Flow investigation of cross-flow turbine using CFD method

Flow investigation of cross-flow turbine using CFD method

Joshua Aditya Sardjono[1], **Steven Darmawan**[1], **Harto Tanujaya**[1]

[1]Mechanical Engineering Department, Faculty of Engineering
Universitas Tarumanagara, Jakarta, Indonesia

joshua.515160021@stu.untar.ac.id (J. A. Sardjono)

Abstract. Cross-flow turbine is one of prime mover type that is simple, capable to extract water power and easy to be manufactured. The runner is working at 2 stages. The need of high momentum has lead researchers to design water nozzle, blades, as well as the set-up between nozzle and runner. High momentum water nozzle and cross-flow runner has been designed individually with the same design criteria. Unfortunately, experimental testing may not gain specific flow pattern inside the nozzle and the runner. The purpose of this study is to investigate the flow pattern inside the nozzle, runner using CFD method, Ansys 2019 R3 academic version. The CFD simulation done 3-dimensionally with 5 nozzle inlet velocity variations: 2 m/s, 3 m/s, 4 m/s, 5 m/s and 6,487 m/s. The fluid is assumed to be 1 phase, using mesh with 17.750 nodes, inlet fluid pressure 1,05 Pa, and temperature 24,85^{o}C. The results showed that the lowest velocity and pressure for variation 1 at 1st stage (v1) = 6,65 m/s and (p1) = -11931,5 Pa, and the highest velocity and pressure at 1st stage (v5) = 21,72 m/s and (p5) = -127521 Pa. At 2nd stage, variation 1 and 5 resulting velocity (v1) = 1,78 m/s and (p1) 593,264 Pa, (v5) = 5,83 m/s and (p5) = 6360,32 Pa respectively. The main different of the flow pattern between 5 inlet velocity variations is the final flow thickness inside the runner get thicker at higher velocity, which is assumed to gain more rotational velocity.
Keywords: Cross-flow turbine, nozzle, CFD

1. Introduction

Today energy are the most important thing for human life, energy from the micro-hydro scale needed because clean, sustainable and renewable energy source [1]. Cross-flow runner have a simple construction, low cost and maximum efficiency which converting kinetic energy to mechanical energy [2]. Elbatran et al., 2015b, turbine is one of the most costly parts, it can even reach up to 30% of the total budget for micro power scheme, so the cost depends on the types of turbines. Olgun, 1998, Cross-flow turbines are more preferable for the micro hydro power scale compared to other turbines [3]. At present, Cross-flow turbines are gaining popularity, this is due to their simple structure and ease of manufacturing [4]. Akerkar, the jet angle from vertical position of the nozzle more greater than horizontal position of the nozzle [5]. Rotational vortices enchane transfer of the flow, that make transient vortices more faster than stationary vortices [6]. Cross-flow turbine system is design as well as the nozzle inlet by the previous researchers [7], [8]. In order to gain the optimal set-up of the nozzle and the runner, CFD simulation is needed to visualize the flow pattern inside runner. This study use water-liquid as fluid and fluid condition assumed to be one phase with temperature 24,85°C. The CFD simulation were done 3-dimensionally with boundary condition as follows: inlet pressure at 1,05 Pa and 5 varitions of nozzle inlet and shaft speed are, 2 m/s (300 rpm shaft speed), 3 m/s (450 rpm shaft speed), 4 m/s (600 rpm shaft speed), 5 m/s (750 rpm shaft speed) and 6,487 m/s (1000 rpm shaft speed). Inlet pressure 1,05 Pa. The result of this simulation will be analyzed the flow that occurs in Cross-flow turbine, and can be a reference for further research, especially the system set-up between the nozzle and runner.

2. Method and materials

2.1. Design

The geometry models of nozzle and runner from previous study [7], [8]. Nozzle geometry, total length = 390.98 mm, width = 84 mm, inlet = 78 mm and 20 mm, and outlet = 78 mm and 20 mm. Geometry of runner, total diameter = 149 mm, shaft diameter = 15 mm, length of blades = 65 mm, total blades = 15 and $\alpha_1 = 16^o$ [7], [8].

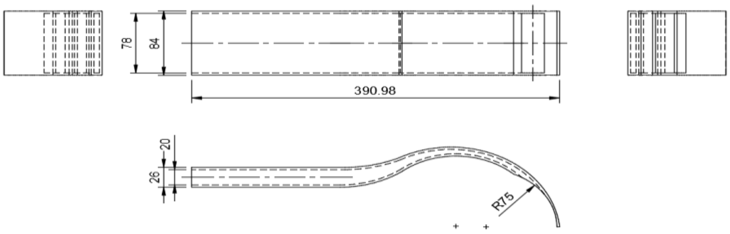


Figure 1. Nozzle geometry [7]

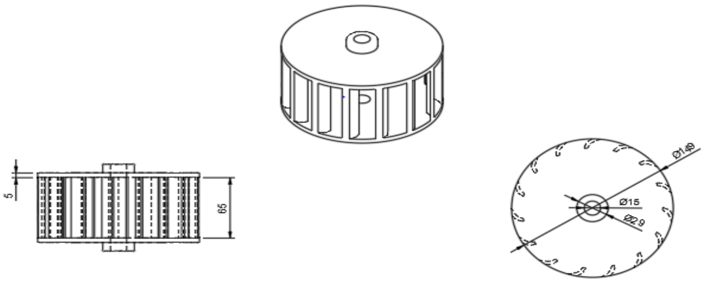


Figure 2. Runner geometry [8]

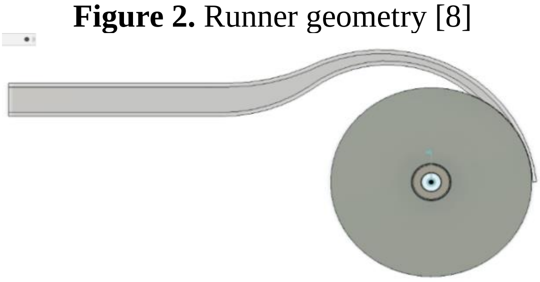


Figure 3. Assembly of nozzle and runner [7]

2.2. Method

Geometry design of nozzle and runner already explain before, this simulation using Ansys 2019 R3 (education licensed). There are 3 CFD methods, pre-processor, solver and post processor [9]. First is pre-processor, making computational domain by using "enclosure, and fill the nozzle using "fill". Next is meshing, this needed for analyzed the models, nodes and element, by using general mesh type with nodes 17.750 and elements 76.406 that are shown at Figure 4. Last step is set up, input data from viscous models (k-epsilon), fluid material (water-liquid), and boundary condition see at Table 1. Second is solver, after input all data click initialization and run calculation. Third is result, the result can show by velocity and pressure.

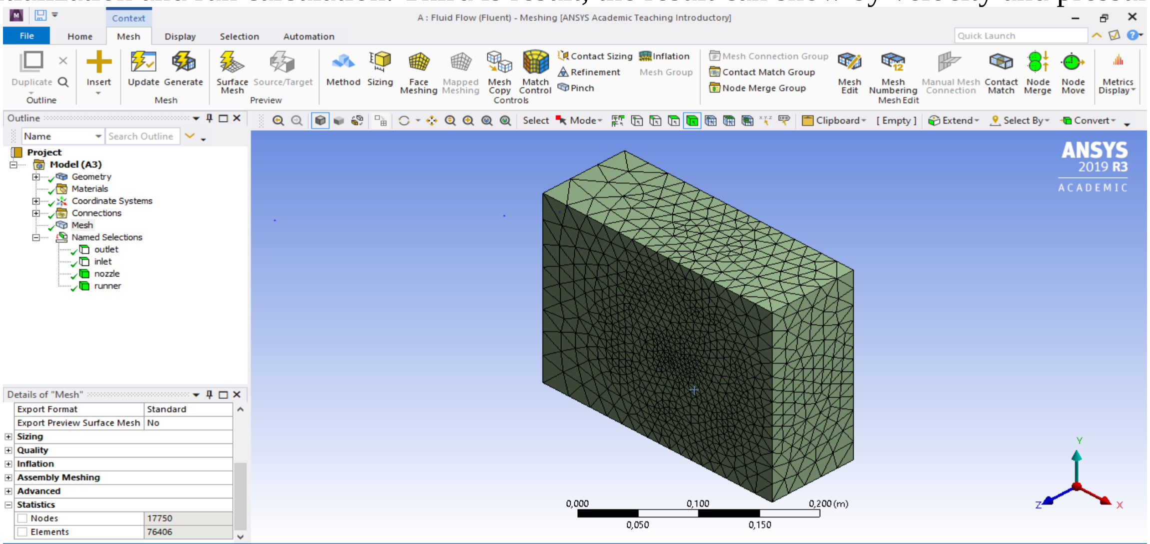


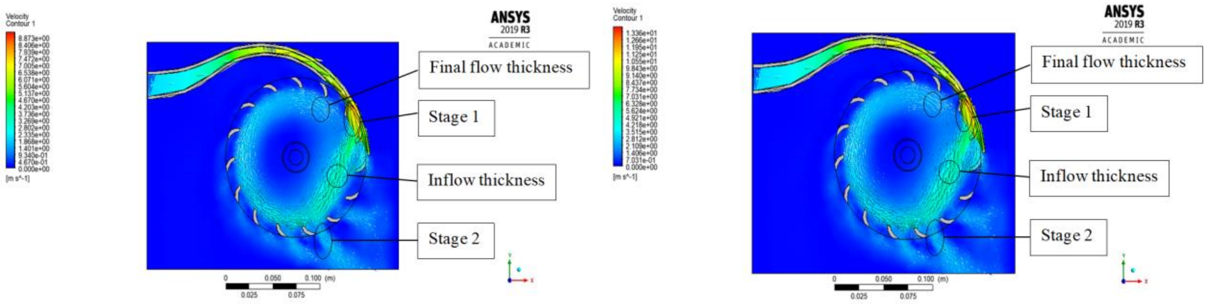
Figure 4. Meshing

Table 1. Boundary condition

Inlet Velocity (m/s)	Inlet Pressure (Pa)	Shaft Speed (Rpm)	Fluid Temperature (^{o}C)
2		300	
3		450	
4	1,05	600	24,85
5		750	
6,487		1000	

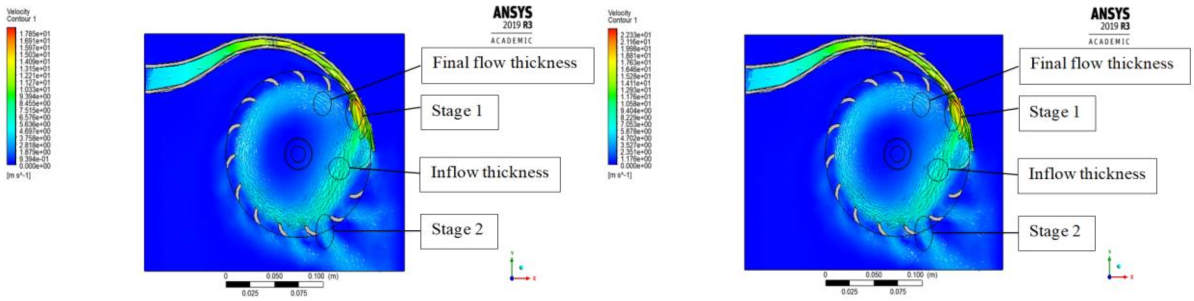
3. Results and discussion

The results shown by contour and vector for velocity and contour only for pressure, pressure doesn't have vector because pressure is scalar and pressure doesn't have direction like velocity. The parameter contour show on the left of each simulation results. For velocity show by meter per seconds (m/s) at Figure 5 – Figure 7 and pressure show by Pa (Pascal) at Figure 8 – Figure 10. With 5 variations are, variation 1 (2 m/s 300 rpm), variation 2 (3 m/s 450 rpm), variation 3 (4 m/s 600 rpm), variation 4 (5 m/s 750 rpm) and variation 5 (6,487 m/s 1000 rpm)



(a) (b)

Figure 5. Velocity variation 1 (a), velocity variation 2 (b)



(c) (d)

Figure 6. Velocity variation 3 (c), velocity variation 4 (d)

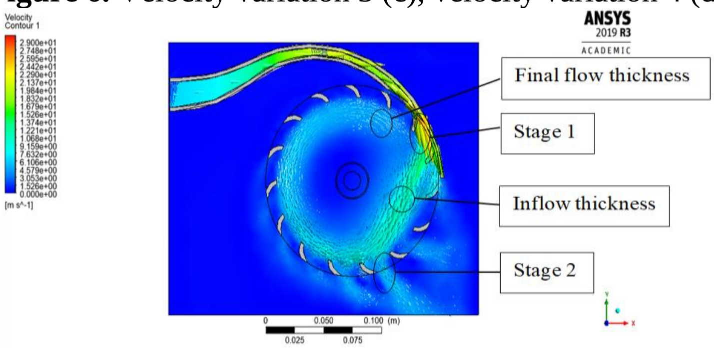
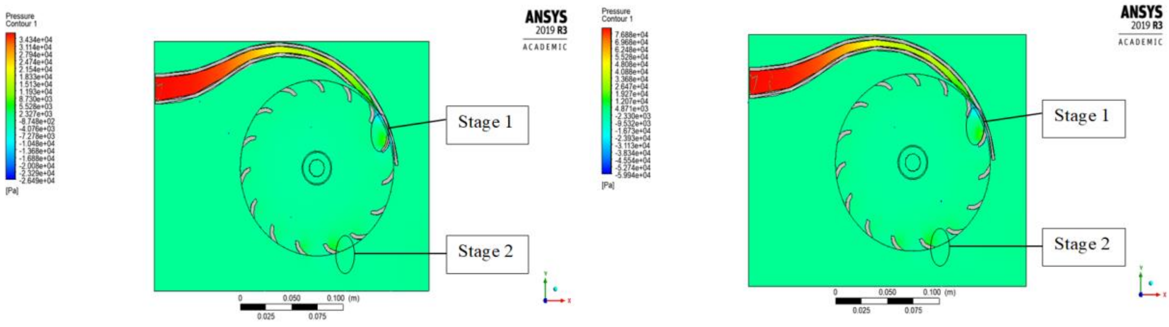


Figure 7. Velocity variation 5

Table 2. Velocity in the area of each variations

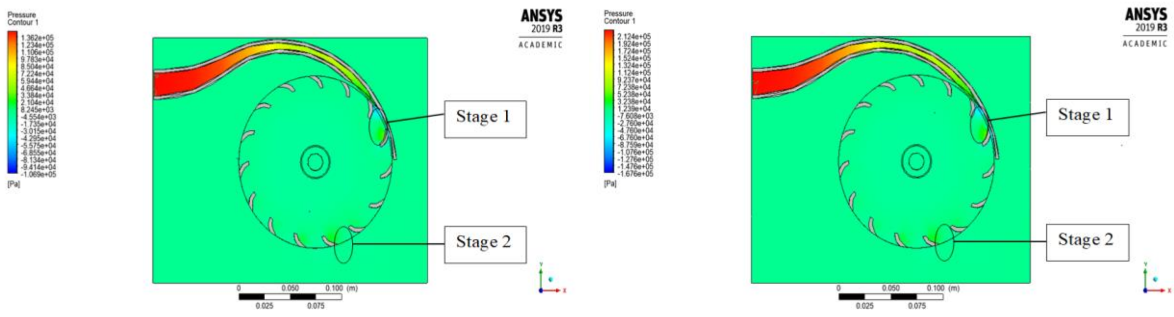
Variation	Stage 1 (m/s)	Stage 2 (m/s)	Inflow thickness (m/s)	Final flow thickness (m/s)
1	6,65	1,78	3,46	1,02
2	10	2,66	5,20	1,54
3	13,37	3,61	6,95	2,05
4	16,72	4,44	8,68	2,54
5	21,72	5,83	11,27	3,32

Velocity from nozzle inlet slower than nozzle outlet, because the geometry of nozzle. At nozzle outlet narrowing occurred, this make pressure from nozzle inlet decrease when flowing at nozzle outlet. So this phenomenon affecting stage 1 and stage 2 for runner, the velocity from stage 1 more faster than stage 2, because stage 1 receive flow from nozzle outlet and stage 2 receive flow from stage 1. The flow profile show that the water flowing through runner at clockwise for each variation, rotational vortices can be seen at the flow of water goes with the flow and recycling at final flow to inflow shown by vector velocity see Fig. 5 – Fig. 7. At stage 1, not all the water flow through runner so this make rotation of runner not maximum. Stage 1 make inflow thickness after receive flow from outlet nozzle and stage 2 make final flow thickness, the velocity at inflow more faster than final flow. The rotation of runner affecting the thickness of flow, so the faster runner rotate the more thicker the flow. The result of variations show at Table 1, variations 5 is the fastest velocity from all variations.



(e) (f)

Figure 8. Pressure variation 1 (e), pressure variation 2 (f)



(g) (h)

Figure 9. Pressure variation 3 (g), pressure variation 4 (h)

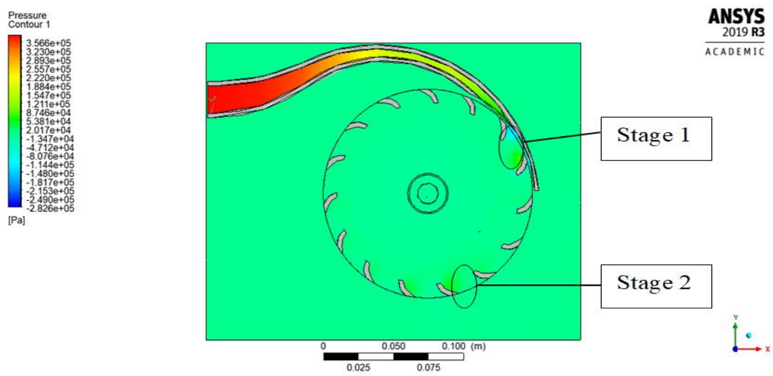


Figure 10. Pressure variation 5

Table 3. Pressure in the area of each variations

Variation	Stage 1 (Pa)	Stage 2 (Pa)
1	-11931,5	593,264
2	-27017,1	1384,33
3	-48203,1	2414,89
4	-75523,7	3823,02
5	-127521	6360,32

The comparasion of velocity in stage 1 and stage 2 from Table 1 with pressure from Table 2, the faster velocity make pressure lower. It happened at stage 1 and stage 2, the flow from outlet nozzle to stage 1 for each variations decreased pressure as the flow through to runner and same condititions at stage 2 too. The pressure of stage 1 have minus result it can't be conclude has no pressure, the pressure at nozzle inlet make flow can through from nozzle to runner. The different pressure from huge to small, make the water can flowing to runner. It is because Bernoulli's law, speed is inversely proportional to pressure, the faster of speed the slower of pressure and vice versa.

4. Conclusion

From result of this study can be concluded inlet velocity and shaft speed influenced the flow in Cross-flow turbine. Velocity triangle of Cross-flow turbine have 2 stage, stage 1 (runner inlet) and stage 2 (runner outlet). Which each stage have different velocity and pressure, stage 1 more faster than stage 2 for velocity but pressure at stage 2 more bigger than stage 1. Variations 5 have biggest velocity and pressure than other variations, that made the inflow thicknesss and final flow thickness in runner wider than other variations, including the vector velocity, which the most important flow that will be converted into shaft power. The experimental method of this set up is needed to gain the actual performance of the system.

5. References

[1] Rantererung C L, Soeparman S, Soenoko R and Wahyudi S 2016 *Dual Nozzle Cross Flow Turbine as an Electrical Power Generation,* vol. *XI*, pp. 15-19.
[2] Darmawan S, Siswantara A I, Budiarso, Daryus A, Gunawan A T, Wijayanto A B and Tanujaya H 2015 *Turbulent flow analysis in auxiliary cross-flow runner of a Proto X-3 Bioenergy micro gas turbine using RNG K-ε turbulence model*, vol. **X**, no.16, pp. 7086-7091.
[3] Elbatran A H, Yaakob O B, Ahmed Y M and Shehata A S 2018 *Numerical and Experimental Investigations on Efficient Design and Performance of Hydrokinetic Banki Cross Flow Turbine for Rural Areas*, vol. **159**, no. Desember 2017, pp. 437-456.
[4] Patel M and Oza N 2016 *Design and Analysis of High Efficiency Cross-flow Turbine for Hydro-Power Plant*, vol. **V**, no. 4, pp. 187-193.
[5] Oliy G B and Ramayya A V 2017 *Design and Computational Fluid Dynamic Simulation Study of High Efficiency Cross Flow Hydro-power Turbine Gutu*, vol. **V**, no. 4, pp. 120-125.

[6] Griffani D, Rognon P, Metzger B and Einav I 2013 *How rotational vortices enchane transfers*, vol. **25**, no. 9, pp. 1-8.
[7] Frandy 2019 *Pengembangan Nosel Turbin Cross Flow dengan Menggunakan Simulasi CFD* (Jakarta: Universitas Tarumanagara)
[8] Christianto R 2019 *Proses Manufaktur dan Unjuk Kerja Hydro Turbine Jenis Cross-Flow dengan Material Al6061 dan SS304* (Jakarta: Universitas Tarumanagara)
[9] Versteeg H K and Malalasekera W 2007 *An Introduction to Computational Fluid Dynamic: the finite volume method* Pearson Education Limited.