Numerical investigation of draft tube flows using a hybrid RANS-LES turbulence model and a low-dissipation scheme

Y Jiang, Y Liu, S Nadarajah
Computational Aerodynamics group, Department of Mechanical Engineering, McGill University, Montreal, H3A 0G4, CA
E-mail: yao.jiang@mail.mcgill.ca, liu.yang4@mail.mcgill.ca, siva.nadarajah@mcgill.ca

Abstract. Hydro-turbines are operated at loads above or below the best efficiency point (over- or part-load), where turbulent flow structures are formed in the draft tube. The goal of the project is to investigate complex turbulent structures present in the draft tube through a thorough numerical investigation. In the context of numerical simulations, a focus is made on the turbulence model and the numerical scheme. Transient simulations with the k-ω SST and SAS (Scale Adaptive Simulation)-SST turbulence models are applied with a novel eddy-preserving limiter scheme for draft tube flows. The application of both the SAS-SST turbulence model and the eddy-preserving limiter scheme allows for the resolution of more turbulent structures. The impact of the inlet modification is also studied. Numerical results are validated through a comparison to experimental data for the BulbT test case.

1. Introduction
Hydro-turbines are designed to operate within an operating range of conditions that provide the maximum power output. At such conditions, the flow passes through the guide vanes at a specific flow rate and enters the runner at a radial inflow direction where the kinetic energy is extracted and leaves the runner in an axial direction. In an attempt to maintain a constant power throughout the electrical grid system, various power plants must be made to operate either below or above their best efficiency point (BEP) range. Thus a hydro-turbine operator must run the power plant at loads above or below the BEP (over- or part-load), where strong swirling flow structures are formed below the runner outlet in addition to massive separations within the draft tube. At part load conditions the energy is not fully utilized by the turbine and high residual swirl flow encounters the lower momentum fluid in the inlet conical zone to form highly turbulent structures along with synchronous pressure oscillations within the draft tube, which leads to a fluctuation of the complete pressure level at the draft tube inlet. The pressure fluctuations will not only cause variations in the power output, but also endangers hydro-turbine components, especially if the excitation frequency corresponds to one of the eigenfrequencies of the components. As a consequence, a restriction on the range of operation might be necessary; limiting the required power output. Hence, it is necessary to investigate complex turbulent flows present in draft tubes using both experimental and numerical approaches.
Computational Fluid Dynamics (CFD) provides one avenue for investigating the complex flow field within hydro-turbine draft tubes. One main challenge to realize the objective is the large range of turbulent structures to solve. In order to accurately capture the flow features, efforts have been made by employing more advanced turbulence models [1, 2, 3], improving the grid quality [4] and the numerical schemes [5] during the past years.

The BulbT project was initiated in 2011 within the framework of an international research partnership at the Hydraulic Machines Laboratory (LAMH) of Laval University [6]. The goal of the BulbT project was to experimentally and numerically characterize the flow field in a low head model-scale bulb turbine. To perform measurements, a model-scale of the bulb turbine (BulbT) was constructed and installed on the LAMH test bench. LDV and PIV measurements [7, 8, 9] were available in different plan positions. Recent trends [4, 10] have demonstrated the need to adopt LES-based turbulence models; however, in this work we reaffirm the role of the numerical scheme and show the importance of employing low-dissipation schemes to better resolve the flow field.

2. Methodology

The simulation was carried out with SYN3D, which is an in-house compressible flow solver in the Computational Aerodynamics Group at McGill University. Several modifications on the equation of state, numerical scheme and turbulence model are made in order to carry out simulations for draft tube flows.

2.1. Stiffened equation of state

In the present simulation, the stiffened gas law is employed as the equation of state (EOS), which provides a reasonable approximation for liquids under high-pressure conditions, which is given by:

\[ p = (\gamma - 1) \rho e - \gamma p_\infty, \]  

(1)

where \( p \), \( \rho \), and \( e \) are respectively the pressure, density and specific internal energy. The parameters \( \gamma \) and \( p_\infty \) are two empirically determined constants used to define liquids with different properties. The speed of sound is then obtained by:

\[ c = \sqrt{\gamma \frac{p + p_\infty}{\rho}}. \]  

(2)

In the present paper, both \( \gamma \) and \( p_\infty \) are chosen to be 7.15 and \( 3.1 \times 10^8 \) Pa respectively for water simulation. A low-speed preconditioner [11] is employed to improve the convergence rate.
2.2. SAS-SST Turbulence Model

Introduced by Menter et al. [12, 13], the SAS falls under the class of hybrid RANS/LES schemes. The model introduces a new length scale, the von Karman scale $L_{vk}$, which is a function of the second and first order spatial derivatives of the velocity field,

$$L_{vk} = \kappa S / |\nabla^2 U|,$$

(3)

where, $\kappa$ the von Karman constant and $S = \sqrt{S_{ij}S_{ij}}$ the strain rate magnitude. With the introduction of $L_{vk}$ as another length scale, the SAS approach can be applied to existing RANS models [13]. Introduced by Menter et al. [14] based on the $k$-$\omega$ SST, the SAS-SST model contains an additional source term $Q_{SAS}$ involving $L_{vk}$ in the transport equation of $\omega$. This additional term dominates in turbulent flow regions and reduces the turbulent viscosity in RANS models [13]; thus, enabling an LES-like resolution of unsteady structures. The SAS approach can resolve turbulent structures down to the mesh scale and switch automatically to operate in RANS mode in steady regions or when the mesh resolution or time step is insufficient. The natural switch between RANS and LES allows SAS to be less dependent on the mesh and time resolution, which appears to be an advantage over the classical DES family of techniques. The application of SAS-SST to hydro-turbine draft tube [1, 2] have demonstrated comparable results to LES.

In the SAS-SST model, the eddy viscosity $\mu_t$ is defined in the same way as the $k$-$\omega$ SST does:

$$\mu_t = \frac{a_1 \kappa}{\max(\sigma_1, \omega, SF_2)},$$

(4)

where $a_1$ is a constant and $S = \sqrt{S_{ij}S_{ij}}$ is the strain rate magnitude. The variables $k$ and $\omega$ are solved from:

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} [(\nu + \sigma_k \nu_t) \frac{\partial k}{\partial x_j}],$$

(5)

$$\frac{\partial \omega}{\partial t} + u_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} [(\nu + \sigma_\omega \nu_t) \frac{\partial \omega}{\partial x_j}] + C + Q_{SAS},$$

(6)

where

$$C = 2(1 - F_1) \sigma_2 \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i},$$

(7)

$$Q_{SAS} = \max[\rho \zeta_2 S^2 \frac{k}{L_{vk}} \frac{L_{vk}}{(L_{vk})^2} - C \frac{2 \rho k}{\sigma_\Phi} \max(\frac{\nabla \omega}{\omega^2}, \frac{|\nabla k|^2}{k^2}), 0],$$

(8)

where $\zeta_2$ and $\sigma_\Phi$ are constants, and $L$ is the length scale of the modeled turbulence:

$$L = \sqrt{k/(c_\mu^{0.25} \omega)}; \quad c_\mu = 0.09.$$  

(9)

The additional source term in the dissipation equation of $\omega$ depends on the ratio between the turbulent length scale $L$ and the von Karman length scale $L_{vk}$. The term allows for an increase of $k$ dissipation in regions where $L_{vk}$ is relatively small, which leads to a decrease of modeled turbulent kinetic energy (TKE) and a subsequent reduction in the eddy viscosity, $\mu_t$, such that the RANS equation exhibits LES type features. Figure 3 and figure 4 show the instantaneous velocity field for the periodic hill test case [15] captured by the $k$-$\omega$ SST and SAS-SST models with the same mesh and boundary condition and illustrates the ability of SAS-SST to capture detailed flow structures. Table 1 shows the separation and reattachment points for the periodic hill case compared to a refined LES result in reference [15]. It is shown that the URANS with the $k$-$\omega$ SST model fails to capture the correct length of the separation bubble with an overestimation of 66%, while SAS-SST provides excellent estimation of the separation bubble even on a coarse grid.
Figure 3. Instantaneous stream wise velocity contour using $k - \omega$ SST model.

Figure 4. Instantaneous stream wise velocity contour using SAS-SST model.

### Table 1. Separation and reattachment points for the periodic hill case.

| Case    | Mesh     | $(x/h)_{sep}$ | $(x/h)_{reat}$ | $L_{bubble}$ |
|---------|----------|---------------|----------------|-------------|
| LES [15]| 13.1M    | 0.19          | 4.69           | 4.5         |
| URANS   | 1.6M     | 0.24          | 7.69           | 7.45        |
| SAS     | 1.6M     | 0.16          | 4.61           | 4.45        |

#### 2.3. Eddy-Preserving Limiter Scheme

Turbulent flow is characterized by the presence of vortices. The traditional second-order finite volume schemes usually provide too large artificial viscosity, thus preventing the turbulence model from well capturing the vortical structures. The eddy-preserving limiter scheme [16] (EDDY) is proposed to control artificial viscosity specifically in regions of strong in the vortical flows. The scheme was derived from the MUSCL scheme by employing an eddy-preserving limiter within the MUSCL flux reconstruction. The state variables for the MUSCL scheme are reconstructed at the interfaces using:

$$u_i^{+} = u_i + \frac{\Phi_i}{4}[(1 - \beta)\Delta u_i + (1 + \beta)\Delta c_{ij}], \quad (10)$$

$$u_j^{+} = u_j + \frac{\Phi_j}{4}[(1 - \beta)\Delta u_j + (1 + \beta)\Delta c_{ij}], \quad (11)$$

where $\Phi_i$ and $\Phi_j$ are slope limiters, $\beta$ is a parameter to balance the use of the upwind and central terms, while $\Delta u_i$ and $\Delta c_{ij}$ are respectively the upwind and central increments.

The concept of the eddy-preserving limiter is to prevent the slope limiter to be activated (or to use a less dissipative limiter) during the reconstruction of the velocity component along the tangential direction of a vortex. To identify vortical flow regions, the enhanced swirling strength criterion of Chakraborty et al. [17] is employed, whereby in a vortical region, the velocity gradient tensor $\Delta v$ possesses a conjugate pair of complex eigenvalues,

$$\sigma(\Delta v) = \lambda_r, \lambda_{cr} + i\lambda_{ci}, \lambda_{cr} - i\lambda_{ci}, |\lambda_{ci}| > \epsilon, \quad (12)$$

where $\epsilon$ is a small positive real number. A local measure of the compactness of the vortical motion is added to further limit the vortical regions to areas where the following condition is satisfied,

$$-\zeta \leq \frac{\lambda_{cr}}{\lambda_{ci}} \leq \delta, \quad (13)$$
where $\zeta$ and $\delta$ are positive thresholds for verifying the compactness of the vortex. The velocity gradient tensor, $\nabla v$ can be decomposed into the following form,

$$\nabla v = \begin{bmatrix} \hat{u}_r & \hat{u}_{cr} & \hat{u}_{ci} \end{bmatrix} \begin{bmatrix} \lambda_r & 0 & 0 \\ 0 & \lambda_{cr} & \lambda_{ci} \left| \frac{u_{cr}}{u_{ci}} \right| \\ 0 & -\lambda_{ci} \left| \frac{u_{ci}}{u_{cr}} \right| & \lambda_{cr} \end{bmatrix} \begin{bmatrix} \hat{u}_r & \hat{u}_{cr} & \hat{u}_{ci} \end{bmatrix}^{-1},$$

where $\hat{u}_r$, $\hat{u}_{cr}$, and $\hat{u}_{ci}$ are normalized eigenvectors of $\nabla v$.

The mapping and transformation matrix from the original Cartesian system $S_0$ to the local vortex system $S_{\omega}$ spanned by $\hat{u}_r$, $\hat{u}_{cr}$, and $\hat{u}_{ci}$, are given by:

$$[\hat{M}] : S_0 \mapsto S_{\omega} \quad [\hat{M}] = [\hat{u}_r \quad \hat{u}_{cr} \quad \hat{u}_{ci}]^{-1},$$

The algorithm for the eddy-preserving limiting procedure is described below:

1. Calculate eigenvalues of the velocity gradient tensor $\Delta v$. If the velocity gradient tensor has only real eigenvalues, then exit and employ the conventional van Albada limiter.
2. Verify the compactness of the vortical motion using Eqn. 13. If the flow lacks vortical compactness, then exit and employ the conventional van Albada limiter.
3. Calculate eigenvectors of the velocity gradient tensor $\Delta v$, and define the transformation matrix $[\hat{M}]$.
4. Transform velocity components into the vortex system $S_{\omega}$.
5. In the axial direction, reconstruct variables using the conventional van Albada limiter with $\beta = 1/3$.
6. In the swirl plane, reconstruct variables using a higher $\beta$ which leads to less artificial dissipation and inactivate the van Albada limiter by setting $\Phi_i = \Phi_j = 1$.
7. Transform interpolated velocity components back to the original system $S_0$ and evaluate the fluxes:

$$f = f_c + f_v + f_d,$$

where $f_c$, $f_v$ and $f_d$ are convective, viscous and artificial dissipation fluxes.

3. Numerical Setup

When applied to the SAS-SST turbulence model, the artificial dissipation of the EDDY scheme should be reduced to a lower level to ensure the capture of small turbulence structures, while there should be sufficient artificial dissipation to guarantee robustness of the scheme. The artificial dissipation term is tuned for the current SAS-SST turbulence model using the Decaying Isotropic Turbulence(DIT) test case [18].

![Figure 5. Energy spectrum for DIT case compared with experimental data [18].](image-url)
The computational domain is \([-\pi, \pi]\) with 64 isotropic cells in each direction. The initial condition is a divergence-free field adjusted by the measured spectrum at non-dimensional time \(T_{\text{non-dim}} = 42\). The energy spectrum at \(T_{\text{non-dim}} = 98\) is shown in figure 5. In the comparison, SAS-EDDY uses a standard value of artificial viscosity for RANS, while SAS-EDDY-reduced-VIS and SAS-MUSCL-reduced-VIS use a reduced value of artificial viscosity. It is shown that the artificial viscosity plays an important role. The SAS simulation using the MUSCL scheme, even with an reduced artificial viscosity, is too dissipative in the high wave-number region. SAS-EDDY-reduced-VIS has the ability to resolve the high and middle wave-number structures and the resolved spectrum is comparable to the LES model.

3.1. Geometry and Grid

The computational domain is illustrated in figure 6. The runner blades are not included while a part of the runner hub is included.

![Figure 6. Computational domain.](image)

The grid is composed of 14 million cells divided into 512 blocks for the multi-block structured solver SYN3D. The first layer cells satisfy \(y^+ \approx 1\), since both \(k-\omega\) SST and SAS-SST turbulence models are designed to resolve up to the wall. As shown in figure 7, the central part connecting the runner cone and the outlet is refined in order to better capture the swirling feature in the region.

![Figure 7. Mesh topology around the runner hub.](image)

3.2. Boundary condition and inlet velocity correction

The operating point OP5 is studied, whose position in the efficiency hill chart is shown in figure 8. Among all the operating points in the studied zone, OP5 has the largest mass flow rate as shown in figure 9, circumferential velocity and the most feature rich regions such as high velocity gradients at the border of the backflow zone.

The original inlet profiles for the draft tube simulations are extracted from the time-averaged URANS simulation of the whole runner-draft tube system performed by Hydro-Quebec Research Institute (IREQ), using a \(k-\omega\) SST turbulence model and the commercial flow solver ANSYS CFX. However, for OP5, the inlet profiles from the simulation deviates in the vicinity of the runner hub when compared against the experimental data. In order to provide an improved inlet profile we employ a technique based on a corrected one-dimensional profile. A modified inlet profile is obtained by matching the inlet profile with experimental data in the measured zone while keeping the original near-wall value. Since the modification of the axial velocity will change the mass flow rate entering the draft tube only the circumferential and radial velocities are altered. Figure 10 and figure 11 show the influence of the inlet correction. In figure 10, it is shown that noticeable differences exist between the CFD data from IREQ and the experimental data for both the circumferential velocity \(C_t\) and the radial velocity \(C_r\). Figure 10 shows that
after the modification, the CFD data recover the experimental data for both $C_t$ and $C_r$ on the measured plan 4A. The turbulent variables $k$ and $\omega$ are read directly from IREQ’s inlet data.

Since the IREQ simulations were based on URANS; we propose a second modification to the inlet through the addition of artificial fluctuations to construct turbulent velocity signals for the SAS turbulence model. The method employed [10] is to simply add a random noise to the velocity variables:

$$u_i(t) = \bar{u}_i + r_i \sqrt{\frac{2}{3}k},$$

where $\bar{u}_i$ is the averaged velocity and $r_i$ is an uncorrelated white-noise signal with a normal distribution (zero mean and unit variance). The method retains the original averaged velocity and produce an inflow turbulent kinetic energy $k$.

4. Numerical Results

Two steady simulations with $k-\omega$ SST model are carried out using the MUSCL and EDDY schemes to study the impact of the low-dissipation scheme on the simulation. The same inlet condition using the corrected velocity profiles is used for both RANS simulations. Five unsteady runs with the URANS $k-\omega$ SST and SAS-SST models are carried out using the MUSCL and EDDY schemes. The simulations are listed in table 2. URANS, SAS2 and SAS3 use the same inlet condition with corrected circumferential and radial velocities, while SAS1 uses the original IREQ data as the inlet condition. SAS4 obtains the inlet condition for velocity by adding the
artificial fluctuation to the corrected inlet profiles. All the steady and unsteady simulations are initialized by a steady simulation with the corrected inlet profiles.

### Table 2. Case summary.

| Simulation | Turbulence model | Scheme   | Inlet Time step |
|------------|------------------|----------|-----------------|
| RANS1      | $k - \omega$ SST | MUSCL    | IREQ+$C_t, C_r$ correction $-$ |
| RANS2      | $k - \omega$ SST | EDDY     | IREQ+$C_t, C_r$ correction $-$ |
| URANS      | $k - \omega$ SST | EDDY     | IREQ+$C_t, C_r$ correction $5^\circ$ |
| SAS1       | SAS-SST          | MUSCL    | IREQ            $2^\circ$ |
| SAS2       | SAS-SST          | MUSCL    | IREQ+$C_t, C_r$ correction $1.25^\circ$ |
| SAS3       | SAS-SST          | EDDY     | IREQ+$C_t, C_r$ correction $1.25^\circ$ |
| SAS4       | SAS-SST          | EDDY     | IREQ+$C_t, C_r$ correction+fluctuation $1.25^\circ$ |

4.1. Velocity and TKE profiles at plan 4BY0

The location of plan 4B is shown in figure 13, which is close to the runner hub and slices through a backflow region. Experimental data are available on the horizontal line $y = 0$. Figure 12 shows the comparison of the velocity profiles for RANS simulations. It shows clearly that the low-dissipation feature helps the EDDY scheme capture better the swirling flow after the runner hub. RANS1 with the MUSCL provides for a wrong estimation for the direction of the circumferential velocity in the backflow region, while RANS2 with the EDDY scheme captures well the flow structure in the backflow region. Figure 15 and figure 16 show the comparison of the velocity profiles for unsteady simulations. It could be noticed that in the central region SAS1 with the MUSCL scheme and the original inlet data fails to predict the swirling flow feature and the backflow, and SAS2 with the MUSCL scheme and the corrected inlet profiles also wrongly predict the circumferential velocity direction of swirling flow, mainly due to the large artificial dissipation, while URANS with the EDDY scheme slightly underestimates the magnitude of the circumferential velocity $C_y$ in the central region close to the runner hub. It is shown that both SAS3 and SAS4 show great agreement with the experimental data for the circumferential velocity profile. Figure 16 shows that SAS3 and SAS4 have a larger overestimation of the backflow velocity after the runner hub than URANS does.

![Figure 12. Zoomed in velocity profiles comparison at 4BY0 for RANS simulations.](image-url)
region. \textit{URANS} has an overestimation of TKE in the central region after the runner hub and an underestimation in the other regions. The simulation \textit{SAS2} and \textit{SAS3} without inlet velocity fluctuation underestimate the TKE in the whole region since the SAS-SST model provides for a low level of eddy viscosity. With the inlet velocity fluctuation, the simulation \textit{SAS4} is able to show two TKE peaks near the edge of the runner hub.

Figure 13. Location of plan 4B.

Figure 14. Location of plan 5A.

Figure 15. Velocity profiles comparison at 4BY0 for unsteady simulations.

Figure 16. Zoomed in velocity profiles comparison at 4BY0 for unsteady simulations.

Figure 17. TKE profiles comparison at 4BY0 for unsteady simulations.

Figure 18. Zoomed in TKE profiles comparison at 4BY0 for unsteady simulations.
4.2. Velocity and TKE profiles at plan 5AX0
Another plan where experimental data are available is the plan 5A, whose location is shown in figure 14. The plan is situated in the transition part of the draft tube and LDV data are available in the central region. Figure 19 shows the comparison of the axial velocity profile on the line $x = 0$. It is shown that all the simulations overestimate the axial velocity on 5AX0. The simulation $SAS_1$ has the lowest maximum value among the simulations. $SAS_2$, $SAS_3$ and $SAS_4$ with the corrected inlet profiles predict a larger velocity deficit in the central region than $URANS$ and $SAS_1$, which is closer to the experimental data. The TKE profile on 5AX0 is shown in figure 20. $URANS$ has a large overestimation of TKE in the central region while SAS simulations provide for a low level of the TKE in the central region. The plan 5A is further from the inlet such that the fluctuation only has small influences on the TKE profile.

Figure 19. Axial velocity profiles comparison at 5AX0.

Figure 20. TKE profiles comparison at 5AX0.

5. Conclusion
A low-dissipation scheme was applied for turbulent flows in hydraulic turbines. The methodology is based on employing the novel eddy-preserving limiter within a MUSCL reconstruction. The EDDY scheme was adapted to the $URANS$ $k - \omega$ SST model and the SAS-SST model to simulate unsteady flows in a draft tube. Numerical simulations were performed for the BulbT case. The inlet velocity correction and the inlet artificial fluctuation were studied and applied to the simulation. Preliminary numerical results show that the EDDY scheme demonstrates strengths over the MUSCL scheme in the unsteady draft tube flow simulation with its low-dissipation feature. The SAS-SST model is able to better capture the circumferential velocity profile than the URANS does. The SAS-SST model shows its advantage over the URANS by showing its scale-adaptive ability in the turbulent zone to capture more TKE instead of modeling it. Further simulation and analysis will be carried out to compare the influence of the low-dissipation numerical scheme and turbulence model on the simulation through a draft tube.

Acknowledgements
The authors would like to thank the participants on the Consortium on Hydraulic Machines for their support and contribution to this research project: Andritz Hydro Canada Inc., GE Renewable Energy, Hydro-Quebec, Laval University, NRCan, Voith Hydro Inc. Our gratitude goes as well to the Canadian Natural Sciences and Engineering Research Council who participated in the funding for this research and to Compute Canada who provided the necessary computing allocations resources.
References

[1] Jošt D and Lipej A 2011 Strojniški vestnik-J. Mechanical Engineering 57 445456
[2] Krappel T, Ruprecht A, Riedelbauch S, Jester-Zuerker R and Jung A 2014 Investigation of Francis turbine part load instabilities using flow simulations with a hybrid RANS-LES turbulence model IOP Conf. Series: Earth and Environmental Science vol 22 (Philadelphia: IOP Publishing) p 032001
[3] Foroutan H and Yavuzkurt S 2014 J. Appl. Mechanics 81 061010
[4] Magnan R, Cupillard S, Gauthier G, Giroux A, Page M and Deschênes C 2014 Challenges in assessing the grid sensitivity of hydro-turbine cfd simulations IOP Conf. Series: Earth and Environmental Science vol 22 (Philadelphia: IOP Publishing) p 022010
[5] Yang L and Nadarajah S 2016 A low-dissipation monotonicity-preserving scheme for turbulent flows in hydraulic turbines IOP Conf. Series: Earth and Environmental Science vol 49 (Philadelphia: IOP Publishing) p 062010
[6] Deschênes C, Ciocan G D, Henau V D, Flemming F, Huang J, Koller M, Naime F A, Page M, Qian R and Vu T 2010 IOP Conf. Series: Earth and Environmental Science 12 012043
[7] Lemay S 2014 Étude expérimentale de l’écoulement dans le canal inter-aube d’une turbine de type bulbe Master’s thesis Université Laval
[8] Longchamp Q 2014 Analyse expérimentale et numérique de l’écoulement dans le canal d’entrée d’un modèle de turbine bulbe Master’s thesis Université Laval
[9] Vuillemard J 2015 Étude expérimentale de l’écoulement en entrée d’aspirateur d’une turbine bulbe Master’s thesis Université Laval
[10] Taheri A 2015 Detached Eddy Simulation of Unsteady Turbulent Flows in the Draft Tube of a Bulb Turbine Ph.D. thesis Université Laval
[11] Turkel E 1987 J. computational phys. 72 277298
[12] Menter F and Egorov Y 2009 Development and application of sst-sas turbulence model in the desider project Atomic Molecular and Optical Physics (Notes on Numerical Fluid Mechanics and Multidisciplinary Design vol 97) ed Dunning F B and Hulet R (Berlin: Springe)
[13] Menter F and Egorov Y 2010 Flow, Turbulence and Combustion 85 113138
[14] Menter F and Egorov Y 2005 A scale adaptive simulation model using two-equation models 43rd AIAA Aerospace Sciences Meeting and Exhibit p 1095
[15] Breuer M, Peller N, Rapp C and Manhart M 2009 Computers & Fluids 38 433457
[16] Mohamed K, Nadarajah S and Paraschivoiu M 2012 AIAA journal 50 429446
[17] Chakraborty P, Balachandar S and Adrian R J 2005 J. fluid mechanics 535 189214
[18] Comte-Bellot G and Corrsin S 1971 J. Fluid Mechanics 48 273337
[19] Vuillemard J, Aeschlimann V, Fraser R, Lemay S and Deschênes C 2014 Experimental investigation of the draft tube inlet flow of a bulb turbine IOP Conf. Series: Earth and Environmental Science vol 22 (Philadelphia: IOP Publishing) p 032010