# Detached-Eddy Simulations of the Flow Structures Underneath a Francis Turbine Runner

Carl-Anthony Beaubien<sup>1</sup>, Guy Dumas<sup>1</sup>, and Guillaume Boutet-Blais<sup>2</sup>

<sup>1</sup> Laboratoire de Mécanique des Fluides Numérique Department of Mechanical Engineering, Laval University, Québec, QC, GIV 0A6, Canada

> <sup>2</sup> Alstom Power & Transport Canada Inc 1350 chemin St-Roch, Sorel-Tracy, QC, J3R 5P9, Canada

> > Email: carl-anthony.beaubien.1@ulaval.ca

# ABSTRACT

The effect of the vortical flow structures ejected by a Francis turbine runner on the flow downstream, in the draft tube, is evaluated in this investigation. Calculations are performed with the commercial code *ANSYS CFX 13.0* and with the turbulence modeling approach DES-SST as proposed by Menter and Kuntz [8].

First, the grid and time step requirements are assessed on a simplified geometry, which includes the draft tube cone and a straight extension. It is shown that a very fine mesh and time step resolution are necessary to capture adequately the flow structures without their premature diffusion underneath the inlet plane, even if the modeled turbulence at the inlet is neglected.

Then, two draft tube flow simulations are compared. The first one includes the unsteady flow structures ejected by the runner and the second one has steady circumferentially averaged velocity profiles imposed at the inlet plane. At the operating point investigated, the flow topology and the performance of the diffuser are found to be only slightly affected by the coherent flow structures expelled by the turbine runner.

# **1** INTRODUCTION

The flow in the draft tube of hydraulic turbines is particularly complex and can present a significant challenge for Computational Fluid Dynamics (CFD). For turbine refurbishment, the existing draft tube may not be optimal, leading to flow separation and significant efficiency losses. In such cases, performance predictions may be unreliable. Furthermore, the adaptation of a new runner design to the already existing draft tube can be a difficult task if draft tube predictions provided by CFD are inaccurate. Main components of the turbine are shown on Figure 1.

One suspects that some form of vortex breakdown and/or flow



Figure 1: Components of a Francis hydraulic turbine.

separation may occur in the adverse pressure gradient internal flow of the draft tube. Such features are particularly difficult to predict with turbulence modeling methods which are used to solve the Reynolds Averaged Navier-Stokes (RANS) equations. Indeed, it is widely known that these models produce much less reliable results in flows where there is strong streamlines curvature, flow rotation or boundary layer separation [10] [11].

Therefore, Detached-Eddy Simulation (DES) is tested in this work, in order to make an attempt at improving the prediction of those phenomena in the draft tube. However, this turbulence modeling approach, which implies simulating large turbulent eddies outside the boundary layers, requires inlet boundary conditions which are consistent with the method. Large unsteady vortical structures ejected by the runner could be needed at the draft tube computational domain inlet plane situated below the turbine runner. It is not the case in RANS simulations where a steady-state solution is produced. The velocity field at the inlet plane is then time averaged. Gagnon et al. [3] recently performed experimental measurements of the flow underneath a propeller turbine. These authors attribute the large scale unsteady fluctuations in the velocity fields and associated vortical flow structures to two distinct phenomena. The first one is the velocity gradient between the runner blades and the second is the wake behind the trailing edge created by the boundary layers leaving the blades.

In this work, the effect of inlet vortical flow structures on draft tube performance is assessed.

### 2 METHODOLOGY

Commercial software *ANSYS CFX 13.0* is used to solve the Navier-Stokes equations. For an incompressible flow with uniform viscosity, they take the following form:

$$\frac{\partial u_i}{\partial x_i} = 0,\tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + v \frac{\partial^2 u_i}{\partial x_j \partial x_j},$$
(2)

where  $u_i$  is the velocity vector, p the static pressure, v the kinematic viscosity, and  $\rho$  the density.

#### 2.1 Turbulence Modeling

The convective term  $u_j \frac{\partial u_i}{\partial x_j}$  in the Navier-Stokes equations is responsible for turbulence which is characterized by flow structures of various scales. Unfortunately, at the Reynolds number of the studied flow ( $\approx 2.5 \times 10^6$  based on the inlet diameter of the draft tube), it is too costly to resolve the smallest turbulent flow structures. Therefore, a turbulence modeling approach needs to be used.

In the present investigation, the turbulence modeling approach employed is DES-SST, as proposed by Menter and Kuntz [8]. Near the wall, in the boundary layers, a RANS approach is used. The Reynolds decomposition is therefore applied on the Navier-Stokes equations, which gives :

$$\frac{\partial U_i}{\partial x_i} = 0, \tag{3}$$

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mathbf{v} \frac{\partial U_i}{\partial x_j} + \overline{u'_j u'_i} \right), \quad (4)$$

where  $U_i$  is the ensemble averaged mean velocity vector,  $u'_i$  is the fluctuating part of the fluid velocity vector ( $u_i = U_i + u'_i$ ) and P is the mean pressure. The term  $\overline{u'_i u'_i}$  is the Reynolds

stress tensor which represents the time-averaged rate of momentum transfer due to turbulence.

Outside the boundary layers, a Large-Eddy Simulation (LES) approach is rather employed which enables the simulation of large scale turbulent structures. In this flow region, a spatial filter is imposed on the Navier-Stokes equations, which leads to :

$$\frac{\partial \tilde{U}_i}{\partial x_i} = 0,\tag{5}$$

$$\frac{\partial \tilde{U}_i}{\partial t} + \tilde{U}_j \frac{\partial \tilde{U}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \tilde{P}}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial \tilde{U}_i}{\partial x_j} + \tau_{ij}^{sgs} \right), \quad (6)$$

where  $\tilde{U}_i$  is the filtered velocity,  $\tilde{P}$  the filtered static pressure and  $\tau_{ij}^{sgs}$  the subgrid-scale Reynolds stress tensor. The latter represents the rate of momentum transfer generated by turbulent motions whose size is inferior to the size of the spatial filter.

If the length scale of the largest turbulent structures  $l_t$  is greater than the size of the grid cells  $\Delta$ , they are resolved and consequently, the LES approach is used. In the opposite situation, where  $l_t < \Delta$ , the RANS approach is used.

The similarity between the RANS and LES equations facilitates the transition between the two turbulence modeling methods. Both the Reynolds stress tensor and the subgridscale Reynolds stress tensor are computed through Menter's SST model, which blends the two-equation models k- $\varepsilon$  and k- $\omega$  [7]. To only model subgrid-scale turbulence in the LES region, the modeled turbulence length scale is limited through the rate of dissipation of the turbulent energy  $\varepsilon$  [9]. Indeed,

$$\boldsymbol{\varepsilon} = \boldsymbol{\beta}^* \, k \, \boldsymbol{\omega} \, F_{DES}, \tag{7}$$

$$F_{DES} = \max\left(\frac{l_t}{C_{DES}\Delta}(1 - F_{SST}), 1\right), \qquad (8)$$

$$l_t = \frac{\sqrt{k}}{\beta^* \omega},\tag{9}$$

where  $\beta^*$  is a constant of the SST model ( $\beta^* = 0.09$ ), *k* is the modeled turbulent kinetic energy,  $\omega$  is the modeled turbulence specific dissipation rate and  $C_{DES}$  is a constant equal to 0.61.  $F_{SST}$  can be equal to 0,  $F_1$ , or  $F_2$ .  $F_1$  and  $F_2$  refer to the SST blending functions described in [7]. In our case,  $F_{SST} = F_1$ . This is used to insure that the RANS approach is employed in the boundary layers, even if  $l_t$  is slightly larger than the grid size. This is to protect the boundary layers against the "Grid-Induced Separation" phenomenon discussed by Spalart [11]. B.-Vincent [1] showed that for the kind of flow considered in this study, DES formulations protecting the boundary layer produce far better results.

# 2.2 Numerics

The solver uses an element based finite-volume method [4]. The pressure and diffusive terms are discretized with finiteelement shape functions which are linear in term of parametric coordinates [4].

In the LES region of the domain, the convective term is also discretized with shape functions. Since a linear interpolation is done in the element, this method corresponds to a Central Difference Scheme (CDS). However, in the boundary layers, where a RANS turbulence modeling approach is used, *ANSYS CFX's* "High Resolution" scheme is preferred. This scheme corresponds to a first order upwind differencing with a second order correction that varies in the domain, in order to be as close as possible to a formal second-order-accurate scheme but without introducing non-physical oscillations.

The transient term is discretized using the second order backward Euler scheme.

Finally, the solver is a coupled one in which conservation of mass is treated in the same way as conservation of momentum.

## 2.3 Boundary Conditions

A no-slip wall condition is applied to all solid boundaries of the domain. At the outlet, an average reference pressure is set. At the inlet plane, velocity profiles and turbulence characteristics are imposed. They were obtained by a RANS simulation of one guide vane and one runner blade passage. The obtained velocity field was then copied over all the inlet plane circumference and was put in rotation at the runner rotation speed  $\Omega$ . This strategy is illustrated on Figure 2. For some draft tube simulations where an axisymmetric condition was desired, the velocity fields were circumferentially averaged instead.

As for the inlet modeled turbulence, it was chosen to be ne-

glected outside the boundary layers, in the LES region. Indeed, since the object of this work is to assess the effect of the large scale flow structures ejected by the runner on the draft tube behavior, it is wanted to minimize their diffusion, even if it may be in a exaggerated fashion. Therefore, if their effect on the flow downstream is not perceptible, too important artificial diffusion from the turbulence model could not be held responsible. In order to identify the boundary layers where the turbulence model quantities from the RANS simulation have to remain untouched, a similar approach from the one proposed by Spalart et al. [12] for DDES is used. First, the squared ratio of the modeled turbulent length scale to the wall distance is computed:

$$r_d \equiv \frac{\mathbf{v}_t + \mathbf{v}}{\sqrt{U_{i,j}U_{i,j}} \, \mathbf{\kappa}^2 \, d^2}.\tag{10}$$

The velocity gradient tensor is  $U_{i,j}$ ,  $\kappa$  is the Kármán constant and *d* is the distance to the closest wall. The parameter  $r_d$  is therefore equal to unity in the logarithmic region and gradually falls towards 0 as it gets further away from the wall. It is then injected into the following equation :

$$f_d \equiv 1 - \tanh([8r_d]^3).$$
 (11)

The parameter  $f_d$  is then equal to 1 in the LES region where the ratio of the modeled turbulence length scale to the wall distance is a lot smaller then unity. Everywhere else,  $f_d$  is null.

As discussed previously, it was chosen in this work to neglect the modeled part of the turbulent spectrum in the LES region. Therefore, small values of  $v_t/v = 0.1$  and I = 0.1%are imposed in the LES region of the inlet plane. To do so, the turbulent kinematic viscosity  $v_t$  and turbulent intensity Iobtained from the RANS simulations of the upstream components, designated by the subscript <sub>RANS</sub>, are adjusted with the following formulas:



Figure 2: Strategy used to impose the boundary conditions at the inlet of the draft tube.

$$\mathbf{v}_t = (\mathbf{v}_t)_{RANS} (1 - f_d) + 0.1 \mathbf{v} f_d,$$
 (12)

$$I = I_{RANS} (1 - f_d) + 0.001 f_d.$$
(13)

The effect of this operation can be visualized on Figure 3.



**Figure 3:** Ratio of  $v_t/v$  and turbulent intensity a) from the RANS simulation of the upstream components and b) after the attenuation of the modeled turbulent quantities in the LES region.

# 2.4 Grid Resolution

The flow structures ejected by the runner have small dimensions and rotate at a high velocity. Furthermore, many authors, such as Mauri [5], Cervantes et al. [2] and B.-Vincent [1], observed a rapid diffusion of these structures underneath the inlet plane of the draft tube. It is suspected that a too coarse spatial and time resolution may be the cause.

In order to establish the meshing and time step requirements to capture adequately the vortical flow structures at the inlet of the draft tube, simulations have been performed on a simplified geometry. It includes the draft tube cone and a straight extension which has the same length as the cone.

Five grids were tested. The number of nodes varied between 2 and 15 millions. The time step corresponds to  $0.5^{\circ}$  of rotation of the runner. However, time discretization will be discussed separately in the next section. For each mesh tested, the evolution of the vortical structures ejected by the runner

is shown on Figure 4. They are visualized with the q-criterion  $q = \frac{1}{2}(||R_{ij}||^2 - |||S_{ij}||^2)$  where  $R_{ij}$  is the rotation tensor and  $S_{ij}$  is the strain-rate tensor. As shown, it is clear that the grid resolution has a strong effect on their propagation in the draft tube.

Furthermore, one can observe these large coherent anisotropic structures through the off-diagonal components of the Reynolds stress tensor. Figure 5 shows the Reynolds stress tensor component  $\langle u'_{\theta}u'_{z} \rangle$  as a function of the radial position on the inlet plane. It is compared to the experimental measurements of Tridon et al. [14], who measured the axial and tangential velocity by LDV 0.13D under the runner at a similar operating point. D refers to the runner diameter. The velocity profiles and fluctuations were measured on three diameters and then averaged to give a profile on a single radius. Further details on their experimental methodology can also be found in [13].

Once again, the effect of the grid resolution is clearly visible. The finest mesh, which counts nearly 15 M nodes is in reasonable agreement with the experimental data, especially when it comes to the amplitude of the fluctuations. However, the slightly coarser mesh, which contains nearly 12 M nodes seems to be a good compromise between computational cost and precision. It is felt that the effect of the large scale structures under the runner should be well captured with this mesh. Therefore, this grid resolution will be transposed to the complete draft tube and used for further investigations.

## 2.5 Time Discretization

A similar approach is used to evaluate the necessary time step resolution to adequately capture the vortical structures ejected by the runner. Four time step resolutions are tested in the simplified geometry. They correspond to  $0.25^{\circ}$ ,  $0.5^{\circ}$ ,  $0.75^{\circ}$ and  $1^{\circ}$  of rotation of the runner. The equivalent normalized time steps are shown in Table 1.

**Table 1:** Time steps tested in terms of runner rotation angle and normalized by the axial average velocity and runner diameter.

$\Delta^\circ$	$\Delta t^* = \frac{4Q\Delta t}{\pi D^3}$
0.25	$0.75  imes 10^{-3}$
0.5	$1.50 \times 10^{-3}$
0.75	$2.24 \times 10^{-3}$
1	$2.99 \times 10^{-3}$

All simulations here were performed with the mesh containing 11.7 M nodes. First, the vortical structures are shown with the q-criterion on Figure 6. As for the the grid resolution, the time step has a strong influence on the way the flow structures



**Figure 4:** Turbulent structures shown with the q-criterion for different meshes and  $\Delta t^* = 1.5 \times 10^{-3}$  ( $q = 1500 \text{ s}^{-2}$ ).



**Figure 5:** Circumferential average of Reynolds stress  $\langle u'_{\theta}u'_{z} \rangle$  as a function of the radial position 0.13D under the runner. The time step corresponds to 0.5° of rotation of the runner ( $\Delta t^{*} = 1.5 \times 10^{-3}$ ).

propagate downstream.

This can also be felt when observing the Reynolds stress tensor component  $\langle u'_{\theta}u'_{z} \rangle$  in Figure 7. The amplitude of the fluctuations are reduced with the two coarser time steps. However, the difference between the two finer time steps, of  $0.5^{\circ}$  and  $0.25^{\circ}$ , is very subtle. Therefore, a time step of  $0.5^{\circ}$  of rotation of the runner will be used for the simulation of the complete draft tube.

## **3 RESULTS**

The necessary grid resolution, once transposed to the complete draft tube, generated a grid with nearly 31 M nodes. The quality criteria of the mesh are shown in Table 2.

In order to evaluate the effect of the vortical structures ejected

 Table 2: Quality criteria of the draft tube mesh.

Min. Angle	> 33.5°
Determinant	> 0.65
Aspect Ratio	< 1500
Expansion factor	< 2.5

by the runner on the flow in the draft tube, two simulations have been performed. The first one includes the structures at the inlet, while for the second one, steady axisymmetric velocity profiles are imposed. In both cases, modeled turbulent quantities are attenuated outside the boundary layers as described in section 2.3. This comparison is performed at an operating point near the best efficiency point, at  $Q/Q_{Opt} = 0.91$ , where Q is the flow rate and  $Q_{Opt}$  is the flow rate at the best efficiency point. In both cases, the time step is set to  $0.5^{\circ}$  of rotation of the runner. It is worth noting that this time step provides a CFL number well below unity.

First of all, one can observe that the flow structures ejected by the runner propagate in the draft tube cone on Figure 8.

To grasp their effect on the flow downstream and on momentum transfer to the boundary layers, Figure 9 shows the skin friction lines on the draft tube solid boundaries.

In both simulations, the boundary layer seems to have a very similar behavior. Indeed, the saddle point on the back of the draft tube cone and the separation lines emerging from it are almost identical. However, the saddle point is slightly higher in the simulation where an unsteady velocity field is imposed at the inlet, indicating that boundary layer separation occurs a little earlier.

No major differences appear on the flow topology in the outlet bays. This can be seen on Figure 10, which shows the timeaveraged velocity contours in the draft tube.

Finally, it is of great interest to observe if these flow structures affect the performance of the draft tube. To do so, the evolution of the static pressure recovery coefficient  $\chi$  in the



**Figure 6:** Turbulent structures shown with the q-criterion for different time step sizes and 11.7 M nodes ( $q = 1500 \text{ s}^{-2}$ ).



**Figure 7:** Circumferential average of Reynolds stress  $< u'_{\theta}u'_z > as$  a function of the radial position 0.13D under the runner. The grid used has 11.7 M nodes.

draft tube is compared for both simulations on Figure 12. The static pressure recovery coefficient definition used is the one proposed by McDonald et al. [6] :

$$\chi = \frac{\frac{1}{A_{out}} \int p_{out} \, dA_{out} - \frac{1}{A_{in}} \int p_{in} \, dA_{in}}{\frac{1}{2} \rho \frac{1}{A_{in}} \int |u_{in}|^2 \, dA_{in}}, \qquad (14)$$

where A is the area of the section and the subscripts <sub>out</sub> and <sub>in</sub> refer respectively to the outlet and inlet sections.

Once again, the effect of the vortical structures ejected by the runner is rather small. However, the earlier separation of the boundary layer with the unsteady inlet velocity field can be observed. Indeed, the static pressure recovery is slightly better in the draft tube bend when steady axisymmetric velocity profiles are imposed at the inlet plane.

The off-diagonal Reynolds Stresses generated by the vortical



**Figure 8:** Coherent turbulent structures in the draft tube shown with the *q*-criterion.

structures ejected by the runner could explain this behavior. The profiles, shown in Figure 11, display a negative  $\langle u'_{\theta}u'_r \rangle$  and positive  $\langle u'_z u'_r \rangle$  (axial velocity is negative) near the wall, indicating that momentum is being "pulled" away from the boundary layer. This phenomenon is of course lost when velocity profiles are circumferentially averaged. However, one should remember that the effect of this seems rather limited when comparing flow topology and performance.

### **4 CONCLUSION**

The objective of this work was to evaluate if the inclusion of large scale vortical structures ejected by the runner in the draft tube inlet plane (which is more consistent with the DES turbulence modeling approach) can improve the prediction of the flow in the draft tube.

It was first shown that the flow structures ejected by the runner require a very fine grid and time step resolution to avoid their premature diffusion underneath the inlet plane, even if negligible modeled turbulence  $(v_t)$  is imposed. However, even with an adequate mesh and time step, their effect on the flow downstream appears to be very limited. Indeed, the flow topology and the static pressure recovery coefficient



**Figure 9:** *Skin friction lines on the draft tube solid boundaries a) with the vortical flow structures injected at the inlet plane and b) with steady axisymmetric velocity profiles.* 



**Figure 10:** *Time-averaged normal velocity field in the draft tube outlet bays a) with the vortical flow structures injected at the inlet plane and b) with steady axisymmetric velocity profiles.* 



**Figure 11:** Circumferentially averaged off-diagonal Reynolds Stress tensor components 0.13D under the runner with and without the vortical structures injected at the inlet plane (2D inlet vs Axisymmetric).



**Figure 12:** Evolution in the draft tube of the static pressure recovery coefficient  $\chi$  with and without the vortical flow structures injected at the inlet plane (2D inlet vs Axisymmetric).

were very similar to those obtained with a simulation where steady axisymmetric velocity profiles were specified at the inlet plane.

However, this investigation was performed at an operating point close to the best efficiency point. Even though the vortical flow structures do not directly transfer a significant amount of momentum to the boundary layers, they could have a more significant interaction with the central vortical structure at partial discharge, where it is stronger. Therefore, in future works, a similar investigation could be performed at different operating points, in order to generalize the present conclusion.

#### **ACKNOWLEDGEMENTS**

The authors would like to thank Alstom Power & Transport Canada inc, the FQRNT and the NSERC for their financial support throughout this research project. Computations were performed on the Colosse supercomputer at Université Laval, under the auspices of Calcul Québec and Compute Canada. Finally, the authors would like to gratefully thank Sylvain Tridon, of ALSTOM Hydro France, for sharing his experimental data.

#### REFERENCES

- P. B.-Vincent. Simulations avancées de l'écoulement turbulent dans les aspirateurs de turbines hydrauliques. Master's thesis, Université Laval, 2010.
- [2] M. J. Cervantes, U. Andersson, and H. M. Lövgren. Turbine-99 unsteady simulations - Validation. In Proceedings of the 25th IAHR Symposium on Hydraulic Machinery and Systems, Timişoara, September 2010. IAHR.
- [3] J.-M. Gagnon, V. Aeschlimann, S. Houde, F. Flemming, S. Coulson, and C. Deschenes. Experimental investigation of draft tube inlet velocity field of a propeller turbine. ASME Journal of Fluids Engineering, 134:10102– 12, October 2012.
- [4] ANSYS Inc. ANSYS CFX-Solver theory guide, November 2010.
- [5] S. Mauri. Numerical Simulation and Flow Analysis of an Elbow Diffuser. PhD thesis, École Polytechnique Fédérale de Lausanne, 2002.
- [6] A. T. McDonald, R. W. Fox, and R. V. Van Dewoestine. Effects of swirling inlet flow on pressure recovery

in conical diffusers. *AIAA Journal*, 9(10):2014–2018, October 1971.

- [7] F. R. Menter. Improved two-equation k ω turbulence models for aerodynamic flows. Technical Memorandum 103975, NASA, October 1992.
- [8] F. R. Menter and M. Kuntz. Adaptation of eddyviscosity turbulence models to unsteady seperated flow behind vehicles. In Rose McCallen, Fred Browand, and James Ross, editors, *The Aerodynamics of Heavy Vehicles : Trucks, Buses, and Trains*, pages 339–352. Springer, 2004.
- [9] F. R. Menter, M. Kuntz, and R. Langtry. Ten years of industrial experience with the SST turbulence model. In K. Hanjalić, Y. Nagano, and M. Tummers, editors, *Turbulence, Heat and Mass Transfer 4*. Begell House, Inc., 2003.
- [10] P. E. Smirnov and F. R. Menter. Sensitization of the SST turbulence model to rotation and curvature by applying the Spalart-Shur correction term. In *Proceedings* of ASME Turbo Expo 2008: Power for Land, Sea and Air, Berlin, Germany, June 2008. ASME.
- [11] P. R. Spalart. Detached-eddy simulation. Ann. Rev. Fluid Mech., 41:181–202, 2009.
- [12] P. R. Spalart, S. Deck, M. L. Shur, K. D. Squires, M. Kh. Strelets, and A. Travin. A new version of detached-eddy simulation, resistant to ambiguous grid densities. *Theor. Comput. Fluid Dyn.*, 20:181–195, 2006.
- [13] S. Tridon. Étude expérimentale des instabilités tourbillonnaires dans les diffuseurs de turbomachines hydrauliques. PhD thesis, Institut Polytechnique de Grenoble, June 2010.
- [14] S. Tridon, S. Barre, G. Dan Ciocan, and L. Tomas. Experimental analysis of the swirling flow in a Francis turbine draft tube: Focus on radial velocity component determination. *European Journal of Mechanics B/Fluids*, 29(4):321 335, 2010.