Modification of the shell type trifurcation shape for penstock design in a hydraulic plant by numerical simulations

Jung Jae Hyun1, Jong Woong Choi1, Won Guk Jang1, Yong Cho1, Young Il Kim1, Tai Young Cho1, In Jun Cho1, and Young Joon Kim1*

1 K-water Institute, Korea Water Resources Corporation, Daejeon, Korea

Abstract. The branching of pipes is commonly used for fluid distribution in penstock of hydro-power plants to reduce construction cost. A trifurcation allows flow of water to provide three turbines operating at the same time, and it is usually designed for uniform distribution of discharge rate and minimum hydraulic loss. A symmetric trifurcation not considered fluid dynamics occasionally may make unequal power production of each hydro turbine and its severe fluctuation. The paper focuses on the cause of fluctuating power production and suggests a solution for stable operation by numerical simulations. The discharge rate distribution for existing and modified shapes of the trifurcation has been investigated by steady state computations, and quasi-steady state calculations have been carried out in order to find the cause of the flow fluctuations in the penstock for rated plant discharge (Q = 153.66 m³/s). In addition, the time dependent flow structures and the head losses in the penstock have been compared for the existing and modified trifurcation shapes, and it is also verified that the fluctuating power production can be reduced significantly with modified trifurcation.

Keywords: hydro-power plant, trifurcation, fluctuation power, headloss, CFD

1. Introduction

Hydro-power plant is the system that produces electricity using water from high altitude to low altitude. In the hydro power generation system, in order to achieve high efficiency and stable operation of the turbine, it is necessary to secure stable energy sources in the turbine inlet section. The power from the source of the hydro-power plant is expressed by a function of \( f(\rho, g, h, Q) \). The water density and gravitational acceleration have constant values, so fall head and flow rate can affect the power quality of the hydro-power plant. By the design of the penstock, the losses of fall head and flow rate are occurred and it can cause loss fall head and fluctuation of flow rate. The penstock facility cost, which is one of the equipment of the water turbine inflow section, accounts for a considerable proportion of the total cost of civil engineering for hydro power projects and by optimizing the penstock, we can obtain maximum energy at minimum cost. In order to secure a stable capacity of the turbine, an appropriate penstock shape that can minimize the loss due to the change in flow supplied to the turbine is necessary. The trifurcation shape of Penstock is influenced by installation environment, construction cost and flow quality. Therefore, loss of penstock shape and evaluation of flow characteristics are positively necessary. Serre [1] presented the headloss coefficient experimentally with the ratio of pipe area and flow rate for the pipe junction shape fastened at 90°. In Ruprecht [2], the power fluctuation from the power plant is cause by the penstock internal flow characteristics of the hydro power plant, that is, the vortex starts from the top of the spherical Trifurcation shape, swirling flow occurs after a certain time, it flows into the branch pipe. This swirling flow is repeatedly introduced to the expanded trifurcation according to the time change and causes the power fluctuation and head loss of the turbine. In order to prevent such a phenomenon, a flat shape was applied to the upper and lower end on a spherical trifurcation. It makes
the output fluctuation of the turbine to be reduced. Liu [3] compared pressure loss to experiment and CFD when two ducts were fitted. Anakiya [4] verified the optimum shape according to the design of the penstock installed in front of the turbine by CFD. Mallik [5] conducted a flow analysis on 3D shape of the trifurcation to find the most efficient value for given pressure, velocity and trifurcation shape. Schmandt [6] analyzed the headloss change in energy dissipation inside the T-junction using CFD, according to the Reynolds number. In Pattnashetti [7], when the rate of inflow into the trifurcation is constant, we derived the flow rate change, the speed size and the pressure change from angular variation using the CFD from trifurcation to branches. Meti [8] presented that pressure loss increases as the angle connected from asymmetric trifurcation to the branch with experiment and CFD. We confirmed that the optimal flow rate in the branch is displayed when middle branch is 40% and side branches are 30%.

In this study, for the trifurcation with three branches, the power output of the turbine fluctuated in about 18% unstable state in the high power section when operating the three turbines. This shows that the dominant cause is the influence of the trifurcation shape rather than the influence of other hydraulic components. We have grasped the cause of the flow rate fluctuation which causes the turbine output fluctuation by a numerical approach. Then, the trifurcation shape for suppressing the fluctuation in the flow rate was analyzed and compared with the steady state and unsteady state method. Analyzing the flow characteristics and headloss of the trifurcation for operating conditions of the turbine, we examined the appropriate turbine operation method.

2. Numerical Analysis Objects

2.1 Shape
Numerical analysis is applied to the trifurcation shape. Water flows into the upstream pipe (diameter of 5,500mm) according to the trifurcation standard and is discharged to each branch through trifurcation. This outflowing water flows into the turbine installed at the end of the branch. It generates the power of the turbine. The maximum diameter of Trifurcation is 8,000mm and it is connected to branch with pipe of shell type. Trifurcation and branch have a symmetrical structure, and both branches are connected to trifurcation at an angle of 66.5 °. Each branch’s diameter connected to the trifurcation is 3,200 mm, and the diameter of the pipe connected to the turbine is 3,000 mm. The other pipe which has different diameter is connected to the reducer. Then, it’s a deflector device installed horizontally that suppresses vortex generation in the upper and lower parts of the trifurcation shape, and the area for applying CFD analysis was applied from the inlet pipe to the pipe forward turbine with trifurcation as the center.

2.2 Numerical Analysis Boundary Condition and Approach Method
Figure 1. shows the trifurcation shape of the flow region for the application of CFD. In this research, the shape to uncover the cause of output fluctuation of the existing model and the shape to reduce output fluctuation were applied. The existing model is the flow passage that horizontal deflector is installed on the top and bottom of the round trifurcation. In the modified model, the deflector is diagonally installed compared with the existing model.

![Figure 1. Trifurcation geometry for CFD.](image)

A grid is required to perform CFD with the grid distribution on the surface. The number of used lattices was 200 million elements. The lattice shape is tetrahedral and the hybrid lattices applying to prism lattice shape are used near the wall. In order to increase the resolution of the wall, compact grids are distributed near the wall and y (+) is set to 300 or less. Lattices are generated in the same way for the change model and existing model. Output fluctuation of hydro power is caused by fluctuation in flow rate according
to time change at the exit of the branch end connected to the front of the turbine.

In order to copy such a phenomenon, an unsteady state analysis method of CFD must be applied. There is an example which is the steady state and unsteady state analysis result of CFD for the spherical shape with Re = 100,000. In the steady state analysis result, symmetrical vortex is displayed around the sphere and the behavior of vortex with time change cannot be known. However, in the unsteady state analysis result, it is possible to observe the karman vortex series which is the behavior of the vortex generated around the sphere. These vortex behaviors can cause hydraulic power output fluctuations as the cause of flow fluctuations and pressure drops. Therefore, in order to confirm the cause of output fluctuation of hydro power generation, an unsteady state analysis method is necessarily required for flow fluctuation and pressure fluctuation of trifurcation.

The boundary conditions are applied to the CFD simulation. The operating flow rate which came from the each turbine connected to the exit of branch and the volumetric flow rate is 51.22 m$^3$/s when operating one turbine. In the entrance condition, a constant velocity profile is applied at a volumetric flow rate according to the number of operating turbine. In the exit condition, a steady state calculation was performed applying an exit pressure condition. In order to replicate the fluctuation of the flow, which is the cause of the power fluctuation of hydropower generation, the unsteady state analysis method was used at the exit of the analysis area to observe the flow fluctuation with the change of time. ANSYS CFX, a commercially available code widely used in domestic and overseas, was used to apply CFD using mass conservation and momentum equations. Considering the economical efficiency of the turbulence model, the k-ε turbulence model that supposes a constant velocity profile in the sublayer was applied. This CFD analysis is carried out by applying the same boundary conditions to the modified model and existing model. We performed steady and unsteady state calculation by applying entrance conditions of analysis domain as volumetric flow 153.66 m$^3$/s, 102.44 m$^3$/s and 51.22 m$^3$/s when operating 3, 2 and 1 turbine. And we compare headloss to existing and new shape via steady state calculation.

3. CFD analysis results

3.1 Cause of Hydro-Power Output Fluctuation

Figure 2. shows the result of the flow rate fluctuation with the time change at each branch outlet under the Case 01 condition which applied 100% of volumetric flow 51.22 m$^3$/s for the existing shape. The most ideal trifurcation design makes the water flow out with same flow rate at each branch and with flow rate of 51.22 m$^3$/s when the total flow rate is 153.66 m$^3$/s. When we assumed 100% of flow rate 51.22 m$^3$/s at Branch 1, a flow variation value was observed about ± 8.05% with the maximum flow rate of 132.4% and 116.3% at the minimum. The peak-to-peak time was 81s and the repetition interval was 0.0125Hz. In Branch 2, the fluctuation value of ± 40.3% appeared at the maximum of 124.5% and the minimum of 43.9%. The peak-to-peak time was 40.05s and the repetition interval was 0.025Hz. In Branch 3, flow fluctuation occurred with the same period with the 40.5 s phase difference for branch 2 result. These fluctuations in flow rate with time change flow into the turbine connected to the tip of the branch causing the output fluctuation of the turbine, the same tendency as the output fluctuation graph of the same turbine as that proposed by Ruprecht [2] was shown.

Figure 2. Unsteady results of case 01.
In order to ascertain the cause of flow rate fluctuation occurring under the [Case01] condition, the flow characteristics according to the time change are shown in Figure 3. (a~d). In Figure 3. (a), due to the difference in the diameter of the pipe and branch at the previous portion of the trifurcation, a stagnation region of water flow occurs in the cavity region at the upper and lower parts of the trifurcation. At the upper and lower parts of the trifurcation, the vortex occurs at the top of the trifurcation as shown in Figure 3. (b) due to the shear force caused by the difference in the water flow. The generated vortex has developed at the center of the trifurcation in Figure 3. (c). As shown in Figure 3. (d), the developed vortex disappears with the lapse of time and the stagnation region occurs again at the upper and lower parts of the trifurcation. As shown in Figure 3. (e) and (f), the vortex is generated and developed at the bottom of the trifurcation. With this flow, water is repeatedly flowed into branch 1, 2, and 3 during the vortex creation, development, and extinction process. Particularly in branch 1 and 3, flow rate fluctuation occurs by inflow of swirling flow developed by the vortex development due to time change. By flow rate fluctuation such a series of process, output fluctuation of hydro power occurs.

3.2 The Result of Turbine Combination Operation
Figure 4. shows the calculation result of the steady state flow characteristics executed under the condition Case 01 of the existing and corrected shape by speed distribution, speed vector and streamline. The flow velocity appears most rapidly at the end of branch 2 and sudden speed change at the tip of each branch is due to the change of euro area by reducer. In the trifurcation, it can be seen that the stagnant region and the rapid velocity change region are displayed in the existing and modified shape in the same way and the water stagnation region decreases in reduction of the trifurcation upper and lower cavity regions of the modified shape. These stagnant areas develop into swirl flow which flows into branch [2]. Then you can see some of the water in the stagnant area flowing into the branch.

Figure 3. Source analysis of the flow fluctuation for case 01.
Figure 4. Flow characteristic of the steady state CFD for the original and modified geometry.

Figure 5. shows the results of the outflow rate at the branch by calculating the steady state of the condition of Case 01 for the existing shape and modified shape. For the existing shape, we can get 109.6% by comparing ideal flow to flow at branch 2 and 95.2% by comparing ideal flow to flow at branch 3. The difference of these flow rate was 14.4%. For modified shape, we can get 107.3% by comparing ideal flow to flow at branch 2 and 96.4% by comparing ideal flow to flow at branch 3. The difference of these flow rate was 10.9%. Due to the reduced flow resistance flowing into the branches 1 and 3 of the modified shape, the flow rate difference decreased by 3.9% in the modified shape of Branch 1, 2 (branch 2, 3) as compared with the existing shape.

Figure 6. compares fluctuations in the flow rate according to the analysis result of the unsteady state of the existing and modified shape. Based on the time change in the modified shape, the ideal flow rate ratio of branch 2 was 107.4% and the ideal flow rate ration of branch 3, 1 was 96.3% as constant flow rate. The difference in flow rate between branch 1 and 2 (or branch 2, 3) occurred 11.1%. In order to judge that the interpretation near the wall is correctly performed by using the k-ε turbulence model, the grid near the wall is more densely generated. After that the SST turbulence model[9] which directly solves the viscosity sublayer region is applied. As a result, it is thought that constant flow rate flows out to the branch outlet by flow rate fluctuation of ± 0.05%. By eliminating the source of the flow rate fluctuation which is the cause of the output fluctuation of the hydro power generation, flow rate fluctuation did not occur.
Figure 6. Comparison of the flow fluctuation for the case 01.

Figure 7. shows the velocity distribution and speed vector of the steady state and the unsteady state as a result, in order to grasp the flow characteristics of the modified shape in the combined operation of the 1st and 2nd turbines. The flow characteristics of the modified shape appeared similar to the results of the existing shape, but the stagnant area decreased. And in trifurcation central cross section, eddy current caused by vortex creation and development is not observed at entrance of each branch, so it is judged that fluctuation in flow rate does not occur.

(a) steady state results
(b) unsteady state results

Figure 7. Flow characteristic of the modified geometry for the case 02.

Figure 8. shows the flow rate values at exit of branch 1 and 2 at steady state and unsteady state under Case 2 condition (simultaneous operation of the 1st and 2nd turbines). From the calculation result of steady state, 97.9% flow at branch 1 and 102.1% flow at branch 2 flowed out at existing shape and the flow difference of branch 1 and 2 occurred 4.2% s higher at branch 2. In the corrected shape, branch 1 flowed out at a flow rate of 98.2% and branch 2 at a flow rate of 101.8%, and the flow rate difference of branch 1 and 2 occurred 3.6% higher at branch 2. It is conceivable that the flow rate rises at branch 2

(a) flow rate by steady state
(b) unsteady state results

Figure 8. Comparison of CFD results for the case 02.
of the existing modified shape and the resistance by the fluid is high at branch 2. The calculation result of the unsteady state in Figure 13. (b) was of a corrected shape, no flow rate fluctuation occurred, and flow rate fluctuation did not occur even as a result of the existing shape. Figure 9. shows the results of the steady state and the unsteady state in order to grasp the flow characteristics of the corrected shape in the case of simultaneous operation of the 1st and 3rd turbines. The flow characteristics of the existing shape resembled the result of the modified shape and there was no large vortex region of the trifurcation central cross section which causes flow rate fluctuation. No eddy current into the branch inlet due to the development of vortex was observed. However, due to the continuous inflow into the branch by vortex generation in the upper and lower trifurcation areas, several swirling flows occur within the branch and the headloss following this swirling flow seems to rise.

![Figure 9. Comparison of CFD results for the case 03.](image)

Figure 10. shows the flow rate values at branches 1 and 3 with steady state and unsteady state under Case 3 condition (simultaneous operation of turbines 1 and 3) for the modified shape. Branches 1 and 3 are discharged at a flow rate of 100.0% by the symmetrical structure of the analysis region from the calculation result of the steady and unsteady state. In unsteady state calculation of the modified shape, a low flow rate fluctuation of 0.005% occurred by the generated vortex in the upper and lower parts of trifurcation.

![Figure 10. Comparison of CFD results for the case 03.](image)

Figure 11. shows the flow characteristics of the modified shape under case 04, 05 as a result of velocity distribution and velocity vector. In Figure 11. (a), we see that a large vortex region occurs in the trifurcation section as a result of the unsteady state of single operation of the turbines for branch 1 or branch 3. This vortex is flowed into the branch and has a large speed swirling flow in pipe. In other words, the flow accompanied by a high rotational speed passes through the tip of the branch and flows into the turbine. Flow fluctuation does not occur due to continuous inflow of swirling flow, but headloss
increases because of these phenomena. It can be seen in Figure 11. (b) that the stagnant region and the vortex are not generated as a result of the unsteady state of the branch 2.

3.3 Headloss of the branch

The following formula applied for deriving quantitative values of headloss and loss coefficient from inlet to outlet for existing shape and modified shape. Equation (1) is the each headloss from the entrance to branch 1, 2, 3 exit, equation (2) is the total headloss from the entrance to the exit of the branches and equation (3) is the headloss coefficient from the entrance to the each branch exit.

\[ H_{L\text{in-}out} = \frac{(p_{t,\text{in}} - p_{t,\text{out}})}{\rho g} \]  

\[ H_{L\text{total}} = \sqrt{\frac{\sum(H_{L\text{in-}out})^{2}}{n}} \]  

\[ K_{\text{in-}out} = \frac{(p_{t,\text{in}} - p_{t,\text{out}})}{\frac{1}{2} \rho V_{\text{ref}}^{2}} \]

In the above equation, \( p_{t,\text{in}} \) is the total pressure [Pa] at the inlet, \( p_{t,\text{out}} \) is the total pressure [Pa] at the outlet of branch 1, 2, 3. \( \rho \) is the density of water [kg/m³], \( g \) represents the gravitational acceleration [m/s²], and \( V_{\text{ref}} \) represents the average inflow velocity [m/s] from the inlet.

Table 1. quantitatively shows the each headloss, headloss coefficients of branches 1, 2, 3, and the total headloss coefficients of the analysis area at the entrance of the analysis region of the existing shape and modified shape. All headlosses and loss factors due to the combination of management appeared lower from the modified shape than the existing shape due to the vortex generation and development suppression of the upper and lower part of trifurcation. This shows that the resistance of the fluid decreases by the shape change. By changing the flow path of trifurcation to modified shape in the full flow rate of 153.66 m³/s, under Case 01 operating condition, the headloss decreased by 0.248 m from 1.207 m to 0.959 m and under Case 02 operating condition, it decreased by 0.24 m from 0.846 m to 0.606 m. Under Case 03 operation condition, it decreased by 0.040 m from 1.209 m to 0.805 m and under Case 04 operation condition, it decreased by 1.662 m from 5.502 m to 3.390 m. And last under Case 05 operation condition, it decreased by 0.012 m from 0.381 m to 0.369 m. Especially, the headloss occurs to a relatively high value in case 04, because the swirling flows are generated by flowing into the branch inflow portion due to the generation and development of vortex occurring above and below trifurcation. In order to reduce the loss fall head when operating two turbines, the operation of the turbine including the branch 2 is advantageous, and the operation of the turbine using the branch 2 is advantageous in order to reduce the loss fall head when operating one turbine.
Table 1. Comparison of the head loss.

| Numbers of operating turbine | Different Total Pressure [Pa] | Head Loss [m] | Head Loss Coefficient |
|-----------------------------|-------------------------------|---------------|-----------------------|
|                             | $P_{t,in} - P_{t,out1}$    | $HL_{in-out1}$ | $K_{in-out1}$         |
| Case 01 (original)          | 13,991                        | 1.430         | 0.697                 |
| Case 01 (modify)            | 11,018                        | 1.127         | 0.840                 |
| Case 02 (original)          | 7,489                         | 0.766         | 0.840                 |
| Case 02 (modify)            | 6,900                         | 0.705         | 0.774                 |
| Case 03 (original)          | 9,661                         | 0.988         | 1.084                 |
| Case 03 (modify)            | 7,873                         | 0.805         | 0.883                 |
| Case 04 (original)          | 49,412                        | 5.052         | 22.182                |
| Case 04 (modify)            | 33,160                        | 3.390         | 14.857                |
| Case 05 (original)          | -                             | 0.381         | 1.671                 |
| Case 05 (modify)            | -                             | 0.369         | 1.167                 |

3.4 On-Site application for the modified geometry

As shown in Figure 12, the trifurcation shape for the original deflection plate (black line) and modified deflection plate (blue line) was analyzed for the cause of output fluctuation using CFD. By reducing the vortex generation regions of the upper and lower triple points, flow fluctuations caused by vortex generation, development, and extinction due to the triangulation pattern were suppressed. The final deflection plate of the red line was applied as Figure 12. and applied to the on-site as the right one.

![Deflection plate application in the trifurcation](image)

Figure 12. Deflection plate application in the trifurcation.

Figure 13. shows the output variation from low flow rate to 153.66 m$^3$/s flow rate in the existing and applied trifurcation shape of the on-site hydropower plant. Figure 13. (a) applied to the existing trifurcation shape shows the severe output variation at high flow rate. Figure 13. (b) shows the output variation of the turbine for trifurcation with the final deflection plate. The output fluctuation of the turbine hardly occurs at the high flow rate.
4. Conclusion
In this study, the conclusions that were carried out using CFD on the cause and solution of output fluctuation of the turbine operated from On-site are as follows.

- Vortex occurs in the upper and lower cavity trifurcation shapes as cause of output fluctuation of hydro-power plant and flow rate fluctuation. Flow rate fluctuation occurs with vortex creation and development and it caused output fluctuation of turbine.
- Fluctuation in flow rate could be reduced by decreasing the cavity area of the upper and lower part of trifurcation which resulted in the generation of vortex. (from ± 40% to ± 0.1%)
- The loss fall head decreased compared to the original shape under all conditions of the modified shape due to the loss reduction by the fluid for shape change.
- In the case of operating two waterwheels in collaboration, there is no output fluctuation from the unsteady state calculation, and driving including Unit 2 is advantageous in order to reduce the loss fall head.
- Operation of Unit 2 is advantageous in order to reduce the loss fall head during single operation.

5. References
[1] Serre M, Jacob A, and Rex A, 1994, Energy loss at combining pipe junctions, J Hydraul. Eng., Vol.120(7), pp.808-830.
[2] Ruprecht A, Helmarich T, and Buntic I, 2003, Very large eddy simulation for the prediction of unsteady vortex motion, Conference on Modeling Fluid Flow(CMFF’03) The 12th International Conference on FluidFlow Technologies Budapest, Hungary.
[3] Liu W, Long Z and Chen Q, 2012, A procedure for predicting pressure loss coefficients of duct fittings using CFD(RP-1943), HVAC&R Research, Vol.18(6), pp.1168-1181.
[4] Anakiya A, Prajapati H and Prajapati S, 2014, Analysis of the penstock for pit turbine using computational fluid dynamics, IOSA Journal of Mechanical and Civil Engineering, Vol.11(3), pp.36-42.
[5] Mallik R K and Paudel P, 2015, 3D flow modeling of the first trifurcation made in Nepal, Journal of Advanced College of Engineering and Management, Vol.1
[6] Schmandt B and Herwig H, 2015, The head change coefficient for branched flows: Why “losses” due to junctions can be negative, International Journal of Heat and Fluid Flow, Vol.54, pp.268-275.
[7] Pattanashetti N and Manjunatha S S, 2016, Energy loss analysis of 3D Asymmetric trifurcations using CFD, International Journal of Engineering Research & Science, Vol.2(6), pp.129-136.
[8] Meti B and Sitaram N, 2017, Determination of optimum pressure loss coefficient and flow distribution at unsymmetrical pipe trifurcation using experimental and numerical technique, Applied Engineering, Vol.1(3), pp.74-80.
[9] Menter F R, 1992, Influence of freestream values on k-w turbulence model predictions, AIAA Journal, Vol.30(6), pp.1657-1659.

Acknowledgments
This research was supported by a grant(code 17IFIP-B128598-01) from Plant Program funded by Ministry of Land, Infrastructure and Transport of Korean government.