NUMERICAL SIMULATIONS OF COUNTER-ROTATING PUMP-TURBINE WITH A NEW HEAD-LOSS PRESSURE BOUNDARY

Citation for the original published paper (version of record):
Fahlbeck, J., Nilsson, H., Salehi, S. (2021). NUMERICAL SIMULATIONS OF
COUNTER-ROTATING PUMP-TURBINE WITH A NEW HEAD-LOSS PRESSURE
BOUNDARY
CONDITION. OpenFOAM Workshop (OFW16) Book of abstracts

NUMERICAL SIMULATIONS OF COUNTER-ROTATING PUMP-TURBINE WITH A NEW HEAD-LOSS PRESSURE BOUNDARY CONDITION

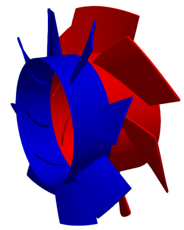
JONATHAN FAHLBECK[1,*], HÅKAN NILSSON[1], SAEED SALEHI[1]
[1] *Department of Mechanics and Maritime Sciences, Chalmers University of Technology, Sweden*
*Email for correspondence: fahlbeck@chalmers.se

Keywords: *Pump-turbine, pump-storage, counter-rotating, headLossPressure, head-losses*

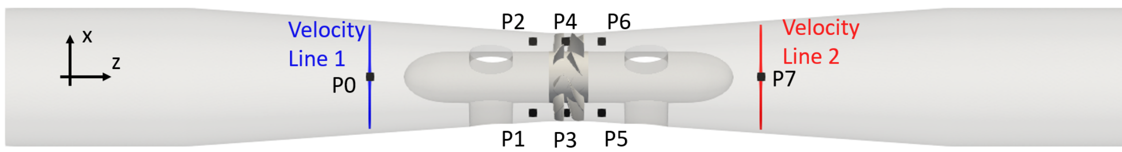
With an increasing amount of energy from renewable sources, such as wind and solar, the need of complementary controllable energy sources increases. Hydropower plays a key role to provide a stable and flexible electrical grid. By storing large amount of water when there is excess power in the grid, and later utilise the stored water when there is a lack of power, hydropower is a stabilising unit for the electrical grid [1]. The ALPHEUS EU project has the aim to develop a low-head to ultra low-head seawater based Pumped Hydropower Storage (PHS) solution with a pump-turbine unit [2, 3]. The main goal of the ALPHEUS project is that the pump-turbine unit should have a round-trip efficiency of 70 - 80 % and a switching time of about 120 s. PHS use the potential energy by pumping water to a reservoir. The potential energy is later extracted by reversing the pump to a turbine. Three pump-turbine concepts are to be investigated, a counter-rotating shaft-driven, a counter-rotating rim-driven, and a positive-displacement alternative. A rigorous optimisation process will be applied to maximize the round-trip efficiency for a wide range of operating conditions. In this work an initial design of a counter-rotating shaft-driven alternative is considered. In the ALPHEUS project an optimised counter-rotating shaft-driven pump-turbine will be experimentally evaluated in model scale, in the hydraulic laboratory at TU Braunschweig. The experiments are partly made in order to generate experimental test data. The numerical models are later going to be evaluated with the experimental test data. The experimental test facility consists of a two open reservoir surfaces, upper and lower. In turbine-mode, water flows from the upper to the lower reservoir, and it is pumped from the lower to the upper in pump-mode. The reservoirs are connected with a series of pipes, including bends and other obstacles in the flow path. The machine is going to be tested at different operating conditions and it is thus hard to estimate the flow rate for any given case. This is because head, or pressure, losses scale quadratic to a change in flow rate. An option to overcome this problem in a numerical framework is to include head losses at the boundaries of the computational domain. The flow rate in the simulation is calculated as a balance between the available pressure and the losses in the system. In OpenFOAM there is not any available boundary condition that can include up-/down-stream losses at a patch. This present work demonstrates a new pressure boundary condition, `headLossPressure`, developed by Fahlbeck [4]. The boundary condition is an extension of the available `totalPressure` boundary condition. It uses the volumetric flux to adjust the static pressure on the patch according the Bernoulli equation [5]. The `headLossPressure` is an incompressible pressure boundary condition for in-/outflow patches. If the patch has inflow the losses are subtracted, and for an outflow patch the losses are added.

The basic functionality of the `headLossPressure` boundary condition is evaluated on a simple test case by Fahlbeck [4]. In this work the boundary condition is used together with the initial design of a model scale counter-rotating shaft-driven pump-turbine in the ALPHEUS project. The blade geometries shown in Figure 1a were designed by the Advanced Design Technology Ltd (ADT) company. The diameter of the runners is 27 cm, runner 1 (red) has eight blades, and runner 2 (blue) has seven blades. Runner 1 has a rotational speed of 1453 RPM in pump mode and 832 RPM in turbine mode, runner 2 rotates at 90 % of the speed of the first runner in each mode.

The numerical simulations are made on the computational domain shown in Figure 1b. The numerical simulations are made with unsteady CFD at one operating condition in both pump and turbine modes. The numerical framework includes the two rotating runners, hub, support-struts, and contraction/extraction parts. The simulations utilise the unsteady incompressible `pimpleFoam` solver and the k-ω SST model is used to account for turbulence. The convective terms of the momentum equations are discretised using the LUST scheme, and temporal discretisation with the backward scheme. The `pressureInletOutletVelocity` and `headLossPressure` are used as boundary conditions for velocity and pressure, respectively, at both the inlet and the outlet. The pressure boundary condition is set to operate with a total height difference of 8 m, the full pipe length is roughly 16 m, one 90° bend, and some additional flow obstacles are included.



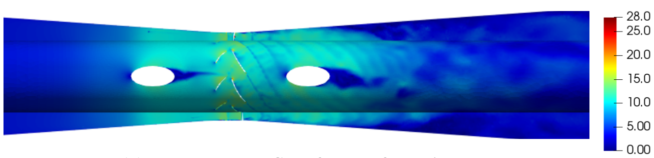
(a) Runners 1 (blue) and 2 (red).



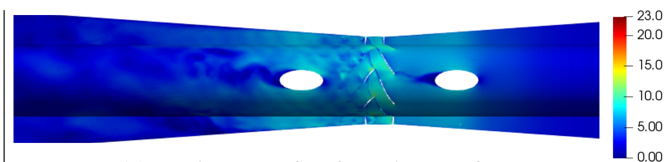
(b) Computational domain with the location of velocity lines (Line 1 and 2) and pressure probes (P0 - P7) shown. The flow is from left to right in pump-mode and from right to left in turbine-mode. The total length of the domain is 12.8 m.

Figure 1: Geometry and computational domain.

The results from the unsteady simulations, shown in Figure 2, resolves the unsteady wakes of the runners and the support-struts. The complex flow pattern produced by the runners is caused by the downstream runner cutting the wakes of the upstream runner. The machine is operating at a high efficiency in both modes as the flow is rather axial after the runners. This is seen by that the vortex shedding of the support-strut is rather axial. A frequency analysis, not shown here, uncover that the pressure pulsations in the system are strongly connected to the blade passing frequencies and linear combinations of it.



(a) Pump-mode, flow from left to right.



(b) Turbine-mode, flow from right to left.

Figure 2: Velocity magnitude snapshots on a plane and a cylinder cutting the domain.

The `headLossPressure` boundary condition can be used to produce a plausible flow field as the solver calculates a flow rate that is not totally unphysical. The question still remains if the flow rate is correct and if the boundary condition can be used even for transient simulations. The numerical model and this new boundary condition will later be compared against experimental test data of an optimised counter-rotating pump-turbine.

Acknowledgments

The authors thank all those involved in the organisation of OFW16 and to all the contributors that will enrich this event.

This project has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No. 883553.

The computations were enabled by resources provided by the Swedish National Infrastructure for Computing (SNIC) at NSC and C3SE partially funded by the Swedish Research Council through grant agreement No. 2018-05973.

References

[1] IEA, *Will pumped storage hydropower expand more quickly than stationary battery storage?* IEA, Paris, 2019. [Online]. Available: https://www.iea.org/articles/will-pumped-storage-hydropower-expand-more-quickly-than-stationary-battery-storage

[2] ALPHEUS H2020. Accessed: 2021-02-12. [Online]. Available: https://alpheus-h2020.eu/

[3] M. Qudaih and et al., "The contribution of low-head pumped hydro storage to a successful energy transition," in *Proceedings of the Virtual 19th Wind Integration Workshop*, 2020.

[4] J. Fahlbeck, "Implementation of an incompressible headlosspressure boundary condition," in Proceedings of CFD with OpenSource Software, 2020, Edited by Nilsson. H., http://dx.doi.org/10.17196/OS_CFD#YEAR_2020.

[5] F. M. White, *Fluid mechanics*, ser. McGraw-Hill series in mechanical engineering. McGraw-Hill, 2011.