Available online at www.sciencedirect.com
ScienceDirect
IFAC PapersOnline Papers Online CONFERENCE PAPER ARCHIVE
IFAC PapersOnLine 51-2 (2018) 109–114

# Simulation and Discussion of Models for Hydraulic Francis Turbine Simulations

**Pål-Tore Storli***
**Torbjørn K. Nielsen****

*Associate Professor at *** (e-mail: pal-tore.storli@ntnu.no)
**Professor at *** (e-mail: torbjorn.nielsen@ntnu.no)
***Waterpower Laboratory, Dept. of Energy and Process Engineering, Norwegian Uni. Of Science and Technology

Abstract: The paper presents simulations using three different models for a hydraulic turbine. The models are too simple and unrealistic to be used for actual simulations, but the intention is to discuss behaviour intrinsic to the models that provide insight into the physics involved. The presented results point out several behaviours that are contradictory to actual turbine behaviour, and suggest what can be done to avoid this. The models can be used as a basis for later modifications that make the models applicable for simulations, modifications the authors are currently working on.

© 2018, IFAC (International Federation of Automatic Control) Hosting by Elsevier Ltd. All rights reserved.

*Keywords:* Hydraulic turbine, characteristics, model simulations

## 1. INTRODUCTION

Simulations are an important tool for investigation of transients and dynamic behaviour of systems. For hydraulic systems, the characteristics of hydraulic machinery is important for the simulation results and their reliability. If experimental results are available, they can be used by creating so-called Suter curves (Wylie and Streeter, 1983). If no results are available, a model must be used. For hydraulic turbines, the two most important properties to model correctly are the torque acting on the machine from the moving fluid and the head the machine requires to be subjected to for a certain flow to be maintained at a certain rotational speed and guide vane opening. Traditionally, both the torque and the head has been modelled using the Euler equation (Nielsen, 2015, Nielsen and Storli, 2014, Nielsen, 1996, Giosio et al., 2017), but there are issues which should be addressed related to the use of this equation as a basis for simulation models for hydraulic turbines. The purpose of this paper is to provide insight and un-ambiguous derivation of equations describing models for torque and head for a hydraulic turbine, in this case a Francis turbine, and to point out and discuss problems concerning the validity of the models.

## 2. TORQUE

The torque is an important property to model, because it will be linked to accelerations or decelerations of the unit. If there is not a balance between the torque acting on the runner from the hydraulic domain and the torques acting on the runner dry mechanical parts, the unit will experience a change in rotational speed. This can be seen in the Newton 2nd law for rotational motion. It says that there must be a balance between the sum of all torque vectors, and the moment of inertia times the acceleration vector. For a hydraulic turbine, the moment of inertia is constant for the mechanical parts, but there is also water flowing inside the runner which also has moment of inertia, and it is not easy to intuitively conclude whether this is constant or not. To complicate matters further, the flowing water is responsible for the torque acting on the runner, and this is highly dependent on the flow conditions, as can be seen in the Euler equation (5). However, there is one equation combining all these effects, and this equation is known as the angular momentum equation.

### 2.1. *The Angular Momentum Equation*

The Angular Momentum (AM) equation is a powerful tool for analysis of rotational motion. It is obtained by using Reynolds Transport Theorem with 'torque' as the extensive property under subject. In words, it describes that there must be a balance between the change of angular momentum for a flow crossing a Control Surface (CS), the change in angular momentum for the contents in the Control Volume (CV), and sum of external torques $\overrightarrow{T_{ex}}$ acting on the CV. Mathematically, it can be written as (Cengel and Cimbala, 2014),

$$\sum \vec{T}_{ex} = \underbrace{\frac{d}{dt}\int_{CV} \rho\left(\vec{r}\times\vec{V}\right)d\forall}_{\text{time rate of change of AM in CV}} + \underbrace{\int_{CS} \rho\left(\vec{r}\times\vec{V}\right)\left(\vec{V}_r\cdot\vec{n}\right)dA}_{\text{Net flux of AM out of CV}} \qquad (1)$$

ρ is water density, $\vec{r}$ is the position vector, $\vec{V}$ and $\vec{V}_r$ are the absolute and relative velocity vectors, respectively. Including the effect of the time change in AM related to the water inside the CV is something the authors are currently investigating, but it is outside the scope of this paper. In this paper, we will only consider the flux part of (1). So we evaluate only the last integral of (1). The integral is a vector integral, and the vector integrated is the cross product of the radii vector and the velocity vector; the angular momentum. For a hydraulic machine it is easiest to evaluate this in a cylindrical coordinate system described by unit vectors $\vec{e}_r$ , $\vec{e}_\theta$ and $\vec{e}_z$ , where the z-

2405-8963 © 2018, IFAC (International Federation of Automatic Control) Hosting by Elsevier Ltd. All rights reserved.
Peer review under responsibility of International Federation of Automatic Control.
10.1016/j.ifacol.2018.03.019

axis is pointing downwards in the direction of the axis of the unit (This will make is easier to deal with the sign in the vector integrals). This is seen in Fig. 2. The CV compose of all rotating parts of the axle. This is seen in Fig. 1. Executing the cross product using the nomenclature seen in the velocity diagrams in Fig. 2, it becomes:

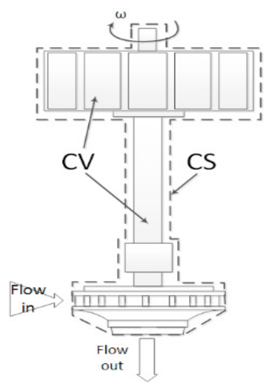



Fig. 1: Rotor as CV

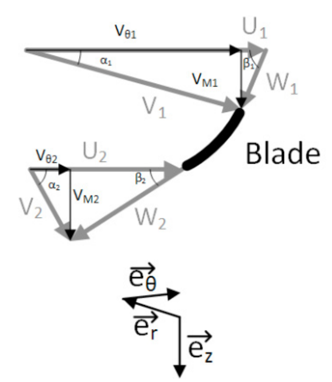



Fig. 2: Velocity diagram and coordinate system

$$\vec{r} \times \vec{V} = -rV_z \vec{e}_\theta + rV_\theta \vec{e}_z \tag{2}$$

$V_z$ and $V_\theta$ are the absolute velocity components in axial and angular direction, respectively. The only term contributing to rotational motion is the z-component, so this is the only one we will continue to investigate.

$$\sum T_{ex,z} = \sum\left(\vec{T}_{ex} \cdot \vec{e}_z\right) = \int_{CS} \rho r V_\theta \left(\vec{V}_r \cdot \vec{n}\right) dA \tag{3}$$

Equation (3) is the steady state version of (1), which for the case of; *the control volume being a Francis runner; uniform angular momentum distributions on inlet and outlet of the control volume*; reduces to

$$\sum T_{ex,z} = \sum \vec{T}_{ex} \cdot \vec{e}_z = -\rho Q\left(r_1 V_{\theta 1} - r_2 V_{\theta 2}\right) \tag{4}$$

$Q$ is the flow, index 1 and 2 denotes the inlet and outlet properties. *In the case of only one external torque T opposing the rotational movement* it reduces to the well-known Euler turbine equation (A torque $T$ opposing the movement has a negative sign on the left hand side of (1)/(4)) (Cengel and Cimbala, 2014):

$$T = \rho Q\left(r_1 V_{\theta 1} - r_2 V_{\theta 2}\right) \tag{5}$$

For a 1D-system simulation the flow parameter is the discharge Q. This is linked to the peripheral velocity component at inlet and outlet by

$$V_{\theta 1} = \frac{Q}{A_1 \tan \alpha_1} \quad , \quad V_{\theta 2} = \omega r_2 - \frac{Q}{A_2 \tan \beta_2} \qquad (6), (7)$$

$A_1$ and $A_2$ are the inlet and outlet runner areas, respectively. ω is the angular velocity, $\alpha_1$ is the inlet velocity angle, $\beta_2$ is the blade outlet angle. The velocity components can be seen in the inlet and outlet velocity triangles, in Figure 2. Substituted back into (4) we get the momentum flux contribution to the torque:

$$\sum T_{ex,z} = -\rho Q\left(Q\left(G_1 r_1 + G_2 r_2\right) - \omega r_2^2\right) \tag{8}$$

Where $G_1 = \dfrac{1}{A_1 \tan \alpha_1}, G_2 = \dfrac{1}{A_2 \tan \beta_2}$

A term appears in (8), $Q(G_1r_1+G_2r_2)$, which represents the angular momentum at zero angular velocity, ω=0. This is the torque needed to balance the starting torque which is acting on the stationary runner at a given flow Q, equivalent to the term $\rho Q t_s$ in the paper by Nielsen (Nielsen, 2015). Equation (8) is equivalent to (4), only written in an alternative form to include the simulation parameters Q, $\alpha_1$ and ω.

We have now completed the work with the left-hand side of (1), and are ready to analyse the right-hand side. It states we should sum all external torques acting on the control volume. By external, we mean torques related to mechanic, magnetic, pressure- forces and so on, torques that don't originate from the flow of water. Generally, it is important to have control of the sign of these torques when summing them, because there might be cases where the torques don't act in the same direction on the rotational CV. For a turbine, all external torques are acting to slow down the rotational speed of the CV. They can be described by

$$\sum T_{ex,z} = -T_{magnetic} - T_{bearings} - T_{disk\ friction} - \ldots \tag{9}$$

By adding appropriate terms in (9) one includes all effects that is desired to simulate. In this paper, the magnetic torque acting on the rotor from the generator is the only one we like to include, and in this case it is the same as the torque commonly denoted $T_{axle}$/$T_{shaft}$. This means that in our case (1) is written like:

$$T_{shaft} = \rho Q\left(Q\left(G_1 r_1 + G_2 r_2\right) - \omega r_2^2\right) \tag{10}$$

To conclude this section, the use of the angular momentum equation reveals that we obtain the same expression for the steady state torque as have been reported earlier, but also that there are some transient effects included in (1) that are omitted by using the classical Euler equation. This is being investigated by the authors in on-going work. For all models presented in this paper, (10) is used to describe the torque.

## 3. HEAD

Establishing a model for the difference in head, $H_t$, between the inlet and the outlet of the runner will make it possible to generate the characteristics. Physically based models can be found using different approaches. One approach is to use the Euler turbine equation multiplied with the angular velocity to obtain the power, which can be extracted from an ideal machine. This power is then said to be the same as the power extracted from the pipe flow, by energy conservation principles, mathematically seen as (11):

$$\rho g Q H_t = T_{shaft}\,\omega = \rho Q\left(U_1 V_{\theta 1} - U_2 V_{\theta 2}\right) \tag{11}$$

U is the peripheral velocity of the runner. Using the law of cosines, we can rewrite this as

$$gH_t = \frac{\left(V_1^2 - V_2^2\right)}{2} - \frac{\left(W_1^2 - W_2^2\right)}{2} + \frac{r_1^2 - r_2^2}{2}\omega^2 \quad (12)$$

$W$ is the relative velocity between the water and the runner. The same expression found if combining the Bernoulli equation for a linear system with the Bernoulli equation for a rotating frame of reference. The expression is the same as the one used to describe how velocities generate the lifting height of a pump, if index 1 is the pump outlet and index 2 is the pump inlet (Cengel and Cimbala, 2014). Using geometrical relations found in the velocity triangles, we could develop this further so that the only variables are the simulation variables Q, $\alpha_1$ and $\omega$. The recent model by (Giosio et al., 2017) use the model represented by (12), added different losses based on empirical formulae. They used the model in simulations, but no turbine characteristics are provided, so it is difficult to compare qualitatively with experimental characteristics.

A different approach has previously been used by Nielsen (Nielsen, 1990),(Nielsen, 2015), where it is stated that at nominal speed, the head must be described by a valve equation. So, the head should relate to flow Q and opening degree $\kappa$ of a valve as

$$gH_t = gH_R\left(\frac{Q}{\kappa Q_R}\right)^2 \quad (13)$$

Index 'R' denotes Rated values. Equation (12) with *rated and steady angular velocity* should also be valid, so we can set the expressions of head equal to each other and get

$$gH_R\left(\frac{Q}{\kappa Q_R}\right)^2 = \frac{V_1^2 - V_2^2}{2} - \frac{W_1^2 - W_2^2}{2} + \frac{r_1^2 - r_2^2}{2}\omega_R^2 \quad (14)$$

Rearranged to express the absolute and relative velocity terms, we can substitute this back into (12), and we remove these velocities from the equation and remain with

$$gH_t = gH_R\left(\frac{Q}{\kappa Q_R}\right)^2 + \frac{\left(r_1^2 - r_2^2\right)}{2}\left(\omega^2 - \omega_R^2\right) \quad (15)$$

Nielsen then adds different terms to the equation to correct for the discrepancies between results obtained using the model and experimental results, but this is not done in the work presented here.

In this paper, (12) and (15) are used in simulations. As can be seen in the results, there are issues regarding both these models, which make their use unphysical. It was decided to modify (12) to include a shock loss (incipient loss), to see the effect this had on simulations. A term was added that included the stagnation pressure from the component of the relative velocity normal to the blade inlet. This can be seen in (16):

$$gH_t = \frac{V_1^2 - V_2^2}{2} - \frac{W_1^2 - W_2^2}{2} + \frac{r_1^2 - r_2^2}{2}\omega^2 + \frac{\left(W_1 \sin\left(\beta_1 - \beta_{1R}\right)\right)^2}{2} \quad (16)$$

Where $\beta_1$ denotes the inlet relative flow angle. The model used by (Giosio et al., 2017) also incorporates this kind of loss, called an incipient loss. However, their representation is dependent on the peripheral component of the relative velocity, and not the component normal to the blade inlet angle.

To conclude this section, three models of head are used in simulations presented in this paper; (12) called 'The Euler equation'; (15) called 'The valve equation substitution'; and (16) called 'The new model'.

## 4. THE EFFICIENCY

The efficiency is normally defined as the energy or power output from a process divided by the energy or power input to the process. For a hydraulic turbine, the efficiency is typically found by evaluating the ratio of mechanical power extracted on the generator to hydraulic power extracted from the flow at steady state operation. Mathematically it is described as

$$\eta = \frac{T_{shaft}\omega}{\rho g Q H_t} \quad (17)$$

We see that the efficiency will account for any discrepancy between left and right-hand side of (11), and since the energy source is the energy in the water, the highest possible efficiency that any machine can have is one; this is an ideal machine without losses.

Calculating the efficiency in a laboratory experimental setup is simple using (17), because the needed properties are quite simple to measure. The head $H_t$ is measured as the difference in head between a measurement section in the uniform pipe upstream the turbine unit and a measurement section slightly down into the draft tube. In this paper, the efficiency is calculated by the same equation, (17), but no losses in wicket gates, draft tube, losses due to disc friction, volumetric losses et.c. are included.

## 5. SIMULATIONS AND EXPERIMENTAL RESULTS

Simulations have been performed to establish the turbine characteristics and the efficiency curves. In the numerical setup, the turbine is simulated as the Francis test rig in the Waterpower laboratory at NTNU in Trondheim, Norway. The runner simulated is the Francis99 runner (Trivedi et al., 2016), only considering main dimensions and no details regarding blade thickness, blade loading and so on. The properties needed from the runner model as input to the simulations are presented in table 2. Three turbine models are simulated and the results presented in this paper. They all utilize the torque as described by (10), they differ by using descriptions of head. The simulation has been performed using MATLAB R2012B, and an in-house code. This code is using Newton's method

*Table 1: Parameters for simulations*

| Property(not all properties used in equations) | Value | Unit |
|---|---|---|
| Inlet diameter | 0.6216 | m |
| Inlet height | 0.0585 | m |
| Inlet blade angle | 71.5 | deg |
| Outlet diameter | 0.3531 | m |
| Outlet blade angle | 17 | deg |
| Rated rotational speed | 539.5 | rpm |
| Rated head | 30 | m |
| Rated flow | 0.336 | $m^3/s$ |

to solve the equations. The Newton's method code has been downloaded from MathWorks (MathWorks). The code uses the symbolic toolbox in MATLAB to solve the equations symbolically. The equations that are solved are the Newton's 2nd law for linear and angular steady state systems:

$$gH = gH_t \quad , \quad T_{runner} = T_{shaft} \qquad (18),(19)$$

gH is determined from setting H equal to 30 meters as the experimental results are obtained at, and the different models for head are used to describe $gH_t$. $T_{runner}$ is the symbolic variable that is solved for, and $T_{shaft}$ is described by (10).

*To establish the characteristics, the guide vane angle is fixed in the simulations, and the rotational speed is increased from zero for each time the code goes through a "for"-loop. In each loop, the solver finds the flow and torque that balance all equations. Having computed for all rotational speeds in the "for"-loop, the guide vane angle is increased and the procedure of increasing the rotational speed from zero is repeated. This is very similar to how the characteristics are obtained in an experimental test campaign, and how the experimental results presented in this paper are obtained. They were obtained from measurements on the Francis99 test rig April 16th 2007 as a part of reference measurements during a model acceptance test at the Waterpower laboratory at NTNU. The main dimension are identical to the ones presented in*
Table 1, since the Francis99 runner has been used as the template for the numerical simulations.

## 6. RESULTS AND DISCUSSION

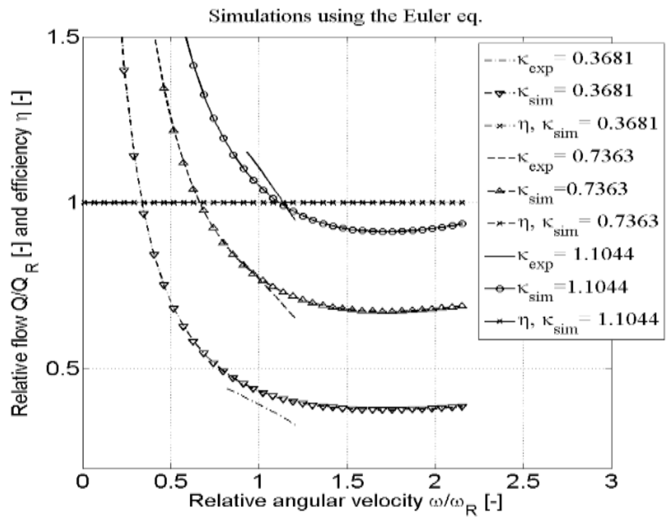
The results from simulations of the different model are presented and discussed individually as the results are presented. Following this is a section containing a general discussion.

### *6.1. The Euler equation*

The turbine characteristics and the efficiency obtained from simulations by using (12) can be seen in Fig. 3. The reduced flow q=Q/Qnom has a very high value at low rotational speeds. What this means is that in order generate the head when the contribution from rotational speed and geometry is low, the velocities need to be high, resulting in a high flow. In fact, the necessary flow is much higher than the maximum flow possible when applying Torricelli's theorem for maximum velocity $V_{max} = \sqrt{2gH}$ (Cengel and Cimbala, 2014), and for the simulations presented here, the maximum value is represented by 2.38 $m^3/s$, if outlet area is multiplied with

*Fig. 3 Simulations using the Euler equation*

the maximum velocity. Clearly, this must be wrong. At




rotational speed equal to zero, the runner should just work as a valve, which will find equilibrium with its surrounding at a flow that generate the head Ht. This head has to be created by flow passing through the geometry of the runner. There is a huge mismatch between the direction of the relative velocity (which for rotational speed equal to zero is coinciding with the absolute velocity of the flow), and the direction the flow must comply to because of the inlet blade angle. This directional mismatch is not represented at all in the original equation, and the authors considers using the equation imply a runner with a fully variable inlet blade geometry that always is in the direction of the relative velocity. Even if such machine would exist, the flow could never be greater than what is dictated by the Torricelli theorem. This contradiction is possibly overcome by using the full description of the head, which includes the pressure, rather than the aggregate property "head" itself. Not accepting solutions that result in an inlet pressure lower than atmospheric pressure would limit the solutions to have a lower flow than dictated by the Torricelli theorem.

The efficiencies accompanying the characteristics are lying on top of each other, and all are identical to one for all rotational speeds. This back up the simulations, since head is found from (11). This imply an efficiency equal to one, which is reproduced in the simulation results.

The experimental results are lying above the lines for the Euler equation results. The Euler equation results represents an efficiency equal to one, and the experimental results should therefor represent efficiencies above one. The efficiencies obtained in the experimental test are not presented here, but it should come as no surprise that the measured efficiencies are below one. The net head in the experimental campaign was 30 meters, the same as in the simulations. Discrepancies between simulation parameters and experimental settings should not translate to the discrepancies seen in Fig. 3. The experimental results include many sources of loss, which the numerical simulations do not include. The discrepancies should intuitively be in the other direction, with the experimental results being below the Euler model simulation results for all curves. The most likely reason for the discrepancy is that performing a loss free simulation at 30 m head would mean that the experimental head must be higher than 30 m to take into account the head loss due to losses in spiral casing, stay vanes, guide vanes, draft tube, et.c. Simulations have been performed with $H_t$=30/$\eta_{exp}$ and the simulated characteristics are then above the experimental results for the case of simulations using the Euler equation.

*6.2. The Valve equation substitution model*

The characteristics using the valve equation substitution and the accompanying efficiencies are seen in Fig. 4.

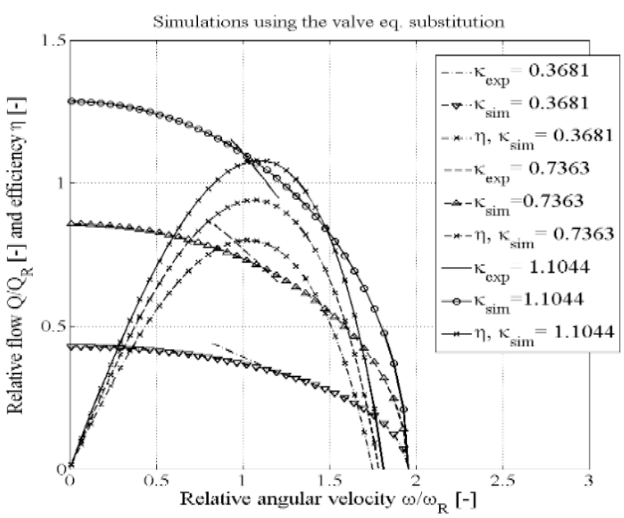
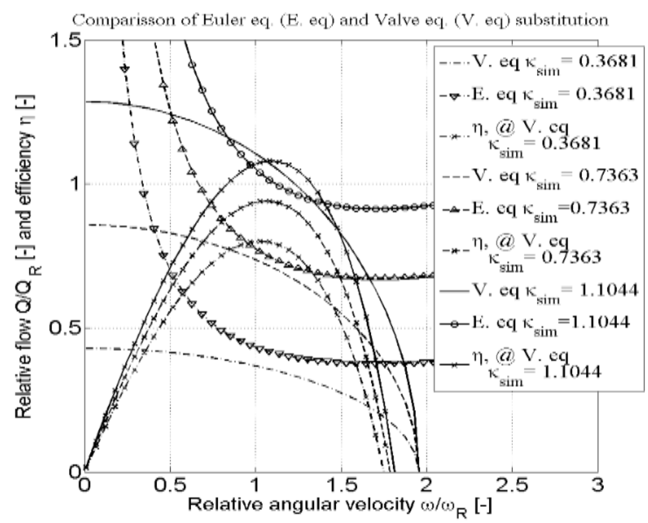



*Fig. 4: Simulations using the valve eq. substitution*

The characteristics look much better compared to empirical knowledge. The efficiencies are also looking more like actual efficiency curves, starting at zero at zero rotational speed, increasing for to decrease after a max efficiency. No efficiency has been introduced in (17) when using the head described by the valve eq. substitution (15), so the losses that make the efficiency drop from one comes from the substitution of the velocity terms with the valve equation. Furthermore, we see that the max efficiency increase with increasing guide vane angle, but the efficiency do not reach a global maximum efficiency of one. It keeps increasing, and increases beyond the maximum possible value one. Clearly, this is wrong. Again, the reason must be found in the substitution used to obtain the model using valve equation substitution. To be honest *any* term *not* containing the angular velocity in (14) would be replaced by the valve equation by such a substitution, and it is not certain that the valve equation is able to capture the head/flow/opening degree relation correctly for a turbine. As an example, it is completely decoupled from the runner geometry. To exemplify this, the flow/head relation for a Reversible Pump Turbine in turbine mode is quite different from a Francis turbine, which has different geometries even if main dimensions are the same.

One interesting observation that is made is if the two characteristics from the Euler equation model and the valve substitution model are plotted against each other. This can be seen in Fig. 5.

*Fig. 5: Comparison of the Euler eq. and valve eq. substitution results*




The point where the efficiency goes above one coincides with the point where the characteristic from the valve equation substitution model crosses the characteristics from the Euler model. Further investigations show that this coincides with where the torque characteristics cross each other; the torque from the Valve equation substitution model becomes greater than the torque from the Euler model. This means that the valve substitution model implies velocities that are unrealistically high, near the peak efficiency. Looking at (15), we see that at for $\omega$=1; $\kappa$>1; $H_t$=$H_R$, the flow must be higher than rated. This means that there are higher velocities than rated, and a mismatch with relative velocity direction and blade inlet direction will occur. This should contribute with a pressure increase, which would result in the flow being reduced. Such effect is not included in the valve equation, and might be a possible reason why the efficiency overshoots one.

In Fig. 4 we see that the experimental results are partly above the characteristics for the valve equation substitution model as well. However, the experimental results are more aligned with the simulations than for the Euler model. The same argument can be made regarding the correctness of comparing loss free simulations at H=30 m to experimental results at H=30, but correcting for any such argument would lift the valve eq. substitution model even higher, and still with an efficiency more than one.

### 6.3. The New model

The results obtained from the simulations using the new model can be seen in

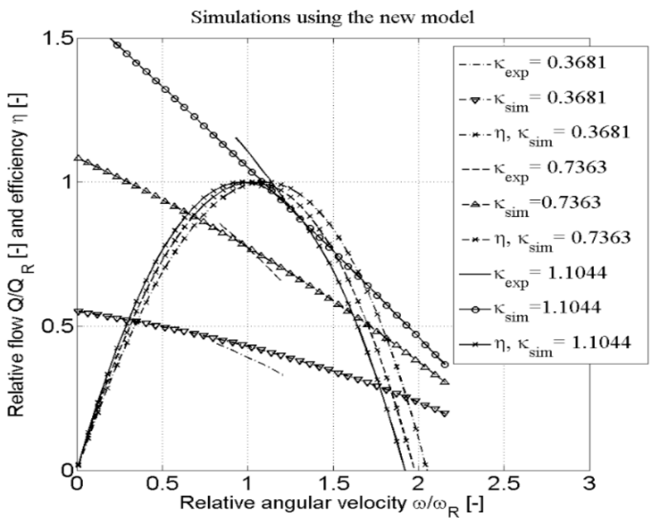



*Fig. 6.*

*Fig. 6: Simulations using the new model*

The new model is simply adding a head term to the Euler model, and the term is the stagnation pressure of the relative velocity component normal to the inlet of the blade. No other physical effect is included, but the effect on the simulations are significant. The efficiencies are no longer one for all rotational speeds, and the efficiency curves exhibit the empirical correct behaviour of having the peak efficiency point moving to a different rotational speed when opening degree change. The flow is still higher at low rotational speeds than the valve equation substitution model results, and lacking experimental results for this region, this model serve as the reference for what a real characteristic should look like in this region.

### 6.4. General discussion

The Euler equation is containing no information about the geometry of the runner or any kind of representation of the losses in an actual turbine runner. The efficiency should thus be one, and simulations show that this is the case. However, the results show an unphysical high flow at low rotational speeds of the runner, and the authors consider this the effect of using the aggregate property 'head' rather than the individual terms the head consist of. Doing so, a limitation on the pressure can be imposed, and the flow is restricted to the maximum possible value it can have, found using the Torricelli theorem. However, this maximum flow is also far too high, compared to experimental results. Therefore, the Euler equation must suffer from another shortcoming, compared to the actual physics involved. The authors consider that the most important shortcoming is the fact that there are no effects due to mismatch between the relative flow angle and the runner inlet angle included in the Euler equation. A term has been added to the Euler equation representing the pressure due to stagnation of the relative flow velocity component normal to the inlet of the blade. The simulation results show a significant improvement, with respect to obtaining efficiency curves that look like actual measured efficiency curves.

## 7. CONCLUSION

Three models are investigated in this paper. None of them can be used as a full model as they are presented in this paper, but investigating them without many different loss models obscuring the results might provide insight into the actual physics involved. The authors conciders the effect due to the mismatch beween geometry and flow at runner inlet, often called *incipient losses*, to be the significant inlet effect, shown to significantly move the efficiency from 1 for all rotational speeds and opening degrees, to something that looks like an actual efficiency curve. Even if improved compared to the Euler model, the flow/rotational speed characteristics of the new model indicates that the new model is unable to fully capture the significant physics involved.

More work is needed to include other sources of loss in the energy transformation process occuring inside a hydraulic turbine, as well as including terms that is present at transient conditions. The authors are currently working on this.

## REFERENCES


Cengel, Y. A. & Cimbala, J. M. 2014. *Fluid Mechanics, Fundamentals and Applications*, McGraw Hill education.

Giosio, D. R., Henderson, A. D., Walker, J. M. & Brandner, P. A. 2017. Physics-Based Hydraulic Turbine Model for System Dynamic Studies. *IEEE Transactions on Power Systems,* 32**,** 1161-1168.

Nielsen, T. K. 1990. *Transient characteristics of High Head Francis Turbines.* PhD, NTH.

Nielsen, T. K. Dynamic Behaviour of Governing Turbines Sharing the Same Electrical Grid. 18th IAHR Symposium on Hydraulic Machinery and Cavitation, 1996 Valencia. Kluwer Acafdemic Publisher, 769-778.

Nielsen, T. K. 2015. Simulation model for Francis and Reversible Pump Turbines. *International Journal of Fluid Machinery and Systems,* 8**,** 169-182.

Nielsen, T. K. & Storli, P.-T. 2014. Measurements and simulations of turbines on common grid. *IOP Conference Series: Earth and Environmental Science,* 22**,** 042018.

MathWorks (accessed 06.10.17): https://se.mathworks.com/matlabcentral/fileexchange/29370-newton-method-in-n-dimensions?s_tid=srchtitle

Trivedi, C., Cervantes, M. J. & Dahlhaug, O. G. 2016. Experimental and numerical studies of a high-head francis turbine: A review of the francis-99 test case. *Energies,* 9**,** 74.

Wylie, E. B. & Streeter, V. L. 1983. *Fluid transients,* Ann Arbor, Mich., FEB Press