Performance Evaluation of Penstock By-Pass System Modification on HEPP (Hydro Electric Power Plant) with CFD Analysis to Optimize Power Consumption

Harpendi¹, Sugianto², Dwi Wijanarko³ and Wahyu Caesarendra⁴

¹Power Plant Governance Experts, Indonesia Power, PT, Bandung, Indonesia
²Aeronautical – Mechanical Engineering Department, Bandung State Polytechnic, Bandung Indonesia
³Assistant Analyst, Indonesia Power, PT, Bandung, Indonesia
⁴Faculty of Integrated Technologies, Universiti Brunei Darussalam, Brunei Darussalam

Abstract. This paper proposes a feasibility test for the performance modification of the penstock by-pass system using CFD (Computational-Fluid-Dynamic) simulations to optimize the energy use of the water-cooling system. The main advantage of this modification is the power savings of 45% -50%. This modification of the by-pass system is carried out with the aim that the final flowrate and pressure of modification can replace the Main-Water-Supply-Pump (MWSP) operation function in the unit's main cooling system. CFD simulation is carried out on the impact of a pressure drop of 40-4 bar with a flow of 0.3 m³/s on the by-pass pipe flow of the existing penstock system and the condition of the pipe modification to be carried out. The numerical simulation results show that the MWSP network structure can accept load flow pressures varying 40-4 bar. This simulation refers to the conditions in HEPP Saguling Indonesia and can be applied to others.

1. Introduction

The main function of the penstock by-pass system in a hydroelectric power plant aims to minimize the occurrence of water hammer due to the large pressure between the spaces before and after the main inlet valve (MIV), and the penstock by-pass system to maintain pressure balance during operation.

However, on the other hand, the main cooling system for generators, lube oil, and others, uses a Heat Exchanger system where cooling water is supplied by the MWSP (Main Water Supply Pump), and the energy use of this pump requires 45% -50% of the total auxiliary power consumption (referring to operational data of Indonesia Power, PT. In 2019, it was found that the total MSWP consumption was 2.06 GWh or 52.4% of the total auxiliary power consumption) [1].
Table 1. TOP 3 SEU (Significant Energy Uses) List 2019

| No | Equipment      | Capacity | Quantity | Cycle of Operations | Usage Energy |
|----|----------------|----------|----------|--------------------|--------------|
|    |                | kW       | pcs      | hours              | MWh          | %            |
| 1  | MWSP           | 132      | 4        | 15,592.05          | 2,058.15     | 52.4         |
| 2  | Governor Pump  | 22       | 4        | 15,592.05          | 343.02       | 8.7          |
| 3  | Drainage Pump  | 45       | 2        | 382.40             | 17.20        | 0.4          |
| 4  | Inlet Valve    | 30       | 4        | 95.12              | 2.85         | 0.1          |
| 5  | Total Others   | 2030     | >110     | Total Energy Uses  | 1,505.31     | 38           |

The idea of modifying the piping system on the penstock by-pass system still takes into account the balance of the penstock operating system as a major consideration in modeling. Modeling is done using analytical calculations and numerical simulations using the Computational Fluid Dynamics (CFD) method, to obtain the results or recommendations required whether the safety of the Penstock by-pass piping system can still be guaranteed after modification. With some consideration of the initial parameter conditions as follows, the flow rate of the penstock pipe is 54 m³/s with a pressure of 39 bar to be 0.3 m³/s with a pressure of 4-6 bar according to the requirements of the main cooling system of the unit, where this was previously supplied by MSWP as main supply cooling system unit.

The analysis calculation is carried out to obtain the geometry and material configuration of the pipe with the required piping components (elbow, flange, orifice, and valve) based on the calculation of flow losses. The calculation process is continued using numerical CFD simulations to obtain the optimal configuration for power reduction in this paper which is described in the parameters of pressure drop and flowrate [2].

2. Practical Analysis & Calculations

The configuration of the penstock system and the MWSP pipeline is shown in Figure 1.

Before conducting a simulation with CFD, the following steps are carried out:

- Determination of pipe thickness.
- Determination of class of fittings and other piping components.
- Calculation of flow velocity and head loss manually for various pipe diameters, as a basis for determining the diameter of the pipe design to be simulated.
- Calculation of head loss on the selected pipe diameter with orifical configuration, to get an idea of the orifical configuration that will be used.

The type of pipe chosen is a type of carbon steel that is widely available in the market, namely A53 Grade A (S = 30 ksi, Table A-1 ASME B31.1 2014) [3]. The initial assumption of pipe thickness is assumed to be Schedule 80 because the water pressure in the pipe is quite large, the maximum pressure is 40 bar.
In the following discussion, we will calculate pipe thickness, flow speed, and head loss for pipes with nominal diameters, NPS 6, 8, and 10.

2.1. Determination of Pipe Thickness, Pipe Diameter, and Orifice Configuration Manually

Pipe thickness is determined based on ASME Code B31.1, Power Piping which is commonly used as a reference piping design for Power Plants. The equation for pipe thickness according to the code is (ASME B31.1 2014) [3]:

\[
t_m = \frac{PD_0}{2(SEW + Py)} + A
\]

**Table 2.** Initial parameters as a reference for calculation

| Variable | Value | Unit | Information |
|----------|-------|------|-------------|
| P        | 40    | bar  | Design Pressure |
|          | 584   | psi  |             |
| S        | 30 ksi*1000 | psi | Table A-1 ASME B31.1 2014 for Seamless Pipe and Tube |
| E        | 1     | -    |             |
| y        | 0.4   | -    | Table 101.1.2.(A) Values of y |
| W        | 1     | -    | for Carbon Steel (1) |
| Sch      | 80    |      |             |
A (Additional Thickness)

|   | Mill | 12.50% | Mill Tolerance |
|---|------|--------|---------------|
| CA | 0.0625 | in | Corrosion Allowance |

Table 3. The calculation results of Pipe Thickness, Pipe Diameter.

| No. | NPS | $D_0$ | $t_m$ | $t_{(sch 80)}$ | Criteria |
|-----|-----|-------|-------|--------------|----------|
|     |     | inch  |       | inch         | mm       |
| 1   | 6   | 6.625 | 0.251 | 10.97        | Fulfilled|
| 2   | 8   | 8.625 | 0.271 | 12.70        | Fulfilled|
| 3   | 10  | 10.750| 0.291 | 15.06        | Fulfilled|

2.2. Determination of Class of Other Piping Components

For other piping components, such as fittings, gaskets, are determined by the pipe schedule (sch 80), namely ANSI Class 600 [3], with a maximum pressure of 1480 psi (hydrostatic test pressure).

2.3. Determination of Pipe Diameter

The design discharge is determined as much as the required cooling water discharge based on measurements and field surveys, which is: $Q = 0.3 \text{ m}^3/\text{s}$, The following is an example of calculating the flow rate and head loss for NPS 8 pipes.

2.3.1. Determine the amount of flow velocity.

$$ v = \frac{Q}{A} = \frac{0.3 \text{ m}^3/\text{s}}{0.02946 \text{ m}^2} = 10.1832 \text{ m/s} $$ (2)

2.3.2. Determine the Reynold Number

The Reynold number is obtained from the following equation [4 - 7]:

$$ Re = \frac{\rho v d}{\mu} = \frac{998 \times 10.1832 \text{ m}^3/\text{s} \times 0.1937 \text{ m}}{1.03 \times 10^{-3}} = 1.96 \times 10^6 \quad \text{(Turbulence)} $$ (3)

2.3.3. Determine the number “f”:

How to determine the number "f" is to use the moody chart [5] with the Reynold number and by determining the value of "e" is the roughness of the carbon steel pipe which is 0.046 and the diameter of the pipe is 8 in [4 - 7].

$$ k = \frac{e}{d} = \frac{0.046}{193.675 \text{ mm}} = 0.00024 $$ (4)

$$ f = 0.0148 $$
2.3.4. Determining the Parameter of Resistance (“k”)

The next step is to determine the resistance parameters, with the following parameters:

- Globe Valve assumes a 60% opening with a value of \( K = 5 \) (according to the flow resistance graph) [4]

![Figure 2. Average loss coefficients for partially open valves.](image)

- Elbow 90° with a total of \( 11 = 11 \times 0.2 \), based on a NPS 8 long radius (12 in. = 308.4 mm). [6,7]

![Figure 3. The Resistance coefficients for elbow 90°.](image)
Based on previous preliminary data and referring to the curve above, the total resistances are as follows:

**Table 4. The Total Resistance.**

| No. | Type of Resistance | Data                     | “k” Value |
|-----|-------------------|--------------------------|-----------|
| 1   | Globe Valve       | 40% closed (60% opened)   | 5         |
| 2   | 11 x Elbow 90° (8x0.2) | R = 304.8               | 2.2       |

**2.3.5. Determining the Head loss**

The determination of the total head loss is obtained from the following equation [4 - 7].

\[
\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \left( \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2 \right) + h_f + \sum h_m - h_p
\]

The equation for the total head loss is

\[
h_p = Z_A - Z_B + h_f + \sum h_m
\]

\[
= 5.5 m + \frac{v^2}{2g} \left( \frac{fL}{d} + \sum K \right)
\]

- Head loss Mayor

\[
h_f = \frac{v^2 fL}{2gd} = \frac{10.183^2 \times 0.0148 \times 31.9}{2 \times 9.8 \times 0.1937} = 12.85 \text{ m}
\]

- Head loss Minor

\[
\sum h_m = \sum K \frac{v^2}{2g} = 7.2 \times \frac{10.183^2}{2 \times 9.8} = 38.09 \text{ m}
\]

By using the height difference data (ZA-ZB = 5.5 m), the total head loss is as follows:

\[
h_p = Z_A - Z_B + h_f + \sum h_m = 5.5 \text{ m} + 12.85 \text{ m} + 38.09 \text{ m} = 56.45 \text{ m}
\]

Then the total loss of head loss is equal to: P = 56.45 x 998 x 9.8 x 100.000\(^{-1}\) = 5.52 bar

By using the same method, a Comparison Table with 3 different diameter pipe size comparison parameters is obtained as follows:
Table 5. Comparison table of different diameter

| No | Q [m³/s] | NPS | hₑ [m] | Σhₑ [m] | hₚ [m] | ΔP [bar] | v [m/s] |
|----|--------|-----|--------|---------|--------|----------|--------|
| 1  | 0.3    | 6   | 54.6   | 118.7   | 178.8  | 17.5     | 17.8   |
| 2  | 0.3    | 8   | 12.9   | 38.0    | 56.5   | 5.5      | 10.2   |
| 3  | 0.3    | 10  | 4.0    | 15.2    | 24.7   | 2.4      | 6.5    |

Regarding the above comparison, it can be seen that the pressure drop for NPS 6 pipe is 17.5 bar, it is close to the desired pressure drop, but the water flow rate reaches 17.8 m/s, this water velocity is too high, it can cause erosion of the pipe and noise, and not according to good practice, commonly used between 2 - 6 m/s. Therefore, the recommended pipe for use is NPS 10.

2.3.6. Determining Head loss After Addition of Orifice

Furthermore, on this NPS 10 pipe 4 orifices will be installed, with a specification comparison of the bore diameter and the pipe diameter of β = 0.5. Referring to the literature reference, the “k” minor loss coefficient is obtained, for orifice like this is 30 - 32 [6 - 11].

According to the calculation, the pressure drop with the addition of 4 orifices is 28.1 - 29.9 bar. Thus, the pressure will be obtained at the entry conditions of the cooling system of 6 - 8 bar.

3. CFD Simulation & Analysis Results

The stages of the numerical simulation of fluid flow in MWSP are carried out based on references [1,14,15] especially the flow through the orifice refers to [10 - 21]. The numerical simulation results of water flow in a penstock pipe and branch pipes with a diameter of 10 inches can be shown as a whole in the distribution of absolute pressure, velocity, and intensity of turbulence. As well as the final discharge flow required for the cooling system (through the strainer).

3.1. Velocity Distribution in Pipe Networks

The results of the numerical simulation of the flow velocity along the MWSP pipe starting from the penstock pipe to the elbow pipe 14 (towards the strainer) are shown in Figure 4 to Figure 7 in the form of the velocity contour distribution.
Figure 4. Velocity distribution in the cross-section along the penstock pipe and branch pipe.

In Figure 4, it can be seen that the flow velocity in the penstock pipe ranges from 14 m/s - 20 m/s which is shown in green. A significant decrease in flow velocity occurs after passing through a branch pipe with a variation of velocity between 10 m/s - 14 m/s and decreases after passing through a series of elbows and the speed varies between 7 m/s - 10 m/s which is shown in blue light.

Figure 5 above, shows a clearer difference in velocity distribution that occurs along the T-reduce pipe. It can be seen that there is a fairly high distribution of velocity values (24 m/s - 27 m/s) which is concentrated in the joint area. This must be observed because of the distribution of high-velocity values accompanied by the emergence of low-velocity distribution which indicates the concentration of flow turbulence that occurs in the T-reduce pipe connection area.

Figure 6. The distribution of velocity in the cross-section XY along the T-junction pipe.
The above conditions are made clear by Figure 6 where it can be seen clearly that the distribution of flow velocities that is quite high (27 m/s - 34 m/s) is also accompanied by a low-velocity distribution (3 m/s - 7 m/s) which occurs simultaneously. With this difference in speed in the area, there will be significant flow turbulence.

3.2. Simulation results of water flow in pipes related to turbulence conditions.

![Figure 7. Streamline flow in penstock pipes and branch pipes, elbows, T-junctions](image)

![Figure 8. Streamline flow in branch pipes, T-junction reducers, and elbows 1 and 2](image)

![Figure 9. Streamline flow in T-junction pipes and elbows 10, 11 and elbow long radius 12, 14](image)

The simulation results are also in a streamlined form which can be seen from Figure 7 to Figure 9. In Figure 7 there is flow turbulence as previously mentioned. The turbulence flow that occurs in the branch pipe and then at the first and second elbows is shown by the movement of the flow pattern from a speed of 0 m/s - 8 m/s that occurs simultaneously with a flow pattern of the velocity of 8 m/s - 17 m/s.
The difference in speed will result in a pressure difference, which will result in turbulence. The greatest turbulence occurs in the T-reduce area as shown in Figure 8 below. And likewise, the possible turbulence pattern can be seen on the elbow shown in Figure 9.

3.3. Analysis of CFD Simulation Results

3.3.1. Analysis of Water Flow in Pipes

Referring to the CFD simulation results described in Figures 4 to 10. It can be seen that the simulation results obtained a penstock pipe inlet pressure of 39.486 atm with a speed of 13.588 m/s resulting in a distribution of fluid properties along the penstock pipe (“pipa_pesat”) and branch pipe. The exit pressure from the penstock pipe (“pipa_pesat”) is 39.477 atm with a speed of 13.476 m/s, and the outflow condition to the cooling system strainer has a pressure of 33.400 atm with a speed of 9.794 m/s as shown in Figure 10 results of the average simulation output data.

| Static Pressure | Velocity Magnitude | Volumetric Flow Rate |
|-----------------|---------------------|---------------------|
| inlet_cooler_1 | 9.7935524 | -0.4407916 |
| inlet_pipa_pesat | 13.58811 | inlet_pipaPesat |
| outlet_pipaPesat | 13.475702 | -53.551088 |

Figure 10. Screen Capture CFD initial Simulation Output.

Referring to the CFD simulation results in Figure 10, the penstock pipe (“pipa_pesat”) flow has an inflow of 53.99 m3/s, an outflow of 53.55 m3/s, and discharge flow to the cooling system strainer has a discharge of 0.44 m3/s, where this is still greater than the cooling system requirement of 0.3 m3/s (18000 liter/minute).

3.3.2. Solutions to overcome the Excess Flow Discharge and Inlet Pressure of Cooling system Strainer.

Referring to the results of the analysis on the simulation results of the outflow from the MWSP pipe to the cooling system strainer which has an excess discharge of 0.44 - 0.3 = 0.11 m3/s and an overpressure of an average of 33.400 - 5 = 28.4 atm.

- Use of the T-junction Reduce Pipe

The use of a T-junction reduction pipe, from a size of 10 inches to a size of 6 inches, succeeded in reducing the excess flow rate by 0.11 m3/s (Figure 11).
- Use of Orifice to reduce excess flow pressure.

To overcome the overpressure of 28.4 atm, it is done by installing 4 orifices (according to the results of the analysis calculation) along the pipe between pipe 4, pipe 5, and pipe 6 as shown in Figure 12.

The simulation results of pressure drop using a series of orifices along a 5-meter pipe are as shown in Figure 13 to Figure 15.
Figures 13 to 15 show the differences in the distribution of velocities at each orifice level. The velocity distribution occurs in the range of 37 m/s - 42 m/s and is concentrated in each orifice diameter area. Turbulence flow can also be seen due to the simultaneous high velocity distribution (37 m/s - 42 m/s) and low velocity distribution (3 m/s - 7 m/s).

The configuration of the 4 levels of the orifice causes a gradual pressure drop ($\Delta p = 8$ atm) on each orifice so that the final pressure desired before entering the cooling system strainers can be achieved at 5 atm with a volumetric flow of 0.3 m$^3$/s can be met, as seen in Figure 16.
Regarding the results of the analysis and flow simulation on the MWSP pipe, the simulation results of water flow entering the penstock pipe and out of the penstock pipe and then leading to the strainer which then enters the cooling system, as shown in Figure 17, where the desired requirement for the main cooling system is a flow rate of 0.3 m/s and a pressure of 5 atm can be achieved.

| Velocity Magnitude | u-Weighted Average | Static Pressure | Volumetric Flow Rate |
|--------------------|--------------------|-----------------|----------------------|
| inlet_cooler_1     | 8.0909979          | inlet_cooler_1  | 5.00000941           |
| inlet_pipes_1      | 10.30011           | inlet_pipes_1   | 39.497332            |
| outlet_pipes_1      | 13.491992          | outlet_pipes_1  | 39.47633            |

**Figure 17.** Screen capture CFD final Simulation output regarding output pressure, velocity, and flow rate as required by the cooling system.

### 4. Result & Discussion

Regarding the results of the analysis, it is obtained that the geometry and material of the pipe are 10 inches in diameter with schedule 80 made of A106 steel and other pipe components are T-reduce (from 10 inches to 6 inches), T-junction, Orifice with a ratio of 0.5 and 0.6, Elbow short radius and long radius and Gate valve. The numerical simulation results show that the MWSP network structure can accept a water flow pressure load that varies from 39 bar - 5 bar which is accompanied by turbulence of the flow which is concentrated especially in the bending component (T-reduce, T-junction, and elbow). The simulation results also show that the use of T-reduce can overcome excess water discharge from 0.44 m³/s to 0.36 m³/s (from a requirement of 0.3 m³/s), a slight excess of 0.04 m³/s is overcome by using a reducing control valve which is placed in the modified pipe end position before entering the cooling system strainer so that it can easily maintain stable pressure during operation. The overall overpressure of 34 bar (39 bar - 5 bar) can be overcome with the elbow-pipe network configuration with 4 orifice configurations on the modification of the piping and is proven to be able to reduce pressure in the range of 4 bar - 6 bar.
5. Conclusion

The numerical simulation results show that the MWSP network structure can accept a water flow pressure load that varies from 39 bar - 5 bar which is accompanied by turbulence of the flow which is concentrated especially in the bending component (T-reduce, T-junction, and elbow).

The optimization that can be obtained by this modification is a savings of up to 52% of the total use of auxiliary power or the equivalent of 2.06 GWH in 2019.

To improve future calculation results, the process of calculating CFD is combined with erosion simulations to obtain the depth of abrasion of the pipe walls by solid particles flowing with water fluid, followed by further calculations with CSS (Computational Static Structural or known as structural stress analysis) to ensure the safety level (safety factor) of the pipe structure, namely the existing penstock pipe and the modified pipe, to obtain an overview of the level of the structure's ability to accept changes in the load acting on the pipe wall in the water supply system network for the unit cooling system.

And for further developments to reduce penstock air pressure for the main cooling system, a deeper study can be carried out in the use of the PAT (Pump As Turbine) concept, where in addition to maintaining the desired cooling water supply (pressure and flow), the pressure expanded in PAT can generate additional electrical energy.

References

[1] Indonesia Power, PT., Report Data Energy Uses and Renewable Energy 2019, Internal Report Corporate, Unpublished Report

[2] Anderson, J. D. J. (1995) 'Computational Fluid Dynamics', Mc Graw Hill International Edition. Mc Graw Hill International Edition.

[3] The American Society of Mechanical Engineers., (2014), Power Piping: ASME Code for Pressure Piping, B31, ASME B31.1-2014.

[4] White, Frank M, (2011), Fluid mechanics, 7th ed, ISBN 978-0-07-352934-9, The McGraw-Hill Companies, Inc

[5] L. F. Moody, 1944, "Friction Factors for Pipe Flow," ASME Trans., vol. 66, pp. 671-684.

[6] Munson, Young, and Okiishi’s, (2016), Fundamentals of Fluid Mechanics, 8th ed, ISBN 978-1-119-08070-1, John Wiley & Sons, Inc.

[7] Brian Silowash (2010), Piping System Manual, ISBN: 978-0-07-159277-2, The McGraw-Hill Companies, Inc

[8] Benedict, R. (1984) Fundamental of Temperature, Pressure, and Flow Measurement. 3rd Edition. John Wiley and Sons.
[9] E N ISO, 5167-1 (2003) 'Measurement of Fluid Flow by Means of Pressure Differential Devices Inserted in Circular-Cross Section Conduits Running Full-Part 1: General Principles and Requirements'. (ISO, 5167-1:2003).

[10] ISO, 5167-2 (2003) 'Measurement of Fluid Flow by Means of Pressure Differential Devices Inserted in Circular-Cross Section Conduits Running Full-Part 2: Orifice Plates, International Standard'.

[11] Babu, M.K.J, Gowda G.C.J, R. K. (2017) 'Discharge Coefficient Prediction for Multi hole Orifice Plate in a Turbulent Flow through Pipe: Experimental and Numerical Investigation', International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE), 2.

[12] Ebrahimi, B. et al. (2017) 'Characterization of high-pressure cavitating flow through a thick orifice plate in a pipe of constant cross section', International Journal of Thermal Sciences. Elsevier Masson SAS, 114, pp. 229-240. doi: 10.1016/j.ijthermalsci.2017.01.001.

[13] Campos, S. R. V. et al. (2014) 'Orifice plate meter field performance: Formulation and validation in multiphase flow conditions', Experimental Thermal and Fluid Science. Elsevier Inc., 58, pp. 93-104. doi: 10.1016/j.expthermflusci.2014.06.018.

[14] Inc, F. (2006) 'Guideline Document', Centerra Resource Park, 10 Cavendish Court, Lebanon, NH 03766, 3766.

[15] Sugianto (2011) Simulasi Numerik Pemisahan Aliran Dua Phase Kerosene-water di dalam T-junction.

[16] Dhairyashil Dhumal, Yashwant More and Ujwal Gawai (2017) 'Design, Fabrication CFD Analysis of Multi-Hole Orifice Plate', International Journal of Engineering Research and, V6(06), pp. 353-357. doi: 10.17577/ijertv6is06161.

[17] Gohil Haikrishna V., A. M. A. (2014) 'Finite Element Analysis of Pressure Drop in Orifice Meter', International Journal of Engineering Research & Technology (IJERT).

[18] Karthik, G. et al. (2015) 'Prediction of Performance Characteristics of Orifice Plate Assembly for Non-Standard Conditions Using CFD', International Journal of Engineering and Technical Research, 3(5), pp. 2321-869.

[19] Kiš, R. et al. (2013) 'The impact of the orifice plate deformation on the differential pressure value', EPJ Web of Conferences, 45, pp. 2-5. doi: 10.1051/epjconf/20134501049.

[20] Singh, R. K., Singh, S. N. and Seshadri, V. (2010) 'Performance evaluation of orifice plate assemblies under non-standard conditions using CFD', Indian Journal of Engineering and Materials Sciences, 17(6), pp. 397-406.

[21] Bunyamin et al. (2019) 'Challenges in Turbine Flow Metering System: An Overview', Journal of Physics: Conference Series, 1198(4). doi: 10.1088/1742-6596/1198/4/042010.