Application of transient CFD-procedures for S-shape computation in pump-turbines with and without FSI

E. Casartelli, L. Mangani, O. Ryan, A. Schmid
Lucerne University of Applied Sciences & Arts, Department of Mechanical Engineering, Technikumstrasse 21, 6048, Luzern, CH.
E-mail: ernesto.casartelli@hslu.ch

Abstract. CFD has entered the product development process in hydraulic machines since more than three decades. Beside the actual design process, in which the most appropriate geometry for a certain task is iteratively sought, several steady-state simulations and related analyses are performed with the help of CFD. Basic transient CFD-analysis is becoming more and more routine for rotor-stator interaction assessment, but in general unsteady CFD is still not standard due to the large computational effort. Especially for FSI simulations, where mesh motion is involved, a considerable amount of computational time is necessary for the mesh handling and deformation as well as for the related unsteady flow field resolution. Therefore this kind of CFD computations are still unusual and mostly performed during trouble-shooting analysis rather than in the standard development process, i.e. in order to understand what went wrong instead of preventing failure or even better to increase the available knowledge.

In this paper the application of an efficient and particularly robust algorithm for fast computations with moving mesh is presented for the analysis of transient effects encountered during highly dynamic procedures in the operation of a pump-turbine, like runaway at fixed GV position and load-rejection with GV motion imposed as one-way FSI. In both cases the computations extend through the S-shape of the machine in the turbine-brake and reverse pump domain, showing that such exotic computations can be perform on a more regular base, even if quite time consuming. Beside the presentation of the procedure and global results, some highlights in the encountered flow-physics are also given.

1. Introduction

The actual electricity-production is characterized by a dynamic request, and hydropower generation needs to take up a more and more flexible role. This leads to an increased number of start and stops, enlarged operational range and consequent transient load-variation of the machines. This affects not only the hydraulic performance but also the mechanical integrity of the plant, as exemplarily summarized in [1] for Francis turbines. Occurring effects need to be investigated in detail in order to increase the understanding and adapt design procedures to develop machines able to face future production strategies without problems. In the last years various researchers presented different methodologies to investigate transient operation like start-up, runaway and load-rejection and related mechanical stresses ([2], [3], [4], [5], [6]). The different approaches use various strategies for model and mesh resolution, in order to reduce the computational effort. The machine speed is often computed by solving an ODE equation with the torque out the CFD computation.

In the present paper a runaway and load-rejection simulation have been performed using the...
full machine from spiral casing inlet to draft tube outlet. The boundary conditions are taken from 1D hydroacoustic simulations, imposing machine speed, flow rate and for the load-rejection case also the GV position. The novelty in the present paper is that for both simulations the operation starts in turbine mode and moves through turbine-brake into reverse-pump and back. From a numerical point of view this is a particularly challenging task, which was tackled with a novel, particularly robust coupled solver with FSI capabilities (see for example [7] and [8]).

In [9] similar simulations were performed from pump through pump-brake to turbine mode. One important result of that investigation was that transient results in the kcm-ku-plane do not follow the steady state results. This indicates that unsteady analysis seems to be necessary for the investigation of such severe transients.

2. Numerical procedure

2.1. CFD Solver

Speed and robustness are among the most important requirements for any software that has to be used for the time-accurate numerical analysis of complex highly unsteady phenomena. When programming a CFD code the best way of combining both requirements is to couple the governing equations implicitly, since resolving efficiently the pressure-velocity coupling is essential for the performance of any CFD code. However up until today the SIMPLE family of algorithms [10], which couples the governing equations only by means of sub-looping, solving sequentially each governing equation, still remains the predominant methodology used in the CFD community. Compared to block coupled implicit algorithms, segregated algorithms lack scalability with mesh size, robustness and calculation speed, which is inherent due in part to the under-relaxation needed to stabilize the algorithm. This is even more critical for unsteady simulations, as a set of inner non-linear iterations must be accounted for, in order to drive the solver to the desired accuracy level in time.

In order to overcome these shortcomings, Mangani et al. [7] developed a block coupled incompressible solver using the open-source CFD library OpenFOAM® as programming platform. Therein the algebraic system resulting from the Navier-Stokes equations are solved simultaneously with the following system of equations, where ‘C’ are the cell-values and ‘NB’ the neighbour contributions:

\[
\begin{bmatrix}
  a_{uu}^C & a_{uw}^C & a_{uw}^C & a_{up}^C \\
  a_{uv}^C & a_{vv}^C & a_{uw}^C & a_{vp}^C \\
  a_{wu}^C & a_{vw}^C & a_{vv}^C & a_{wp}^C \\
  a_{wu}^C & a_{wu}^C & a_{vp}^C & a_{wp}^C 
\end{bmatrix}
\begin{bmatrix}
  u_C \\
  v_C \\
  w_C \\
  p_C 
\end{bmatrix}
+ \sum_{NB}
\begin{bmatrix}
  a_{uu}^{NB} & a_{uw}^{NB} & a_{uv}^{NB} & a_{up}^{NB} \\
  a_{uv}^{NB} & a_{vv}^{NB} & a_{uw}^{NB} & a_{vp}^{NB} \\
  a_{wu}^{NB} & a_{vw}^{NB} & a_{vv}^{NB} & a_{wp}^{NB} \\
  a_{wu}^{NB} & a_{wu}^{NB} & a_{vp}^{NB} & a_{wp}^{NB} 
\end{bmatrix}
\begin{bmatrix}
  u_{NB} \\
  v_{NB} \\
  w_{NB} \\
  p_{NB} 
\end{bmatrix}
= \begin{bmatrix}
  b_u^C \\
  b_v^C \\
  b_w^C \\
  b_p^C 
\end{bmatrix}
\] (1)

Segregated algorithms, as implemented for example in OpenFOAM® built-in unsteady solver pimpleFoam, operate using many sub-loops to account for inter-equation coupling, continuously updating the right hand side of the discretized governing equations, which contains field values of previous iteration steps. Additionally under-relaxation of the governing equations is needed to assure that the solution process remains numerically stable.

In block-coupled algorithms sub-looping is also applied in order to update second order derivative terms and non-linear terms. However, in contrast to segregated algorithms, the intrinsic inter-variable coupling of the approach is much stronger, thus needing less sub-loops. Additionally under-relaxation can be completely avoided using so-called false transient time stepping. Therefore the solution of such a discretized system of equations results to be numerically much more stable than that of segregated algorithms and also turns out to be significantly faster in terms of calculation time, which has been demonstrated by Mangani et al. [7].

For the moving mesh and deformations, once the structural displacements and velocities are described on the relevant boundary patches, it is necessary to tackle the problem of adjusting the mesh to the new configuration in such a way that grid quality is not degraded significantly, e.g.
with non-negative cell volumes, moderate levels of stretching and non-orthogonality. This task of moving what can count several millions of nodes has to be performed thousand of times during an unsteady FSI computation (for both one-way or two-way coupling). Therefore the availability of efficient, massively parallel and robust mesh deformation tools and topology modifiers is crucial. In order to achieve the best compromise between those opposing requirements, rather than using algorithms that require solving a systems of equations (computationally expensive), we choose an explicit mesh deformation technique, as described in [8].

2.2. Numerical setup
The CFD computations were performed using an operational, full size pump-turbine prototype connected to a hypothetical hydraulic system. The hypothetical hydraulic system was used only for non-disclosure reasons, while the resulting entire system still retains a realistic behavior. 1D transient simulations were performed in order to generate appropriate boundary conditions for the 3D CFD computations. For this investigation it was not retained necessary to determine the machine speed from the CFD torque and mechanical inertia of the runner-generator as often seen in recent publications, considering that other simplifications of the numerical model are taken into account, like neglecting leakage flow and disk friction.

For the present study the computations were performed using the SST turbulence model and fully second order discretization for both space and time. The latter is achieved by means of inner non-linear iterations (for the present simulations 4-6 sub-iterations were sufficient to achieve a drop of 5 orders of magnitude of the inner residuals) and using a fixed time-step of 0.0016667s, corresponding to an angular resolution between 4 and 6 degrees, depending on the instantaneous, variable machine speed imposed according to the 1D results.

Both simulations were computed on 96 cores during 50s real time, which corresponds to approx. 420 revolutions. The runaway simulation lasted 2.5 weeks. For the load rejection case a 20% increase in computational time was observed, due to the additional effort involving the mesh motion (one-way FSI).

2.3. Mesh
As indicated above, the computational-mesh for the simulations corresponds to the prototype. The mesh count is 5 million cells and includes the entire machine from the spiral casing to the draft tube. The resolution corresponds to that of a typical industrial mesh. Particular attention
was dedicated to the guide vane topology, which moves, in the load-rejection case, from 24 to 6 degrees. A sketch of the mesh and computational domain is given in Figure 1.

### 2.4. Boundary conditions

Two simulations, one for runaway and one for load-rejection, were performed starting from the same stable condition at 24 degrees GV opening and at design speed. The load rejection simulation includes one-way FSI for the GV motion, which setting’s changes from 24 to 6 degree. The boundary conditions were taken from the 1D transient system simulations, imposing the resulting angular velocity, GV position and mass-flow as a function of time (see Figure 2). For both cases the operating conditions extend through the S-shape region of the machine-characteristic into the turbine-brake and reverse-pump domain.

While in turbine and turbine-brake mode the domain inlet is in the spiral casing, during reverse-pump operation it has to be switched to the draft tube exit. From a numerical point of view this is quite a severe change, where the BCs switch in one time-step from inlet to outlet and vice versa at the other end. This leads to a sort of numerical water-hammer which, thanks to the high code robustness, damps out in few iterations and does not affect the overall result.

In the load-rejection case no GV control was used, which results in a linear closing-law for the gates. This leads to an undamped oscillation in the hydraulic system after the final GV position is reached.

![Simulations](image)

**(a) Runaway**

**(b) Load rejection**

*Figure 2: Imposed boundary conditions for the simulations*

### 3. Results

The global results obtained for both simulations are head and torque over time. Additionally virtual pressure probes were introduced locally in selected locations to monitor the transient behavior.

Experimental data for an independent validation of the results is not available, since the hydraulic system is not existing, as previously indicated. Nevertheless a direct comparison with experimental results can be performed for the starting point of the simulations, where constant boundary conditions were imposed for 2s before starting with the dynamic BC’s. Here the differences between CFD and experiments is less than 2%. Additionally it can be observed that the 1D simulations include the measured turbine characteristic: the comparison with the 3D CFD results can be therefore considered as a further validation, showing that the proposed methodology can be applied for this kind of investigations.
Figure 3 shows both the 1D and 3D results on the ku-kcm plane, giving an overview of the machine behavior during the simulated transient operation. In this kind of representation it is clearly visible that for both simulations the operation moves from turbine mode to turbine-brake into reverse pump and back during the subsequent oscillatory behavior.

During the first 2 real-time seconds the conditions are kept constant. Then dynamic BC are imposed according to the 1D results. This point in time is considered as reference for the times given in the following discussion and therefore defined as $t = 0$.

First the results for runaway will be discussed. Due to the constant GV position, the observed phenomena are completely defined by the changes in machine speed and volume flow.

Next the results for load rejection will be presented. Imposing a GV moving-law, additional transient effects are introduced in the simulation and will partly shift the behavior observed in the runaway simulation.

**3.1. Runaway Simulation**

As mentioned above, the runaway simulation is performed at a constant GV angle of 24 degrees. This corresponds to a larger opening than at design conditions. The results are depicted in Figures 4a) and 5a).

In general it can be stated that both head and torque from the CFD simulation follow consistently the 1D prediction over the complete simulated time-frame, despite the highly transient behavior. The filtered head signal presents deviations of less than 3% during turbine and turbine-brake operation, while in reverse-pump mode local differences up to -10% occur, as it can be seen in both the representation over time as well as in the ku-kcm plane. Corresponding details will be given below. The filtered torque is in general overestimated by 5 to 10%, except in the reverse pump-mode, where the difference nearly vanishes.
In the first 1.5s after start of the runaway situation, the behavior of head and torque is very smooth, while speed has increased by 10% and flow rate has reduced by 8%. From this point both head and torque start to show an increase in amplitude of the signal, which remains similar up to 5s after start, when the maximum speed is reached, which corresponds to the entrance in the S-shape. Just after that the amplitude of both signals increases drastically, while the machine is running through the S-shape, until about 9.2s, when the flow rate is reduced to nearly 10% of the original value. From this moment the oscillations reduces quickly, while the flow rate vanishes at 10.2s.

Figure 4: Head characteristic: instantaneous and filtered CFD-result with 1D-Simulations. Square-dots refer to specific postprocessing times in Fig. 6 and 7

Figure 5: Torque characteristic: instantaneous and filtered CFD-result with 1D-Simulations

As mentioned above, at this point inlet and outlet are switched, leading to a short oscillation (less than 0.1s, i.e. about 1 runner revolution) with very high peak values. This is due to the instantaneous change in flow direction. From a numerical point of view this is quite a severe situation, which can be dealt in the simulation thank to the robustness of the proposed flow solver.

In reverse pump mode the oscillations are significantly reduced. The filtered CFD head drops
faster than predicted by the 1D model over time as long as the flow-rate is decreasing (since negative, i.e. increasing in reverse-pump mode). As soon as the flow-rate starts to increase in direction of turbine-brake, the filtered CFD head flattens and moves toward the 1D solution. After returning in turbine-brake mode the amplitude of the oscillations quickly increases, while the filtered CFD signal moves closer to the 1D result, again in the 3% range. The machine moves then through turbine-brake operation in the S-shape, with large amplitudes of head and torque, up in a more stable regime when the flow rate reaches its maximum at around 22.7s. After this the flow rate reduces, the machine operates again in the S-shape region and the same behavior as above described is repeated.

3.2. Load Rejection

As mentioned above, the load rejection simulation is performed using a linear moving law for the guide vanes, starting from 24 down to 6 degrees in a realistic time-frame of just less than 14 s. After the closing-phase, the GV position is kept constant, i.e. no particular control law is applied, so that an oscillation persists in the hydraulic system.

Also in this case it can be generally stated that both head and torque from the CFD simulation follow consistently the 1D prediction over the complete simulated time-frame, as can be seen in Figures 4b) and 5b).

The deviations of the filtered CFD signals compared to the 1D simulation present a different picture than at runaway. In turbine and turbine-brake mode the head is underestimated between 1% and 5%, while the torque is overestimated by 2 to 6%. After entering reverse-pump mode the behavior switches, with an overestimation of the head and an underestimation of the torque of up to 10%.

Also during load rejection both head and torque are very smooth in the first phase, until the speed has increased by nearly 10% (as at runaway) and flow rate reduced by 17%. From this point, similarly to runaway, both head and torque start to show an increased signal-amplitude, which remains similar up to 4s after start. By this time the speed as reached more than 95% of the maximum speed. After this point in time the amplitude of the oscillations of both head and torque increase considerably and the machine passes from turbine in turbine-brake mode.

Approximately 3.5s later (i.e. 7.5s after starting to close the GVs) the no-flow condition is reached and the GV are at 13.4 degrees. Just before this point the amplitude is quickly reduced and in reverse-pump operation it stays quite small, similarly to the runaway simulation. At 14s the GV are closed to 6 degrees and the machine enters in turbine-brake mode, moving along the S-shape of the 6 degree curve. In this region, in contrary to what observed at runaway, the oscillations of the instantaneous head and torque remain small. This is attributed to the narrow setting of the GV, which leads to large dissipation in the vaneless space between GV and runner, thus strongly damping possible local instabilities like rotating stall, even in the S-shape region.

3.3. General remarks on global quantities

The analysis and comparison of the two simulations at runaway and load rejection shows that CFD is able to predict the general behavior of the 1D model even when the operation leads into the reverse-pump mode. Local differences are of course present due to simplifications in the CFD model (no leakage flow, no disk friction, . . . ). For the load rejection case the similarity with the 1D results is very high, with only a shift in magnitude for both head and torque which mainly depends on the flow rate and direction. At runaway there is a clear difference between 3D CFD and 1D model in the reverse-pump mode. The phenomenon seems a sort of dynamic hysteresis when changing flow direction, which can not be taken into account by the 1D model based on the static turbine characteristic. In this respect it has to be noticed that the CFD results are based on the 1D boundary-conditions which are themselves a result of the 1D computation.
(based on the static pump-turbine characteristic). This means that the actual, real behavior will be different and somewhere in-between. A way to avoid this would be a coupled simulation of the hydraulic and mechanical system with the 3D CFD.

Figure 6: Surface streamlines at midplane in Stay- and Guide-Vanes Section for the runaway-Simulation at different timesteps, indicated in Fig. 3

3.4. Flow features
In order to have a better understanding of the machine behavior, surface streamlines in the midplane in the stay- and guide-vane region are plotted at different selected times, at which particular flow structures can be observed.

The first analysis is performed for the runaway simulation, as shown in Fig. 6. The first selected time-step is at 1.5s, when the amplitude of the oscillations of the instantaneous head and torque increase compared to the values at constant conditions. The flow is well ordered and no particular effects can be noticed. The next picture presents the streamlines at 5s, just before the machine
enters the S-shape and the amplitude of the instantaneous signals increases considerably. It can be observed that at different locations at the interface to the runner the streamlines are nearly tangential, indicating that soon part of the GV-channels will be blocked. This is exactly the case at 9s, when the flow rate has reached nearly 10% of the original value. By this time more than half of the GV passages present a disturbed flow, with clear tangential pattern and vortices starting to block selected channels. One and a half second later, at 10.5s, the flow rate is practically zero. At midspan every GV passage is clearly blocked by a vortex which spans over the complete width. Vortical structures are also present in the stay-vanes, even if not so ordered and clear as in the GV region. This flow pattern is also characteristic for the reverse-pump mode, where only a minimal flow opening is present in each passage.

![Surface streamlines at midplane in Stay- and Guide-Vanes Section for the Simulation load rejection at different timesteps, indicated in Fig. 3](image)

Figure 7: Surface streamlines at midplane in Stay- and Guide-Vanes Section for the Simulation load rejection at different timesteps, indicated in Fig. 3

Fig. 7 presents the surface streamlines for interesting points during load rejection. Also for
In the present paper a runaway and a load-rejection simulation were performed, both ranging from turbine into reverse-pump mode. The proposed computational strategy allows to capture the relevant flow physics consistently, as comparisons of global data (head and torque) from 1D hydroacoustic results have shown. This is the case also when flow reversal occurs during reverse-pump operation. When moving through the S-shape, increased oscillations of the instantaneous global values are clearly detected, indicating unstable behavior of the machine. At runaway condition a clear difference in the behaviour of the 1D and 3D CFD results was detected. This should be due to a hysteresis when changing flow direction. This effect can not be taken into account by the 1D model based on the static machine characteristic. Future steps include a refinement of the numerical model, the investigation of the hysteresis and the application of a GV control-law in order to damp the oscillations of the hydraulic system. The methodology will then be applied not only to analyse the transient behavior of the turbine under various operating conditions, but also to assess control-strategies to reduce the effect of operational transients on mechanical integrity.

Acknowledgement
The authors would like to thank Mr. C. Widmer and Mr. 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.

References
[1] Trivedi C, Gandhi B and Michel C J 2013 Journal of Hydraulic Research 51 121–132
[2] Cherny S, Chirkov D, Bannikov D, Lapin V, Skorospelov V, Eshkunova I and Avdushenko A 2010 IOP Conference Series: Earth and Environmental Science vol 12 (IOP Publishing) p 012071
[3] Li J, Yu J and Wu Y 2010 IOP Conference Series: Earth and Environmental Science 12 012001 URL http://stacks.iop.org/1755-1315/12/i=1/a=012001
[4] Nicolle J, Morissette J and Giroux A 2012 IOP Conference Series: Earth and Environmental Science vol 15 (IOP Publishing) p 062014
[5] Nicolle J, Giroux A and Morissette J 2014 IOP Conference Series: Earth and Environmental Science vol 22 (IOP Publishing) p 032021
[6] Nennemann B, Morissette J, Chamberland-Lauzon J, Monette C, Braun O, Melot M, Coutu A, Nicolle J and Giroux A 2014 IOP Conference Series: Earth and Environmental Science vol 22 (IOP Publishing) p 032055
[7] Mangani L, Buchmayr M and Darwish M 2014 Numerical Heat Transfer, Part B: Fundamentals 66 1–20
[8] Casartelli E, Mangani L, Romanelli G and Staubli T 2014 IOP Conference Series: Earth and Environmental Science vol 22 (IOP Publishing) p 032029
[9] Liu J, Liu S, Sun Y, Jiao L, Wu Y and Wang L 2013 Journal of Mechanical Science and Technology 27 1305–1312
[10] Patankar S 1980 Numerical Heat Transfer and Fluid Flow (Hemisphere Publishing, Washington)