Abstrak

Turbin *Cross-flow* merupakan salah satu penggerak utama yang simpel, dapat mengekstrak daya air dan mudah dibuat. *Runner* bekerja dalam 2 *stage*. Keperluan momentum yang tinggi membuat peneliti mendesain *nozzle* air dan *runner Cross-flow* telah di desain sendiri dengan kriteria desain yang sama. Akan tetapi, uji coba eksperimen tidak mendapatkan pola aliran yang spesifik pada *nozzle* dan *runner*. Tujuan dari penelitian ini untuk menginvestigasi pola aliran didalam *nozzle*, *runner* dengan menggunakan metode CFD, *Ansys* 2019 R3 *academic version*. Simulasi CFD dilakukan secara 3 dimensi dengan 5 variasi kecepatan *inlet nozzle*; 2 m/s, 3 m/s, 4 m/s, 5 m/s and 6,487 m/s. Fluida di asumsi sebesar 1 fase, menggunakan *mesh* dengan *nodes* 17.750 dan *elements* 76.406, tekanan fluida di *inlet nozzle* 1,05 Pa, dan temperature $24{,}85^{o}$C. Hasil dari simulasi pada kecepatan dan tekanan terendah di variasi 1 pada *stage* 1 (v1) = 6,65 m/s dan (p1) = -11931,5 Pa, dan kecepatan dan tekanan tertinggi di variasi 5 pada *stage* 1 (v5) = 21,72 m/s dan (p5) = -127521 Pa. Pada *stage* 2, variasi 1 and 5 menghasilkan (v1) = 1,78 m/s and (p1) 593,264 Pa, (v5) = 5,83 m/s and (p5) = 6360,32 Pa. Perbedaan utama pada pola aliran antara 5 variasi kecepatan *inlet* terletak pada ketebalan aliran akhir yang berada didalam *runner* lebih tebal pada kecepatan tinggi, yang diasumsikan mendapatkan lebih banyak kecepatan rotasi. Perbedaan kecepatan dan tekanan pada variasi 1 dan variasi 5 sebesar $^{-}/_{+}$ 325% dan $^{-}/_{+}$ 1070%.

Kata kunci: turbin *cross-flow*, *nozzle*, CFD

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 and 76.406 elements, nozzle inlet fluid pressure 1,05 Pa, and temperature 24,85^{o}C. The results show 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 for variation 5 at 1st stage (v5) = 21,72 m/s and (p5) = -127521 Pa. At 2nd stage, variation 1 and 5 resulting (v1) = 1,78 m/s and (p1) 593,264 Pa, (v5) = 5,83 m/s and (p5) = 6360,32 Pa. The main different of the flow pattern between 5 inlet velocity variation is the final flow thickness inside the runner get thicker at higger velocity, which is assumed to gain more rotational velocity.The difference in speed and pressure at variation 1 and variation 5 are $^{-}/_{+}$ 325% *and* $^{-}/_{+}$ 1070%.

Keywords: *cross-flow turbine, nozzle, CFD*