Proceedings of IOE Graduate Conference, 2016
pp. 271–276

Flow Analysis and Structural Design of Penstock Bifurcation of Kulekhani III HEP

Dipesh Thapa [1], Mahesh Chandra Luintel [2], Tri Ratna Bajracharaya [3]

[1, 2, 3] *Department of Mechanical Engineering, Pulchowk Campus, Institute of Engineering, Tribhuvan University, Nepal*
Corresponding Emails: [1] dipesh.tac@gmail.com, [2] mcluintel@ioe.edu.np

Abstract

The development of hydropower is possible with the development of sound knowledge about all aspects. Various guidelines are available for the design of hydropower plants which are based on experience and theoretical basis. With the development of modern techniques and availability of enhance computational devices, various problems can be solved using these techniques. One of these areas is the design optimization of penstock manifold and bifurcation. With the proper design of penstock bifurcation, the head loss incurring in the mixed flow condition can be minimized the output from both the units can be maximized. The conventional technique of design based on codes results in high thickness and overall increase in material quantity. The structural design can be optimized using Finite Element Method by accurately determining the three dimensional stress condition. Application of Computational Fluid Dynamics and Finite element analysis in the field of hydropower projects is the current industrial practice. However, it has found very limited use in context of Nepal. The research aims to enhance the theoretical knowledge base for the application of Computational Fluid Dynamics and Finite Element Method for the design and analysis of penstock bifurcation. The manifold arrangement of Kulekhani-III Hydropower Project was chosen for the optimization. The proposed manifold arrangement was modelled and flow analysis was performed. The flow and head loss were reviewed and the manifold arrangement was revised successively to achieve acceptable geometry. The bifurcation was given thickness and reinforcements and the solid model for the same was prepared which was then subjected to Finite Element Analysis. The result of stress and deformation was observed and checked against prevailing design codes. Finally the acceptable design of bifurcation was recommended for fabrication and installation.

Keywords

Computational Fluid Dynamics, Bifurcation, Finite Element Analysis

1. Introduction

Penstock is the pressure conduit between the turbine inlet valve and the first open water upstream from the turbine. The open water can be a surge tank, forebay or a reservoir. The penstock is mostly made up of welded carbon steel. In some low head applications, HDPE pipes are used for this purpose. Penstocks should be optimized with respect to the head loss and the material requirement.In the hydropower plant, a single generating unit is seldom chosen. The turbines and generators needs periodic repair and maintenance. The shut down time required for maintenance purpose is the time the generation will be lost. In case of single unit, the plant generation loss for the maintenance will be huge. Hence, most of the plant will have at least two generating units. In many cases the number of units are optimized based on the transportation limitation.So when there are more than one generating units, each one of them will be required to be feed up by penstock.

Unless the head is very low, it is not economical to use separate penstock for each units. So mostly a single penstock will carry water from free water surface near the powerhouse. Then it will be branched depending upon the number of units. When there is two generating units, the penstock is branched into two segments. This branching is called penstock bifurcation.

The profile of the manifolds affects the loss in the available water head significantly. This loss can decrease the potential plant capacity. The profile selection process can be done either by experimental analysis on reduced scale manifolds model test at lab or by numerical modeling of the fluid flow [1] [2]. The former option is rather expensive and may not be feasible every time. It is preferable to select best profile by tuning it with CFD solver and then follow reduced scale model test for the confirmation of flow parameters.

The flow analysis through pipe under pressure is simple and can be described by the one dimensional and two dimensional flow equations precisely. But the flow near the junction of the branches is difficult or some time impossible to describe by the closed form mathematical solution. In such case either model analysis will capture the flow pattern or the Computational fluid dynamics (CFD) can best model for the flow. Finite element model of the control volume just replaces the water volume by the discrete tetrahedral or hexahedral elements. The flow parameters are assigned to each node. The nodal parameters known at the boundary are known as a boundary conditions. Mathematically this process converts the flow differential equations by the set of simultaneous linear equations for each fluid element. The coefficient matrix of the linear equations of each element is well known by the 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.

The result validation can be done by doing experimental analysis. Another way to validate the result is to compare the result with the result of similar experimental research.While doing so, exact values cannot be compared. So, the comparision can be done in terms of some coefficient calculated for both experimental model and the computational model. Therefore, the loss coefficient of the experimental model and the loss coefficient of computation model shall be compared for validation.

2. Methodology

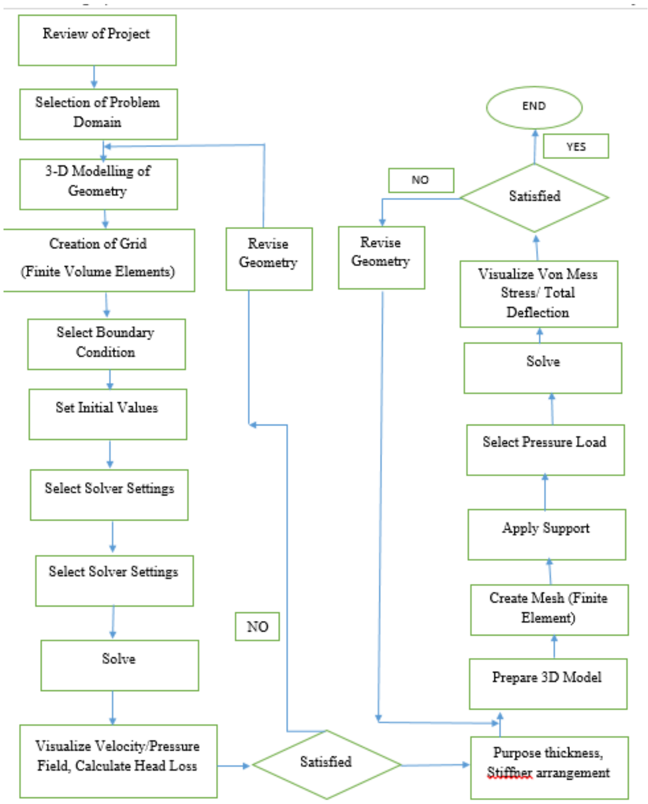


Figure 1: General Methodology

Computational Fluid Dynamics shall be used for the analysis of the velocity and pressure distribution in penstock pipes with branches. The velocity and pressure distribution can be used as criteria for choosing the best option to get maximum possible efficiency. The option with minimum loss or maximum discharge carrying capacity with same head loss should be selected for the recommendation. Following steps shall be followed for the hydraulic analysis of the bifurcation:

1. The geometry of the bifurcation will be purposed and the flow field is calculated for it.
2. The result of pressure and velocity distribution as well as calculation of head loss will be studied.
3. Improvement in the geometry is purposed and step 2 is repeated till an acceptable geometry is achieved.

K-E turbulence model shall be used to model the flow turbulence. A turbulence intensity of 0.02 to 0.05 had been examined and had been assumed which is fair for such flow condition [3]. High resolution solver option with convergence criteria of 10e-4 shall be selected.

The selected geometry will be recommended for structural design. Analysis of bifurcation geometry needs to be carried out in order to check its structural capacity to withstand the given loading condition. This can be performed using conventional analytical method. However, due to the complicated geometry of the bifurcation, this method does not yields accurate result. So, to optimize the design works, finite element method needs to be employed to calculate the structural stress of the bifurcation in the given condition.Following methodology shall be employed for structural analysis:

1. Creation of solid model of the bifurcation with an initial guess of all thickness and sizes.
2. Creation of mesh based on this solid model.
3. Application of pressure load, Nominal pressure @ t=0 s and gradually increasing upto upsurge pressure from t=0 to t=3 s to the inner wall of the bifurcation.
4. Calculation of equivalent von misses stress.
5. Reviewing of the von miss stress to check if it is within limit or not.
6. Review of overall factor of safety.
7. Change the geometry, if required.
8. Repeat steps 2 to 8, if required.

3. Flow Analysis

Option 1 is the option initially purposed by the project.

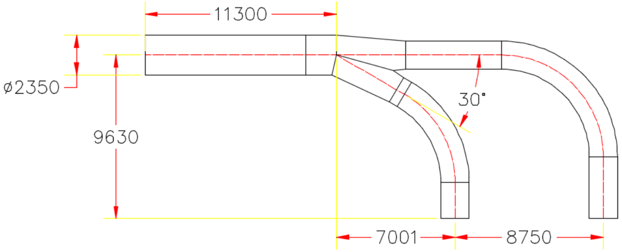


Figure 2: Geometry Proposed by Client(insert source)

The manifold lies in the single plane and hence the dimensions shown are true measurements. However, the bifurcation bend angle purposed is 45degrees which is slightly high [**?**](E.Mosonyi, 1991). As there are plenty of space available in the penstock alignment, the bifurcation is purposed to be shifted towards upstream side in order to reduce the branching angle to 30 degrees. It is well known to us (E.Mosonyi, 1991) that this will improve the flow behavior significantly. The requirement of increased structural strength will be well justified by the savings in the head loss and improvement of flow behavior.

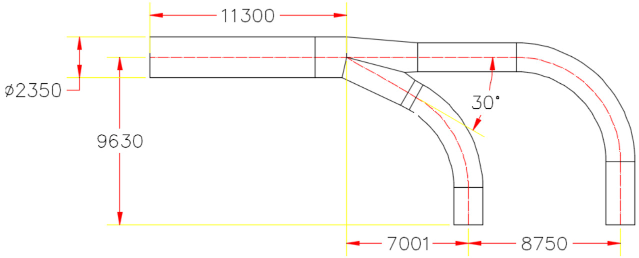


Figure 3: Geometry of Opt-2

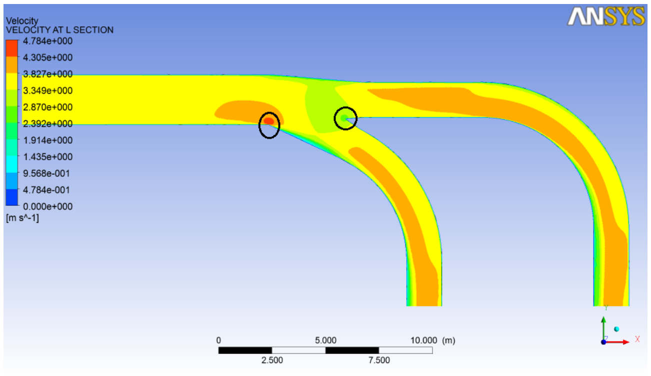


Figure 4: Velocity Distribution at Mid Plane

The manifold is pushed 3.5m towards upstream side as purposed in the initial layout. The branch angle is now 30 degrees as we have decided. Other arrangements are as left as per the initial layout.The cad model is exported to ICEM CFD and a tetrahedral mesh is generated. The mesh file is then exported to Ansys Fluent. Reference pressure: 1 atm; Boundry Condition at inlet: pressure inlet with total pressure equivalent to water head of 110m; Boundry Condition at Outlets: flow rate of each outlet is 8cums; Wall: No Slip Wall; Turbulence: K-E turbulence model with T I 5%, Solution Method: P-V Copuling; Maximum No of Iteration: 500; Solver: Second Order; Convergence Criteria: 0.00001Flow field is calculated and is post processed. Pressure and velocity distribution at mid plane,inlet and outlet as well as the pressure and velocity streamline is visualized.

The average flow velocity is in the range of 3.5m/s.
There are two stagnation points(indicated by circles0 in the geometry.
The loss coefficient is calculated to be 0.44 and 0.43.
The shape of bifurcation is further improved by increasing the flair angle.

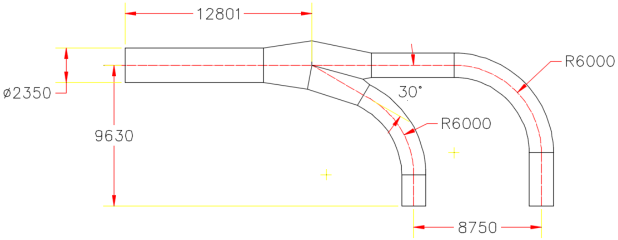


Figure 5: Geometry of Opt-3

Similar analysis was performed for this option.

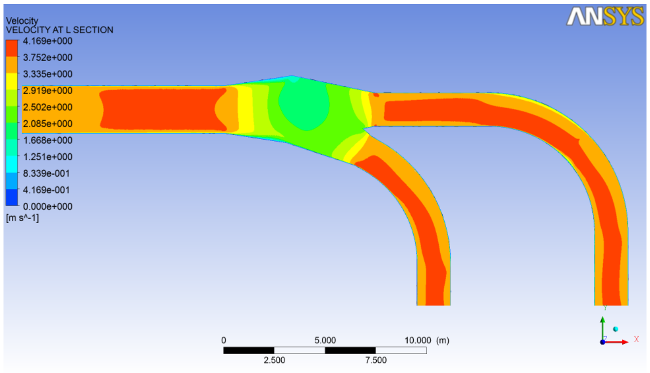


Figure 6: Velocity Distribution at Mid Plane

The average velocity at which the actual bifurcation is happening is in the range of 2m/s. The topmost part of the bifurcation has not been utilized by the flow field. Therefore this part needs to be modified for further improvement in geometry.The loss coefficient for option 3 is calculated to be 0.26 and 0.23. The top edge of option 3 has been replaced by a curved part as shown in the figure below:

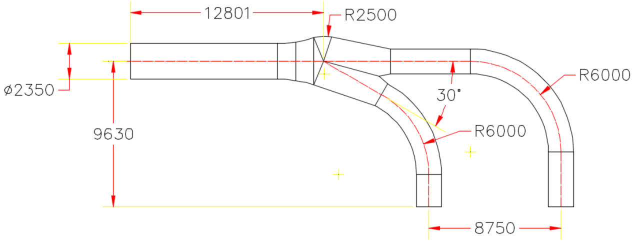


Figure 7: Geometry of Opt-4

Similar analysis was performed for this option.

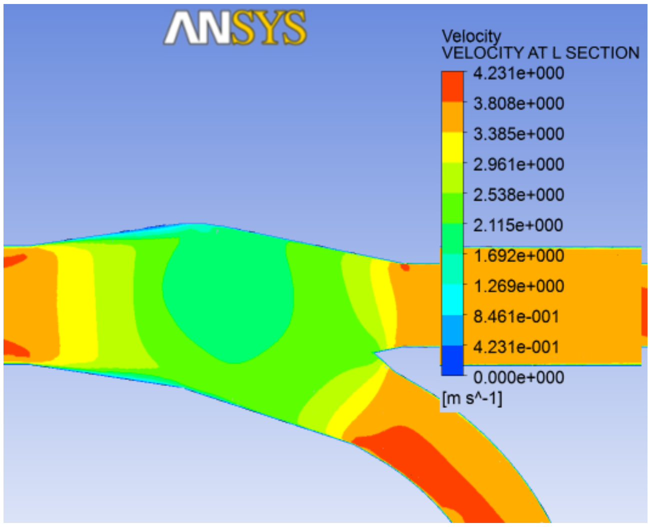
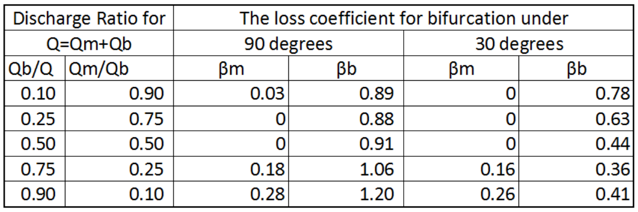


Figure 8: Velocity distribution at mid plane

The loss coefficient for this option is improved to 0.23 and 0.22. The value of loss coefficient predicted by experimental method is



Discharge Ratio for		The loss coefficient for bifurcation under			
Q=Qm+Qb		90 degrees		30 degrees	
Qb/Q	Qm/Qb	βm	βb	βm	βb
0.10	0.90	0.03	0.89	0	0.78
0.25	0.75	0	0.88	0	0.63
0.50	0.50	0	0.91	0	0.44
0.75	0.25	0.18	1.06	0.16	0.36
0.90	0.10	0.28	1.20	0.26	0.41

Figure 9: Bifurcation Loss Coefficient [4]

Hence this value is satisfactory and hence the geometry of option 4 is recommended for further analysis.A comparision of all three options is summarized in figure 10.

Figure 10: Velocity distribution at mid plane

The loss coefficient of option 2 is comparable to the loss coefficient provided in figure 1. With the change in the shape of bifurcation, the loss coefficient have improved significantly in option 3. However, the improvement in loss coefficient from option 3 to option 4 is not very significant. Therefore, further improvement in the geometry is expected to decrease the loss coefficient. But the increased cost of fabrication will not be justified. Therefore, option 4 has been recommended for the structural analysis.

4. Structural Analysis

Three dimensional model of the bifurcation is proposed for the given bifurcation geometry with a suitable guess values for sickle plate thickness,pressure vessel thickness and the thickness and arrangement of the stiffners.

Material for the bifurcation is selected to be ASTM A36 having yeild stress of 250MPa. The geometry is imported to the ANSYS and mesh is generated. The designed pressure is calculated by adding surge pressure to the static pressure.The total design pressure results to be 1.54MPa. The appropriate structural boundary condition is very important factor to obtain realistic result during the simulation process. Application of exact nodal boundary condition was very complicated due to geometrical complicacy. So, the branching section considered as simply supported beam and fixed supports are applied to the free ends of inlet and branch outlets. To decrease the error due lack of exact boundary condition, the whole domain was taken about 20x Diameter times longer than the conical transition (critical) section for the analysis.

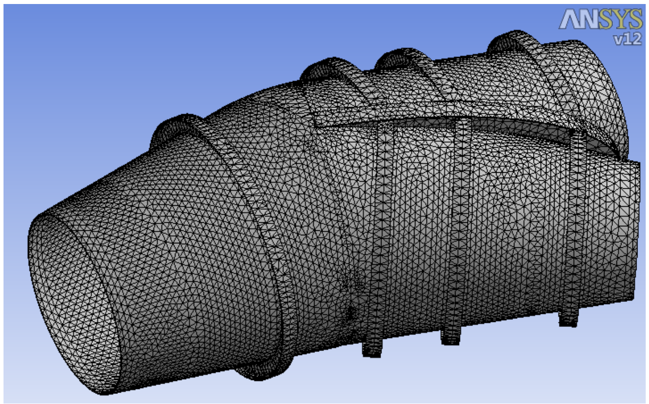


Figure 11: Tetrahedral Mesh of Bifurcation

The structural simulation was then carried after applying design pressure and boundary condition. The von-Mises stress, total deformation and Safety factor were selected as key parameters to describe the simulation results.

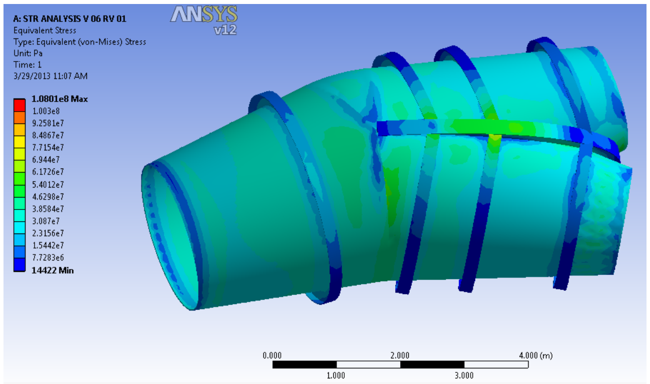


Figure 12: Equivalent Stress Distribution

The equivalent Equivalent Von Mises stress was calculated. The stress diagram below was produced by the program.The maximum stress is 108MPa which is within the allowable limit [5]. Hence given geometry is accepted for fabrication.

Figure 13: Bifurcation installed at site

5. Conclusion

It was observed that the loss coefficient for bifurcation has reduced from 0.44 to 0.21. This will add up in the overall plant performance in long term. Furthermore, with the help of Finite Element Analysis, we are sure about the performance of the designed structure. The weak parts are identified during the design phase and changes had been made to make it acceptable.

Acknowledgments

The authors are grateful to Nepal Electricity Authority and the engineering team of Kulekhani III Hydropower Project for providing us necessary data and information.

References

[1] S. Ahmad. Head loss in symmetric bifurcation. Master's thesis, UNIVERSITY OF BRITISH COLUMBIA, 1943.

[2] R. G. C. A. AGUIRRE. Head losses anslysis in symmetrical trifurcation of penstock-high pressure pipeline systems cfd. *IGHEM*, 2016.

[3] Ansys Inc. Ansys cfx theory guide. *IGHEM*, 2016.

[4] E.Mosoniya. *High Head Power Plant Volume IIA and IIB*. AKADIMIA KIADO, Bdapest, 1991.

[5] ASME. *Boiler and Pressure Vessel Code*. ASME, 2004.