Simulations for Effect of Surface Roughness on Wind Turbine Aerodynamic Performance

Yong Su Jung and James Baeder
University of Maryland, College Park, MD, USA
E-mail: jung9053@umd.edu

Abstract. In the present study, blade surface roughness effects on the aerodynamic performance of NREL Phase VI wind turbine rotor are investigated using Reynolds-Averaged Navier–Stokes (RANS) simulation. The rotating rotor component is modeled using an overset mesh system and computations are conducted using an in-house heterogeneous CFD framework based on both CPUs and GPUs. The effect of laminar-turbulent boundary layer transition on the blade is included using the $\gamma-\theta-\Re_{e}tSA$ model. The surface roughness effects on both the turbulent boundary layer and boundary layer transition are modeled through extensions of turbulence and transition models. The locally distributed roughness at the leading edge zone is applied for both S809 airfoil and NREL Phase VI rotor simulations. The roughness effects on the airfoil performance are captured, and the prediction shows reasonable agreement with the experimental data. In the case of the rotor simulation, the surface roughness affects the performance differently depending on the wind speed. At the operating wind speed with attached flows, the roughness decreases the performance due to the roughness-induced transition occurring at the leading edge. On the other hand, at the higher wind speed, this transition induces less flow separation on the leeward side of blade, which is favorable to the performance. As the wind speed increased further, the flow become fully separated and the roughness effect is minor.

1. Introduction
Wind turbines are an effective means of obtaining environmentally-friendly sustainable energy, and the number of turbines operating around the world has dramatically increased in recent years. Typically, wind turbines are required to operate for many years after initial installation due to high production and installation costs. Therefore, it is important to maintain the desired component performance during the turbine’s life cycle for reliable energy production. The horizontal axis wind turbine (HAWT) is most prevalently used for both offshore and onshore applications. Because the HAWT can be installed and operated in various environments, the wind turbine blades are easily contaminated by various sources, such as insects, dirt, dust and erosion, depending on the operational environment.

These contaminants near the leading edge zone act as a distributed roughness on the surface, and the aerodynamic performance of the blade deteriorates. The surface roughness at the leading edge can influence not only the laminar-turbulent boundary layer transition process but also modify the flow characteristics in the turbulent boundary layer. As a general effect, the lift of a blade section decreases due to the modification of the log-law profile of the boundary layer and the drag increases due to the increased skin friction. Therefore, it is important to consider the surface roughness effects on the performance analysis for the design of wind turbine blades.
Low fidelity models, such as BEM (blade element momentum) or panel methods, have been used in the wind turbine industry as a main design tool [1]. Due to its low computational cost, these methods have advantages especially for large scale wind farm calculations. However, these models inherently assume two-dimensional flow and are limited for accurate prediction of the shear force and three-dimensional flow features, such as cross flow, laminar-turbulent boundary layer transition, and surface roughness characteristics. Therefore, high-fidelity CFD (Computational Fluid Dynamics) methods have been used widely for wind turbine simulations for more accurate prediction of the boundary layer on the rotating blades and various unsteady flow features including aerodynamic interference between the components [2, 3, 4, 5].

The NREL unsteady aerodynamics experiment [6, 7] is the most comprehensive experiment for CFD validation. The experiment was carried out in the NASA-Ames wind tunnel on a NREL Phase VI wind turbine. This experimental data set has been used extensively in many other works to compare with CFD simulation results [2, 3, 4, 5]. The CFD studies have been conducted for the isolated rotor or full configuration using either Reynolds-Averaged Navier–Stokes (RANS) or hybrid RANS-LES methods. Laminar-turbulent boundary layer transition models are coupled with the turbulence model to improve the rotor torque prediction, by capturing transition along the blade [3, 4, 5].

More recently, surface roughness effects have begun to be considered in fully turbulent wind turbine airfoil/blade simulations. Ferrer and Munduate [8] showed a capability of RANS simulation for the leading edge roughness effect on the aerodynamic performance of a NREL S814 airfoil. The distributed roughness has been modeled using a concept of equivalent sand grain roughness height, \( k_s \), through the modified Wilcox \( k - \omega \) turbulence model. As a result, the RANS simulation result showed good correlation with experimental data by predicting the roughness effect at moderate angles of attack. Otherwise, the low-fidelity Xfoil [9] based on the panel method was not able to predict the roughness effect. Bouhelal et al. [10] also conducted RANS-based CFD simulation for the uniform roughness over the entire surface of the MEXICO (Model Experiments in Controlled Conditions) blade. The modified \( k - \varepsilon \) turbulence equation using \( k_s \) was used for the simulation. Overall, the roughness decreases total power of the rotor at multiple wind speeds, and the power loss reached 35% of total for the most severe case.

However, as far as the authors know, there is no simulation study which considered the effect of the surface roughness on the boundary layer transition process in addition to the turbulent boundary layer characteristics. Especially for the case of leading edge roughness, the accurate prediction of transition onset affects the flow history downstream on the blade. Therefore, the use of fully turbulent flow assumption can be inaccurate depending on the flow conditions and roughness height. Therefore, the main objective of the current study is to predict the leading edge roughness effect on the turbine aerodynamic performance by considering the roughness effect on both the transition process and the turbulent boundary layer through RANS simulation.

As a representative HAWT model, the NREL Phase VI wind turbine model is used.

2. Methodology

In the present work, a heterogeneous CPU/GPU computing is used for the wind turbine simulation. A line based solver on unstructured grids on CPUs (HAMSTR) [11] is coupled to a structured grid solver on GPUs (GARFIELD) [12] through a lightweight Python-based framework. These separate in-house flow solvers and the CPU/GPU heterogeneous framework has been previously validated through various rotary wing simulations [13].

2.1. Reynolds-Averaged Navier–Stokes Solver

Reynolds-Averaged Navier–Stokes (RANS) method has been applied in the current simulations, which is widely used for rotary wing aerodynamics as a practical tool for high Reynolds number flow conditions. The computations were performed using in-house flow solvers (HAMSTR and
GARFIELD) which were developed at the University of Maryland [11, 12]. Each flow solver is a parallel solver for the solution of the compressible Navier–Stokes equations based on the finite volume formulation. Roe’s approximate Riemann solver is used to compute the inviscid fluxes at the cell interface. Viscous fluxes are calculated using second-order central difference discretizations. For the unsteady simulation, second-order backwards difference implicit method (BDF2) is used with dual time-stepping to allow for a practical time step size for the large scale simulation. The turbulent boundary layer is modeled using the one equation Spalart–Allmaras (SA) model [14] and a delayed detached eddy simulation (DDES) method ([15]) is integrated to the SA model framework, which provides more accurate prediction in the wake region as a hybrid RANS-LES method.

The near-body domain (turbine rotor) is based on unstructured grids and computed using HAMSTR; otherwise, the off-body domain (background) based on structured grids is computed using GARFIELD. Each flow solver computes the flow solutions independently for its domain. Specifically, HAMSTR uses third-order MUSCL (Monotone Upstream Conservative Limited) scheme for the reconstruction and the DDLGS (diagonally dominant line Gauss-Seidel) method for the implicit operator inversion. GARFIELD uses fifth-order WENO (Weighted Essentially Non Oscillatory) scheme and the DADI (Diagonalized Alternating Direction Implicit) method with up-wind dissipation, respectively. Communication between the multiple codes, HAMSTR and GARFIELD, is accomplished through TIOGA, a Topology Independent Overset Grid Assembler [16]. Each flow solver is wrapped in Python to allow for ease of integration with other codes written in various languages (C and CUDA). Within the Python framework, the communication between flow solvers is performed at a sub-iteration level.

2.2. Laminar-Turbulent Boundary Layer Transition

The use of transition models allows for the inclusion of boundary layer transition effects in the RANS simulations. In the near-body domain, a two equation $\gamma - \text{Re}_{\theta_t} - \text{SA}$ transition model is used, which is coupled with the SA turbulence model. The form of the current transition model is briefly presented in this paper and a detailed description can be found in the Refs. [17, 18]. The baseline transition model can predict natural transition, separation-induced transition, and bypass transition and has been validated through various canonical problems. The transition model uses the concept of intermittency, $\gamma$, in order to trigger transition locally. The intermittency is a scalar transport variable that varies between 0 (pure laminar) and 1 (pure turbulent).

The transport equation for the intermittency is given by:

$$\frac{D(\rho \gamma)}{Dt} = P_{\gamma} - D_{\gamma} + \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \frac{\partial \gamma}{\partial x_j} \right]$$

(1)

This transport equation is based only on the local flow field variables and gradients, along with the nearest wall distance, $d$. Both production and destruction of intermittency terms are included.

The transport equation for transition momentum thickness Reynolds number, $\text{Re}_{\theta_t}$, is used to account for history effects of pressure gradient on determining the onset of transition. This equation is given by:

$$\frac{D (\rho \text{Re}_{\theta_t})}{Dt} = P_{\theta_t} + \frac{\partial}{\partial x_j} \left[ 2.0(\mu + \mu_t) \frac{\partial \text{Re}_{\theta_t}}{\partial x_j} \right]$$

(2)

Once the distribution of $\text{Re}_{\theta_t}$ is solved, the critical momentum thickness Reynolds number, $\text{Re}_{\theta_c}$, is obtained, which determines the value of the vorticity Reynolds number, $\text{Re}_\omega$. Then, the intermittency production can be triggered based on the vorticity Reynolds number. When it is
coupled with the SA turbulence model, the intermittency is used to control only the production term of the transported variable, \( \tilde{\nu} \), in the SA turbulence model as follows:

\[
\frac{D\tilde{\nu}}{Dt} = \gamma P_\nu - D_\nu + \frac{1}{\sigma} \left[ \nabla \cdot ((\nu + \tilde{\nu})\nabla \tilde{\nu}) + c_{\nu 2}(\nabla \tilde{\nu})^2 \right]
\]  

(3)

2.3. Modeling Roughness Effect

The surface roughness effect has been considered in the SA turbulence model by implementing the Boeing roughness model as proposed by Aupoix and Spalart [19]. As a ‘macroscopic’ roughness strategy, the model assumes that the roughness-element size in any direction is small compared with the boundary layer thickness and the exact location of each roughness element is not important. It assumes the roughness induces a shift in the log-layer of the turbulent boundary layer using the equivalent sand grain roughness height, \( k_s \).

As a simple way to impose the shift, the wall distance function from the SA turbulence model is augmented as given by:

\[
d_{\text{new}} = d + 0.03k_s
\]  

(4)

The turbulent viscosity, \( \nu_t \), is linked to the transported variable, \( \tilde{\nu} \), in the SA turbulence model as given by:

\[
\nu_t = f_{e1}\tilde{\nu}, \quad f_{e1} = \frac{\chi^3}{\chi^3 + c_{e1}^3}, \quad \chi = \frac{\tilde{\nu}}{\nu}
\]  

(5)

where the definition of \( \chi \) is modified to achieve good prediction for smaller roughness as given by:

\[
\chi = \frac{\tilde{\nu}}{\nu} + 0.5\frac{k_s}{d_{\text{new}}}
\]  

(6)

The wall boundary condition \( \tilde{\nu}_{\text{wall}} = 0 \) is also replaced for the roughness effect by:

\[
\left( \frac{\partial \tilde{\nu}}{\partial n} \right)_{\text{wall}} = \frac{\tilde{\nu}_{\text{wall}}}{0.03k_s}
\]  

(7)

Surface roughness has been reported to cause an increase in the momentum deficit over the rough sections, which results in an increase of momentum thickness. Since the behavior of the \( \gamma - Re_{\theta t} \)-SA transition model is strongly dependent on the momentum thickness, it becomes crucial to incorporate the effects of surface roughness in the transition modeling. In this study, an additional transport equation for the roughness amplification, \( A_r \), parameter is used which was originally developed to be coupled with \( \gamma - Re_{\theta t} \) model as proposed by [20, 21].

\[
\frac{\partial (\rho A_r)}{\partial t} + \frac{\partial (\rho U_j A_r)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \sigma_{ar} (\mu + \mu_t) \frac{\partial A_r}{\partial x_j} \right]
\]  

(8)

where the diffusion coefficient, \( \sigma_{ar} \), is chosen to be 10; the same as previous references used.

The roughness amplification variable is treated as an additional non-physical quantity that will be produced at rough surface boundaries. The quantity is transported through the flow field by the convection and diffusive terms of Eq. 8. This behavior enables the flow history effects to be taken into account. Through interaction of \( A_r \) with the \( Re_{\theta t} \) transport equation, the transition onset is triggered by reducing \( Re_{\theta t} \). The distribution of \( A_r \) is determined with a boundary condition at rough walls as given by:

\[
A_r = 8.0 \times k_s^+\]

(9)
where the user inputs a non-dimensional equivalent sand grain roughness height.

\[ k_s^+ = \sqrt{\frac{\tau_w}{\rho_w}} k_s \] (10)

Thus, the current model is able to simulate different roughness heights on the object and to capture the effect on transition due to varying roughness height over the surface.

The final production term of the transport equation for transition momentum thickness Reynolds number, \( \overline{Re_{\theta t}} \), is given by:

\[ P_{\theta t} = 0.03 \frac{\rho}{t} \left[ (\overline{Re_{\theta t}} - \overline{Re_{\theta t}})(1.0 - F_{\theta t}) - F_{A_r} \right] \] (11)

where the \( F_{A_r} \) is the correction for roughness-induced transition.

It should be noted that \( \overline{Re_{\theta t}} \) was limited with a minimum value of 20.0 to avoid nonphysical overshoots of shear stress for the cases with high Reynolds number and high roughness height [20].

Once \( A_r \) is solved throughout the entire computational domain, the \( F_{A_r} \) expression is given by:

\[ F_{A_r} = \begin{cases} \frac{c_{A_r}^2}{A_r} \left( \frac{A_r}{C_{A_r}} \right)^3, & \text{if } A_r < C_{A_r} \\ \frac{c_{A_r} C_{A_r}}{A_r} + \frac{c_{A_r}^2}{A_r} \left( \frac{C_{A_r}}{A_r} \right)^3, & \text{if } A_r \geq C_{A_r} \end{cases} \] (12)

The function is switched at \( C_{A_r} = \sqrt{\frac{c_{A_r}^3}{3c_{A_r}^2}} \) to allow for a smooth transition between the cubic and linear functions.

The same model parameters are used as Langel et al. [21] which is given by:

\[ c_{A_r} = 0.0005 \quad c_{A_r^3} = 2.0 \] (13)

3. Validation

3.1. Fully Turbulent Flow over Flat Plate

Fully turbulent flow over a zero pressure gradient flat plate simulation was conducted using the SA turbulence model. For this test case, a structured grid (137 × 97) was used, which was provided at the NASA Turbulence Modeling Resource (TMR) site [22]. Initial wall normal spacing is \( 2 \times 10^{-6} \) which corresponds to \( y^+ = 0.1 \).

Current skin friction predictions for turbulent flow over both smooth and uniform rough surfaces were compared with the experiment by Blanchard and other simulation results using the same model [23] as shown in Fig. 1. The skin friction of turbulent flow over a smooth surface at zero pressure gradient is given by the empirical relation as a function of Reynolds number as follows:

\[ C_f = \frac{0.025}{Re^{1/7}} \] (14)

In the simulation, the equivalent sand grain roughness, \( k_s \), was \( 9.5 \times 10^{-4} \) which is the same value as in the reference. The corresponding roughness Reynolds number, \( Re_{k_s} = U k_s / \nu \), is 4,750. It is observed that the skin friction is increased due to the uniform roughness on the surface compared to the result on the smooth surface. Overall, the current prediction shows good agreement with experiment and other simulation results with rough surfaces. Also, the boundary layer profile was changed due to the roughness and the thickness was increased as shown in Fig. 1 (b).
Figure 1. Surface roughness effect on fully turbulent flow over flat plate at $k_s = 9.5 \times 10^{-4}$.

3.2. Boundary Layer Transition from Surface Roughness

The roughness amplification model in the current work has been validated through the same zero pressure gradient flat plate test case as for the original validation of the model by Dassler et al. [20]. In the experiment, the dependency of transition onset location on the roughness height has been measured. According to Dassler et al. [20], the freestream turbulence intensity (FSTI) was set in a way that the predicted transition onset on the smooth wall is the same as in the experiment because FSTI was not given in the experiment. In the current study, FSTI was set in the same way and 1.0% was chosen for the simulations, which is somewhat higher than the value (0.91%) from Dassler et al. Various equivalent sand grain roughness heights have been applied along the no-slip boundary, which ranges from $Re_k = 0$ (smooth surface) to $Re_k = 381$.

Figure 2 (a) demonstrates the effect of varying $Re_k$ on the skin friction ($C_f$) distribution along the wall. As expected, the transition onset location moves towards upstream as roughness height increases. It should be noted that the transition onset location is indicated by a sharp increase in the skin friction value. The predicted transition onset locations were compared with experiment and other numerical simulation results [20, 21] as shown in Fig 2 (b). The transition onset location, $Re_{st}$ was defined in this case as the point of minimum skin friction as in Dassler et al. [20]. The current model can predict the overall trends of onset location when the results are compared with the scattered experimental data although it seems to predict an earlier transition at lower $Re_k$. Further study is required for better predictions at the lower $Re_k$.

4. Results And Discussion

4.1. NREL S809 Airfoil

The current method has been applied to the two-dimensional S809 airfoil; which is a 21% thick airfoil and the primary section of the NREL Phase VI wind turbine blade. The airfoil was tested at the Ohio State University (OSU) 3m × 5m subsonic wind tunnel under steady and unsteady conditions [24]. In the experiment, the 457 mm (18 inch) chord airfoil model was tested with a thickened trailing edge of 1.25 mm (see Fig. 3 (b)). The leading edge grit roughness (LEGR) was applied on the airfoil. The average grit particle size divided by airfoil model chord length ($k/c$) was 0.0019 which corresponds to $Re_k$ of 315 at the end of the rough surface. It should be noted that the roughness was extended about 11% chord length (51 mm) along both upper and bottom surfaces of the model from the leading edge. In the current simulation, it is assumed that the $k/c$ value in the experiment corresponds to the $k_s/c$ value used for the current model.
The rough surface is applied uniformly on the same amount of extension with the experiment from the leading edge.

In the current simulation, the Reynolds number is 1.5 million based on chord length, which was the largest Reynolds number in the experiment. The FSTI is assumed as 0.05 %. For the two-dimensional computational mesh, the structured O-grid topology is generated with finite thickness trailing edge as shown in Fig. 3. 400 points are used in the wrap-around direction on the airfoil surface, while 200 points are used in the wall-normal direction. The initial wall-normal spacing is $1 \times 10^{-5}$ chord length, which corresponds to $y^+ = 0.8$. The domain is extended for about 100 chord lengths away from the airfoil to eliminate the far-boundary effect.

A grid refinement study for the S809 airfoil is performed by using three different initial wall normal grid spacings as shown in Fig. 4. Based on the current mesh, the initial wall normal grid spacing is either increased or decreased 10 times as the coarser and the finer mesh, respectively. The same grid points in the wrap-around direction is kept because it is sufficiently fine. In both smooth and rough surface simulations, all of the coefficients are converged using the medium resolution mesh which has $y^+ = 0.8$. It is also observed that more difference occurs
in the results for rough surface between the coarse and medium meshes. This indicates that rough surface simulation might require finer wall normal grid spacing than the spacing for clean surface simulation.

![Grid refinement study for clean and rough surface S809 airfoil simulations.](image)

Figure 4. Grid refinement study for clean and rough surface S809 airfoil simulations.

Figure 5 (a) shows the comparison for lift over drag ratio against the angle of attack. For the clean case, the current prediction is well matched with the experimental results in terms of the curve slope at the moderate angles of attack. At the beginning, around 6 to -6 degrees, the current prediction is over-predicted because of the limitation of two-dimensional steady RANS simulation to predict the unsteady trailing-edge flow separation zone on the suction side of the airfoil. For the rough case, both prediction and experiment show the reduced curve slope as well as reduced maximum lift over drag ratio at around 6 degree compared to the results from the clean surface. In Fig. 5 (b), the drag coefficient is increased for the rough case at the current range of angles of attack. This is mostly because the transition onset location moved towards upstream due to the roughness at the leading edge as shown in Fig. 5 (d). The roughness also affects the pitching moment coefficient as shown in Fig. 5 (c). In the prediction, the nose down moment is reduced in the rough case, which agrees with the experimental data. However, the effect of roughness on the moment is under-predicted in the current result which may be due to the assumption of the current sand grain roughness height. It should be noted that the sand grain size \( k_s \) can be several times larger than the real-life roughness to have the same effect on the velocity profile [19].

4.2. NREL Phase VI Wind Turbine Rotor
The NREL Phase VI turbine is a HAWT two-bladed rotor which has a diameter of 10.06 m (32.99 ft). It rotates with a constant rotational speed of 72 RPM. As shown in Fig. 6, the rotor consists of a cylindrical cross-section at the root, and the transition region connects the circular section to the root airfoil section. The blade has a linear taper and has a non-linear twist with the tip pitch angle set at 3°. The pitch and twist axes are located at 30 % chord [6]. In the current simulation, the spinner part of the rotor is also modeled which connects each blade at the center. The surface mesh of the rotor is generated using a hybrid mesh which comprises a total of 98,729 quadrilateral elements. In the main body of the blade region, the structured-type mesh consists of 240 points in the wrap-around direction and 90 points in the spanwise direction. For the spinner part, an unstructured-type mesh is used for efficient mesh generation.

Figure 7 (a) shows the cross-section of volume mesh at 75% span of the blade including both near-body and off-body domains. A total of 51 layers of strands are generated for the near-body domain by extruding the surface node points in the wall normal direction. The initial wall
Figure 5. Comparison of S809 airfoil performance at various angles of attack.

Figure 6. NREL Phase VI turbine rotor surface mesh.

spacing is $1 \times 10^{-5}$ of the root chord length, which corresponds to a $y^+$ value of less than 1. The
dimensions of the background domain is determined based on the NASA Ames wind tunnel test section as shown in Fig. 7 (b). A slip wall boundary is applied to the test section wall faces and a freestream boundary condition is used for the inlet and outlet faces. The Cartesian nested mesh is used to capture the wake structure downstream of the rotor, which is indicated as blue lines. The finest cell spacing is approximately 8% of root chord length.

The near-body rotor domain is rotated at an equivalent computational time step of 1° around the rotational axis and connected with the off-body stationary domains using overset technique. The mesh connectivity between the near-body and nested domains are computed every time step. It should be noted that the flow is steady unless the flow is largely separated on the blade because the interference with the tower is not accounted for in the current simulation.

![Overset mesh system for NREL Phase VI turbine rotor simulation.](image1.png)

(a) Strand near-body mesh at 75% spanwise section

![Overset mesh system for NREL Phase VI turbine rotor simulation.](image2.png)

(b) Cartesian background mesh and nested domain

**Figure 7.** Overset mesh system for NREL Phase VI turbine rotor simulation.

The test condition has zero coning angle with a rigid hub, which corresponds to the test sequence ‘H’ in the experiment test [6]. The wind speed is varied from 7 m/s to 20 m/s. The FSTI value is assumed as 0.05% in all of the simulations. For the rough surface simulations, a similar roughness is applied on the blade as with the two-dimensional S809 airfoil case. The rough surface is assumed only from the leading edge to 10% of the local chord length on both
upper and lower surfaces and the equivalent sand grain roughness height \( (k_s/c) \) is 0.0019. At the three different stations of \( r/R = 0.25, 0.5, \) and 1.0 along the span, the corresponding \( Re_{k_s} \) at the end of the rough surface is 90, 150, and 175, respectively. The remaining part of the blade surface and spinner are assumed as a clean surface. It should be noted that the experiments (sequence ‘H’) were conducted using a clean surface blade.

To study the effect of the leading edge roughness on the turbine performance, integrated rotor thrust and torque predictions are compared both with and without roughness at various wind speeds as shown in Fig. 8 and Table 1. As references, experimental data and other transition flow predictions [3, 4] for the clean surface are compared at the same wind speeds. For the current simulations, the freestream flow condition is applied as an initial condition. From the initial condition, the rotor rotates 3 revolutions for the cases at 7, 10, 15 m/s wind speeds and 4 revolutions for the case at 20 m/s wind speed. The mean rotor thrust and torque values are converged during the last rotor revolution.

Overall, good agreement between the experiment and current prediction from the clean surface is found from 7 to 15 m/s wind speeds. At an operating wind speed of 7 m/s, stall does not occur anywhere on the blade surface. At 10 m/s, the stall starts occurring on the leeward side. At the wind speed of 20 m/s, most of the blade is operating in stall and the rotor thrust and torque are over-predicted by 16 \% and 12 \%, respectively, which is a similar trend with other predictions. This might be due to both limitations of current models and standard deviation of the highly unsteady flow in the experiment. A large change in predicted torque is observed between the clean and rough simulation results at 10 and 15 m/s because the onset of massive flow separation on the blade occurs in this range of wind speeds. Thus, the prediction is very sensitive to small variation [2].

At 7m/s, the performance (torque) of the rotor with roughness is decreased by 8.1\% compared to the result with a clean surface. This trend was also observed in the previous study for the blade with roughness on the entire surface [10]. At 10 m/s, on the other hand, the performance is increased by 23.3 \% due to the roughness. As the wind speed increased further, the roughness effect becomes minor on the performance, because the flow is fully separated from the leading edge.

![Figure 8. NREL Phase VI rotor thrust and torque predictions.](image)

At three different wind speeds, the instantaneous streamlines with skin friction coefficient...
Table 1. Roughness effect on thrust and torque predictions of NREL Phase VI rotor.

| Cases (m/s) | Thrust (N)     | Torque (N-m)  |
|------------|----------------|---------------|
|            | Clean Rough Difference | Clean Rough Difference |
| 7.0        | 1,143 1,104 -3.4 % | 753 692 -8.1 % |
| 10.0       | 1,677 1,805 7.6 %  | 1,405 1,733 23.3 % |
| 15.0       | 2,258 2,273 0.7 %  | 1,136 1,283 12.9 % |
| 20.0       | 3,519 3,459 -1.7 % | 1,248 1,245 -0.24 % |

contour on the leeward side are shown in Fig. 9. At a wind speed of 7 m/s, the presence of a separation bubble around the mid-chord is observed near the blade root and tip regions in the clean case. The transition onset occurs right after the separation and the turbulent flow re-attaches. In the rough case, well attached flow with higher skin friction are observed over the whole surface. This is because roughness-induced transition occurs and the turbulence starts to build-up from the leading edge. The increased region of turbulence results in the loss of turbine performance as shown in Table 1. At a wind speed of 10 m/s, the flow is considerably separated in the clean case, except for the blade tip region. On the other hand, the flow separation is confined to the trailing edge in the rough case. This is also because the turbulent boundary layer from the leading edge roughness has higher resistance than the laminar flow in the clean case. The better performance in the rough case can be explained by this reduced separation region on the leeward side. At the wind speed of 20 m/s, both results of clean and rough cases show complete separation from the leading edge of the blade and the roughness does not affect the solution very much.

Figure 9. Instantaneous streamlines and skin friction magnitude contour on the leeward side of NREL Phase VI blade.
The instantaneous chordwise surface pressure coefficients are compared for both the results from the clean and the rough surface simulation to experimental data (for the clean surface) at three spanwise stations as shown in Fig. 10. The sectional pressure coefficient is defined as:

\[
C_p = \frac{P - P_\infty}{0.5 \rho (W_\infty^2 + (r\omega)^2)}
\]

At a wind speed of 7 m/s, although higher suction pressure in the clean surface is observed over the span, the roughness effect is minor and both predictions are close to the experimental data. At a wind speed of 10 m/s, a noticeable difference is observed at the outer stations. The higher suction peak occurs with the rough surface, which results from the smaller separated region on the leeward side. At 20 m/s, the flow is fully separated on the leeward side and the flat pressure distributions are observed in both simulation results. The effect of the roughness is not consistent along the span. It should be noted that the flow is highly unsteady at 20 m/s and the instantaneous result might not represent the flow features completely.

Figure 10. Chordwise distribution of surface pressure at three wind speeds for NREL Phase VI blade.
The sectional normal and tangential forces are extracted along the spanwise direction as shown in Fig. 11. In this result, only the pressure term is considered for the calculation of sectional forces. At 7 m/s, the rough case shows less sectional forces at the outer stations in both normal and tangential directions. This result corresponds with the reduced turbine performance due to the roughness in Table 1. The opposite result is observed at a wind speed of 10 m/s. At the mid-span where the stall occurs mostly on the leeward side, both normal and tangential forces are increased due to the roughness. This results from the reduced stall region in the rough case. The comparison at 20 m/s is not shown in this study because the flow is highly unsteady and the roughness effect on the flow characteristic is minor.

Finally, the wake structures are visualized using the Q-criterion iso-surfaces colored by vorticity magnitude around the rotor at three different wind speeds. Figure 12 (a) shows the result of the clean surface at 7 m/s wind speed. The blade tip vortex is clearly visible without any flow separation along the span of the blade. Figure 12 (b) and (c) compares the results from the clean and rough surfaces at 10 m/s. The wake from the flow separation at the mid-span is observed in the clean case, otherwise the wake structure is significantly reduced at the mid-span in the rough case. At 20 m/s, the fully separated flow over the span induces highly unsteady wake structures as shown in Fig. 12 (d). The results of the rough case at 7 and 20 m/s are omitted because there is no noticeable difference with the results of the clean case.

5. Conclusion
In this paper, the effect of surface roughness near the leading edge of turbine blade on aerodynamic performance is studied. The flow simulation for a NREL Phase VI HAWT rotor is conducted using in-house RANS flow solvers which are based on both CPUs and GPUs. The laminar-turbulent boundary layer transition on the blade is accounted for using the $\gamma - Re_{\theta t} - SA$
transition model. The surface deterioration acts as distributed roughness on the blade and the localized rough surface effect is modeled through the extension of both turbulence and transition models. The use of wall roughness correction in the SA model induces a shift in the log-layer of the turbulent boundary layer which results in both higher skin friction and a thicker boundary layer. The roughness-induced transition is predicted by incorporating a transport equation for the roughness amplification parameter into the transition model.

First, the current method is applied to a two-dimensional S809 airfoil, which is a primary section of the NREL Phase VI turbine blade. Comparison with experimental data has shown that the current method is able to provide reasonable roughness effects on aerodynamic coefficients; reduced lift-to-drag ratio and less nose-down moment, etc.

Then, the rotating wind turbine rotor has been simulated at various wind speeds from 7 to 20 m/s. For the clean surface, the flow behaves quite differently on the leeward side of the blade within the range of wind speeds. Once the roughness is applied at the leading edge zone, transition onset is triggered due to the roughness. The roughness-induced transition has an opposite effect on the turbine performance at 7 and 10 m/s wind speeds. At 7 m/s wind speed, the torque is decreased in the rough case mostly due to the extended turbulent flow region which...
has higher skin friction than laminar flow. On the other hand, at 10 m/s, the flow separation along the span is noticeably disappeared in the rough case, which results in the higher airloads at the mid-span. At the higher wind speeds, the blade is operating in stall and the effect of roughness on turbine performance becomes minor.

Further study is obviously required to rigorously validate the current model due to the lack of information for computing sand grain roughness height accurately from the experiment. Also, the current approach can be applied to modern wind turbine blades which typically have much larger span than the NREL Phase VI wind turbine and are pitch controlled at high speeds rather than stall controlled, to compare the surface roughness effect on its performance.

Acknowledgments
The authors would like to acknowledge Dr. Roger Strawn (Army AFDD) and Dr. Rajneesh Singh (ARL) for their continued support in HAMSTR development. The transition modeling work was supported under the Vertical Lift Research Center of Excellence Grant at the University of Maryland with Dr. Mahendra Bhagwat as Technical Mentor. The authors would also like to acknowledge the University of Maryland Deep Thought2 computing resources and Johns Hopkins ‘Maryland Advanced Research Computing Center’ made available in conducting the research reported in this paper.

References
[1] Marten D, Lennie M, Pechivanoglou G, Nayeri C and Paschereit C Journal of Engineering for Gas Turbines and Power 138 072601–10
[2] Sorensen N 2002 Wind Energy 5 151–169
[3] Sorensen N 2009 Wind Energy 12 715–33
[4] Aranake A, Lakshminarayan V and Duraisamy K 2012 42nd AIAA Fluid Dynamics Conference and Exhibit (New Orleans, LA)
[5] Jung Y, Jude D, Govindarajan B and Baeder J 2017 North American Wind Energy Academy Symp. (Ames, IA)
[6] Hand M, Simms D, Fingersh L, Jager D, Cotrell J, Schreck S and Larwood S 2001 NREL/TP-500-29955
[7] Simms D, Schreck S, Hand M and Fingersh L 2001 NREL/TP-500-29949
[8] Ferrer E and Munduate X 2009 47th AIAA Aerospace Sciences Meeting (Orlando, FL)
[9] Drela M 1989 Low Reynolds Number Aerodynamics (Norte Dame, IN)
[10] Bouhelal A, Smaili A, Masson C and Cuerr O 2017 25th Annual Conf. of the Computational Fluid Dynamics Society of Canada (Ontario, Canada)
[11] Jung Y, Govindarajan B and Baeder J 2017 AIAA Journal 55 1986–2001
[12] Jude D and Baeder J 2016 Extending a Three-Dimensional GPU RANS Solver for Unsteady Grid Motion and Free-Wake Coupling 54th AIAA Aerospace Sciences Meeting (San Diego, CA)
[13] Jude D, Lee B, Jung Y, Petermann J, Govindarajan B and Baeder J 2018 74th Annual Forum of American Helicopter Society (Phoenix, AZ)
[14] Spalart P and Allmaras S 1992 30th AIAA Aerospace Sciences Meeting (Reno, NV)
[15] Spalart P, Deck S, Shur M and Squires K 2006 Theoretical and Computational Fluid Dynamics 20 181–95
[16] Brazell M, Sitaraman J and Mavriplis D 2016 Journal of Computational Physics 322 33–51
[17] Medida S 2014 Ph.D. thesis University of Maryland
[18] Jung Y and Baeder J 2018 Journal of Aircraft 56 1040–55
[19] Aupoix B and Spalart P 2003 Int. Journal of Heat and Fluid Flow 24 454–62
[20] Dassler P, Kozulovic D and Fiala A 2012 European Conf. on Computational Methods in Applied Sciences and Engineering (Vienna, Austria)
[21] Langel C, Chow R, Dam C, Maniaci D, Ehrmann R and White E 2014 52nd AIAA Aerospace Sciences Meeting (National Harbor, MD)
[22] NASA Langley Research Center Turbulence Modeling Resource http://turbmodels.larc.nasa.gov
[23] Medida S, Baeder J, Nigam N and Chen P 2014 52nd AIAA Aerospace Sciences Meeting (National Harbor, MD)
[24] Ramsay R, Hoffmann M and Gregorek G 1995 NREL/TP-442-7817