and the manifold arrangement was revised multiple times to achieve better geometry. The loss coefficient for bifurcation has reduced from 0.44 to 0.21 with such revisions in the bifurcation geometry. A computational research has been carried out by Kandel and Luitel[11], to determine the most efficient branching manifold which has three units of turbine. More than 20 models were prepared to tested through simulation, to minimize the head loss and mass flow variation in the 3 different units. The research finally concluded to go towards a single trifurcation instead of successive two bifurcation or individual branching form main branches. R. Saheed and H. Gildeh[10]

compared the performance of standard k-ε and realizable k-ε turbulence model in curved and confluent channels to conclude that standard k-ε performed better in the curved channel and the realizable k-ε model performed better in the confluent channel. Dangi[12] performed numerical analysis in manifold of Phukot Karnali Hydroelectric Project. Initially, branching angle in the manifold was designed to be 10°. The optimized profile was created by combining best branch angle, best cone length and sickle plate. The head loss at outlet-1, outlet-2 and outlet-3 for the optimized profile were computed as 0.13 m, 0.46 m and 0.31 m, respectively which were were 37%, 15% and 24% less as compared to the base case.
The flow analysis is done in this study with the help of CFD, mainly focusing the flow difference in two outlets and the head loss for the two outlets of the bifurcation and compare the results while using and not using the upstream bend .

2. Methodology

The research methodology followed for this study is shown in figure 1.

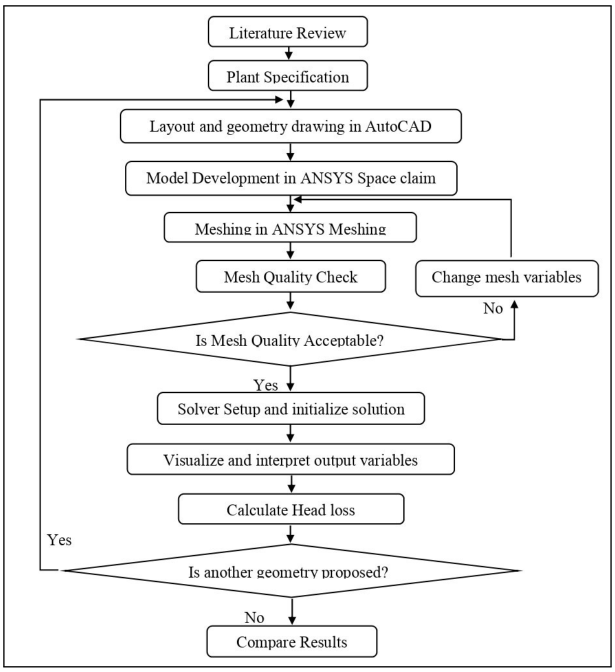

Figure 1: Methodology

The fluid flow simulation was done by performing the simulation in ANSYS platform. First of all, the geographical constraints of the site were studied.

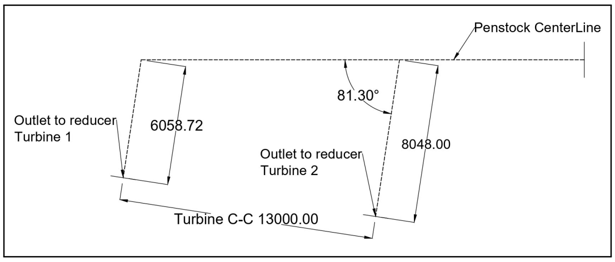

Figure 2: Geographical constraints of site

A layout was developed totally guided by these constraints, having an asymmetric bifurcation with angle of bifurcation 81°. The layout drawing was done in AutoCAD and 3D model was developed in ANSYS SpaceClaim. Mesh was generated in the fluid domain in ANSYS meshing. Tetrahedral mesh elements were used for simplicity. Inflation layers were generated in the near the wall areas so that the actual fluid behavior near the wall are properly calculated and errors are minimized. The quality of the mesh generated was checked by monitoring skewness and orthogonal quality. The maximum value of skewness was kept below 0.90 and the mesh exceeding that value was refined by further processing and change in mesh sizing, inflation layers, etc. The first layer thickness of the inflation layer was given to be 2mm and a total of 10 layers with a growth rate of 1.2 were adopted. The setup was carried out in ANSYS Fluent and steady state pressure based solver was used. Other relevant parametres are presented in table 1.

Table 1: Setup Parameters

Platform	ANSYS Fluent
Fluid Domain Properties	Water (Liquid)
Solver	Steady State, Pressure based
Turbulence Model	k- ε Turbulence Model
Wall Roughness Height	0.1mm
Residual	0.00001
Iterations	Enough Until Convergence
Solution Scheme	SIMPLE

K-ε turbulence model was used to model the flow of turbulence in the fluid domain. Standard wall function was incorporated with the K-ε turbulence model. The wall roughness height of 0.1mm was selected to account for friction loss. The boundary condition at inlet is mass flow inlet equal to 10320kg/s, and the boundary conditions at both the outlets is pressure outlet to the atmosphere. No slip wall boundary condition is provided for the wall. The fluid domain was defined as water and a residual value of 0.00001 was adopted. Hybrid initialization was done and the iterations were run until convergence to the specified residual value. The variables such as Pressure, velocity, turbulence, mass flow rate etc. are monitored to analyze the flow of fluid. Mass flow rate is calculated at the two outlets and compared. The sum of mass flow rate at the two outlets was compared with the mass flow rate at inlet to determine flux imbalance. Area weighted average pressure and area weighted average velocity at inlet and outlets were calculated (result provided by ANSYS) to determine the head loss in the bifurcation layout. Different contours of variables like pressure, velocity, turbulence etc. with color mappings were analyzed to determine the flow behavior and consider the need for revision in the layout and geometry of the bifurcation. After analyzing the results obtained from the simulation of first layout, a revision is made and another asymmetric bifurcation with angle of bifurcation 45° is developed refereeing to previous literature. The position of bifurcation was shifted upstream by 3185mm to conform to the geographic constraints. The whole process from developing the 3D model to analyzing the results is performed with same boundary condition for the revised layout. The layout is further revised to incorporate a symmetric bifurcation. A bend is introduced in the inlet pipe just upstream of the bifurcation. The angle of the upstream bend was measured to be 28.23° while placing a bifurcation of angle 60° downstream to it. It was then rounded off to 28° for simplicity and further layouts were developed by changing the bend angle by 1° in each revision, both in increasing and decreasing direction, keeping other geographical constraints and the angle of bifurcation constant. Fluid flow simulation and analysis was carried out for each layout. The bend angle was decreased upto 24° and halted there. On the other side, the bend angle was increased upto 32° and halted. Results for each layout were calculated and compared.

3. Simulation Results

3.1 Mesh Independence Test

Mesh independence test was carried out to ensure that the result obtained from the simulation does not depend on the mesh size. It was performed in the symmetric layout with a bend angle of 24°. The mesh size is decreased from 400 mm to 50 mm in steps as shown in table 2. The mesh size of 60mm was chosen to be used further in this study since the outputs which are inlet pressure and mass flow rate at outlet one are stable in that range. The values of inlet pressure and mass flow rate at outlet 1 for different mesh sizes are presented in table 2 and the graphical representations are in figure 3 and figure 4.

Table 2: Mesh independence test results

Mesh Size (mm)	Inlet Pressure (Pa)	Mass Flow rate at outlet 1 (kg/s)
400	107569.87	4896.1831
300	105650.05	4949.6271
200	104017.13	5045.2255
100	103387.91	5061.6505
90	103368.53	5062.5522
80	103377.05	5064.7138
70	103367.05	5063.2377
60	103358.65	5060.8665
50	103366.91	5061.4394

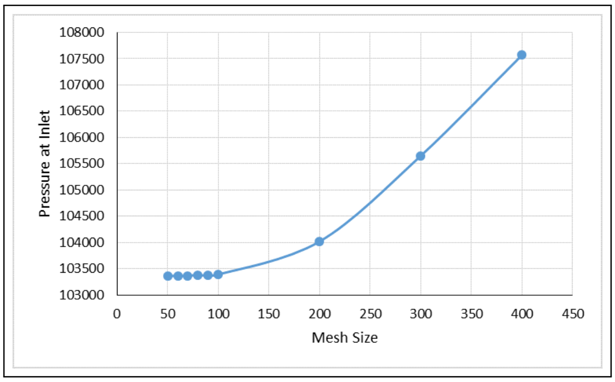

Figure 3: Variation of inlet pressure with mesh size

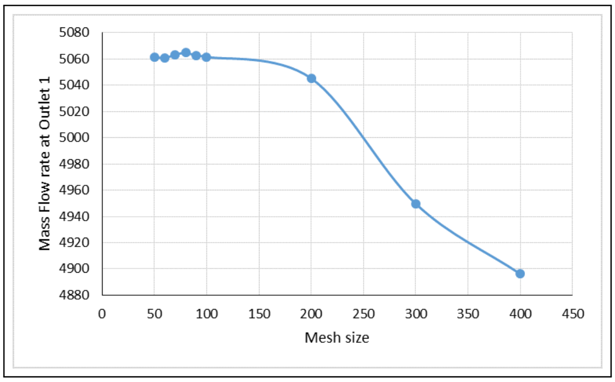

Figure 4: Variation of flow rate with mesh size

3.2 Flow Analysis

3.2.1 Asymmetric Layout 1

The layout developed is totally guided by the geographical constraints. The pressure, velocity and turbulence contour plots are shown in figures 6, 7 and 8 respectively.

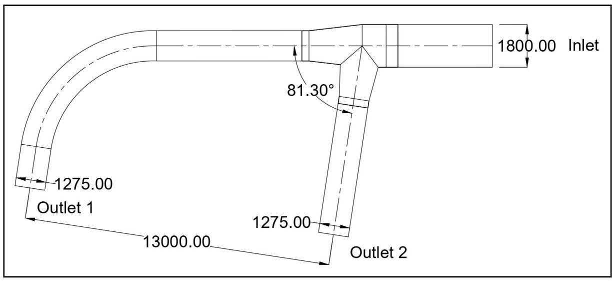

Figure 5: Asymmetric Layout 1

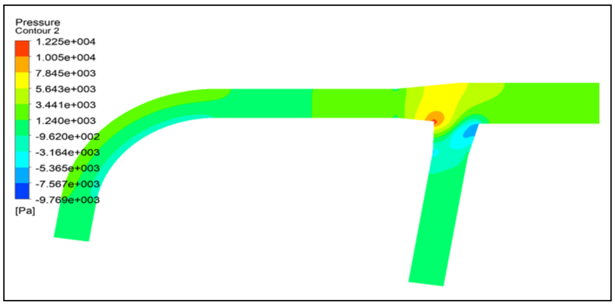

Figure 6: Pressure distribution at midplane

It was observed that there is high pressure accumulation in the crotch region of bifurcation and low pressure region was formed at the intersecting point of branch pipe and the inlet pipe.

Figure 7: Velocity distribution at midplane

Low velocity region is observed at the crotch region and another low velocity region is developed in the intersecting region of the angled branch pipe and inlet pipe.

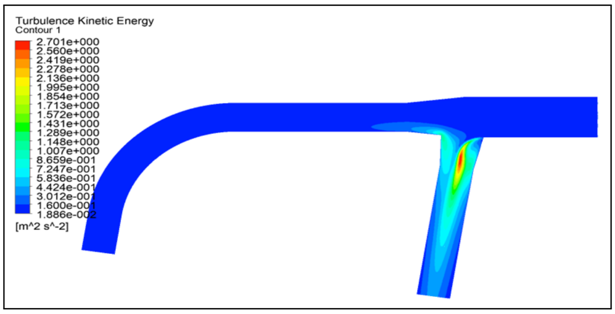

Figure 8: Turbulence(k) distribution at midplane

A significant region with high turbulent kinetic energy is observed in the branch which is at an angle with the inlet pipe. The large value of turbulent kinetic energy gets dissipated when the eddies interact and the viscous force converts kinetic energy into heat. This results in head loss.

Table 3: Results from simulation

Parameter	Inlet	Outlet 1	Outlet 2
Mass flow rate (kg/s)	10320	5606.41	4713.58
Pressure (Pa)	104637	101325	101325
Velocity (m/s)	4.0649	4.4184	3.7028
Head Loss (mm)	-	154.52	571.51

3.2.2 Asymmetric Layout 2

Another asymmetric layout is developed decreasing the angle of bifurcation to 45 degree and shifting it upstream.

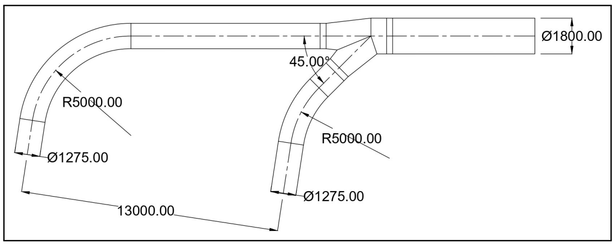

Figure 9: Asymmetric Layout 2

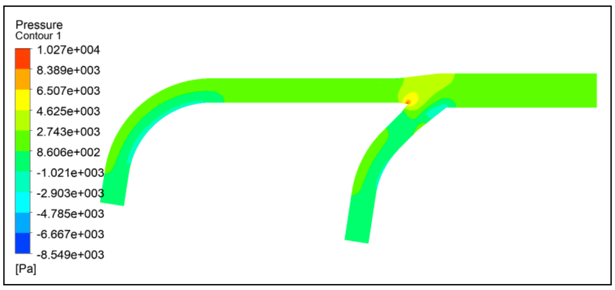

Figure 10: Pressure distribution at mid-plane

Figure 11: Velocity distribution at mid-plane

High pressure at crotch of bifurcation and low pressure at the starting of angled branch pipe was still observed.The most notable low velocity region was observed at the starting of angled branch pipe.

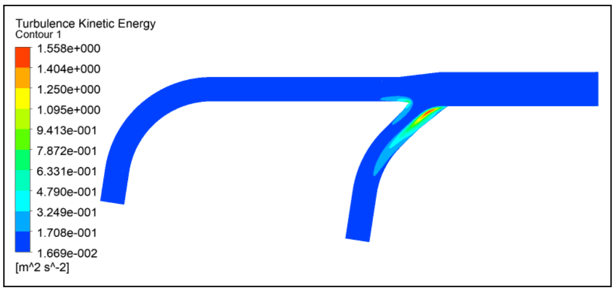

Figure 12: Turbulence distribution at mid-plane

A significant region with high turbulent kinetic energy was observed similar to layout 1 but with less intensity.

Table 4: Results from simulation

Parameter	Inlet	Outlet 1	Outlet 2
Mass flow rate (kg/s)	10320	5179.04	5140.95
Pressure (Pa)	103808	101325	101325
Velocity (m/s)	4.0658	4.0901	3.7028
Head Loss (mm)	-	236.86	263.58

3.2.3 Symmetric Layout

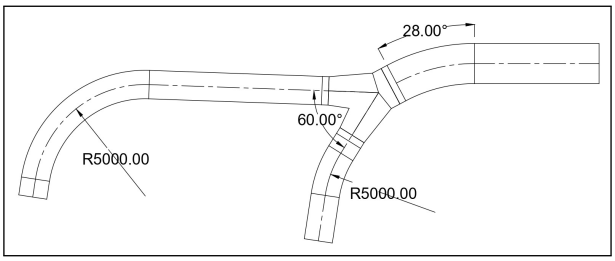

Figure 13: Symmetric layout (Bend angle 28 degree)

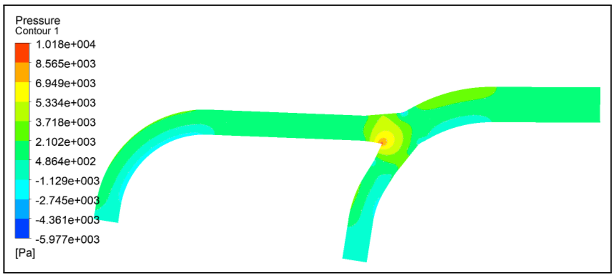

Figure 14: Pressure Distribution at mid-plane

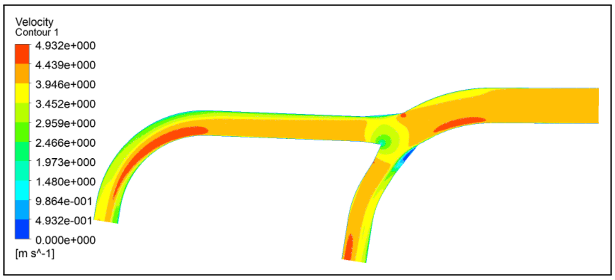

Figure 15: Velocity Distribution at mid-plane

High pressure region is still observed in the crotch of bifurcation and the pressure is distributed more or less uniformly in the domain with some small low pressure regions at the inner curvature of bends. High velocity is observed in those inner side of bends.

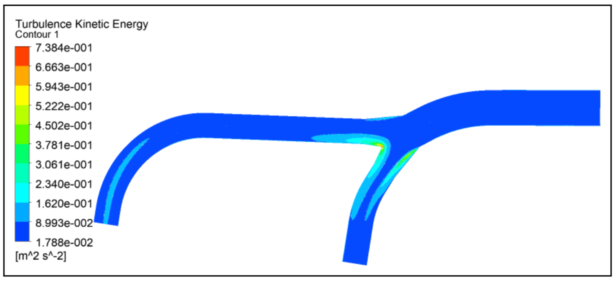

Figure 16: Turbulence Distribution at mid-plane

The Turbulent kinetic energy is high at the crotch which is the main area of head loss in this case. Another high turbulence is seen in the branch pipe towards outlet 2. The computed values of variables and calculated head loss are presented in table 5.

Table 5: Results from simulation

Parameter	Inlet	Outlet 1	Outlet 2
Mass flow rate (kg/s)	10320	5079.63	5240.37
Pressure (Pa)	103332	101325	101325
Velocity (m/s)	4.0659	4.0002	4.1204
Head Loss (mm)	-	235.86	174.64

The symmetric layout with bend angle 28° was successively revised by changing the bend angle by 1° in each revision up to 24° at lowest and 32° at highest. Other geometrical correction in the bend angle of branch pipes were done according to requirement. The angle of bifurcation was kept constant at 60°. Simulation was performed in each case and the results were visualized as well as the head loss was calculate for each case. The pressure and velocity contours similar to that of the 28° bend layout and were not distinct visually on normal observation. The values of different parameters computed and head loss calculated for each layout are presented in the table. The 2D drawing of the extremities cases are shown in figure 16 and figure 17

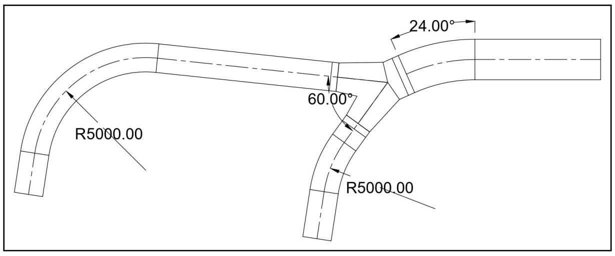

Figure 17: Symmetric Layout (decreased bend angle)

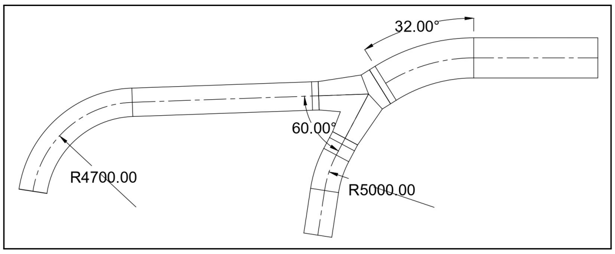

Figure 18: Symmetric Layout (increased bend angle)

Table 6: Mass flow rate for different bend angles

	Mass Flow Rate(kg/s)		
Angle	Outlet 1	Outlet 2	Difference
24	5060.867	5259.134	198.267
25	5069.738	5250.262	180.523
26	5072.401	5247.599	175.198
27	5078.851	5241.149	162.297
28	5079.626	5240.374	160.748
29	5082.402	5237.598	155.195
30	5087.489	5232.511	145.022
31	5089.586	5230.414	140.826
32	5088.526	5231.474	142.947

Table 7: Velocity at different bend angles

	Velocity		
Angle	Inlet	Outlet 1	Outlet 2
24	4.06828	3.98293	4.13499
25	4.06591	3.99039	4.12795
26	4.06591	3.99296	4.12593
27	4.06591	3.9988	4.12087
28	4.06591	4.0002	4.12038
29	4.06591	4.00362	4.11835
30	4.06591	4.01034	4.11446
31	4.06591	4.01129	4.11288
32	4.06591	4.02327	4.11385

Table 8: Pressure and head loss at different bend angles

	Pressure	Head Loss	
Angle	Inlet	Outlet_1	Outlet_2
24	103358.65	248.922	171.081
25	103358.04	243.826	173.497
26	103348.19	241.418	173.52
27	103338.48	237.283	175.031
28	103332.01	235.866	174.637
29	103316.40	232.428	174.098
30	103314.51	228.681	175.813
31	103316.38	228.372	176.769
32	103328.35	223.309	177.47

The Variation in different output parameters monitored with the change in bend angle are plotted in graphs.

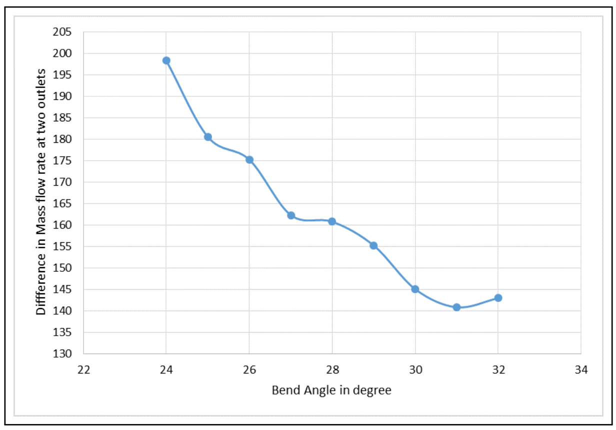

Figure 19: Change in flow imbalance with bend angle

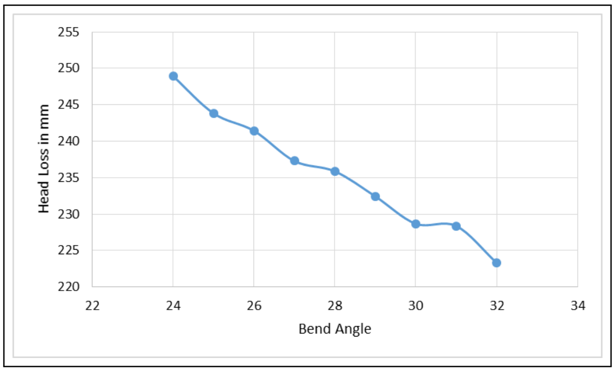

Figure 20: Head loss v/s bend angle(Outlet 1)

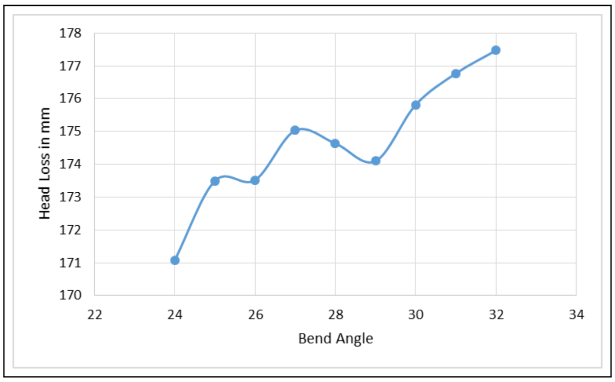

Figure 21: Head loss v/s bend angle(Outlet 2)

While comparing the head loss for two outlets, it was observed that the head loss for outlet 1 decreased with the increase in bend angle and that for outlet 2 increased with the increase in bend angle.

4. Conclusion

It was observed that the head loss due to bifurcation was reduced while using a symmetric bifurcation with upstream bend while compared to the asymmetric bifurcation. While changing the bend angle, the head loss in one outlet decreased (outlet 1) while increasing the bend angle and that in another outlet increased (outlet 2) while doing the same. However, the change amount was low. The best possible flow distribution was observed at a bend angle of 31° and the head loss in the two outlets are also close to the lowest head loss in the considered range. Hence, it is concluded to be the optimum angle. While comparing the head loss of asymmetric layout (45 ° bifurcation) and optimum symmetric layout (31 ° bend) the head loss at outlet 1 has decreased by 3.6% and the head loss at outlet 2 has decreased by 32.9%.

5. Acknowledgement

Authors would like to acknowledge Daram Khola Hydropower Company and Units Engineering and Design Consultancy for providing base data of Daram Khola Hydroelectric Project. The authors would also like to thank Department of Mechanical and Aerospace Engineering of Pulchowk campus for necessary support.

References

[1] Ravi Koirala, Sailesh Chitrakar, Hari Prasad Neopane, Balendra Chhetri, and Bhola Thapa. Computational Design of Bifurcation: A Case Study of Darundi Khola Hydropower Project Ravi. *International Journal of Fluid Machinery and Systems*, 10(1):1–8, 2017.

[2] BIS. Penstock Branch Design Manual. Technical Report November, 2009.

[3] D.S. Miller. *Internal Flow Systems 2nd Edition*. 1990.

[4] Ms Kasturi Sukhapure, Mr Alan Burns, Mr Tariq Mahmud, and Mr Jake Spooner. CFD Modelling and Validation of Pipe Bifurcations. *13th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics*, pages 489–495, 2017.

[5] H K Versteeg and W Malalasekera. *An Introduction to Computational Fluid Dynamics*, volume 6. 2005.

[6] Sourav Dhungana. Flow analysis in eccentric bucket of Micro Pelton turbine: Multiphase modelling with transient state condition. 2507(February):1–9, 2020.

[7] Dipesh Thapa, Mahesh Chandra Luintel, and Tri Ratna Bajracharaya. Flow Analysis and Structural Design of Penstock Bifurcation of Kulekhani III HEP. (May):271–276, 2016.

[8] U Lasminto and R Klasinc. Comparative Similarity Study on Hydraulic Losses of a Y-bifurcation. (June 2014), 2016.

[9] M Gisselbrecht and Emmanuel Plaut. High Reynolds number K - epsilon model of turbulent pipe flows with standard wall laws : first quantitative results. 2017.

[10] Rawaa Shaheed, Abdolmajid Mohammadian, and Hossein Kheirkhah Gildeh. A comparison of standard k–ε and realizable k–ε turbulence models in curved and confluent channels. *Environmental Fluid Mechanics*, 19(2):543–568, 2019.

[11] Bipin Kandel and Mahesh Chandra Luitel. Computational Fluid Dynamics Analysis of Penstock Branching in Hydropower Project. *Journal of Advanced College of Engineering and Management*, 5:37–43, 2019.

[12] Bardan Dangi, Tri Ratna Bajracharya, and Ashesh Babu Timilsina. Numerical Analysis of Manifold : A Case Study of Phukot Karnali Hydroelectric Project. (July), 2022.