Tidal Effect on Sea Water Intake of Power Plant using CFD Model

Puspa Devita Mahdika Putri* and Suntoyo
Department of Ocean Engineering, Faculty of Marine Technology, Institut Teknologi Sepuluh Nopember (ITS), Surabaya, 60111, Indonesia

Keywords: Computational Fluid Dynamics, Intake Channel, Turbulence, Vortex.

Abstract: To develop the capacity of electricity production in Indonesia, supporting infrastructure such as water intake channel is necessary. By using water intake channel system, power companies can utilize seawater as a cooling power plant. Water from the ocean is pumped into the cooling system to cool the generating engine. In practice, the construction of intake channel often has a problem, especially in the pump section. One of the most common problems is the vibration of the pumps caused by vortex flow. Based on research conducted by Kim et al (2012), one of the causes of vortex flow is the speed difference. The free sea water surface has several characteristics, one of which is sea tides. The tides can cause an acceleration that allows the vortex to occur. For this reason, this paper perform numerical testing to determine how the effect of these tides on the possibility of vortex flow in the intake channel. Moreover, the direction of vortices flow and shape that may occur due to differences in elevation caused by tides is also examined.

1 INTRODUCTION

Grati Block 2 Power Plant with minimum net dependable capacity of 150 MW is located in Desa Lekok, Kabupaten Pasuruan, Indonesia. It utilize circulating-water cooling systems that typically require a number of large-scale, hydraulic pumps to withdraw water from the sea. The system, as sketched in Figure 1, comprises two pump-intake structures, two stop logs, claw screen and revolving chain screen. Warm waters enters each bay through a claw screen and revolving chain screen, and is pumped into a common discharge header.

Many of the large-scale vertical pumps installed in power plants and various pumping stations have experienced some sort of vibration, impeller damage due to local cavitation, or loss of pumping efficiency (Nakato, 1990). It caused primarily by nonuniform pump-approach flow conditions in pump sumps that known to produce prerotation and air-entraining free-surface vortices. These nonuniformities in the intake flow promote vibrations and excessive bearing loads. Low pump intake submergence depths could result in the formation of air-entraining free-surface vortices, a phenomenon that significantly complicates the flow field and promotes cavitation (Constantinescu and Patel, 1998).

Tides are the rise and fall of sea levels caused by the combined effects of the gravitational forces exerted by the moon, the sun, and the rotation of the earth. Most places in the ocean usually experience two high tides and two low tides each day, called semidiurnal tide. Grati Pasuruan East Java sea waters had mixed prevealing semi diurnal tide (Wijaya et al., 2016). The difference of sea levels created by tides can produce the difference velocity magnitude in varies space. Thus it can impacts the flow conditions inside the sea water intake. Fluid velocity becomes faster when the sea levels is higher. It is considered that the unstable flow develops the free surface vortex (Kim et al, 2012).

Factors affecting the formation of vortices at pump intakes have been known in general terms for quite some time, there is no theoretical method for predicting their ocuurence. Hence, it demands the full power of modern computational fluid dynamics (CFD) to solve the equations of motion and turbulence models in domains that involve multiple surfaces.

* Graduate Student

Putri, P. and Suntoyo, .
Tidal Effect on Sea Water Intake of Power Plant using CFD Model.
DOI: 10.5220/0009859102090211
In *Proceedings of the 6th International Seminar on Ocean and Coastal Engineering, Environmental and Natural Disaster Management (ISOCEEN 2018)*, pages 209-211
ISBN: 978-989-758-455-8

2 METHOD AND NUMERICAL SIMULATIONS

For perceiving and describing flow conditions in a system, aside from physical observations, numerical methods are also available. Flow fields can be simulated by using Computational Fluid Dynamics (CFD) techniques. To form the numerical model, the first step is to construct a 3D model of the system in computer environment. In CFD applications, the flow environment is limited by boundary conditions to simulate the surrounding effects on the particular investigation area. The steps of a problem solution in using Autodesk CFD can be listed as follows :

- The flow domain is defined
- Boundary conditions are defined
- Simulation start is given

Three models with different water elevation was simulated to describe the difference result caused by tides. The first water elevation that used in this case is mean sea level (MSL) and the lowest low water level (LLWL). For MSL, the water elevation is 6.67 m from bottom of the intake channel and LLWL is -2.0 m from MSL.

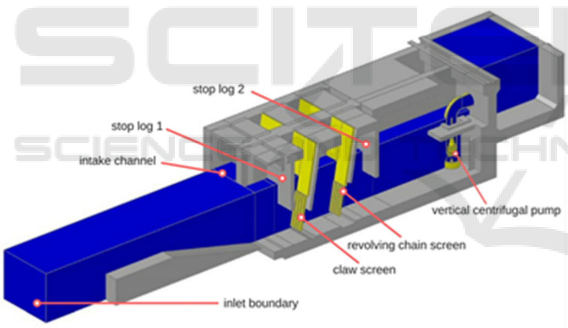

Figure 1: Design of Intake Channel of Grati Power Plant.

2.1 Boundary Conditions

Boundary conditions have important role to create similar flow conditions with the physical system. There are three kind of boundary conditions used for this design. The first boundary conditions is intake boundary that placed in the inlet of the channel. The intake boundary is defined with flowrate 50,000 m^3/s and water temperature 30° C. Then, all region of wall and screen was defined as slip/symetry boundary condition. Rotating region with with a speed of 424 RPM also set for pump sump area. For the outlet boundary, specified pressure 0 Pa boundary condition was set.

2.2 Turbulence Model

In computation of turbulent flows, various turbulence model options are avilable to solve Navier Stokes equations. Prandtl mixing length, one equation turbulent energy model, two equation (k-ε) model, two equation (k-ω) model, Renormalized Group Model (RNG) and Large Eddy simulation model are possible options. In this case, the flow in this model is constrained by a solid wall. The wall no-slip condition ensures that, over some region of the wall layer,viscous effects on the transport processes must be large. The particular turbulence model such as the *k-ε* model are not valid in the near-wall region as shown in Suntoyo et al, 2008, Suntoyo and Tanaka, 2009, Suntoyo et al., 2016 where viscous effects are dominant. Thus, the two equation (k-ω) SST turbulence model works well in the wall-bounded region but needs fine mesh close to the wall (Andersson et al., 2012).

3 RESULT AND DISCUSSION

The result of CFD model using Autodesk CFD can be analyzed by 2 variables, velocity magnitude and vorticity magnitude. Two models with different sea levels was simulated. The difference of velocity magnitude and vorticity magnitude of two models showed in Figure 2 - Figure 5.

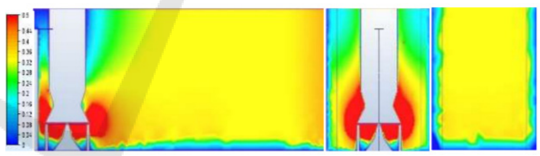

Figure 2: Velocity Magnitude LLWL.

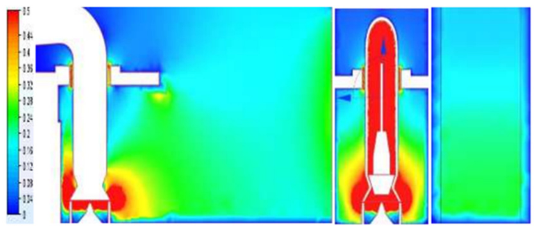

Figure 3: Velocity Magnitude HHWL.

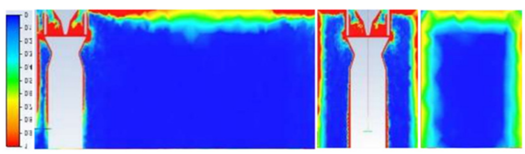

Figure 4: Vorticity Magnitude LLWL.

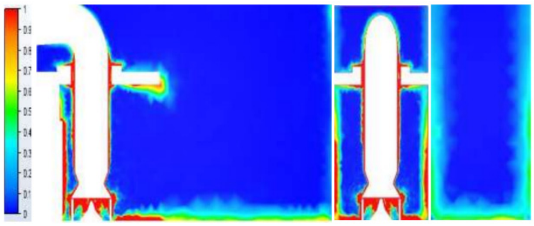

Figure 5: Vorticity Magnitude HHWL.

Figure 2 and 3 show that the average velocity magnitude near pump sumps were increased at lower sea levels (LLWL). The average velocity at HHWL and LLWL is about 1.36 m/s and 1.42 m/s. It was observed that, due to the changed of sea levels, the average velocity magnitude also changed. It was also happened to vorticity at intake. Figure 4 and 5 show that the average vorticity near pump sumps were increased at smaller sea levels (LLWL). The average vorticity at HHWL and LLWL is about 18.33 spin/s and 19.1 spin/s. It was also observed that, due to the changed of sea levels, the average vorticity also changed.

4 CONCLUSIONS

According to the results from the CFD model, flow velocity at LLWL is higher than HHWL. It means, the lower sea level, the faster flow velocity. The fastest flow velocity occured arround pump suctions, at both HHWL and LLWL. Fluid vorticity also become wider when the sea level is lower. It can concluded that tides can impacts the flow characteristics inside the intake and can produce the difference of flow velocity that caused vortex flow.

REFERENCES

Andersson, B., Andersson, R., Hakansson, L., Mortensen, M., Sudiyo, R., L. B. van Wachem, Hellstrom, 2012. *Computational Fluid Dynamics for Engineers.* Cambridge University Press. UK.

Kim, C.G., Choi Y.D., Choi, J.W., Lee, Y.H., 2012. A Study on the Effectiveness of an Anti Vortex Device in the Sump Model By Experiment and CFD. *IOP Conference Series Earth and Environmental Science* 15(17): 1-11.

Constantinescu, G.S., Patel, V.C., 1998. Numerical Model for Simulation of Pump-Intake Flow and Vortices. *Journal of Hydraulic Engineering*. 124 (2): 123-134.

Nakato, T., 1990. A Hydraulic Model Study of the Proposed Pump-Intake and Discharge Flume: Florida Power Corporations Crystal River Helper Cooling-Tower Project. *IIHR Technical Report No*. 339.

Suntoyo, Tanaka H., Sana A., 2008. Characteristics of turbulent boundary layers over a rough bed under saw-tooth waves and its application to sediment transport, *Coastal Engineering* 55 (12): 1102-1112.

Suntoyo, Tanaka H., 2009. Effect of bed roughness on turbulent boundary layer and net sediment transport under asymmetric waves, *Coastal Engineering* 56 (9): 960-969.

Suntoyo, Fattah A.H., Fahmi M.Y., Rachman T., Tanaka H., 2016 Bottom shear stress and bed load sediment transport due to irregular wave motion, *ARPN Journal of Engineering and Applied Sciences* 11(2): 825-829.

Wijaya, M. M., Suntoyo, Damerianne, H.A.. 2016. Bottom shear stress and bed load sediment transport formula for modeling the morphological change in the canal water intake, *ARPN Journal of Engineering and Applied Sciences* 11(4): 2723-2728.