Evaluation of turbulence models on roughened turbine blades

R Dutta¹, J Nicolle², A-M Giroux² and U Piomelli¹

¹Queen’s University, Kingston (ON) K7L 3N6, CANADA
²Institut de recherche d’Hydro-Québec, Varennes (QC) J3X 1S1, CANADA
E-mail: ugo@queensu.ca

Abstract. The accuracy of turbulence models for the Reynolds-Averaged Navier-Stokes (RANS) equations in rough-wall flows is evaluated by comparing the model predictions with the data obtained from large-eddy simulations (LES). We have considered boundary layers in favourable and adverse pressure gradients mimicking those encountered in hydroturbines. We find that some features of the flow cannot be captured accurately by any model, due to the fundamental modelling assumptions. An example is the flow reversal that occurs in the roughness sublayer prior to separation, which cannot be predicted by the commonly used approaches, which bypass the roughness sublayer while modifying the boundary conditions. In mild pressure gradients most models are sufficiently accurate for engineering applications, but if strong favourable or adverse pressure gradients are applied (especially those leading to separation) the model performance rapidly degrades. A particularly difficult problem (both for rough- and smooth-wall cases) is the return to equilibrium after a strong perturbation, a known limitation of RANS models. Simulations of real configurations using commercial codes are also considered.

1. Introduction
Roughness plays an important role in many fields of study; in geophysical and environmental applications, for instance, most surfaces are rough (hilly terrain, plant canopies, urban environments etc.). In engineering, roughness is particularly important in electronic cooling, turbomachinery, duct and pipe flows. The last two areas of application are of interest to the power industry and to Hydro-Québec. Roughness is especially important in penstocks and in the turbines themselves. The roughness of hydraulic surfaces increases with time, affecting the overall performance of equipment in a significant way: in a Francis turbine, measurements showed a loss of efficiency of 1.25% due to surface roughness [1]. Methods for surface smoothing are available, and it is well known that a small increase of efficiency through smoothing may result in important savings. The cost of smoothing penstocks and turbine surfaces is, however, significant, both because of direct costs and due to the shut-down time needed. The benefit can justify the investment, but an accurate way to evaluate losses related to surface roughness is required as a decision-making tool. An understanding of the effects on turbulence of specific forms of roughness is, therefore, critical for the hydroelectric industry, both to interpret the physical phenomenologies, and to develop predictive models.

At the present time, numerical modelling of flows in turbines relies on the solution of the Reynolds-Averaged Navier-Stokes (RANS) equations. While roughness corrections have been
developed, they were mostly tested in flows close to the canonical ones used to validate the models and set their constants; their accuracy in more realistic applications is therefore difficult to assess. While the real Reynolds numbers (typically of the order of $10^6$ to $10^8$) in hydroturbines still make high-fidelity simulations infeasible, the recent increase in computer power available allows us to develop a roadmap to address the roughness problem making use of such simulations at lower, yet significant, Reynolds numbers. Among these techniques are Direct Numerical Simulations (DNS), in which the grid is fine enough to resolve all the scales of motion, including the dissipative ones, and Large-Eddy Simulations (LES), in which the largest eddies are resolved, and the effect of the smaller, more universal ones, is modelled. In this paper we discuss methods to evaluate and improve turbulence models taking advantage of the availability of data obtained from LES. To obtain these results, simplified geometries are used at first. However, models are gradually extended to include more complex physics, with the comparison to industrial CFD tools in mind. At present, these simulations allow us to validate the models, determine the main sources of error, and the error bars that can be expected. Based on these results, applications to more realistic geometries are finally discussed.

2. Problem formulation

In this work we solve either the filtered or the Reynolds-Averaged forms of the incompressible equations of motion. They have the same form:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0; \quad \frac{\partial \overline{u}_i}{\partial t} + \frac{\partial \overline{u}_i \overline{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \nabla^2 \overline{u}_i + \frac{\partial \tau_{ij}}{\partial x_j};$$

an overline denotes either a filtered or Reynolds-Averaged variable and $\tau_{ij}$ are either the subgrid-scale (SGS) stresses, or the Reynolds stresses. The SGS stresses are modelled using the Lagrangian Dynamic Eddy Viscosity model [2], while various models have been tested to parametrize the Reynolds stresses; they will be discussed shortly.

The equations of motions were solved using two codes; the first one is a finite-difference code that uses second-order central differences on all terms and a staggered grid [3], and a second-order accurate time advancement scheme. The second code is the commercial software ANSYS CFX.

In the LES the inflow conditions were based on the recycling method by Lund et al. [4], the freestream velocity was assigned, periodic boundary conditions were used in the spanwise direction and convective boundary conditions [5] used at the outflow. An Immersed Boundary Method (IBM) was employed to represent the roughness. The rough surface was modelled as a random distribution of ellipsoids, and the volume-of-fluid in each cell was used to reduce the value of the velocity in grid cells partially or entirely occupied by the roughness elements [6]. This method results in a sandgrain-like surface, in which the roughness height is equal to the smallest axis of the ellipsoid. This methodology has been extensively validated in rough-wall flows [7, 8, 9, 10, 11].

ANSYS CFX is a finite-volume solver for fluid-dynamics problems, widely used in industry. It uses body-fitted grids, and can implement a variety of discretization schemes, turbulence models and boundary conditions. To be as representative as possible of industrial application, default parameters were kept. This involves a mixed-order upwind scheme approaching 2nd-order accuracy for pressure and velocity, and 1st-order for turbulent quantities [12].

The RANS calculations employed three turbulence models, with various roughness corrections. All the turbulence models examined do not include the roughness sublayer in the calculation. Rather, the solid surface is replaced with a virtual one in which the velocity vanishes, but the turbulent kinetic energy (TKE) and, in most cases, the eddy viscosity $\nu_T$ are not zero. The logarithmic law is generally used to assign the values of TKE and $\nu_T$ at the virtual wall, but the wall distance is offset by an amount (generally taken equal to $0.003k_s$, where $k_s$ is
the equivalent sand-grain roughness) to account for the presence of turbulence in the roughness sublayer. With ANSYS CFX, the sandgrain roughness length was directly imposed through a wall-function modified for roughness.

The first model tested is the one-equation Spalart-Allmaras (SA) model [13], with the roughness modification developed by Aupoix and Spalart [14] (we use the Boeing correction). Second, we used the two-layer $k-\varepsilon$ model of Chen and Patel [15], extended to rough wall flows by Durbin et al. [16]. Lastly, we employed the SST $k-\omega$ model [17]. Two roughness corrections were implemented: first the one proposed by Hellsten and Laine [18], second a new roughness correction we developed to extend the approach of Durbin et al. [16] to the SST $k-\omega$ model.

3. Results
We now discuss the results of the various simulations. First, we describe the application of LES and RANS turbulence models to the boundary layer on a flat plate with an Adverse Pressure Gradient (APG) strong enough to cause separation. This application highlights some critical issues that may be encountered when applying RANS models to rough-wall boundary layers with separation, and will help understand what the accuracy limits are in flows of this type. Since the pressure gradients encountered in hydraulic applications are only rarely strong enough to cause separation (notable exceptions are the draft tube and off-design conditions) we next consider the flow on a flat plate in which the freestream velocity reflects a typical profile encountered in a hydroelectric turbine. These calculations are carried out at a laboratory Reynolds number, much smaller than that encountered in practical applications. They are, nevertheless, useful to understand how the pressure gradient effects (which propagate from the freestream towards the wall) and roughness effects (which propagate outward from the wall) interact. These cases also allow us to determine the magnitude of the errors expected. We then switch to a more realistic geometry, and perform simulations with turbulence models both at low Reynolds numbers (matching the laboratory-scale ones used for the LES) and at realistic ones, to determine if the Reynolds number plays a role.

3.1. Separating boundary-layer
Figure 1 shows the computational domain used for the separating boundary-layer calculations. The Reynolds number, based on momentum thickness and freestream velocity at the reference location is $Re_\theta = 2,300$. Two roughness heights were considered, in addition to a case with smooth wall: $k_s/\theta_o = 0.5$ and 1; $\theta_o$ is the momentum thickness at the reference plane. Only

![Figure 1. Computational domain for the separating boundary-layer. $\theta_o$ is the momentum thickness at the reference plane.](image)
the smooth case and the one with larger roughness will be shown here. Almost 400 million grid points were used, and each roughness element was resolved using between 1,600 and 16,000 grid cells. For the RANS simulations, 110,000 grid points were used to discretize the 2D domain. The LES model, in this configuration, has been validated by comparison with the DNS results of Na and Moin [19].

Figure 2 shows the skin-friction distribution for the smooth and rough cases obtained from LES and RANS models. The skin-friction coefficient is defined as

$$C_f = \frac{\tau_w}{\rho U_o^2/2}; \quad \tau_w = 2 \frac{dC_D}{dx},$$  \hspace{1cm} (2)

where $C_D(x)$ is the local drag coefficient, the integral of viscous and pressure forces, at a given location, per unit streamwise and spanwise length, normalized by the dynamic pressure $\rho U_o^2/2$ (for more details on the method used to evaluate $\tau_w$ see [9]). Thus, the wall stress includes the contributions of both friction and the form drag around the roughness elements. When the wall is rough, the wall stress becomes zero much earlier than in the smooth-wall case, at $x/\theta_o \approx 142$ rather than 207. This phenomenon, however, is not associated with separation of the flow from the solid surface, but rather with flow reversal inside the roughness sublayer [20]. This can be observed by comparing the contours of streamwise velocity (figure 3). In the rough-wall case the streamline dividing the outer flow from the recirculation region separates much downstream of the point where $C_f = 0$, at $x/\theta_o \approx 190$: the zero-velocity contour also emerges from the roughness sublayer around the same location. In fact, the real separation (as opposed to the flow reversal) correlates much better with the location where the total stress becomes negative. We can define a normalized total stress at the roughness crest as

$$C'_f = \frac{\tau_{crest}}{\rho U_o^2/2}; \quad \tau_{crest} = \left( \mu \frac{\partial U}{\partial y} - \rho \langle u' v' \rangle \right)_{y=k_{max}} \hspace{1cm} (3),$$

where $\langle \cdot \rangle$ represents time averaging. $C'_f$ is also shown in figures 2 and 3, and approaches zero near $x/\theta_o = 210$.,
Since RANS turbulence models do not extend their calculations to the roughness sublayer, but include it only through the increased eddy viscosity, which is non-zero at the virtual wall, the total stress at the crest is an output of the calculation (with \(-\rho \langle u'v' \rangle \approx \nu_T \langle \partial u / \partial y \rangle \)). They cannot be expected to predict \(C_f\) correctly, since it is strongly affected by the flow in the roughness sublayer, but the total stress prediction is more accurate, and is reflected in better prediction of the separation point. Thus, one can expect that, in APG boundary layers, RANS turbulence models would significantly overpredict the friction drag (as in the initial region of the flow in figure 2), but would be more accurate in the estimation of the form drag, since the separation point is better captured; thus, in massively separated flow the accuracy could be expected to be higher than in mildly separated ones. Notice that this is a feature present in any model that takes the present modelling approach, using a virtual wall and bypassing the roughness sublayer entirely; it cannot be easily corrected by changing either the model coefficients or the modelling ansatz.

More significant errors are present in the reattachment region and where the flow returns to equilibrium. The principal source of error comes from the fact that the shear stress along the dividing streamline is due to the liftup of large turbulent eddies in the separation region, which are carried over the separation bubble and impinge on the reattachment zone [19]. Figure 4 shows the dividing streamline predicted by the LES and the various turbulence models, as well as the Reynolds shear stress, \(-\langle u'v' \rangle\), along the dividing streamline. The LES data show that the turbulence from the near-wall region is advected over the recirculation bubble, and significant additional Reynolds stress is only generated near the reattachment point. Note the much higher levels of near-wall shear stress, in the rough-wall case, advected from the upstream region. The turbulence models predict a lower recirculation bubble, but are also unable to predict such unsteady generation of momentum flux, being sensitive to the mean shear, which is large immediately above the dividing streamline (figure 5), as also discussed in [21]. The return to equilibrium is also a situation in which turbulence models fail, and that is the case in this geometry as well (figure 2).

### 3.2. Boundary layer with realistic acceleration

Next, we considered a case with a realistic acceleration of the type that could occur, for instance, in the hydraulic turbine distributor shown in in figure 6. To separate the errors due to the effect of the pressure gradient from others such as streamline curvature or interblade crossflow, we
extracted the velocity along a streamline on the pressure side of the stay vane and guide vane (see figure 6), and used that to generate the pressure gradient at the freestream over the flat plate. We then performed both LES (which used 270 million grid points) and simulations that solved the RANS equations, with the turbulence models described before. We also computed this case using the commercial software ANSYS CFX. In this case, the grid used 240,000 grid points, and the equations of motion were not integrated to the wall, since the first grid point was in the logarithmic region of the velocity profile; wall functions were used to determine the wall stress. The roughness height chosen was $k_s/\delta^* = 0.32$.

Since the LES could not be performed at a Reynolds number comparable to that in actual turbines, determining the pressure gradient at the freestream was somewhat arbitrary. Two dimensionless pressure gradients can be defined,

$$K = \frac{\nu}{U_\infty^2} \frac{dU_\infty}{dx}; \quad A = \frac{\delta^*_o}{U_\infty} \frac{dU_\infty}{dx}. \quad (4)$$

At low Reynolds number, matching $K$ will result in a much lower change in the freestream velocity $U_\infty$ than observed in the real turbine, and might lead to underestimation of the pressure-gradient effects. Matching $A$, on the other hand, will result in more significant changes in $U_\infty$ than observed in the real case, and overestimation of the pressure-gradient effects. We compare simulations in which $A$ is matched with those in which $K$ is matched, and expect these cases to be extrema, with the real configuration lying somewhere in-between.
Figure 7. Flow with realistic freestream velocity profile, case with matched $A$. (a) Freestream velocity $U_\infty$ (circles) and acceleration parameter $K$ (crosses). (b) Skin-friction coefficient.

Figure 8. Flow with realistic freestream velocity profile, case with matched $K$. (a) Freestream velocity $U_\infty$ (circles) and acceleration parameter $K$ (crosses). (b) Skin-friction coefficient.

Figure 9. Flow with realistic freestream velocity profile, case with matched $A$, $k_s/\delta_o^* = 0.32$. Contours of $-10^3 \times \langle u'v' \rangle$.

Figures 7 and 8 show the acceleration parameters and skin-friction coefficient in the two cases. As expected, when $A$ is matched, a significantly stronger acceleration is achieved, and the skin-friction coefficient is much larger (note that $C_f$ is normalized by the reference velocity).

Figure 9 shows contours of the Reynolds stress for the rough-wall case with matched $A$, $k_s/\delta_o^* = 0.32$. Contours of $-10^3 \times \langle u'v' \rangle$. 

7
superposed on the streamlines. The general features of the flow are captured correctly (as indicated by the streamlines); however, the Reynolds stresses, immediately downstream of the high acceleration region, \( x/\delta^* \approx 300 \), are significantly under-predicted by both SA and \( k-\omega \) SST models. The mean velocity profiles, figure 10, show similar trends. The error is maximum in the region following the strong favourable pressure gradient (FPG), and is significantly larger in the case with roughness. Strong FPGs are challenging cases for turbulence models, and it appears that the interaction of roughness with the FPG is not predicted well by any of the models tested.

An additional source of error, in industrial applications in which the Reynolds number is high, and the grid cannot be refined to resolve the wall layer, is in the use of wall functions, which relate the wall stress to the velocity when the first grid point is in the logarithmic region. Wall functions assume that the wall layer is in equilibrium (i.e., production equals dissipation) so that a logarithmic region exists. Such is not the case in flows with strong pressure gradients, three-dimensionality and separation. In this flow, in the region of strong acceleration, the accuracy of the wall functions decreases, and the error in the skin friction prediction (figure 11) increases. The general behaviour is, however, still captured.

3.3. Flow in a hydraulic turbine

Having determined the response of turbulence models in two simplified applications, we then applied them in a more realistic configuration. ANSYS CFX is used to study the flow. No experimental or numerical data is available for this configuration; however, the simplified
Figure 11. Flow with realistic freestream velocity profile, case with matched \( A, k_s/\delta^* = 0.32 \). Skin-friction coefficient obtained using ANSYS CFX with wall functions.

geometries indicate that the numerical model can predict the behaviour of the skin-friction coefficient, except in the regions of very high acceleration. We performed three simulations, two at the Reynolds number encountered in realistic applications, one at the Reynolds number of the LES and RANS described above, one tenth of the realistic one. At the high Reynolds number, two values of sandgrain roughness were used, \( k_s/\delta^* = 0.03 \) and 0.21. The first results in transitional roughness, the second in fully rough flow.

The skin-friction coefficient is shown in figure 12. Several features are noteworthy. First, the general trends match those of the flat-plate boundary layer with realistic acceleration, notably the mild increase of \( C_f \) on the first blade, the peak at the beginning of the second one with a subsequent decrease. Second, as expected, the magnitude of the \( C_f \) is intermediate between the cases with matched \( K \) and matched \( A \). Finally, the Reynolds number dependence is weak. This is due to the implementation of the wall functions, that assume fully rough flow in the range of roughness heights used here.

4. Conclusions

We have carried out large-eddy simulations (LES) of the flow in boundary layers with favourable and adverse pressure gradients, and compared the results with those obtained using turbulence models for the Reynolds-Averaged Navier-Stokes (RANS) equations, both within the framework of a research code (the same used for the LES), and using a commercial CFD solver, ANSYS CFX.

The LES were used in simple geometries, and provided both data for model validation, and also allowed us to determine the intrinsic limitations of the turbulence models. For instance, the LES data showed that flow reversal occurs first in the roughness sublayer. This is due to the fact that the recirculation regions behind roughness elements widen as an effect of the adverse pressure gradient, until they occupy most of the span of the boundary layer. At this point the wall stress \( \tau_w \) becomes negative, but the flow remains attached. Only further downstream the zero-velocity contour separates from the wall, as does the dividing streamline, characterizing the inception of real separation. The change of sign of the total stress (sum of Reynolds and viscous stresses) above the roughness crest correlates better, for the cases examined, with the true separation. The total stress is the quantity actually predicted by RANS turbulence models, and the models tested were able to calculate the real separation reasonably well. The skin-friction coefficient, however, was significantly overestimated because the roughness sublayer was
bypassed.

More significant errors, also due to the fundamental eddy viscosity assumption, appear above the recirculation bubble and in the reattachment region. These errors are due to the fact that turbulence models are only sensitive to the mean shear, and do not account for the contribution of the eddies generated at the separation point and advected over the recirculation bubble, which eventually impinge on the wall near the reattachment point. This issue is present in smooth-wall boundary layers as well, and in rough-wall cases the model prediction is slightly more accurate, as the turbulence level near the wall, upstream of separation, is higher. These errors are difficult to correct within the eddy-viscosity framework, and will probably need to be included in the error bars of the calculations.

For acceleration levels of the type encountered in a distributor the errors are lower, since the flow remains attached. In rough-wall cases, however, the model accuracy suffers. The use of wall functions further degrades the model accuracy. The trends are, however, predicted correctly.

Application of the turbulence models in an industrial code, with a realistic geometry, shows the same trends observed in the simplified cases. This indicates that the approach taken here, to perform careful validations of the model in simple geometries, but with realistic freestream velocity distribution, can be an effective tool to evaluate the validity and accuracy of RANS solutions in hydraulic turbines.

Acknowledgments

RD and UP acknowledge the support from Hydro-Québec and the Natural Science and Engineering Research Council of Canada (NSERC). The simulations were performed at the High Performance Computing Virtual Laboratory (HPCVL) Queen’s University site. UP also acknowledges the support of the Canada Research Chair program.

References

[1] Grenier R, Oryschuk P and Léveillé P 2007 Component surface deterioration and effect on performance Tech. Rep. T032700 0323A CEATI
[2] Meneveau C, Lund T S and Cabot W H 1996 J. Fluid Mech. 319 353–385
[3] Keating A, Piomelli U, Bremhorst K and Nešić S 2004 J. Turbul. 5 20–1–27
[4] Lund T S, Wu X and Squires K D 1998 J. Comput. Phys. 140 233–258
[5] Orlanski I 1976 J. Comput. Phys. 21 251–269
[6] Scotti A 2006 Phys. Fluids 18 031701
[7] Banyassady R and Piomelli U 2014 J. Turbul. 15 186–207
[8] Yuan J and Piomelli U 2014 J. Fluid Mech. 760 R1–1–12
[9] Yuan J and Piomelli U 2014 Phys. Fluids 26 015113–1–28
[10] Yuan J and Piomelli U 2014 J. Turbul. 15 350–365
[11] Yuan J and Piomelli U 2015 J. Fluid Mech. 780 192–214
[12] Barth T J and Jespersen D C 1989 The design and application of upwind schemes on unstructured meshes AIAA Paper 89-0366
[13] Spalart P R and Allmaras S R 1994 La Recherche Aéropatiale 1 5–21
[14] Aupoix B and Spalart P R 2003 Int. J. Heat Fluid Flow 24 454–462
[15] Chen H and Patel V 1988 AIAA J. 26 641–648
[16] Durbin P A, Medic G, Seo J M, Eaton J K and Song S 2001 J. Fluids Eng. 123 16–21
[17] Menter F R 1994 AIAA J. 32 1598–1605
[18] Hellsten A and Laine S 1997 Extension of the k − ω−SST turbulence model for flows over rought surfaces AIAA Paper 1997-3577
[19] Na Y and Moin P 1998 J. Fluid Mech. 370 175–201
[20] Mottaghian P 2016 Effects of Wall Roughness on Adverse Pressure Gradient Boundary Layers Master’s thesis Queen’s University Kingston (ON), Canada
[21] Raiesi H, Piomelli U and Pollard A 2011 J. Fluids Eng. 133 0212203–1—10