Validation of blade-resolved computational fluid dynamics for a MW-scale turbine rotor in atmospheric flow

Christian Grinderslev*, Ganesh Vijayakumar, Shreyas Ananthan, Niels N. Sørensen, Frederik Zahle and Michael A. Sprague

1Department of Wind Energy, Technical University of Denmark, Risø Campus, 4000, Roskilde
2National Wind Technology Center, National Renewable Energy Laboratory, Golden, Colorado, USA
3Computational Science Center, National Renewable Energy Laboratory, Golden, Colorado, USA
E-mail: cgrinde@dtu.dk

Abstract. In this study, simulation results of two different computational fluid dynamics codes, Nalu-Wind and EllipSys3D, are presented for a wind turbine rotor in complex yawed and sheared inflow. The results are compared to measurements from the DanAero experiments, to validate computed pressures and azimuthal trends. Despite different code methodologies and grid setups, the codes agree well in computed pressures and integrated forces along the blade for all blade azimuthal positions, however with some discrepancy in the very yawed case. Additionally, both codes capture well the azimuthal trends and force levels seen in measurements. Investigation into discrepancies shows that expanding grids before the rotor, lead to smearing of the wind profiles, which is likely the main cause of the differences in the results between the codes. Additionally, omission of the ground constraint cause discrepancies in relative velocity seen by the passing blade, due to an over speeding beneath the rotor.

1. Introduction
Simulation of wind turbines operating in atmospheric flow conditions is and has for some time been a widely researched topic, as the influence of shear and yaw are crucial for the estimation of turbine loads and power production. Depending on specific aims of studies, multiple modelling methods are used, from simple blade element momentum (BEM) based codes, to more complex computational fluid dynamics (CFD) actuator disc/line (AD and AL, respectively) methods, to the even more complex rotor-resolved CFD models. For studies of, e.g., wake deficits, the actuator-disc and -line methods are suitable as shown in [1] for non-yawed rigid rotors, as the methods are computationally much more efficient than blade-resolved simulations. If accurate near rotor flow and loads are of interest, however, blade resolved simulations are preferable. These do not depend on predefined airfoil data with empirical corrections for tiploss and 3D effects, as is the case with BEM-based and AD/AL methods [2].

As part of the IEA Wind Task 29 [3], CFD simulations are conducted for the 2 MW NM80 rotor in three different flow conditions, for which experimental measurements from the DanAero field experiments [4] are available for comparison. The aim of the IEA Task 29 is to improve and
validate aerodynamic models for wind turbine design codes at various computational fidelities [3]. In this work, two well-established CFD codes EllipSys3D [5, 6, 7] and Nalu-Wind [8] are used and the results are compared.

Firstly, an idealized axi-symmetric case is conducted followed by two cases considering tilt, yaw and sheared-inflow conditions. Despite using different grid setups and methodologies, good agreement is seen between the simulated results from the codes and also with the experimental measurements.

2. Objectives
The purpose of this study is to validate CFD codes and methods for complex flow conditions and highlight areas of difficulty, which need improvement in the respective codes. The work is part of a larger future study, also part of the IEA 29 task, which will consider turbulent inflow and fluid-structure-interaction in atmospheric flow.

3. Methodology
3.1. Flow solvers
EllipSys3D [5, 6, 7] solves the incompressible-flow Navier-Stokes equations in structured curvilinear coordinates using the finite-volume method with a collocated grid arrangement. The code includes turbulent features through Reynolds-averaged-Navier-Stokes (RANS) and large-eddy-simulation (LES) capabilities. The code is parallel and highly scalable using Message Passing Interface (MPI) and multi-block decomposition, multi-grid method and grid sequencing. EllipSys3D has multiple different convective schemes to choose from, and in the present study the Quadratic Upstream Interpolation for Convective Kinematics (QUICK) scheme is used. For pressure correction, an improved version of the the SIMPLEC algorithm [9] is used to couple the velocity and pressure. Rhie-Chow interpolation is used to avoid odd/even pressure decoupling. Overset capabilities, including grid hole-cutting, are implemented internally in the code (see Zahle et al. [10] for the implementation details).

Nalu-Wind[8] solves the incompressible form of Navier-Stokes equations, on generalized unstructured grids, with appropriate RANS or LES models for turbulence closure. The codebase uses an implicit backward-difference-formula (BDF) family of timestep algorithms with an approximate pressure-projection algorithm. Nalu-Wind uses the linear upwind differencing scheme (LUDS). Overset hole-cutting capabilities are implemented through the TIOGA library [11]. For overset simulations, each equation is solved using a single global linear system where the overset coupling between the meshes are introduced as linear constraint rows. The momentum and scalar transport equations are solved using solvers from Trilinos (Belos, Ifpack2) and the pressure-Poisson system is solved using solvers from hypre (GMRES with BoomerAMG preconditioner). Four Picard iterations are used within each timestep to minimize the error from approximate pressure projection algorithm.

3.2. Computational grids
All simulations are conducted using the overset-grid method to combine purpose-fitted grids (near rotor, wake, and background domain) and to allow relative motion between the rotor and the background domain. To ensure a comparable grid near the blade, both solvers use the same surface representation of the blade along with a hyperbolic O-O grid grown away from the blade. Each blade is \( \approx 40 \text{ m} \) long and is resolved with 128 cells spanwise and 256 cells chordwise per blade. The grid has 96 cells in the normal direction extruding approximately 12 m from the blade surface. The first cell adjacent to the blade is \( 1 \times 10^{-6} \text{ m} \), resulting in a \( y^+ \) less than 1. In this study, it is chosen to not resolve the ground boundary but rather simulate the flow with boundaries placed sufficiently far from the rotor to not influence the forces.
The EllipSys3D setup consists of four grid groups overlapping each other. In the background domain, a Cartesian grid is used, which stretches 2000 m ($\approx 25D$), in the streamwise direction, $D$ being the rotor diameter of $\approx 80$ m, with the rotor placed 800 m ($\approx 10D$) from the inlet. The domain is 1000 m ($\approx 12.5D$) across in width and height, with the rotor placed in the center. The outer domain side boundaries are defined as symmetry boundaries while a velocity inlet is used along with an outlet condition assuming fully developed flow. To resolve the wake, a cylindrical grid is used stretching $\approx 5D$ behind the rotor. Around the aforementioned rotor grid a disc grid is used, which rotates along with the rotor to enable a regular hole-cutting procedure. A total of 31M cells are used in this grid setup.

For the Nalu-Wind setup, only two grid groups are used. The domain consists of an unstructured background grid which stretches 2000 m ($\approx 25D$) in the flow direction, and 1600 m ($\approx 20D$) across, with the rotor placed 800 m from inlet and centered in the cross stream directions. The unstructured grid is attached to a structured cylindrical grid resolving the wake. Overlapping this grid is the aforementioned O-O rotor grid, which matches the one used in the EllipSys3D setup. A total of 26M cells are present in this setup.

3.3. Simulation setups

Three different configurations are considered. Firstly, an idealized axi-symmetric case with no tilt, yaw or shear being applied is considered. Here, measurements used for comparison are based on measurements averaged over a revolution within a time window, where conditions
around the rotor are deemed close to axi-symmetric, i.e., no yaw and very low shear. Two other asymmetric cases with measured conditions are considered. Here, a 5° tilt is included along with yaw and shear. One case has dominating shear and one has dominating yaw. For sheared inflow, a power law corresponding to that found from measurements is used as an inlet condition and initial flowfield in the domain. To constrain the profile in the high domain, a maximum of 12 m/s and a minimum of 4 m/s is used such that a cutoff is made on the profile, as presented in [12] and [13]. The cases conducted are summarized in Figure 3 and a visualization of the shear profiles including cutoff are shown. The Reynolds number of the flow ranges from around $5 \times 10^6$ to $6 \times 10^6$, based on relative wind speed and chord length along the blade. All simulations are conducted as fully turbulent unsteady RANS (URANS) simulations using the \( k - \omega \) SST turbulence model [14]. Time steps are reduced through the simulations until time step insensitivity in thrust and torque is obtained at $6.43 \times 10^{-4}$ sec (0.0625 deg/sec). All simulations have been run for +25 revolutions to develop the wake and converge in thrust and power.

### 3.3.1. Assumptions and uncertainties

In this work, the rotor is considered rigid and the tower, nacelle and spinner are not modelled. The rotor geometry is based on the theoretical geometry of the rotor, meaning that no irregularities on the actual rotor surface are modelled. No ground constraint on the flow is considered either, as the domain expands far from the rotor in all directions. The sheared inflow profile is based on averaged measurements from a met mast placed approximately 200 m (2.5D) from the considered turbine in the DanAero experiments.

| Case | Wind speed at hub height [m/s] | Tilt [°] | Yaw* [°] | Shear exponent \( \alpha \) [-] | Pitch** [°] | Rotor speed \( \omega \) [rpm] |
|------|--------------------------------|---------|----------|-----------------|-----------|-------------------|
| 1    | 6.1                            | 0       | 0        | 0.249           | -0.15     | 12.3              |
| 2    | 9.792                          | 5       | 6.02     | 0.262           | 4.75      | 16.2              |
| 3    | 8.429                          | 5       | 38.34    | 0.262           | 4.75      | 16.2              |

* Yaw is defined as positive when the rotating blade at the top position follows the yawed wind component.
** Pitch is defined as positive when increasing the angle of attack.

### Figure 3.

Left: Velocity profiles used for conducted cases. Black lines represent location of rotor and ground. Right: Input parameters for the three conducted cases.

### 4. Results

#### 4.1. Case 1 - Axi-symmetric flow

For the axi-symmetric case, pressure distributions along the blade are monitored and compared with pressure distributions measured in the DanAero experiment. In Figure 4, the pressure distribution at 33% blade length and 76% blade length are depicted. As seen, visually both EllipSys3D and Nalu-Wind agree well with measured pressure. However, when the pressure is integrated to normal and tangential forces to the rotor plane \( F_{nr} \) and \( F_{tr} \) as seen on Figure 5, larger discrepancies to the measurements appear, especially at the out-board region of the blade. This discrepancy, which at its maximum at 30 m span is approximately 11% for the
mean normal forces, is seen for both EllipSys3D and Nalu-Wind, which mutually agree very well. Reasons for this could likely be differences from modelled geometry to the actual molded geometry of the blade. The lack of modelled blade flexibility will also affect the out-board region the most, and an inclusion of this might twist the blade and change the pressure distribution.

Figure 4. Pressure distributions at 33% and 76% blade length for the axisymmetric Case 1. For clarity, only every third simulation point is shown.

Figure 5. Integrated pressure forces in normal \((F_{n,c})\) and tangential \((F_{t,c})\) directions to the chord plane

4.2. Case 2 - Dominating shear
For the non-symmetric cases, forces are monitored in time for four locations along one blade at blade lengths of 0.33%, 0.48%, 76% and 92%. The forces normal and tangential to the chord depending on azimuthal positions at 0.33% and 92% blade length can be seen on Figure 6 a-b for Case 2 with dominating shear conditions, along with the corresponding measured forces from the DanAero experiment.
As seen, the CFD simulations from both solvers capture well the level of forces and azimuthal trends from the measurements, however with some discrepancy for the outer section of the blade. A phase shift of the minimum between the CFD results and the measurements is seen, especially in Figure 6 b). The CFD results agree on a minimum around $170^\circ$, whereas the minimum is seen around $210^\circ$ for the measurements. The omission of the tower could cause some of the discrepancy, as the present yaw misalignment will shift the tower shadow effect towards higher azimuths. As seen in the EllipSys3D results, the flow is stalling at the inner section at 0.33%. For this reason, the phase average forces are shown for this section, which are matching the measurements. This stall is not present in the results from Nalu-Wind.

Figure 7. Normal ($F_n$) and tangential ($F_t$) force distribution along rotor blade at 0° and 180°, DanAero measurements shown as mean with standard deviation error bars.
Figure 7 depicts the resulting chord normal \((F_{n,c})\) and tangential \((F_{t,c})\) forces based on pressure only along the blade for azimuth positions of 0\(^\circ\) and 180\(^\circ\). As seen the solvers agree well between each other and the DanAero measurements for normal forces. Both solvers, however, tend to overestimate the tangential forces compared to measurements, with a maximum error at the 30.2 m (76\%) section at 0\(^\circ\) azimuth of 30.2\% and 37.9\% for EllipSys3D and Nalu-Wind respectively. This contradicts what was seen for Case 1, where tangential forces were under predicted, as seen on Figure 5. Omission of blade flexibility could likely be the cause to this discrepancy.

4.3. Case 3 - Dominating yaw
As with Case 2, forces are monitored in time for sections at 0.33\%, 0.48\%, 76\% and 92\% blade length, where the first and the last positions can be seen on Figure 8 a-b.

![Figure 8](image)

**Figure 8.** Normal \((F_{n,c})\) and tangential \((F_{t,c})\) forces on blade at 0.33\% (a) and 92\% (b) blade length across azimuthal positions, for Case 3 with dominating yaw. DanAero measurements shown as mean with standard deviation error bars.

The two codes agree on the overall level of forces, however, a large difference in amplitudes of the azimuthal fluctuation is seen. This is emphasized in Figure 9, which depicts the resulting pressure normal \((F_{n,c})\) and tangential \((F_{t,c})\) forces in the chord plane along the blade for azimuth positions of 0\(^\circ\) and 180\(^\circ\). The solvers do not agree between each other and the DanAero measurements at 180\(^\circ\) azimuth. Here the EllipSys3D results overshoot the forces and the Nalu-Wind results on the contrary undershoot the forces. For normal forces, the DanAero results seem to lie between the solvers, whereas for tangential forces, the Nalu-Wind results fit better.
At the 180° azimuth position, the blade sees a lower incoming wind speed due to wind shear. However, at the same position the blade motion component is against the wind direction, which will result in a larger relative wind speed.

![Figure 9. Normal (F_{n,c}) and tangential (F_{t,c}) force distribution along rotor blade at 0° and 180°, DanAero measurements shown as mean with standard deviation error bars.](image)

To investigate the discrepancy further, the velocity profiles at several positions upstream of the rotor have been extracted from the computed flow fields for comparison, as presented on Figure 10. The velocity profiles indicate, that the used grid setups influence differently the shear development through the domain. For the Nalu-Wind unstructured setup, the rapid expansion of unstructured cells outside the cylinder domain seems to cause a smearing of the profile. The smearing is seen already close to the inlet at 10D in front of the rotor and increases when moving closer to the rotor. This is especially clear in the lower part of the prescribed profile, Figure 10 (right). Here, it is seen that the smearing of the Nalu-Wind shear leads to lower velocities seen by the blade when passing 180°, explaining the lower forces here. The grid setup for EllipSys is structured and less expanding which seems to stabilise the profile through the domain. However, when looking close the rotor a speed up is seen in the lower altitudes, which is likely due to the omission of a ground constraint. This explains the high loading seen when passing 180°. At the top position of the blade, the effects of smearing and speed-up are low, as to why the results show better agreement in normal forces with each other and DanAero here.

![Figure 10. Extracted wind speed profiles at 0.5D, 4D and 10D in front of the rotor along the flow direction, for Case 3. Left: The profile in a vertical range of -100 m to 200 m from the hub. Right: Zoom in on the lower part of the profiles. Hub is located at y=0 m](image)
Looking at pressure distributions of the section at 76% blade length for four azimuth positions, the discrepancies are also clear. At 180° the suction is under-predicted by Nalu-Wind and over-predicted by EllipSys. At 0 and 270° the codes compare well, and a good agreement is also seen with measurements.

![Figure 11](image)

**Figure 11.** Modelled and measured pressure coefficient $C_p$ distribution at 76% blade length at different azimuthal positions of 0, 90, 180 and 270°

### 4.4. Recap

For two of the three considered cases, being uniform and dominantly sheared inflow the two codes compare well. This is also seen in the predicted rotor thrust and power, as presented in Table 1. As seen, both solvers predict a higher power than measured for cases 1 and 2, which is, at least partly, due to the fact that the measured power is electrical, while the predicted power is mechanical. This means that no power loss through the gears and generator (usually 5-8%) is considered in simulation results. For Case 3, Nalu-Wind underestimate the power production while EllipSys3D over estimates it. This aligns well with the aforementioned discrepancies in this case, where wind speed is underpredicted in Nalu-Wind due to an altering wind profile, and an overspeed beneath the rotor seen in EllipSys3D.

| Case | Thrust [kN] | Power [kW] | DanAero* |
|------|-------------|------------|----------|
| 1    | 93.88       | 303.72     | -        |
| 2    | 267.51      | 1252.9     | -        |
| 3    | 222.3       | 652.3      | -        |

Table 1. Torque and power. $\mu = \text{mean}$ and $\sigma = \text{standard deviation}$. *Note that power from CFD relates to mechanical power, whereas DanAero relates to measured electrical power.
5. Conclusions

Three flow conditions, one idealized axi-symmetric flow case and two with atmospheric sheared, yawed and tilted flow, have been studied using two different CFD codes, EllipSys3D and Nalu-Wind. Measurements from the DanAero experiments are used to define flow conditions and measured pressure and resulting forces are used for comparison and validation of the CFD codes. In general, good agreement is seen between the simulations of the two codes and also between simulations and measurements, especially considering assumptions omitting blade flexibility, tower shadow, and the constraint to cross flow from the ground. Despite these simplifications, both pressure distributions and forces depending on azimuthal positions, found by simulations, match with measurements in both levels and trends.

Some difference is seen, however, in terms of forces along the blade at varying azimuths, especially in the combined case with high yaw and shear. This is likely due to the shear development throughout the domain of the unstructured setup of Nalu-Wind along with the acceleration of the flow beneath the rotor seen in the EllipSys3D results, as a result of omitting the flow constraining ground condition.

Acknowledgments

The DanAero projects are funded partly by the Danish Energy Authorities, (EFP2007. Journal nr.: 33033-0074 and EUDP 2009-II. Journal nr 64009-0258) and partly by eigenfunding from the project partners Vestas, Siemens, LM, Dong Energy and DTU.

This work is authored in part by the National Renewable Energy Laboratory, operated by Alliance for Sustainable Energy, LLC, for the U.S. Department of Energy (DOE) under Contract No. DE-AC36-08GO28308. Funding provided by the U.S. Department of Energy Office of Energy Efficiency and Renewable Energy Wind Energy Technologies Office. The views expressed in the article do not necessarily represent the views of the DOE or the US Government. The U.S. Government retains and the publisher, by accepting the article for publication, acknowledges that the US Government retains a non-exclusive, paid-up, irrevocable, worldwide license to publish or reproduce the published form of this work, or allow others to do so, for U.S. Government purposes. A portion of this research was performed using computational resources sponsored by the Department of Energy’s Office of Energy Efficiency and Renewable Energy and located at the National Renewable Energy Laboratory.

References

[1] Troldborg N, Zahle F, Réthoré P and Sørensen N 2015 Wind Energy 18 1239–1250
[2] Ehrich S, Schwarz C, Rahimi H, Stoevesandt B and Peinke J 2018 Applied Sciences 8 2513
[3] IEA 2019 Task 29 URL: https://community.ieawind.org/task29/29workplan
[4] Bak C, Madsen H, Paulsen U S, Gaunaa M, Sørensen N, Fuglsang P, Romblad J, Olsen N, Enevoldsen P, Laursen J and Jensen L 2010 European Wind Energy Conf. 2 792–836
[5] Michelsen J 1992 Basis3D - A Platform for Development of Multiblock PDE Solvers. Tech. rep. Risø National Laboratory
[6] Michelsen J 1994 Block Structured Multigrid Solution of 2D and 3D Elliptic PDE's. Tech. rep. Technical University of Denmark
[7] Sørensen N 1995 General purpose flow solver applied to flow over hills. Ph.D. thesis Risø National Laboratory
[8] Sprague M, Ananthan S, Vijayakumar G and Robinson M 2019 ExamWind: A multi-fidelity modeling and simulation environment for wind energy Proc. of NAWEA WindTech vol 1452 (IOP Publishing) p 012071
[9] Shen W, Michelsen J, Sørensen N and Sørensen J 2003 Numerical Heat Transfer Part B 43 221–239
[10] Zahle F, Michelsen J, Sørensen N and Johansen J 2009 Wind Energy 12 594–619
[11] Sitaraman J 2019 Tioga URL: https://github.com/jsitaraman/tioga
[12] Troldborg N, Sørensen N, Mikkelsen R and Sørensen N 2014 Wind Energy 17 657–669
[13] Zahle F and Sørensen N 2008 Rotor aerodynamics in atmospheric shear flow Ewec 2008 Conf. Proc. pp 120–124
[14] Menter F 1993 Zonal two equation kappa-omega turbulence models for aerodynamic flows Proc. of 24th AIAA Fluid Dynamics Conf. Paper no. 93-2906 pp 1–21