Challenges in assessing the grid sensitivity of hydro-turbine CFD simulations

R Magnan¹, S Cupillard¹, G Gauthier¹, A M Giroux¹, M Page¹, C Deschênes²

¹ Institut de recherche d’Hydro-Québec, Varennes, Québec, Canada
² Hydraulic Machine Laboratory (LAMH), Université Laval, Québec, Canada

E-mail : magnan.robert@ireq.ca

Abstract. Despite the great progress made over the last 20 years, making reliable predictions with CFD is still a challenge. In fact the hydraulic machinery community rarely relies entirely on numerical simulations to provide final designs. The core of the problem lies with the estimation of the solution accuracy. Grid convergence studies are a useful way to check for grid independence and can provide some insight into the actual solution accuracy. However, conducting such studies for complex, detached flows involves many challenges among which: creating progressively refined grids and comparing with measurement results. This paper investigates this problem through the practical example of the BulbT experimental setup for which multiple CFD runs were conducted.

1. Introduction

Most practitioners of Computational Fluid Dynamics (CFD) in hydraulic turbines still rely heavily on structured grids for their simulations. Most of us recognize that the grid generation task is time-consuming and involves a high level of expertise to meet the somewhat fuzzy requirements leading to good quality grids. The introduction of unstructured grids in the CFD process is often relegated to the most complex 3D geometries like casings and draft tubes where hand-crafting a multi-block grid is sometimes too costly to justify.

We have previously, in ad hoc tests, used unstructured meshes to compute turbine efficiency but have always found the results to be somewhat unreliable. We have also found recently some cases where similar and (in our judgment) reasonably high-quality block-structured computational grids yielded different results. This prompted us to delve into this matter more rigorously and try to evaluate the discretization error and the accuracy of different CFD solutions obtained on a variety of computational grids. The advent of the BulbT dataset [3] provided us with the necessary data to settle on which grid yielded the best results and try to understand why some grids are better than others.

Since our basic goal here is to evaluate how new grid generation software will impact the numerical results, we wanted to keep with an established and proven methodology. The setup used for the complete turbine simulation is fairly traditional: Ansys-CFX RANS simulation with the k-epsilon model, multi-stage computations with the distributor and runner modeled using periodic boundaries. For the advection modeling we resorted to a second-order scheme rather than the more usual high-resolution. This choice was motivated by our desire to avoid the automatic switching happening in this latter scheme, which could hinder the convergence studies.

In order to draw meaningful conclusions from the study, we initially aimed for a numerical model as faithful to the lab model as possible. This proved to be difficult to achieve. Some of the obstacles
encountered are inherently linked to our limited knowledge (the turbulence level at the inlet or the exact shape of the intake for example), others to limitations in the physical models (turbulence, mixing planes) but still more to deliberate simplifications introduced because the reality was just too complicated (gaps in the distributor, fillets in the runner).

2. The BulbT dataset
One of the prevalent difficulties in assessing the precision of CFD results is gaining access to accurate measurement data (including complete geometry) for comparison. As a utility Hydro-Québec generally does not have access to lab data. The recent set up of Université Laval’s Consortium on Hydraulic Machines (CHM) has thus created a fabulous opportunity for us.

CHM has been working for some seven years on obtaining high-quality, very detailed measurements on model scale turbines in its Québec City laboratory. Over the last two years, through the BulbT project, the research team has put out some of the best results so far. This database comprises velocity and pressure measurements in various areas of a bulb turbine: intake, inter-blade, draft tube, etc. We are interested in making the most of this rich dataset to validate the use of new CFD technology: mesh generators, solvers, turbulence models and post-processing workflow. In this paper, we focus mainly on meshing technology and its impact on CFD results.

![Figure 1. The BulbT test rig global configuration, measurement section 4B (red) and CFD domain interfaces (green).](image)

The Laval University’s test rig is described in [3] and the BulbT database in [2]. The bulb turbine studied was already available along with the intake and distributor ring. A new draft tube with a steeper recovery was designed and manufactured to make sure the flow would exhibit some features of interest. The actual geometry of the blades and draft tube was measured to account for any deviation out of specification [10]. In fact, this highlighted some waviness of the draft tubes walls and slight discrepancies between runner blades (we settled on using blade #1 only).

Multiple test campaigns were conducted by CHM under various operating conditions. Although we eventually intend to cover a larger spectrum, we have currently focused on test campaign #4 with the machine operating at OP1, a condition close to the best efficiency point. According to the lab results and observations, the flow at this point was essentially featureless.

This campaign provides us with LDV velocity measurements in two planes dubbed 4A and 4B located just aft of the runner blades [13], static pressure along the draft tube wall and global information like torque and axial thrust on the runner.

3. Grid generation and comparisons
For each of the four domains (intake, distributor, runner and draft tube), we have generated two sets of grids: structured multi-block grids with Ansys-ICEM and either Numeca-Autogrid (AG) or hybrid-Cartesian grids with Numeca's HexpressHybrid (HH). For each type, we have created grids with
progressive refinement levels allowing us to run grid convergence studies. An example of such refinement is illustrated in Figure 2.

![Figure 2. Three levels of AG grid refinement in the runner.](image)

ICEM is well known; it can generate arbitrarily complex multi-block grids but requires much interactive work when tackling a new geometry and to fine-tune the node distributions. It was used in all domains and is viewed here as the reference grid. Autogrid is a semi-automated multi-block template-based mesher for turbomachinery. It can rapidly create high-quality grids from a few parameters; however, it offers less flexibility to tune the cell distribution precisely and makes it hard to create uniformly refined grids. It was only used for the distributor and the runner. HexpressHybrid is another product from Numeca which can create a Cartesian grid in arbitrarily complex geometries. That grid can be subdivided locally within spatial volumes and progressive wall layers can be added. Transitions between grid levels are managed by inserting pyramids, wedges or tetrahedrons. A benefit of this type of grid is that the core of the domain is made up of perfect cubes, theoretically reducing discretization errors linked to skewness, expansion and non-orthogonal faces. These grids are used for the complex geometry of the intake and the draft tube. In the draft tube, two topologies of grids were generated with ICEM. In the first one, dubbed “topo>”, the O-grid under the runner cone is extended straight in the axial direction. In the second one, dubbed “topo<”, the blocking is spread out to distribute the cells more uniformly. Coarse samples of all grid types are illustrated in Figure 3.

|                     | ICEM                     | Hexpress or Autogrid |
|---------------------|--------------------------|----------------------|
| Intake              | ![Intake ICEM grid](image) | ![Intake Hexpress or Autogrid grid](image) |
The grids used were deemed of decent quality both to our eyes and to the CFX filter. Table 1 shows typical cell quality statistics for the mid-range grids. The “critical” column indicates the percentage of cells that are considered of marginal quality by CFX. However, generating very refined grids, while maintaining appropriate $y^+$ value in the 20-100 range near the wall, proved challenging, especially for
a mesher like Autogrid, which involves complex interactions between the node distribution and the blocking. In fact, for the finest grid we sometimes had to change some meshing parameters resulting in valid but not systematically refined grids.

### Table 1. Grid quality statistics computed by CFX.

| Component     | Mesher     | Min. Orthogonality Angle | Max. Expansion Ratio |
|---------------|------------|--------------------------|----------------------|
|               |            | Angle (deg)              | Critical (%)         | Ratio       | Critical (%) |
| Intake        | ICEM       | 30.6                     | 7                    | 10          | <1           |
|               | HH         | 15.5                     | 3                    | 36          | 9            |
| Distributor   | ICEM       | 36.6                     | 1                    | 5           | 0            |
|               | Autogrid   | 46.4                     | 1                    | 2           | 0            |
| Runner        | ICEM       | 20.9                     | 24                   | 9           | <1           |
|               | Autogrid   | 27.7                     | 24                   | 9           | <1           |
| Draft tube    | ICEM “topo=“ | 25.3                   | 9                    | 3           | 0            |
|               | ICEM “topo<“ | 29.3                   | 11                   | 2           | 0            |
|               | HH         | 35.2                     | <1                   | 23          | 3            |

Generating HH grids in the intake also proved more difficult than expected. The tiny gap between the sharp trailing edge of the large bulb pillars and the downstream interface required such a high cell density to properly capture the edge that wall layers could not be inserted into more refined grids. In the end, little effort has been spent on the HH intake grids because the focus quickly laid on the other components.

### 4. Grid convergence studies

Since the early 1990s and Roache's book [9], grid convergence (GC) studies have made their way into some of the most influential CFD best-practices guidelines [1][4] as a means to ascertain the validity of numerical results. The basic idea is to analyze the evolution of some response quantity of the numerical model (also coined System Response Quantity – SRQ [7]) to the systematic refinement of the grid. These SRQs can be anything of interest – power, force, losses, local or global. In a finite volume scheme, the truncation error is linked to the cell size. If the grid is fine enough, reducing this size should correspondingly reduce the error in a systematic way. If the grid is too coarse, some flow physics may be missing altogether and refining the grid might not show a well-behaved convergence pattern. Without access to the exact solution, the solution error cannot be computed, but convergence toward a value on an infinitely fine grid can be highlighted by tracing the SRQ against the typical grid spacing $h$. On a complex 3D grid, this $h$ value is computed as

$$h = 1 / \sqrt[3]{N_{cells}}$$

It should be noted that the flow physics being different for all operating points, a grid which is deemed satisfactory for one operating point might not be that good for another one. Consequently, GC studies should theoretically be carried out for each point of a numerical hill chart. Similarly, convergence for one SRQ does not imply convergence for all SRQs [8].

This convergence behavior should only happen on systematically refined grids, which can hardly be enforced with unstructured and automatic meshers. Even on block-structured grids, this stipulation is not realistic when using wall functions which have their own incompatible requirements on cell height near solid walls. When creating very refined meshes, we find ourselves in the awkward situation where cells in the core flow can become more refined than the cells at the wall. For the purpose of GC studies, we aimed for the same relative density distribution in each refined grid.

Our analysis of suitable grids for the flow simulations follows a previous study by Guénette [5], who selected the proper level of refinement of multi-block grids by varying one by one the refinement along each direction: axial, radial and tangential. His comparisons, however, were made before any
actual measurement took place and thus without any means to judge whether the accepted grid yielded the right answer or not.

In our specific situation, we ran independent GC studies for each of the four hydraulic components at a single operating condition. We computed the averaged radial (1-D) profile of velocity, pressure, and turbulent quantities at the inlet and outlet of each component from a solution obtained on a mid-range mesh for operating condition OP1 and used these profiles to make independent studies in each component. Comparisons on cross-section 4B under the runner with the measured average velocity profiles (curve “CFD-stage” in Figure 4) show that the global solution is right.

Figure 4. Axial and tangential velocity comparisons on cross-section 4B.

We conducted the GC studies with at least three grid levels with each type of mesh. Results are plotted as a function of the global grid spacing $h$ (see Figure 5).

Figure 5. Grid convergence graphs for all components.

The convergence graphs generally show some stabilization of the SRQs as the grid is refined (grid spacing is reduced). However this does not always happen in the expected, asymptotical way. Moreover, from one grid to the next the limit value when $h=0$ differs significantly, which is somewhat disturbing.

The ICEM intake grids converge nicely. The coarsest grid seems to be almost as good as the finest; a testimony that the flow is already well resolved. Only one HH grid was constructed in the intake and the losses it predicted were of the same order as the ICEM grids. Little more can be said without either a measured value or more grids.

In the distributor, the almost linear curve for the ICEM grids is surprising. It looks as though the asymptotic convergence zone has not yet been reached. This could not be explained. The limit value...
for the losses prediction is around 1.2% of the total head. The AG grids, on the other hand, exhibit a nice convergence behavior to a value around 1.3%. The difference may seem large but, in absolute terms, this represents a tiny fraction of the overall losses.

In the case of the runner, we have analyzed multiple SRQs. Focusing on losses, the convergence behavior is regular, but again we have two curves converging to different limit values. This is hard to explain (even though the ICEM grid has worse orthogonality statistics than the AG grids – see Table 1) because both sets of grids do indeed converge. Looking at power, the pattern is different. The ICEM grids are almost insensitive to refinement while the more sensitive AG grids show a strange, divergent behavior. But, in the end, the finest AG grid predicts a value very close to the measured value. On this grid, the first cell height was already a limiting factor and further refinement was not realistic. One should note that the AG grid refinement is not fully systematic owing to the automated process and some constraints on the shroud tip gap. This could explain the observed behavior. Finally, it should be mentioned that the computed power was obtained for non-coupled simulations (single component with averaged profiles used as boundary conditions). Thus comparison with the power measured in the lab is slightly flawed.

The graph for the draft tube GC study is puzzling. The ICEM “topo=” grids seem to converge nicely but the other grids produce wildly different results. The finest HH grid (HH 27M) for example, despite its 27 million cells, seems way off. In order to address this situation, a more detailed analysis of the flow behavior in the draft tube was realized.

5. Draft tube analysis
The three steady-state solutions computed using the ICEM “topo=” grids all converged to low-residuals (indicated by the circles around the corresponding points) and yielded similar loss values. These grids are all quite dense with acceptable cell quality. The GC study indicated a good asymptotic pattern. Moreover, additional simulations with 11- and 18-million-cells HH grids confirmed the predicted values. The picture changed only when an additional HH grid of higher density became available. We then got a hint that something more complex was happening.

Figure 6 illustrates the flow field on the X-Z mid-plane computed on different grids. In all cases, only the draft tube was computed from a fixed radial inlet profile. We can see that the grids, despite their decent quality and high density, yielded vastly different flow patterns. The corresponding losses (Figure 5) were also quite spread out.

For all grids, the flow field downstream of the runner (cross-section 4B) is relatively well captured, but it subsequently develops differently depending on the grid type. Such behavior is not unusual for detached flow, as was also exhibited in the AIAA sponsored Drag Prediction Workshops of the last decade [6][11]. Refined grids just downstream of the runner hub indicated some unsteadiness, which is visible in the velocity and turbulent kinetic energy (TKE) contours of Figure 6, but which was also felt from the maximum residuals which could not be lowered substantially below $10^{-3}$. We thus switched both the ICEM “topo=” grid and the HH 27M grid to a URANS computation to see how the flow behaved in unsteady computations and found that the initial steady-state solution was slowly flushed out of the domain, to be replaced by some unsteady vortex rope which entailed substantially lower draft tube losses. In fact, we ended up with three families of solutions: the low-residual RANS solutions (the ones which would normally be considered acceptable) had quite large losses, the RANS solution, which stagnated with max residuals above $10^{-3}$, had somewhat lower losses and, finally, the URANS solutions which had still lower losses. In this case, averaging the steady-state monitor values as suggested by Vu et al. [12], did not yield the same result as time-averaging the unsteady simulation values. The lack of a measured global value for comparison is crippling here.
6. Conclusions and future work
The relative insensitivity of the first draft tube grids in picking up the unsteadiness in the flow could easily have led us to predict losses that were way too large. Even the GC study led us to believe that everything was fine. This should warn us of too swift conclusions regarding the suitability of a given grid for detached flow analysis.

| Grid | Cells | Axial velocity | Pressure | TKE |
|------|-------|----------------|----------|-----|
| Icem topo < | 3M | ![Image](image1.png) | ![Image](image2.png) | ![Image](image3.png) |
| Icem topo = | 3M | ![Image](image4.png) | ![Image](image5.png) | ![Image](image6.png) |
| Icem Topo < Unsteady Averaged (coupled) | 3M | ![Image](image7.png) | ![Image](image8.png) | ![Image](image9.png) |
| HH | 18M | ![Image](image10.png) | ![Image](image11.png) | ![Image](image12.png) |
| HH | 27M | ![Image](image13.png) | ![Image](image14.png) | ![Image](image15.png) |
| HH Unsteady Averaged | 27M | ![Image](image16.png) | ![Image](image17.png) | ![Image](image18.png) |

Figure 6. Flow field contours in the draft tube. Uncoupled solution except where noted.
RANS grid convergence studies in hydro turbines are not conducted on a regular basis. They are difficult to carry out because of the huge resources involved but also because systematically refined grids are difficult to obtain and wall functions add some specific constraints on the grid. Moreover, the guidance they offer is not always reliable. However, conducting such studies with more than one grid configuration is instructive. Two different grids (say a multi-block structured and an unstructured) converging toward the same value greatly reinforces the credibility of the results. More probable is the case where the two grids will converge to different values, as we have seen here for the runner. In that situation, the difference can provide some quantitative accuracy indication. The lack of systematic convergence for one grid type warns of something worth digging into.

These are mostly impressions gathered from the analysis conducted around a single operating point. To go further, comparisons with more experimental data are required but only sketchy data is available for OP1 at the time of writing. It is also worth noting that high-precision comparisons with experimental data are only meaningful if the numerical modeling was faultless. Inexact geometry, less than perfect boundary conditions and questionable physical modeling choices are other factors that can explain discrepancies between computation and measurement. These, however, are difficult to quantify without proper sensitivity analysis.

More generally, we have seen that different good-quality grids can yield quite different flow fields and sometimes important differences in engineering quantities. The reason for this has not yet been fully understood and more investigation is warranted. GC studies can be an aid in detecting these situations but in day-to-day practice they are not realistic.

On a different note, it might be instructive to carry out similar GC studies with low-Re turbulence models requiring tiny cells at the wall but at the same time circumventing the difficulties with the cell height constraint at the wall. These turbulence models might also provide better results in the presence of detaching boundary layers. They were voluntarily left out of the present study because they would constitute yet another variable to account for in the comparisons.

The BulbT test case proved to be highly sensitive (actually it was designed that way) and might not be representative of typical installations. It is a good example, though, of the difficulty in designing experimental setups aimed at CFD validation and ensuring meaningful comparisons are made.

Work on this problem will continue with further investigation with new, more systematically refined grids, other operating conditions and comparisons with PIV velocity measurements in the draft tube when they become available. The profiles used for the GC studies would be upgraded to new ones computed from an unsteady simulation as they offer a better agreement to the measured profiles (see unsteady curve on Figure 4). In an ideal world, we would also like to run similar analysis with OpenFOAM, other turbulence models, and other meshers.

Acknowledgments
The authors would like to thank the participants of the Consortium on Hydraulic Machines for their support and contribution to this research project: Alstom Renewable Power Canada Inc., Andritz Hydro Ltd, Hydro-Quebec, Laval University, NRCan, Voith Hydro Inc. Our gratitude goes as well to the Canadian Natural Sciences and Engineering Research Council who provided funding for this research.

References
[1] ASME 2009 Standard for verification and validation in computational fluid dynamics and heat transfer
[2] Consortium on Hydraulic Machines 2013 BulbT database guide
[3] Deschênes C, Houde S, Aeschlimann V, Fraser R and Ciocan G D 2014 Modern challenges for flow investigations in model hydraulic turbines on classical test rig, 27th IAHR Symposium on Hydraulic Machinery and Systems
[4] ERCOFTAC 2000 Best practices guidelines for industrial computational fluid dynamics, Version 1.0
[5] Guénette V 2013 Prédiction numérique de l’écoulement turbulent au sein d’une turbine bulbe par des simulations RANS, Master thesis, Laval University

[6] Murayama M and Yamamoto K 2008 Comparison study of drag prediction by structured and unstructured mesh method, J Aircraft 45, 3, 799-822

[7] Oberkampf W L and Roy C J 2010 Verification and validation in scientific computing, (Cambridge Press)

[8] Roache P J 1997 Quantification of uncertainty in computational fluid dynamics, Ann Rev Fluid Mech 29 123-60

[9] Roache P J 1998 Verification and Validation in Computational Science and Engineering Albuquerque, NM (Hermosa Publishers)

[10] Taraud J-P 2014 Recouvrement de géométries complexes et applications pour l’étude d’une turbine hydraulique de type bulbe, Master thesis, Laval University

[11] Vassberg et al. 2010 Summary of the fourth AIAA CFD drag prediciton workshop, AIAA Paper 2010-4547

[12] Vu T C, Devals C, Zhang Y, Nennemann B and Guibault F 2010 Steady and unsteady flow computation in an elbow draft tube with experimental validation, 25th IAHR Symposium on Hydraulic Machinery and Systems

[13] Vuillemard J, Aeschlimann V, Fraser R, Lemay S and Deschênes C 2014 Experimental investigation of the draft tube inlet flow of a bulb turbine, 27th IAHR Symposium on Hydraulic Machinery and Systems