CFD simulation of transient startup for a low specific-speed pump-turbine

E Casartelli, O Ryan, A Schmid and L Mangani

Lucerne University of Applied Sciences & Arts, Department of Mechanical Engineering, Technikumstrasse 21, CH-6048 Horw
E-mail: ernesto.casartelli@hslu.ch

Abstract. Hydropower plants are very well suited for the modern electricity market which depends on increased flexibility and storage capabilities. Nevertheless, increased dynamic operation leads to new challenges in the design as well as the mechanical integrity of the equipment. In particular, the number of start and stops has significantly increased in recent years. The startup of the machine is a transient procedure with considerable load-oscillations which can affect the lifespan of the machine. Detailed understanding of transient effects in pump-turbines is therefore useful for improved prediction of loads and an important tool in pump-turbine layout and design. This study discusses the application of unsteady RANS CFD to fully transient pump-turbine operations. A low specific-speed pump-turbine during startup is investigated and results are discussed accordingly. The boundary conditions are imposed from 1D hydro-acoustic results of the power plant, using time dependent quantities like guide-vane angle, rotational speed and flow rate. Previous studies of the same authors contain simulations such as runaway or load-rejection, where the same method has been successfully applied. Turbine-startup results in terms of head and torque can be compared to the 1D model data for the validation of the CFD procedure. Instantaneous fluctuation of the same quantities can be observed as runner speed increases. A flow feature analysis shows that partial pumping takes place in all individual runner passages, resulting in significant global pressure fluctuations. This phenomenon can be associated with unsteady vortex formation as the characteristic reaches the speed-no load curve close to the S-shape of the characteristic.

1. Introduction

Electricity production and the operation of pump-storage has changed significantly over the last years and will continue to do so. More specifically, the variable component of electricity delivery is higher than at the time when most pump-storage facilities were built. Today’s changed demands result in pump-turbine operation which is more dynamic and leads to an increased number of start-up, stops and other transient operations.

As mentioned in [1], especially near the S-shape of the pump-turbine characteristic, frequent oscillations and transient operations take place which affect the durability of the machine. As a consequence, the structural load is more dynamic and load changes are more frequent, which ultimately leads to shorter life spans of pump-turbine components.

Modern CFD methods allow to simulate such dynamic operations. They give valuable insight into understanding how loads develop and where these loads affect day-to-day operation. A better understanding of the flow situation in dynamic operation allows to improve process regulation and can increase the lifespan of machines. Similar recent studies for various transient
operations such as [2] and [3] show, that CFD is an important tool for predicting loads for machine operation under dynamic operating conditions.

In a study by Li et al.[4], 3D unsteady RANS using the RNG k-ε turbulence model is applied to runaway and load-rejection simulations. A set of differential equations is used for the modeling discharge, runner speed and guide vane opening. Results are compared to measurements and errors in the range of ± 5% are noted.

Cherny et al.[5] use the angular momentum equation to simulate variable runner speed for turbine runaway and load rejection in 3D unsteady RANS simulations. A 1D elastic equation for the penstock is used to prescribe a time dependent head at the inlet of the 3D domain. Draft tube vortex rope formation could successfully be simulated during runaway.

In Nicolle et al.[6] and [7], two different turbine-startup scenarios are simulated using variable runner speed and moving mesh methods. Starting from a 1 degree guide vane opening, a transient turbine-startup is modeled for a Francis turbine. A moving mesh strategy is applied and structural loads are computed using a quasi-static approach. The results show good agreement with measurements and it is noted that a rotating stall is captured near speed-no load.

The focus of the present study is the simulation of the transient turbine-start operation based on 1D hydraulic system simulation data. A fully coupled in-house CFD solver with moving mesh capabilities is applied here for the unsteady RANS computations. The results are discussed based on global quantities, as well as a detailed flow analysis.

2. Numerical Setup

2.1. Solver

The choice of CFD solver for this study is based on three requirements. The first requirement is code speed, since transient change of operating mode requires significant computational power. The second requirement is overall robustness of the code, since flow conditions change frequently and unexpected flow phenomena can appear. Furthermore, the third requirement is the moving mesh capability of the solver due to guide vanes moving in the turbine start procedure.

The solver used here is the block coupled solver by Mangani et al.[8]. The governing equations are coupled implicitly and therefore the robustness and code speed criteria can be met. Linear scalability of computational effort with mesh size variation is also of great advantage for this study. The system of equations shown in Equation 1 is solved simultaneously. Cell-values are denoted by the subscript "C" and neighbour contributions by the subscript "NB".

The code is also capable of handling mesh deformation which is necessary for the handling of the guide vane movement in the current case. Instead of solving for an additional set of equations which accounts for mesh deformation, explicit mesh deformation using the inverse distance weighting approach is applied. Since no additional system of equations has to be solved, this method is computationally very efficient and therefore suitable for large simulations. More information on the method and the implementation can be found in [9] and other examples can be found in [2] and [3].

$$\begin{bmatrix}
  a_{uu}^{C} & a_{uw}^{C} & a_{uw}^{C} & a_{up}^{C} \\
  a_{wu}^{C} & a_{wv}^{C} & a_{wu}^{C} & a_{wp}^{C} \\
  a_{vu}^{C} & a_{vu}^{C} & a_{vu}^{C} & a_{vp}^{C} \\
  a_{ww}^{C} & a_{ww}^{C} & a_{ww}^{C} & a_{pp}^{C}
\end{bmatrix} \cdot \begin{bmatrix}
  u_{C} \\
  v_{C} \\
  w_{C} \\
  p_{C}
\end{bmatrix} + \sum_{NB} \begin{bmatrix}
  a_{uu}^{NB} & a_{uw}^{NB} & a_{uw}^{NB} & a_{up}^{NB} \\
  a_{wu}^{NB} & a_{wv}^{NB} & a_{wu}^{NB} & a_{wp}^{NB} \\
  a_{vu}^{NB} & a_{vu}^{NB} & a_{vu}^{NB} & a_{vp}^{NB} \\
  a_{ww}^{NB} & a_{ww}^{NB} & a_{ww}^{NB} & a_{pp}^{NB}
\end{bmatrix} \cdot \begin{bmatrix}
  u_{NB} \\
  v_{NB} \\
  w_{NB} \\
  p_{NB}
\end{bmatrix} = \begin{bmatrix}
  b_{u} \\
  b_{v} \\
  b_{w} \\
  b_{p}
\end{bmatrix}$$

(1)

2.2. Computational Domain, Mesh and Simulation Setup

For this transient analysis a prototype scale pump-turbine is used. It is built from four main components which are shown in Figures 1 and 2: A. Spiral casing and stay vanes (SV), B. guide vanes (GV), C. runner and D. exit cone & draft tube. Leakage flow and disk friction are neglected entirely for this analysis.
All mesh components are structured grids with a resolution typical for industrial application (i.e. medium mesh resolution for efficient simulation times). The overall cell count is 5 million, and a boundary layer resolution with an average $y^+$ of approximately 30 is applied.

Turbine start operation requires the GVs to open from 0 degrees. No remeshing procedure is applied here, therefore a minimum GV starting angle of 1 degree is defined. The mesh topology and the moving mesh parameters are optimized for an operating range between 1 to 5 degrees in order to fully capture the startup procedure. Figure 2 shows the GVs at approximately 3 degrees.

The discretization of the flow equations is second order in time and space, with a backward Euler temporal scheme and a high resolution spatial scheme. Turbulence is modeled using the standard $k-\varepsilon$ turbulence model. A recent study shows that the $k-\varepsilon$ model is superior near the S-shape of the pump-turbine characteristic compared to other two equation models such as the SST model [3, 10].

Figure 1. Computational domain with 4 cell zones.  
Figure 2. Mesh detail of stay vanes, guide vanes and runner.

2.3. Boundary Conditions
During turbine start up, the GVs are opened a few degrees and the runner is accelerated to synchronization speed near the speed-no load curve where torque is close to zero. From there, the GV opening can be increased steadily until normal operation is achieved. Here, the first part of this procedure is simulated up to the point where the speed-no load point is reached.

The boundary conditions for the turbine start operation are taken from the 1D simulation of a fictive, though realistic hydraulic system and the measured static machine characteristic at model size. Three parameters as functions of time are changed continuously during the 3D simulations as can be seen in Figure 3: The flow rate at the inlet, the rotational frequency and the GV angle. Originally, the 1D simulation starts at a GV opening angle of 0 degrees. Due to the meshing strategy chosen here, a starting angle of 1 degree is applied as mentioned above. Consequently, the 1D data is cut off at 1 degree. A 2 second starting phase with constant parameters is defined for initialization. After initialization, the GVs are opened and the rotational frequency steadily increases to synchronization speed. The flow rate increases and decreases again once the maximum GV opening is reached. At the outlet, a constant average pressure is defined.

The time step is fixed at 5 degrees runner rotation at maximum rotational frequency. Using this time step, the 50 seconds real time can be simulated in approximately 20 days using 120 CPU
cores in parallel (Intel Xeon E5-2630). Convergence is assured by starting with a steady-state solution and using a maximum of 10 inner iteration loops per time step.

![Figure 3](image)

**Figure 3.** Boundary conditions for the CFD simulations, based on a 1D system simulation.

### 3. Results

Since the turbine start procedure ends in operation near the speed-no load curve, hydraulic instability in the form of flow oscillations or rotating stall can occur. Global quantities are monitored and evaluated accordingly. Based on that, flow features at significant times are examined.

#### 3.1. Global Quantities

In Figures 4 and 5 the results of the turbine start simulation are illustrated in the four quadrant characteristic. The graph shows the $K_{cm1}$ coefficient as a function of the $K_{u1}$ coefficient (cf. Equations 2 and 3) and displays multiple curves of same GV opening angle. Both, the 1D data and the 3D results are shown. Overall, 1D and 3D results agree well with regard to the $K_{cm1}$ and $K_{u1}$ coefficients. The 3D CFD-results curve starts near the origin of the coordinates and moves right, towards speed no-load. Past 16 seconds, the 3D CFD results start to overestimate $K_{u1}$ compared to the 1D data. Near speed-no load, sudden oscillations occur and CFD overestimates the 1D $K_{u1}$ values even further. Consequently, the CFD results also predict a different speed-no load curve than the 1D data.

\[
K_{cm1} = \frac{4Q}{D_1 2\pi \sqrt{2gH}} 
\]

\[
K_{u1} = \frac{n\pi D_1}{\sqrt{2gH}}
\]

In order to analyze this observation, head and torque behavior of the 1D hydraulic system simulation are compared against the 3D CFD results in Figures 6 and 7. The figures show the 1D data, the 3D data and the smoothed 3D data for comparison.

The head distribution as a function of time shows an initial peak which is 15% higher than the 1D baseline due to starting at a GV angle of 1 degree instead of 0 degrees. After the initial peak the CFD and 1D curves converge. After 16 seconds, the CFD results underestimate torque...
and drop below the 1D data. The 3D simulation curve then consistently follows the 1D curve, although with a 3-4% negative offset. Shortly after 26.7 seconds, already before maximum $K_{u1}$ is reached, the head signal starts to oscillate. The amplitude of the oscillation is variable with an average magnitude of approximately 2% of the head.

Taking a look at the torque distribution, the initial overestimation shortly after 6 seconds is also significant, approximately 20-25% compared to the 1D data. Close to 10 seconds real time, peak torque is overestimated by approximately 5% compared to the 1D data. Same as the head near 16 seconds, the 3D torque distribution also crosses the 1D torque curve. Then, it stabilizes itself at a 2% underestimation of the 1D data, relative to the reference torque which is located near the point of best efficiency. The sudden oscillation from the head can also be seen in the torque with a magnitude equivalent to approximately 5% of the reference torque value.

The head and torque oscillations which can be observed in the four quadrant characteristic, can be associated to the approach of the speed-no load close to the S-shape during turbine start.
3.2. Flow Features

For the following discussion, the turbine-startup procedure is split into five phases for which distinct flow features can be observed. From these five phases, six different operating conditions at 10, 16, 18, 20, 26.7 and 28.05 seconds are discussed in further detail. The six operating conditions are also highlighted in Figures 3 through 7.

1. **0 - 16 seconds**: Counterclockwise (sense of rotation same as runner) rotating single vortex symmetric for each runner passage at very low runner speeds (Figure 8 at 10 seconds and Figure 9 at 16 seconds).

2. **16 - 18 seconds**: Primary counterclockwise (sense of rotation same as runner) rotating and secondary clockwise (sense of rotation against runner) rotating vortices form, symmetric for all runner passages (Figure 10 at 18 seconds).

3. **18 - 24 seconds**: Transition phase between counterclockwise (sense of rotation same as runner) and clockwise (sense of rotation against runner) rotating single vortex due to incidence sign change (Figure 11 at 20 seconds).

4. **24 - 27 seconds**: Distinct, clockwise (sense of rotation against runner) rotating single vortex, symmetric in all runner passages (Figure 12 at 26.7 seconds).

5. **27 - 50 seconds**: Frequent and irregular oscillations near speed-no load (Figure 13 at 28.05 seconds).

An evaluation of the relative velocity vector fields is considered for the comparison between the different selected operating conditions. A constant height cut plane at GV mid-span for the two regions is shown in Figures 8-13.

The first operating condition is at 10 seconds, which corresponds to the point of maximum torque in Figure 7. At this point, the flow rate has already reached its maximum and slowly decreases as the startup procedure continues. The flow around the GVs and visible SVs in Figure 8 is steady-state and symmetric over the whole circumference of the machine. It can be observed, that the nearly closed position of the GVs lead to a stagnation point on the GVs far upstream on the pressure side, corresponding to low flow rates. The flow situation in the SVs and GVs remains approximately the same throughout the turbine start, as the overall flow rate remains small. Initially, runner incidence is very high due to low runner speeds and the small GV opening. The stagnation point can be seen as a black dot, located downstream of the leading edge on the pressure side of the runner. This results in a distinct counterclockwise rotating vortex structure due to flow separation on the suction side of the blade. This vortex structure is symmetric for each runner passage and is highlighted in Figure 8.

As mentioned above in the global quantities discussion, the 1D and 3D data diverges after 16 seconds real time. Figure 9 shows the last point where 1D and 3D data still agree very well (less than 0.1% difference). The single counterclockwise rotating vortex structure still persists (cf. highlighted area), though the head has decreased (indicated by the reduced extension of low pressure colored in blue). The runner speed is higher, which results in a smaller incidence angle compared to the previous flow state in Figure 8. The stagnation point moves further downstream compared to the previous figure.

Only 2 seconds later (Figure 10), a slow transition to another stable condition takes place where an additional, clockwise rotating vortex slowly forms at the leading edge of the blade as runner speed further increases. Both, the initial counterclockwise rotating vortex and the second clockwise rotating vortex are highlighted in this illustration. This again occurs simultaneously throughout all runner passages as the incidence angle changes even further. Now, the stagnation point has moved close to the leading edge, though still being located on the pressure side. At this point, the 3D head is underestimated by approximately 2% compared to the 1D data.
Figure 8. LIC of relative velocity on $z = 0$ cut plane; Quasi-steady operating condition at $t = 10$ seconds; Red = High static pressure, Blue = Low static pressure.

Figure 9. LIC of relative velocity on $z = 0$ cut plane; Quasi-steady operating condition at $t = 16$ seconds; Red = High static pressure, Blue = Low static pressure.

Figure 10. LIC of relative velocity on $z = 0$ cut plane; Quasi-steady operating condition at $t = 18$ seconds; Red = High static pressure, Blue = Low static pressure.

Figure 11. LIC of relative velocity on $z = 0$ cut plane; Quasi-steady operating condition at $t = 20$ seconds; Red = High static pressure, Blue = Low static pressure.
Followed by that event, in Figure 11 the flow state at 20 seconds is shown. The time line of flow features shows that the two previously observed distinct counteracting vortex structures change their appearance. From the two spanwise vortices, a streamwise oriented vortex develops in this state of transition which is highlighted with a dashed line. This phenomenon is again symmetric throughout the whole runner and remains stable. This can be associated with the still increasing runner speed which changes the appearance of the flow field. In this span section, the stagnation point is nearly ideal. In addition, at this point, the guide vane opening angle is reduced as can be seen in Figure 3.

Figure 12, after 26 seconds, shows the last non-oscillating operating condition with steady-state-like behavior. When looking at a single runner passage, a large vortex structure can be seen again. On the left-hand side this vortex is highlighted, which again is symmetric for all runner passages. The vortices seen here rotate in a clockwise sense, as opposed to the counterclockwise sense previously noted. This can be associated with the now negative incidence as runner speed increases even further and the flow rate decreases overall. As the stagnation point moves to the suction side, the vortices are formed by leading edge separation on the pressure side of the blade. Again, flow similarity for all passages can be noted, which is also observed in [7] before approaching speed-no load.

Past 28 seconds, the flow field becomes unstable and irregular and non-symmetric vortex formation takes place, as can be seen in Figure 13. This corresponds to the oscillations previously observed in the global quantities analysis. A detailed look at the time series in the oscillating operation shows how the vortices form. It reveals that a clockwise rotating vortex from transient flow separation at the leading edge propagates downstream and interacts with the clockwise rotating main vortex core of the previously stable flow. An example of two vortices interacting with each other in a single passage is highlighted with two red circles in Figure 13. In this state, not every channel has the same number of vortices at the same time. In the current illustration, for example, the upper three passages show two vortices, whereas the others show only one main...
vortex. The number of either 1 or 2 large vortices in each passage continuously and irregularly changes during this phase of operation.

In Figures 14 and 15, the previously highlighted vortical structures of Figures 12 and 13 are shown again with relative-velocity streamlines at the same time steps. In addition, the radial velocity in the vaneless space is shown, which is useful to indicate backflow regions. In Figure 14, the stable, non-oscillating operating condition shows the distinct steady-state flow single spanwise vortex in the runner passage. The radial velocity vectors in the vaneless space mainly point inward, i.e. in the general flow direction.

As the flow rate is further reduced and the flow field moves to the oscillating phase, the same illustration of the oscillating operating condition in Figure 13 is shown in Figure 15. Here, the two vortices previously observed can be seen. The upstream vortex is formed by the main flow separation zone at the leading edge and results in partial pumping of the flow before it propagates downstream as the runner turns. The moment this illustration is captured represents the same time step when the upper amplitude of the oscillation occurs. The partial pumping of the runner can clearly be seen when considering the radial velocity vectors, which point outwards, against the general flow direction. The inward and outward flowing portions in the vaneless space are close to an equilibrium. They cancel each other out, though being in a dynamic, non-stable mode. Therefore, a close to zero flow rate near speed-no load can be correlated with the counteracting inward and outward moving flow. This state changes continuously and irregularly and is the main cause of the oscillations when approaching speed-no load during turbine start.

4. Conclusive Remarks

In this study, a transient turbine start of a pump-turbine is simulated starting at GV angle opening of 1 degree using a block-coupled CFD solver. The boundary conditions are based on 1D hydraulic system simulations. For the present operating conditions, the use of the $k-\varepsilon$ two equation turbulence model yields best results as previous studies have shown. Initially, the flow field remains stable and quasi-steady flow structures are dominant. As the simulation progresses, head and torque are both underestimated by approximately 3% while sudden oscillations start to occur. Differences in the comparison of the 1D to the 3D data occur for to two main reasons. The first reason is that the 1D model is based on the static characteristic of the machine. Therefore, possible effects due to hysteresis are not modeled. However, the transient 3D flow simulation will capture those effects, which potentially leads to differences among those two models. The
second reason is the 1 degree GV starting position in the current simulation. This is originally not the case for the 1D data, which starts at 0 degrees and is simply cut off at 1 degree for the 3D simulation. This explains the initial peak in head when comparing the 1D vs. the 3D data. Tests with a 2 degree starting angle show that the GV position boundary condition is significant for the development of the results. The 1 degree GV starting angle yields better agreement with the 1D system simulation data.

A closer examination of the flow in the oscillating regime shows that partial, oscillating pumping takes place once the simulation reaches speed-no load. Unsteady vortex formation from the runner blade leading edges results in back flow regions in the vaneless space. The backflow region and inflow region per channel passage cancel each other out which results in a near zero average flow rate. The formation of the described vortex occurs irregularly in all runner passages, while the partial pumping phenomenon simultaneously oscillates over the whole circumference of the runner. Due to this synchronous oscillating behavior of the pumping effect, a rotating stall occurrence is ruled out.

The study shows, that the in-house solver used for this simulation is robust and accurate enough to simulate transient turbine start and important flow features can be identified. In theory, the resulting data could be used to understand where problematic regions are located in terms of structural load or load changes. This would ultimately allow to determine how the turbine start procedure could be improved to reduce stress and therefore increase the lifespan of certain machine components. Further investigations could benefit from a detailed frequency analysis and the effect of additional turbulence modeling approaches on the turbine start characteristic and other dynamic pump-turbine operating modes.

Acknowledgements
The authors would like to thank C. Widmer and W. Michler of Andritz Hydro in Kriens, Switzerland, for providing the geometry and the system boundary conditions, thus allowing to test the code capabilities for a realistic scenario. The authors would also like to thank SCCER-FURIES for financial support.

References
[1] Staubli T, Senn F and Sallaberger M 2008 Instability of pump-turbines during start-up in turbine mode. Hydro2008, Ljubljana, Slovenia, Paper, 9
[2] Casartelli E, Mangani L, Ryan O and Schmid A 2016 Application of transient CFD-procedures for S-shape computation in pump-turbines with and without FSI IOP Conference Series: Earth and Environmental Science 49(4) 042008 IOP Publishing
[3] Casartelli E, Del Rio A, Schmid A and Mangani L 2017 CFD computation of transients in Pump-turbines. Hydro2017, Sevilla
[4] Li J, Yu J and Wu Y 2010 3D unsteady turbulent simulations of transients of the Francis turbine IOP Conference Series: Earth and Environmental Science 12(1) 012001 IOP Publishing
[5] Cherny S, Chirkov D, Bannikov D, Lapin V, Skorospelov V, Eshkunova I and Avdushenko A 2010 3D numerical simulation of transient processes in hydraulic turbines IOP Conference Series: Earth and Environmental Science 12(1) 012071 IOP Publishing
[6] Nicolle J, Morissette JF and Giroux AM 2012 Transient CFD simulation of a Francis turbine startup IOP conference series: Earth and Environmental Science 15(6) 062014 IOP Publishing
[7] Nicolle J, Giroux AM and Morissette JF 2014 CFD configurations for hydraulic turbine startup IOP Conference Series: Earth and Environmental Science 22(3) 032021 IOP Publishing
[8] Mangani L, Buchmayr M and Darwish M 2014 Development of a novel fully coupled solver in openfoam: Steady-state incompressible turbulent flows Numerical Heat Transfer B 66(1) 1-20
[9] Casartelli E, Mangani L, Romanelli G and Staubli T 2014 Transient simulation of speed-no load conditions with an open-source based C++ code IOP Conference Series: Earth and Environmental Science 22(3) 032029 IOP Publishing
[10] Deniz S, Del Rio A and Casartelli E 2018 Experimental and numerical investigation of the speed-no-load instability of a low specific speed pump-turbine with focus on the influence of turbulence models IOP Conference Series: Earth and Environmental Science Vol,p tbd IOP Publishing