Numerical investigation of the flow behavior into a Francis runner during load rejection

P Côté\textsuperscript{1}, G Dumas\textsuperscript{1}, É Moisan\textsuperscript{2} and G Boutet-Blais\textsuperscript{2}
\textsuperscript{1} Laboratoire de Mécanique des Fluides Numérique (LMFN), Université Laval, Québec, Canada
\textsuperscript{2} Alstom Renewable Power Canada, Sorel-Tracy, Canada
E-mail: Philippe.Cote.14@ulaval.ca

Abstract. The main objective of the work presented in this paper is to investigate numerically the flow behavior inside a Francis hydro-turbine during the transient event of load rejection. First, a theoretical description of the flow during the event is presented in order to predict the global flow characteristics to be anticipated since no velocity profiles are available for this transient event. The issue of choosing the proper boundary conditions to obtain the absolute pressure and the correct flow characteristics within the runner when using a typical truncated geometry is then discussed. Finally, by using a hypothesis of “quasi-stationarity” and a validated methodology, global flow characteristics within the turbine are highlighted near the no-load operating condition and the unsteady vortical motions within the runner are assessed.

1. Introduction
The recent increase in computer power now allows for the numerical study of complex, highly turbulent and cavitating flows. Hydraulic engineers can indeed nowadays investigate flow behaviors in a hydraulic turbine for many transient conditions such as start-up and load rejection. In this work, the event of load rejection is investigated as it can induce dynamic loads into the runner. One case of interest consists of Francis runner undergoing vibration during a load rejection from its maximal power output regime. In fact, for this particular case, vibrations begin right before the runner reaches its maximal overspeed ratio, corresponding to the no-load operating condition in which the net torque applied on the runner is null. The clear frequency characterizing the phenomenon at full scale is $f = f_v$, defining a vibration timescale $T = 1/f_v$. The main objective of the work partly presented in this paper, and carried out in collaboration with Alstom Renewable Power Canada, is to investigate numerically the potential hydraulic sources of vibration in the runner at a model scale ($D = 0.35 \text{ m}$). This paper describes the path undertaken in order to elaborate a reliable, simple and efficient methodology to gain better knowledge of the global flow characteristics during the event. The complete investigation of the case will be presented in the first author’s master degree thesis.

2. Load rejection theory
The load rejection event consists of the sudden unloading of the generator resistive torque which leads to the runner acceleration. As the runner accelerates, the governor forces the guide vanes to close at a given speed to limit the overpressure into the spiral case and the runner acceleration. During closure, the discharge entering the turbine decreases and the net head increases due to the overpressure into the spiral case associated with the water inertia being slowed down. During the whole process, the rebound of pressure waves on the upstream and downstream free surfaces can generate important water hammer phenomena which can induce strong pressure...
fluctuations within the turbine. In the reference case considered in this work, the manometric pressures at the spiral case inlet and at the draft tube outlet were measured during the event, as well as the guide vane opening and the runner speed. By using a 1-D “Elastic Water Column” theory solver, the evolution of the discharge during the event was obtained, allowing to calculate the net head by equation (1):

\[ H_n = \frac{p_1 - p_2}{\rho g} + \left( \frac{1}{2g} \right) Q^2 + (z_1 - z_2), \]

in which subscripts 1 and 2 correspond respectively to the spiral case inlet and the draft tube outlet. With all the hydraulic and geometric variables available during the load rejection, it is possible to obtain the evolution of the performance parameters. As the load rejection progresses, the reduction in discharge is expected to redress the flow by making it more tangential. At the runner’s inlet, the highly circumferential flow coming from the distributor should induce a strong separation on the pressure side of the blade. As the effective inter-blade channel is altered, the downward passing flow should be projected to the cone’s wall by the effect of the centrifugal forces. At the draft tube inlet, the reduction in discharge added to the increased runner speed are expected to leave a residual circumferential velocity in the draft tube. During the event, a strong pressure drop at the runner’s outlet can produce vapor cavities if the absolute pressure reaches the saturated water pressure. Even under steady flow conditions, it was experimentally demonstrated that cavitation attached to solid walls may produce highly unsteady and energetic phenomena associated to significant vibrations and erosion in hydraulic machinery [1, 2].

3. Proposed methodology, boundary conditions and validation
Some of the most challenging issues regarding the simulation of transients in hydraulic turbines concern the boundary conditions. As mentioned before, the geometrical motion and varying hydraulic conditions impose on us to consider varying boundary conditions during simulations. In order to simulate a load rejection event [3] and a discharge surge [4], Cherny used a Distributor-Runner-Draft Tube (D-R-DT) numerical domain with time varying boundary conditions to simulate the full events. Their boundary conditions, evaluated at each timestep of the 3-D simulation by an “Elastic Water Column” theory solver, allowed for the simulation of the flow through the whole turbine incorporating the effect of the water hammer phenomenon. With a similar methodology, Huang [5] successfully simulated the load rejection event with an interesting approach involving moving meshes and evolving boundary conditions (obtained from 1-D calculations). Furthermore, by using a sequence of meshes, Kolsek [6] was able to correctly simulate the unsteady flow and runner’s motion during the shut-down of an axial turbine with very good agreement. In a similar manner and also with good results, Nicolle [7] simulated a Francis turbine’s start-up with a moving mesh algorithm. Once the no-load operating condition was reached, Nicolle was able to identify several of the flow behaviors which will be discussed in this paper.

The problem with these complex numerical methodologies, as pointed out by Casartelli [8], is that they are not easily applicable in industries. Since the main objective of this investigation is to study a particular hydraulic phenomenon occurring inside the runner, an industrially practical hypothesis is now proposed. The suggested assumption is based on a comparison of the characteristic timescales involved with the runner vibrations and with the guide vane’s closure process. Under this comparison, the timescale \( T \) corresponding to vibration is found to be much smaller than the timescale characterizing the variation of hydraulic conditions. This leads to the hypothesis of “frozen” hydraulic conditions under which one can perform unsteady simulations but with fixed boundary conditions corresponding to a certain operation point on the runaway hill chart. It is argued that the unsteady flow patterns and spectral characteristics monitored by probes on the runner should thus fairly correspond to what would be actually measured on an installed pressure probe. For the complete investigation, several “stationary” operating regimes shall be studied in order to obtain the evolution of the flow characteristics
within the turbine. However, before performing unsteady simulations and investigating multiple operating points, a reliable and efficient CFD methodology has to be developed and validated. Since no detailed experimental data are yet available for validation, an alternative validation approach is here proposed. The next section presents a rigorous investigation of the geometry and boundary conditions needed in order to simulate the load rejection event with confidence and at low computational costs. In this paper, all simulations are carried out at condition right before the no-load operating condition, i.e., at \( n/n_0 = 1.6 \) and \( Q/Q_0 = 0.36 \), just at the beginning of the vibration event.

3.1. Stationary investigation on boundary conditions during load rejection

When numerically simulating the flow field in a hydraulic turbine with an “industrial” perspective, the number of components being included is often reduced in order to minimize computational time. As one might expect, this accent to reduce time costs is even more important when simulations are unsteady. In the case of steady operation, this method has often been proven to give reliable results. However, in the case of transient events and off-design conditions in which backflow regions might be expected, it is not clear whether or not this methodology is usable at all, especially if domain boundaries are within or close to those regions. A typical reduced domain and structural mesh that one would use for unsteady simulations are presented in Figure 1.

![Figure 1: Reduced numerical domain and structural mesh of the runner proposed for unsteady simulations.](image)

This reduced numerical domain, which contains 3/24 of the distributor, 2/17 of the runner and draft tube cone, is considered representative and is chosen for its important industrial versatility. As illustrated in Figure 1, a refined mesh of 2.3 million elements by blade passage is proposed within the runner. To obtain the flow characteristics, the Reynolds Averaged Navier-Stokes (RANS) equations are solved with the commercial solver ANSYS CFX \([9, 10]\). The equations fully describing the mean turbulent flow within the turbine, under the Boussinesq assumption, are the continuity and the Reynolds equations presented below:

\[
\nabla \cdot \mathbf{u} = 0 , \tag{2}
\]

\[
\mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \mathbf{G} + (\nu + \nu_t) \nabla^2 \mathbf{u} , \tag{3}
\]

where \( \nu_t \) is the turbulent eddy viscosity and \( \mathbf{G} \) the gravitational force. In the work presented in this paper, \( \nu_t \) is calculated by using the 2-equations \( k - \omega \) Shear Stress Transport (SST) turbulence model. The latter has been chosen for its good versatility to resolve the mean turbulent
flow characteristics in the core and near the wall with reasonable computational costs. In order to fully resolve the turbulent flow into the runner, a good resolution near the walls with an average $y^+$ lower than unity is enforced. A total of 204 nodes are positioned onto the blade’s perimeter and 135 on the height of the blade. The numerical resolution of equations (2) and (3) is done by using a second order differencing scheme.

For comparison purposes, a reference case is proposed which contains 1/24 of the distributor but the full runner and draft tube plus a straight extension as illustrated in Figure 2. This reference case has a more complete geometry that is more costly but that provides a more reliable rotor flow prediction. In this case, the runner-draft tube frame change is ensured by a “Frozen-rotor” interface which allows a certain interaction between components. This simulation - called “Ref (D-R-DT)” - is used in order to compare various methodologies involving two reduced geometries and various boundary conditions. The first proposed reduced numerical domain is the original geometry presented in Figure 1 and here refereed to as “Ori” in Figure 2. The second and last geometry - here named “Ext” in Figure 2 - contains the same components as the original (“Ori”) plus a straight extension at the cone’s outlet.

Figure 2: Retained geometries for the investigation on boundary conditions to simulate the load rejection. From left to right, the reference simulation geometry with full runner and draft tube, the original geometry containing the distributor-runner and the geometry using a straight extension.

In the present investigation, two main flow characteristics are critical. First, as mentioned earlier, the simulation of cavitation requires the correct level of absolute pressure within the numerical domain. Secondly, the accurate prediction of flow separations and vortical motions which are expected to play a significant role are likely to be quite sensitive to the discharge. For this purpose, the “Ref (D-R-DT)” simulation uses a conventional mass flow rate inlet with flow angle and a zero averaged pressure outlet. For the reduced geometries, many sets of boundary conditions allowing for the absolute pressure and flow characteristics to be obtained are being tested. Three approaches are proposed which have been proven to give reliable results in steady state simulations. The first technique consists in imposing the mass flow rate at the inlet and using an averaged pressure outlet. This method is referred to as “$Q - \bar{p}$”. The second methodology uses a mass flow rate outlet, which implies using a total pressure inlet in order for the pressure to become an implicit result of the calculation. In this case, the inlet total pressure is set as the net head $H_n$. The latter method is referred to as “$H_n - Q$”. One last set of boundary conditions is proposed by the code when back flow might be expected at the domain’s outlet. When it is the case, an opening option is proposed since back flow is explicitly allowed within the domain. Furthermore, with this boundary condition, an “Opening Pressure” option is also used since the flow direction is completely implicit in the simulation. Along with the pressure
opening at the outlet, a mass flow rate inlet is imposed in the “$Q - p$ Ope” methodology. All those cases are listed in Table 1 below.

Table 1: List of simulations used in the investigation of the boundary conditions

| Simulation | Geometry          | Inlet                        | Outlet          |
|-----------|-------------------|------------------------------|-----------------|
| Ref (D-R-DT) | DIST-RUN-DT      | Mass flow rate & Flow angle ($\beta$) | Ave. pressure   |
| Ori $Q - \bar{p}$ | DIST-RUN        | Mass flow rate & Flow angle ($\beta$) | Ave. pressure   |
| Ext $Q - \bar{p}$ | DIST-RUN-EXT    | Mass flow rate & Flow angle ($\beta$) | Ave. pressure   |
| Ori $H_n - Q$        | DIST-RUN        | Total pressure & Flow angle ($\beta$) | Massflow        |
| Ext $H_n - Q$        | DIST-RUN-EXT    | Total pressure & Flow angle ($\beta$) | Massflow        |
| Ori $Q - p$ Ope       | DIST-RUN        | Mass flow rate & Flow angle ($\beta$) | Pressure opening|
| Ext $Q - p$ Ope       | DIST-RUN-EXT    | Mass flow rate & Flow angle ($\beta$) | Pressure opening|

DIST: Distributor, RUN: Runner, DT: Draft Tube

The present series of tests involves only stationary simulations but is relevant for determining the correct methodology to be used in order to obtain reliable unsteady simulations. Since the flow unsteadiness is important during load rejection, numerical convergence for the present stationary simulations was not pushed beyond RMS residuals of the order of $10^{-5}$. The circumferential-averaged velocity profiles under the runner for all simulations are shown below in Figure 3.

Figure 3: Circumferentially-averaged velocity profiles under the runner ($z = -0.54D$; purple plane above) for the stationary investigation of boundary conditions during load rejection.

As shown in Figure 3, the choice of boundary conditions greatly affects the resultant flow field at the runner outlet. First, one can notice that when using the “$H_n - Q$” set of conditions, no pumping within the runner is observed since the axial speed component is purely descending. On the other hand, when using an averaged pressure outlet versus a pressure opening, back flow is being pumped to the runner in a very different manner. When using an averaged pressure outlet, no backflow is found in the core region but is rather concentrated in a medium sized band within $0.2 < 2r/D < 0.8$. However, in the reference simulation and when using an opening condition, flow is being pumped back through most of the section, which is expected to considerably modify the flow characteristics within the inter-blade channel, particularly at the
It can also be noted at Figure 3 that the radial velocity distribution differs when using a mass flow rate outlet. The latter suggests, by considering the continuity equation under the assumption of axisymmetric flow, that the axial velocity profile evolution in the $z$ direction is different as no mass flow rate is being pumped by the runner. However, the most striking discrepancy between results is found when examining the circumferential velocity profiles under the runner. As one can note in Figure 3, the circumferential velocity greatly differs when using different boundary conditions and when using the extended geometry. This last observation suggests that the only boundary condition that allows for a reasonably correct simulation of the back flow at the outlet is the opening. One way to explain some of the important dissimilarities with the reference simulation when using a mass flow rate or an averaged pressure outlet is by examining the reduced axial velocity profile at the cone’s outlet, as shown below in Figure 4.

![Velocity profile at the cone's outlet](image)

Figure 4: Circumferentially-averaged reduced axial velocity profile at the cone’s outlet ($z = -1.27D$; purple plane above) for the stationary investigation of boundary conditions during load rejection.

As one can see, since back flow is being pumped into the domain by the runner, the choice and position of the outlet boundary condition greatly affects the flow field. For instance, when using a mass flow rate outlet, discharge seems to be forced downward through the whole outlet, which would explain the absence of positive axial velocity in Figure 3. On the other hand, when using an averaged pressure outlet, the simulation does not allow backflow and artificially imposes a no-through flow condition over most of the outlet’s section as visible in Figure 4 on the “Ori $Q - p$” curve. When adding an extension to the numerical domain, pushing away those boundary conditions, the effects of the proximity of the outlet on the flow field are lowered, allowing the runner to pump more fluid. However, as shown in Figure 3, using an extension does not allow for the correct simulation of the flow field into the runner at all.

We thus conclude at this point that the only outlet’s boundary condition that allows for a reliable simulation within the runner with a reduced numerical domain is a pressure opening. As demonstrated, an averaged pressure or mass flow rate outlet does not yield the correct flow field to be predicted in the runner and cone under such operating conditions. As for the straight extension, its effect on the flow field varies but is more pronounced when the flow field is not correctly simulated. When using an opening boundary condition, no gain is obtained by using the extension. Finally, since the opening condition allows for the correct simulation of the flow within the runner, it appears that the resultant flow behavior in the whole turbine is mostly driven by the runner.
4. Global flow behavior

Using the reference simulation with the draft tube and the retained reduced geometry with the opening boundary condition, the global flow characteristics within the turbine have been investigated. For reminder, the present simulations are done under the instantaneous conditions when vibrations begin, corresponding to a near no-torque condition. First, the flow field within the runner is presented with the resulting 3-D streamlines in Figure 5. To gain more insight on the flow physics, the reduced axial vorticity field is also plotted on three meridian planes within the runner.

One can notice that two main flow phenomena happen simultaneously within the runner during load rejection. As expected, flow separation occurs at the leading edge. The intensity at which flow separates varies along the height of the blade. When separation occurs, the inter-blade channel becomes altered and the remaining flow is directed downward to the draft tube. At the trailing edge, as pointed out in Figures 3 and 4, back flow is being pumped back to the runner during load rejection. The mainly axial returning flow - in stationary frame - then enters the runner which is still rotating, generating a strong flow separation highlighted by positive axial vorticity at the right of Figure 5. As also presented and circled in Figure 5, a region of strong interaction between flow separations occurring at the leading edge and trailing edge exists near the middle of the blade. As a matter of fact, this interaction is quite unstable and shall only be further investigated with unsteady simulations. It is of great interest in the present research to develop a better understanding of the pumped back flow and from where it originates.

To gain insight into this returning flow, the reduced axial velocity and reduced pressure are plotted below on a $r - z$ plane within the runner and draft tube cone in Figure 6. As noticeable in Figures 5 and 6, the vortical motion at the leading edge caused by separation prevents the flow from reaching the trailing edge. As a result, the flow is projected to the band and cone wall by centrifugal forces. This observation suggests that during load rejection, the reduction in discharge and the increased runner speed change the force balance on the flow field as centrifugal forces become predominant. The resulting flow behavior mimics a flow separation within the cone’s core as high speed mass flow is convected near the wall and no fluid is provided within the core. The observed flow field at the upper blade’s outlet is then a region of very low pressure, as highlighted on the right of Figure 6. This region of low pressure in the core is suspected of suctioning fluid from the draft tube and thus creating the returning flow.
One other way to explain this flow behavior is by the phenomenon of viscous entrainment. As the high speed descending flow is concentrated at the band, the low speed flow within the core is entrained by viscous effects. In order to replace the fluid being entrained by the high speed descending flow, flow is being pumped by the runner in the center of the cone. In addition, one can notice a small flow asymmetry in the low pressure field into the cone by the presence of a strong flow separation occurring in the draft tube. As illustrated below in Figure 7, the fluid being pumped back to the runner originates from the separation occurring at the elbow’s outlet.

As expected, the flow at the draft tube’s inlet is highly tangential, which results in a strong flow separation in the draft tube elbow. Even if the resulting flow is most certainly highly unsteady, it appears from the present stationary simulations that the low speed fluid contained into the recirculation at the elbow’s outlet is being pumped back to the runner. Also, as illustrated in Figure 7, the discharge being rejected to the river is concentrated in an upper region between the elbow and the recirculation which blocks an important section of the draft tube.
In order to validate the global flow topology, one last analysis is proposed as experimental tests have been performed under a similar “quasi-stationary” assumption on a different but analogous turbine. An instantaneous sketch of the experimentally observed flow field is being presented below in Figure 8 as well as iso-surfaces of minimal pressure within the runner in our stationary simulation.

Even if hydraulic conditions are not identical between the picture and the simulation, main flow characteristics are still similar. As illustrated above, the flow being pumped by the runner, which generates the flow separation at the trailing edge, creates a low pressure region near the crown. Under absolute conditions, it is suggested that this region would cavitate as pictured under model test experiments to the right in Figure 8. As observed, a part of the vortical structure generated at the trailing edge is entrained with the discharge passing through the band, leading to the stretching of this vortical tube. As perceived experimentally, the stretched vortical tube passing through the inter-blade channel can also cavitate until it reaches the draft tube. Again, under this comparison, it appears that the presented simulations predict the correct flow behavior within the runner. This also suggests that unsteady flow phenomena should be reproduced reasonably well when performing unsteady simulations with the present model. The next section quickly introduces the principal unsteady flow behaviors observed so far within the runner.

### 4.1. Unsteady flow characteristics

In order to perform reliable unsteady simulations of the flow field within the runner, the reduced numerical domain and mesh presented in Figure 1 along with a mass flow rate inlet and a pressure opening outlet with no extension are used. To calculate the local inertia term $\frac{\partial \mathbf{u}}{\partial t}$ added to equation (3), a second order backward Euler scheme is used. For the study of typical unsteady flow behavior within the turbine, a reasonable time discretization is proposed in order to minimize computational costs. The presented simulation uses a discrete timestep corresponding to 1° of runner rotation at synchronous speed. For the complete investigation of the phenomenon, finer time steps shall be proposed. Below, Figure 9 presents the reduced axial vorticity evolution on meridian planes within the runner, showing the flow dynamic behavior.
As illustrated above, the unsteady flow inside the runner is mainly governed by the evolution of flow separations and vortical motions. First, flow separation at the leading edge begins near the crown as visible at time $t^*$. In fact, it can be deduced that the negative vorticity region generated at the leading edge has already convected a certain distance on the meridian plane near the crown when separation at the blade’s mid height happens. At $t^* + \Delta t^*$, the flow evolves differently near the crown than at the middle of the blade. Encircled in black at the top, the vorticity generated by the flow pumping does not interact with the flow separation at leading edge because vorticity remains relatively close to the leading edge. At the blade’s mid height, a strong interaction area between both vortical motions is encircled in black. As discharge is concentrated near the band, the vorticity generated at the leading edge is convected downward and meets the separation at the trailing edge which is rolling-up on the suction side of the blade. The interaction between these two structures could strongly affect the dynamics in the area, more so if cavitation is present. As observable at $t^* + 2\Delta t^*$, flow separation near the crown remains relatively located near the blade. Yet, the interaction between the two structures at the mid blade is still important, which suggests a stronger interaction leading to higher frequency perturbations. However, since statistical convergence has not yet been demonstrated, spectral characteristics and amplitudes of pressure pulsations will only be presented at the conference.

5. Conclusion
In conclusion, the characteristic flow behavior of a Francis runner undergoing load rejection has been investigated near the no-load transition regime. In order to do so, an efficient, reliable and low computational cost methodology has been developed and validated to perform unsteady simulations of transient events. The flow during load rejection has been proven to be highly tangential, resulting in strong flow separations and important centrifugal forces. The resulting flow pattern can be described as a downward tangential flow near the band and draft tube cone’s wall. Within the flow core, a mainly axial back flow is found which seems to be resulting from a state of equilibrium between torque generation and flow pumping reached by the runner in this particular no torque condition. When the back flow reaches the runner, it generates a strong flow separation at the trailing edge as the runner is still rotating. The two resulting vortical structures then interact in a very unsteady way, more importantly at the blade’s mid height. The resulting unsteady flow pattern, added to the physics of cavitation could induce dynamic loads within the runner.

Acknowledgments
The authors would like to thank Alstom Renewable Power Canada, the FRQNT and the NSERC for their financial support of this research project. Computations were performed on the Colosse supercomputer at Université Laval, under the auspices of Calcul Québec and Compute Canada.
Nomenclature

Alphabetical (variables and performance parameters)

- $f_v$ (Hz) Vibration frequency
- $n$ (rev/s) Runner speed
- $Q$ (m$^3$/s) Discharge
- $u$ (m/s) Velocity field
- $U^*$ (m/s) Reference velocity ($U^* = \pi n D$)
- $C$ (m/s) Velocity component in stn. frame

- $T$ (s) Vibration period
- $H_n$ (m) Net head
- $p$ (Pa) Pressure
- $\omega$ (s$^{-1}$) Vorticity field
- $\Omega^*$ (s$^{-1}$) Reference rotation ($\Omega^* = 2\pi n$)
- $D$ (m) Runner’s diameter

Subscripts and superscripts

- 0 Before load rejection
- 1 Spiral case inlet
- 2 Draft tube outlet
- $z$ Axial component
- $r$ Radial component
- $u$ Circumferential component

References

[1] Coutier-Delgosha O, Stutz B, Vabre A and Legoupil S 2007 J. of Fluid Mech. 578 171–222
[2] Escaler X, Farhat M, Eguquisita E and Avellan F 2007 J. Fluids Eng. 129 886–893
[3] Cherny S, Chirkov D, Bannikov D, Lapin V, Skorospelov V, Esikhunova I and Avdyushenko A 2010 3d numerical simulation of transient processes in hydraulic turbines 25th IAHR Symposium on Hydraulic Machinery and Systems (IOP Conf. Series: Earth and Environmental Science no 12)
[4] Chirkov D, Avdyushenko A, Panov L, Bannikov D, Cherny S, Skorospelov V and Pylev I 2012 Cfd simulation of pressure and discharge surge in francis turbine at off-design conditions 26th IAHR Symposium on Hydraulic Machinery and Systems (IOP Conf. Series: Earth and Environmental Science no 15)
[5] Huang W D, Fan H G and Chen N X 2012 Transient simulation of hydropower station with consideration of three-dimensional unsteady flow in turbine 26th IAHR Symposium on Hydraulic Machinery and Systems (IOP Conf. Series: Earth and Environmental Science no 15)
[6] Kolsek T, Duhovnik J and Bergant A 2010 Journal of Hydraulic Research 44 129–137
[7] Nicolle J, Morissette J F and Giroux A M 2012 Transient cfd simulation of a francis turbine startup 26th IAHR Symposium on Hydraulic Machinery and Systems (IOP Conf. Series: Earth and Environmental Science no 15)
[8] Casartelli E, Widmer C, Ledergerber N and Sallabarger M 2010 Simplified cfd model for flow field investigations at no-load conditions Hydro 2010, (Lisbon, Portugal)
[9] ANSYS 2012 ANSYS CFX-Solver Theory Guide release 14.5 ed (ANSYS, Inc.)
[10] ANSYS 2012 ANSYS CFX-Solver Modeling Guide release 14.5 ed (ANSYS, Inc.)