Simulation Bursting Effects To The Performance Vertical Axiz River Turbine Using Computational Fluid Dynamics

Aank Suseno[1], Ridho Hantoro[1], and Gunawan Nugroho[1]

Abstract-Turbulent flow in the open channel, often exhibit a phenomenon bursting in expansion or inhibition case. This phenomenon disturbing to the flow at the turbine area. In this study, Research to analyze the occurrence of interference bursting in the river turbine area, done by simulating the river flow with a variety of depths ratio y/L, using vertical turbine NACA 0018 in a trapezoidal channel, with k-ε turbulence models in CFD. From the simulation results the smaller ratio of the depth y/L, the force of turbulence flow (bursting flow) at the turbine increases.

Index Terms - Channel, CFD k-ε, Bursting.

INTRODUCTION

If the channel undergoing expansion, there will be a behavior experienced positive pressure gradient and unstable flow, the flow make recirculation zone in the area around the wall. In this zone, the direction opposite to the direction of the main stream flow. These conditions resulted unstable and turbulent nature flow. At the zone of turbulent flow structure will be formed periodically through is called the phenomenon of bursting. At certain recirculation zone velocity fluctuations become large and may exceed the average speed of the flow [1]. The researchers partly reviewing and searching for the best solutions that approach the natural results (experimental) fluid behavior in the various forms of open channels, especially in sea and river cannel [2]. One of the studies apply a 3D numerical models to calculate the flow in a curved open channel, which solves the Reynolds-averages fully 3D Navier-Stokes equations With turbulence models [3,4]. Non linear simulation model of turbulence k-ε introduced in this study [5,7]. Non linear simulation model of 3D turbulence k-ε present good results to investigate the flow structure, the distribution of velocity and mass transport processes in the various forms of surface channels. Kamel Benoumessad's study refers to a solution solving the Navier-Stokes equations in cylindrical coordinates Reynolds turbulence model with standard k-ε [6]. Using the same model of turbulence, this research was conducted simulated river flow on a vertical turbine. The main contribution of this study focuses on the analysis characteristics of turbulent flow, analysis the formation of the bursting phenomenon in trapezoidal channels and bursting phenomenon effect to the performance of the turbine.

METHOD

Geometry models in this simulation are turbine NACA 0018 and trapezoidal cross section channel. Geometry model turbine and the channel has dimensions as shown in Figure 1.

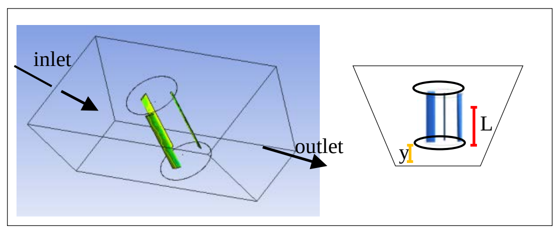

Figure 1. Geometry Model In Simulation.

Geometry model turbine and channel as shown in Figure 3.1 then simulated with some variation of the ratio depth of treatment turbine (turbine placement) y/L and the flow velocity (v). Simulations performed using trapezoidal channel models with a channel length of 270 cm, 140 cm depth flow and free surface width of 120 cm. Turbine used is NACA 0018 with a turbine diameter D = 40 cm of length L = 80 cm, the turbine is located at a distance of x = 110 cm from the beginning of the main channel (inlet). The position of the turbine is placed at a height y from the base channel. Height ratio of the turbine to be simulated by varying the value of y/L and velocity flow (v). In this study used a model of turbulence k-ε because this model has good accuracy to observe the turbulent flow in the turbine wall to areas away from the turbine. Where convergence in this study achieved with 500 iterations. The magnitude of the forced bursting flow is calculated using a turbine wheel that forces fluid flow around the turbine rotates with vortex force (Fc) on the surface of the water within (r) from the center axis of the rotary flow. Forced rounds of this turbine will cause the fluid centrifugal force is proportional to the weight of the fluid particles (w) and the rotation of the particles urgent

[1]Aank Suseno, Ridho Hantoro, and Gunawan Nugroho are with Departement of Physics Engineering, Faculty of Industrial Technology, Institut Teknologi Sepuluh Nopember, Surabaya, 60111, Indonesia. E-mail: mas.aank1@gmail.com; ridho_hantoro@yahoo.com; Gunawan_nugroho@gmail.com.

compulsion (ω). The equation of vortex centrifugal force (Fc) is

$$Fc = \frac{W}{g}\omega r^2 \quad (2.1)$$

RESULTS AND DISCUSSION

Bursting phenomenon arising at turbine has a different value of bursting forces and area (radius of vortex). Differences flow rate also affects the size of the force vortex bursting. High flow rate causes the flow of oncoming fill the area behind the turbine has a higher value, but the style vortex is formed not by force vortex caused by the flow at low speed. This is due to the flow of the turbine rotation speed increases, so there is a lot of fluid mass that rotates behind the turbine due to carry over into the downstream flow. Flow at high speed which has low vortex style also has a sweep radius lower than the flow at low speed. This is due to the rapid recirculation event that caused the flow.

By increasing the ratio of y/L, at the same speed conditions, causing style downward spiral. This decrease was due to the influence of pressure ekstrac. Relations depth ratio y/L with style vortex that occurs is shown in Figure 2.

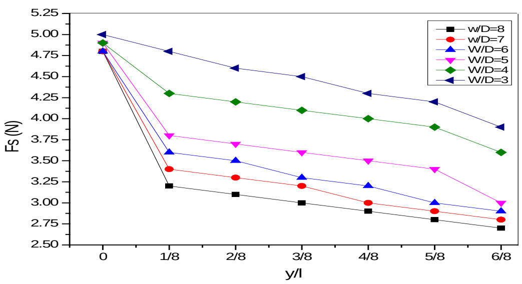

Figure 2. The Influence Of The Ratio Of The Depth Y/L To Force The Vortex (Fs).

decreased flow velocities inlet cause increased flow turbulence as well as the style vortex occurs, vortex style foil blocking flow increased due to the longer time compared to the high flow rate. Figure 3 shows the relationship between the time a turbine wheel with whirlpool style

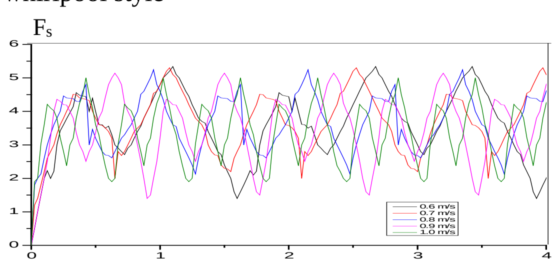

Figure 3. Bursting Force (Fs) Of The Time Vortex Turbine Wheel (t).

Bursting phenomenon that occurs behind the turbine speed in the opposite direction to the direction of rotation of the turbine, so it can be a nuisance drag force on each foil turbine. But this effect does not last long because every bursting released by each foil on the turbine will soon be filled by the next foil.

CONCLUSION

Based on the research concluded that the different pressure distribution flow at the turbine area because of turbin blockage, making the flow turning from high pressure into the direction toward the lower pressure, create bursting which disrupt the performance of turbine. The bursting force against the direction of rotation of the turbine. And obtained the bursting force will be increased by lowering the ratio of the depth and lower flow velocity.

REFERENCES

[1] M. Syahril B. Dkk. Studi Pengembangan Model Turbulen κ-ε untuk Sirkulasi Arus II: Aliran Turbulen Dua Dimensi pada Saluran Ekspansi. PROC. ITB Sains & Tek, 2007;Vol. 39 A No. 1&2:70-96.

[2] H. Shamloo. Analysis of roughness density and flow submergence effects on turbulence flow characteristics in open channels using a large eddy simulation. Applied Mathematical Modelling, 2015, vol. 39, pp. 1074–1086.

[3] Y. Shimizu, H. Three-dimensional computation of flow and bed deformation. Journal Hydraul, vol. 116, pp.1090–1108, 1990.

[4] T. Kawamura. Numerical simulation of 3D turbulent free-surface flows. International Research Center for Computational Hydrodynamics, Horsholm, Denmark, 1998.

[5] L. Shen. Large-eddy simulation of free-surface turbulence. The Journal of Fluid Mechanics, vol. 440, pp. 75-116, 2001.

[6] K. Benoumessad. 3D simulation of velocity profile of turbulent flow in open channel with complex geometry. Physics Procedia, 2014, vol. 55, pp. 119–128.