Proceedings of:
Current Research in Hydropower Technologies, CRHT X
Kathmandu University, 2020
CRHTX-18

Numerical simulation of cavitation on a Reversible Pump Turbine

Johan C Jenssen

Waterpower laboratory, Department of Energy and Process Engineering, NTNU, Trondheim, Norway

E-mail: johancj@stud.ntnu.no

Abstract. The modern power grid instabilities are increasing, arising from renewable power sources. There are a lot of different ways of stabilising the power grid, but pumped storage hydropower plants are considered to be one of the best ways of storing energy in a large scale. In these plants, it is common to have one turbine in addition to a pump. This requires two different waterways in addition to a turbine and a pump which is expensive to build. An alternative is to use a single reversible pump turbine (RPT) that can act as both a turbine and a pump.

In this paper, steady state simulation of cavitation a reversible pump turbines impeller has been successfully performed and validated using experimental data. The break in efficiency was found to happen at about the same cavitation number for both the simulations and experiments. However, the efficiency in the simulations is higher than the experimental data. This was as expected due to simplifications in the geometry in comparison with the full geometry. Further work needs to be done in order to check if the efficiency will correlate better if the full geometry including guide vanes, stay vanes, spiral casing and draft tube. Additional operating points should also be simulated in the future.

Keywords— Reversible pump turbine, cavitation, pump mode, CFD, hydropower

1. Introduction

The demand and supply in electricity does not always correlate. This is especially true for the modern grid requirements arising from renewable power sources, for example wind and solar power [4]. As a result, the modern power grid is becoming more unstable. Pumped storage hydropower plants are tested in large scales and found to be an efficient way of storing energy.

A Reversible Pump Turbine (RPT) can act as both a turbine where the hydro machinery extracts energy from the fluid and a pump where the hydro machinery adds energy to the fluid [7]. In order to contribute to make the power grid more stable, the RPT can pump water into the upper reservoirs when the demand is low and act as a turbine in order to generate power when the demand is high. RPTs are highly adaptable and can respond fast to changes in the power grid [4].

RTPs can be may be run outside its best efficiency point (BEP) which may induce unwanted cavitation. Cavitation can both reduce the RPTs efficiency in addition to increase the deterioration of the impeller thus creation a mechanical failure. Due to the consequences of cavitation, it is of great importance to be able to predict the cavitation behavior of a given RPT during the design process of RPTs. A cost-efficient way to design a pump turbine is to design it by making a 3-dimensional model of it and using numerical simulation in order to test the pump turbine numerically. The alternative it to design a pump turbine, making a physical model and testing it in small scale. Computational fluid dynamics (CFD) reduces the need to make many physical pump turbines, thus reducing the cost of designing a pump turbine.

Cavitation is a phase shifting process which makes the CFD simulation a multiphase problem. This is computational heavy in comparison with single phase simulations due to an extra set of equations which needs to be solved every iteration.

1.1. Objectives and limitations

The objective of the work presented in this paper is to set up and test numerical simulation of cavitation on a Reversible Pump Turbines impeller in pump mode. This simulation is done at close to the RPTs BEP and compared with experimental data provided by Rainpower AS in order to validate the simulation. Limitations of this paper is mainly due to time and computer power restrictions. Consequently, the steady state CFD simulation on the impeller at a rotating frame of reference was chosen.

2. RPT characteristics

RPTs can be run both as a turbine and as a pump. The only difference between RPT in pump mode and turbine mode is the direction at which the impeller is spinning. When designing a RPT, the dimensions needs to be designed for pump mode because the head in pump mode, H_p, is larger than the head in turbine mode, H_t [2]. This is illustrated in Figure 2 and is the main reason this paper covers cavitation on a RPT in pump mode.

Euler's turbo machinery equation for RPTs in turbine mode is:

$$gH_t\eta_{ht} = (u_1 c_{u1t} - u_2 c_{u2t}) = E_t \eta_{ht} \tag{1}$$

where η_{ht} and H_t is the hydraulic efficiency and head in turbine mode and all the velocities is as in Figure 1. Euler's turbo machinery equation for RPTs in pump mode is given by Equation 2

$$gH_p = \eta_{hp}(u_1 c_{u1p} - u_2 c_{u2p}) = E_p \tag{2}$$

where η_{hp} and H_p is the hydraulic efficiency and head in pump mode and the velocities are described in Figure 1. This paper only considers an RPT in pump mode and therefore, the hydraulic efficiency in pump mode will be defined as η_h form here on out.

3. Cavitation

Cavitation is a phenomena where parts of the liquids static pressure p falls below its vapor pressure p_{va}. When cavitation is present in a domain there is two phases flow present in the domain: both liquid and vapor. If the cavitating zone is large, then the head and efficiency of the RPT may be severely compromised [3].

In pump mode, cavitation bubbles are most commonly formed at the inlet of an impeller blade, where the static pressure usually is the lowest. The bubbles are then transported through the RPT to regions where the static pressure is higher and the cavitation bobbles collapse. The collapse of the cavitation bobbles close to the physical surface of the impeller can causes a large local pressure on the surface. This effect can lead to pitting erosion on the impeller, and eventually catastrophic failure of the RPT [7, 2].

The net positive suction head ($NPSH$) is a commonly used way to assess if a hydro machinery is in risk of cavitation or not. $NPSH_A$ is the available NPSH and it is defined in Equation 3 [5].

$$NPSH_A = \frac{p_{\text{abs,in}} - p_{va}}{\rho g} \tag{3}$$

where $p_{\text{abs,in}}$ is the absolute pressure at the inlet.

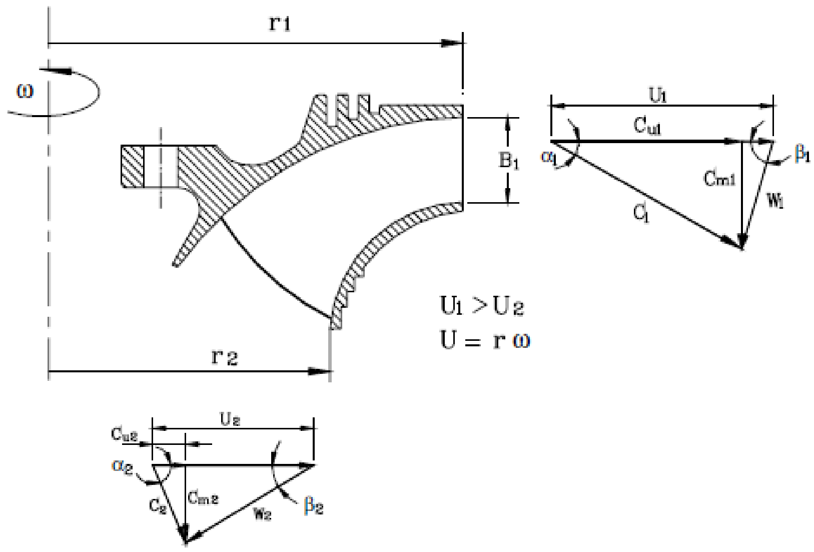

Figure 1. Velocity triangles for RPTs. Collected from [2].

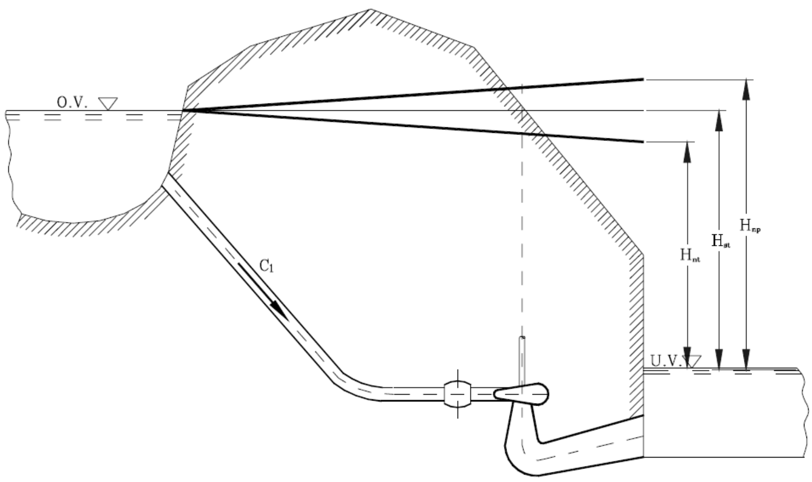

Figure 2. Difference between head in turbine and pump mode. Collected from [2].

In order to avoid cavitation, the following requirement needs to be met:

$$NPSH_A > NPSH_R \tag{4}$$

where $NPSH_R$ is the required $NPSH$ in order to avoid cavitation. There are several ways to define $NPSH_R$, but a common way is to define it at when the efficiency has dropped 1% due to cavitation, $NPSH_1$.

3.1. Sigma break curves

The Thoma cavitation number, σ, is a dimensionless term indicating the conditions of cavitation under which the machine operates [5] and is defined in Equation 5.

$$\sigma = \frac{NPSH_A}{H} \tag{5}$$

where H is the head. Similarly to the $NPSH_R$ can be set to be $NPSH_1$, the cavitation number σ_1 can be defied based on the $NPSH_1$.

The sigma break curve is a chart where usually either the cavitation number, σ, or the $NPSH_A$ is plotted along the x-axis and efficiency is plotted along the y-axis. From this chart it is possible to judge at under which conditions cavitation will appear for a RPT. The sigma break curve will therefore be used in order to validate the CFD results to the experimental data.

4. CFD setup

It was decided to use the CFD software Ansys Turbogrid for the meshing of the impeller and Ansys CFX for the simulations.

4.1. Geometry

The Geometry is kindly provided by Rainpower AS. They have also provided the author with their experimental data for this geometry under different operating conditions for cavitation. This geometry is confidential and all parameters will therefore be relative. The only part of the geometry used in this paper is the impeller. It was decided to only use this part of the geometry in order to reduce computational time and to verify the numerical setup early.

4.2. Mesh

The mesh of the impeller was created using Ansys Turbogrid. The mesh was designed for a $k-\epsilon$ turbulence model with a wall y+ value of 30 and a maximum expansion rate of 1.2. Structured hexahedral mesh was generated and it had $\sim$ 1.1 million cells for a single passage. The mesh statistics can be found in Table 1.

Table 1. Mesh statistics

Mesh attribute	**Value**
Minimum face angle	28.79 [$degree$]
Maximum face angle	151.24 [$degree$]
Maximum element volume ratio	3.22
Maximum aspect ratio	91.68

The mesh statistics in Table 1 is well within the general advise from Ansys [1] and the mesh is therefor considered to be a good mesh. The general advise from Ansys CFX is:

- 10 [degree] $<$ Face angle $<$ 170 [$degree$]
- Element volume factor $<$ 20
- Aspect ratio $<$ 100

4.3. Numerical setup

It was decided to use a robust and computational cheap numerical setup in order to get to some results fast. The following setup was chosen:

- The standard $k-\epsilon$ turbulence model was chosen because of its good reputation from the industry for being a stable turbulence model. This turbulence model was found to be a good compromise between speed and accuracy. The $k-\epsilon$ model uses wall functions thus it requires a less fine mesh than turbulence models like the SST model, that do not use wall functions [6, 3].

- "High Resolution" advection scheme for both continuity and momentum equations. This scheme was chosen because of the precision and stability of this scheme.
- "Upwind" advection scheme for turbulence eddy dissipation and turbulence kinetic energy equations as suggested by Ansys [1].
- Steady-state was chosen in order to simplify the problem and thus making it less computational heavy.
- Single, rotating frame of reference on the impeller. No rotor-stator interactions.
- Single phase initialisation, then multiphase simulations for different cavitation numbers according to the experimental data. p_{va} was computed by linear interpolation for the exact temperature from the experimental data.
- Rotational periodicity interface on a single passage of impeller. This reduces the computational time a lot by simulating only a single passage.
- Physical timescale according to 1 degree of rotation per timescale.
- Boundary conditions:
  - Inlet: constant total pressure in stationary frame of reference, corresponding to the desired cavitation number, 5% turbulence intensity at the inlet.
  - Outlet: constant mass flow based on the averaged mass flow from the experimental data.
  - Walls: No slip, smooth wall.

The numerical simulation was considered to be converged when the root mean square residuals (RMS) was below 10^{-6}.

5. Results

The CFDs hydraulic efficiency, $\overline{\eta}_{h,\text{CFD}}$, will be normalised based on the best efficiency for the current CFD run. Likewise, the experimental hydraulic efficiency, $\overline{\eta}_{h,\text{exp}}$, will be normalised based on the best efficiency for the current experimental data set. The normalised efficiency is defined in Equation 6.

$$\overline{\eta}_h = \frac{\eta_h}{\eta_{h,\text{best efficiency}}} \tag{6}$$

Similarly to the normalised efficiency, a normalised cavitation number, $\overline{\sigma}$, is defined in Equation 7 for both CFD and the experimental data based on the highest σ value of the experiments data set.

$$\overline{\sigma} = \frac{\sigma}{\sigma_{\text{a defined value}}} \tag{7}$$

After running the simulation, it is found that $\frac{\eta_{h,\text{CFD}}}{\eta_{h,\text{exp}}} = 1.0701 \approx 7\%$. It is expected that the CFD simulation was going to have a higher efficiency than the experimental data due to the lack of losses in the draft tube, guide vanes, stay vanes and spiral casing.

Figure 3 shows the preliminary results with the mesh and the numerical setup described in section 4. The break in efficiency happens at about $\overline{\sigma}_{\text{CFD}} \approx 0.251$ for the CFD results and $\overline{\sigma}_{\text{exp}} \approx 0.249$ for the experiments. This is very close to each other and is considered to be the same due to the lack of data points close to this position. A commonly used definition of critical cavitation point of 1% loss in efficiency and described as σ_1. In this case, we get the following approximate results: $\overline{\sigma}_{1,\text{CFD}} \approx 0.23$ and $\overline{\sigma}_{1,\text{exp}} \approx 0.21$.

Figure 4 and Figure 5 illustrates the location at which cavitation is occurring on the impeller. It can be observed that cavitation happens on both sides of the impeller which indicates that the RPT is close to its best efficiency point.

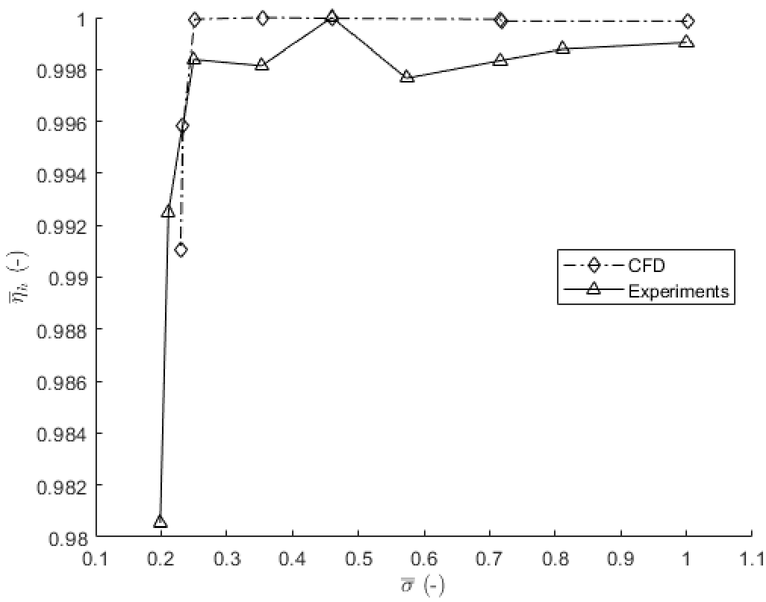

Figure 3. Sigma break curve for CFD and experimental data.

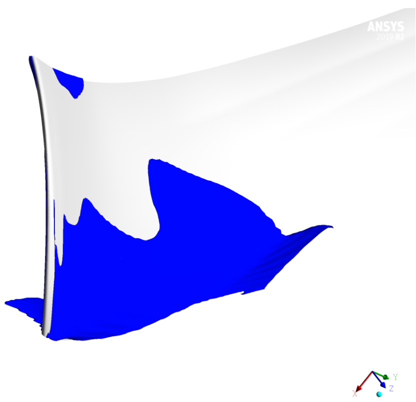

Figure 4. Cavitation on the impeller at the suction side.

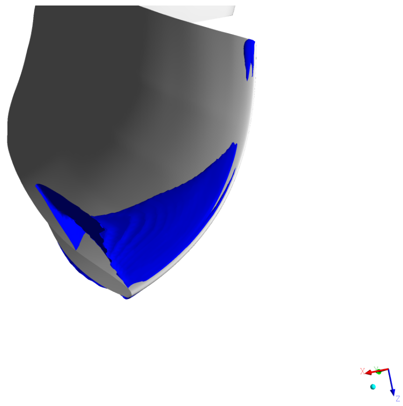

Figure 5. Cavitation on the impeller at the pressure side.

6. Conclusion

The mesh parameters is good according to the general advise from Ansys [1]. Despite of this, a mesh independence study should be performed in order to validate the performance of the mesh. However, because the break point of the sigma break curve in Figure 3 it is shown that the CFD results correlates well compared to the experimental data. Therefore, the mesh is considered to have a sufficiently good quality despite the lack of mesh impedance studies.

The overall results is promising despite the $\sim 7\%$ higher efficiency in the simulations compared to the experiments. The simplified CFD geometry does not take losses between the inlet of the draft tube and the outlet of the spiral casing and setup into account. The increased efficiency in the simulation is as expected due to the reduced losses due to the simplified geometry.

The break in efficiency in Figure 3, happens for about the same $\overline{\sigma}$ values for both the CFD simulations and the experimental data. This means that the CFD simulation is validated, thus the numerical setup is therefore considered to be good for this operation point. Due to the good result, the numerical setup will create a solid base to start increasing the complexity of the simulations in the future.

7. Further Work

A mesh impedance study should be performed in order to validate the mesh. However, this is not a requirement as long as the simulated results are validated using experimental data. More operating points should be simulated and validates. In order to achieve more accurate results at different operating points, the geometry needs to include the entire geometry including the draft tube, guide vanes, stay vanes and the spiral casing. This will probably additionally make the simulated efficiency closer to the efficiency according to the experimental data. Simulations adding pre-rotation to the inlet should also be performed in order to check if pre-rotation has a significant advantage/disadvantage on the RPT in pump mode of operation outside the normal best efficiency operation.

References

[1] ANSYS. Ansys cfx 2019 r2, cfx-solver modeling guide, 2019.

[2] H. Brekke. Pumper & turbiner. *Vannkraftlaboratoriet NTNU*, 2003.

[3] J. F. Gülich. *Centrifugal pumps*, volume 2. Springer, 2008.

[4] IHA. Hydropower status report. 2019.

[5] I. Standard et al. Hydraulic turbines, storage pumps and pump-turbines—model acceptance tests. 1999.

[6] H. K. Versteeg and W. Malalasekera. *An Introduction to Computational Fluid Dynamics*. Pearson Education Limited, 2 edition, 2007.

[7] Y. A. Çengel and J. M. Cimbala. *Fluid Mechanics: Fundamentals and Applications*. McGraw-Hill Education, 3 edition, 2014.