Numerical simulation of propeller hydrodynamics using the open source software

Andrey Britov[1][0000−0002−9930−4594], Sofya Yarikova[1][0009−0006−8936−7088], Andrey Epikhin[1][0000−0002−6249−4283], Stepan Elistratov[1,2][0000−0002−7006−6879], and Qin Zhang[3][0000−0001−5010−9715]

[1] Ivannikov Institute for System Programming of the RAS, Moscow
[2] Shirshov Institute of Oceanology of the RAS, Moscow
[3] Ocean University of China, Qingdao

Abstract. The paper presents the results of numerical simulation of the propeller Ka4-70 using the actuator line model in the OpenFOAM, AMReX and Nek5000 open-source software. The modifications of the tools for wind farm simulation for these packages are carried out. Features of these implementation are described. For numerical calculations the LES and IDDES turbulence models are used. A comparison of the computational costs and accuracy of flow structures are made for the actuator line model using different methods and the arbitrary mesh interface approach. The actuator line model provides force characteristics and flow structures with good enough accuracy.

Keywords: propeller · thrust forces · wake dynamics · OpenFOAM · AMReX · Nek5000

1 Introduction

Enterprises engaged in design and development of various propellers are constantly facing the problems of their numerical modeling and characterization. These problems can be solved using different numerical approaches [1, 2]. In mid-90's sliding mesh interfaces method was developed [3]. This approach turned out to be applicable in many of scientific studies related to rotational motion of solid bodies. Especially wide sliding mesh techniques are being applied in propellers development. Steijl et al. [4] proposed a method for the study of helicopter rotor-fuselage interaction by third-order sliding mesh on block-structured mesh. This technique was successfully applied in LES simulations of tidal-stream turbines, showing great agreement with experimental power and thrust predictions [5]. Ramirez et al. [6] presented a new technique to maintain the high-order stencil across the sliding interfaces. Those solutions are presented in the vast majority of CFD-toolboxes, in particular in OpenFOAM [7] via AMI and GGI techniques [8–10]. However, further investigations showed that using such techniques demand a lot of computational resources [11] . In case of propeller work simulation it turns out that it's enough to replace the propeller's geometry with a simplified model saving computational resources but providing fine accuracy of calculations. In

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_23

order to reduce computational time, several researchers used panel method [12] or the so-called hybrid models which combine CFD solver and blade element model (BEM) [13]. In this type of modelling the aerodynamic forces applied to a blade do not result directly from CFD, but are calculated separately using inflow data and blade geometry. These calculations are carried out in parallel with the numerical simulation. The blade forces are calculated at each iteration and are implemented as source terms in the flow. There are three hybrid models depending on the distribution of the source terms: actuator disk, actuator line and actuator surface [14].

The simplest hybrid model is the actuator disk that replaces the wind turbine rotor with a thin disk volume. Typically the disk has a diameter equal to that of the rotor and its blade forces affected on the fluid are replaced by equivalent source terms. This model related to wind turbines was introduced in by Sørensen & Myken in [15] using axisymmetric Euler solver. In this study the rotor is replaced with equivalent constant intensity sources. This intensity is calculated from the wind turbine thrust. Actuator disk model neglects separate blade geometry details. However, geometry and viscous flow around blades are not defined. The occupied swept area of the rotor is replaced with distributed source terms instead [16]. Three-dimensional calculations using actuator disk were presented by Amara et al [17] in cases of isolated and clustered wind turbines. Although, for detailed representation of near wake or blade tip vortices, a three-dimensional model for each blade must be used.

Sørensen in [18] proposed so-called actuator line model (ALM). In accordance with it a blade is discretized by finite set of points (elements). Drag and lift forces are calculated for each element taking into account local attack angle and relative velocity. Blade elements themselves are defined by aerodynamic and geometric characteristics. Aerodynamic performances corresponds the foil, made by intersection of blade in the point where the element is situated. Geometrical parameters are derived from the geometry of analyzed foil. For each blade element, relative flow velocity and angle of attack are computed using fluid velocity and element radial one. The comparison of the actuator Line model with the experimental data reveals the effectiveness of this model for wind turbines' power characteristic calculations. The reliability of the model for representing near and far wakes was proved in [19, 20].

More complete and complicated hybrid model is the actuator surface model one applied by Dobrev & Massouh [21] and Shen [22] et al. The main advantage of this model is more physically realistic force distribution along the blade. In the actuator surface model the blade geometry is represented by a surface formed by chord lengths distribution at different radial locations of a blade [23]. In fact the actuator surface model considered as an extend actuator line model where blade are no more represented as thin lines but expand along the chords to the surfaces immersed in the flow. According to [13] the distribution of the calculated forces on blade chords improved structure of induced velocities near the wake. Rotating blade effect is represented by pressure and velocity discontinuity, which related to a circulation around the aerodynamic foil. Thus, velocity and pressure

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_23

gradients created by the actuator surface becomes very close to a real turbine rotation case improving the initial conditions for the wake.

Based on computational resources and calculation accuracy, it's expedient to carry out further investigations using actuator line model.

2 Numerical method

2.1 Governing equations

The complex flow fields around the propeller are obtained by solving the 3D incompressible Navier-Stokes equations. For the incompressible single-phase Newtonian turbulent flows, the filtered mass and momentum equations are the following:

$$\begin{cases} \nabla \cdot \boldsymbol{u} = 0, & (1) \\ \frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u} \cdot \nabla)\boldsymbol{u} = -\frac{1}{p} + \nu \nabla^2 \boldsymbol{u} + \boldsymbol{F}_{turb}. & (2) \end{cases}$$

Where $\boldsymbol{u}$ is a three-dimensional velocity vector, t – time, p – pressure, ∇ – del operator, ν – coefficient of kinematic viscosity, $\boldsymbol{F}_{turb}$ – source term.

Simple explanation of actuator line model is presented on Fig.1. Propellers blades are replacing by lines consisting from finite number of elements.

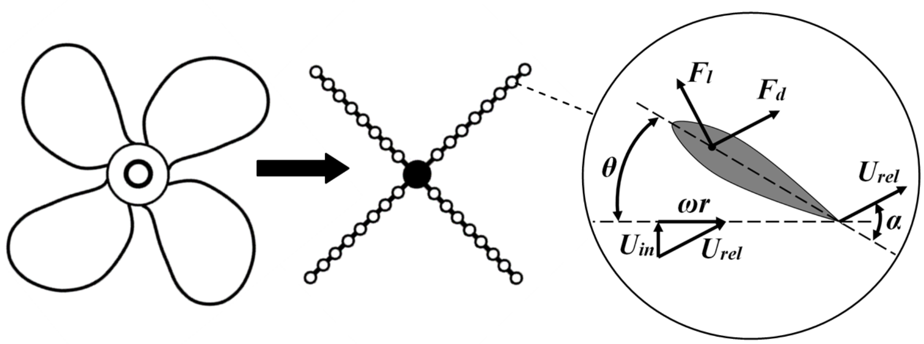


Fig. 1: Actuator line model scheme

Force and moment characteristics for each actuator line element are calculated via the following formulas [24]:

$$F_l = \frac{1}{2} A_{elem} C_l(\alpha) |\boldsymbol{u}_{rel}|^2, \quad (3)$$

$$F_d = \frac{1}{2} A_{elem} C_d(\alpha) |\boldsymbol{u}_{rel}|^2, \quad (4)$$

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_23

$$M = \frac{1}{2} A_{elem} r C_m(\alpha) |\boldsymbol{u}_{rel}|^2, \quad (5)$$

$$\boldsymbol{u}_{rel} = \boldsymbol{u}_{in} + \boldsymbol{w} r. \quad (6)$$

Where A_{elem} – element square, $C_l(\alpha)$ – lift coefficient, $C_d(\alpha)$ – drag coefficient, $C_m(\alpha)$ – pitching moment coefficient, $\boldsymbol{u}_{in}$ – inflow velocity vector, ω – angular velocity of the turbine, r – radius of element.

For each element drag force, lift force and momentum are calculating respectively to local velocity, angle of attack (α) and force coefficient. Every element has aerodynamic and geometric characteristics (chord, twist (θ)). Aerodynamic performances are corresponding to foil in section, where element is placed, assuming that lift and drag coefficients are known. Geometrical characteristics is defined by geometry of corresponding foil.

After the force on the actuator line element from the flow being calculated, it is then projected back onto the flow field as a source term in the momentum equation. To avoid instability due to steep gradients, the source term is tapered from its maximum value away from the element location by means of a spherical Gaussian function [25]:

$$\eta = \frac{1}{\epsilon^3 \pi^{\frac{3}{2}}} \exp\left[-\left(\frac{|\boldsymbol{r}|}{\epsilon} \right)^2 \right], \quad (7)$$

$$\epsilon_{mesh} = 2 C_{mesh} \Delta x, \quad (8)$$

$$\Delta x = \sqrt[3]{V_{cell}}. \quad (9)$$

Where r is a distance from the actuator line element quarter-chord location, ϵ – regularization parameter, V_{cell} – cell volume, Δx – cell length, C_{mesh} - coefficient taking into account the unevenness of the cell faces.

Thus, the source term F_{turb} in momentum equation can be obtained as the following:

$$\boldsymbol{F}_{turb} = (\boldsymbol{F}_l + \boldsymbol{F}_d) \otimes \eta. \quad (10)$$

2.2 Open-source software

OpenFOAM The turbinesFoam library is used for numerical simulation of the propeller hydrodynamics [24, 26]. This library was developed to model wind and marine hydro-kinetic turbines in OpenFOAM using the actuator line method, which was written as an extension library, using the fvOptions functionality for adding source terms to equations at run-time. This allows the ALM to be added to many of the standard solvers included in OpenFOAM without modification. In the original turbinesFoam library the propeller rotation speed is calculated using parameterized coefficient tip speed ratio and inflow velocity. The library is modified in order to being able to run the propeller hydrodynamics simulation in a generator mode (zero flow velocity). If the inflow velocity is different from zero, the original version of the turbinesFoam library is used. Otherwise, rotational

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_23

speed is set by user. The ability to continue the calculation after its suspension is also added. For the numerical simulation of propeller hydrodynamics PIMPLE algorithm is used. The algorithm is implemented in pimpleFoam solver in OpenFOAM. The IDDES turbulence model is used [27]. The LES approach is used in the free stream area and the Spalart - Allmaras turbulence model [28] is used near the hub wall. The filteredLinearM difference scheme proposed in [29] is applied for convective flux discretisation.

AMReX The AMR-Wind [30] library is used to investigate propeller performances based on AMReX framework [31, 32]. AMR-Wind is a parallel, block-structured, finite volume method, incompressible flow solver for wind turbine and wind farm simulations specialized for efficiency and scalability. The solver is built on top of the AMReX library which provides the mesh data structures, performance portable parallel algorithms compatible with different GPU architectures, linear solvers which are a combination of geometric and algebraic multigrid solvers. AMReX supports the development of adaptive mesh refinement (AMR) algorithms for solving systems of partial differential equations, in simple or complex geometries. AMR reduces computational costs and the amount of memory compared to a uniform mesh, while maintaining accurate descriptions of different physical processes in complex multi-physics algorithms. AMReX provides support for both explicit and implicit mesh discretization algorithms. Summing up, AMR-wind achiving the following advantages: an open, well-documented implementation of the state-of-the-art computational models for modeling propellers flow physics at various fidelity's. The numerical simulation is based on the LES approach and actuator line method, which used forces obtained from the OpenFOAM solution.

Nek5000 Nek5000 is an open-source spectrum-element based method [33]. The method unites advantages of finite-element methods and those of spectral element ones. The joining of the solutions in each element through the element edges is made via 'overlap' of the finite element solutions in adjacent element (Lagrangian joining) which allows to calculate solution with 8 order accuracy. The method finds the solution as a Legendre polynomial series on a grid consisting of Gauss-Lobbato-Legendre points. The velocity field obtained can be legally interpolated to a user grid due to high order of the solution. The feature is that the elements used might not be orthogonal, that allows it to be applied to a wider problem class. The main disadvantage is the inability to set pointwise force because of absence of element in sense of which finite element use it (Nek's element are really huge in actuator-line step scales), which make smoothing an obligation. Another one follows the high-order and precision of the method and is a gross time of the calculation (about a week for a case).

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_23

3 Numerical setup

The rotation of propeller Ka4-70 [34] from Wageningen series with propeller diameter $D = 0.1$ m is considered. Rotational speed of propeller is 500 RPM. Inflow velocity is equal to 0.05 m/s.

The simulations are performed inside a domain (height is 10D, wide is 6D and full length equal to 2D + L, where L is local refinement length equal to 3D or 10D) presented in the Fig.2. It is similar to size that used in Ocean University of China (OUC) numerical simulation [35]. For 10D distance simulation an open tank is proposed.

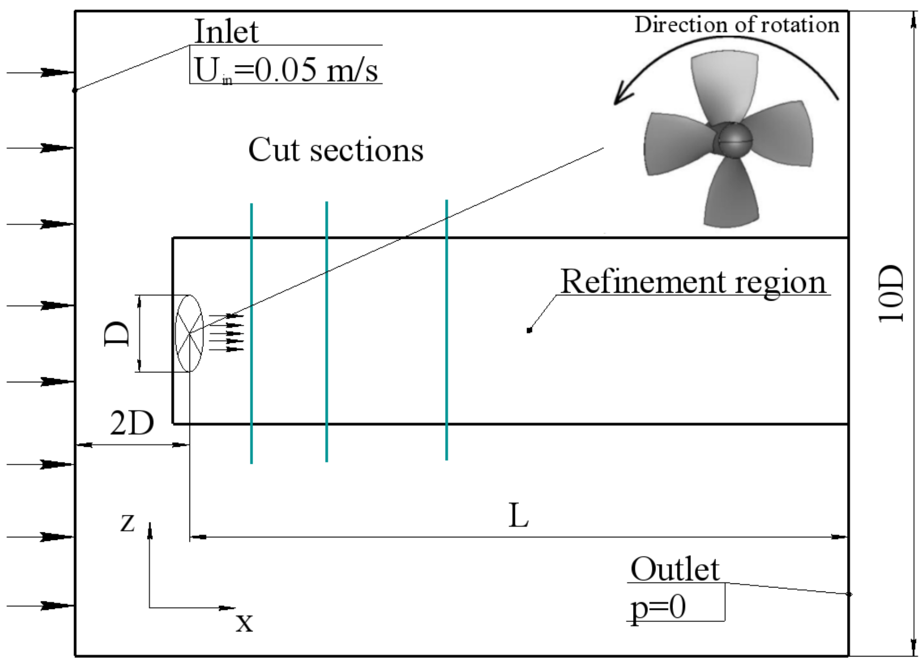


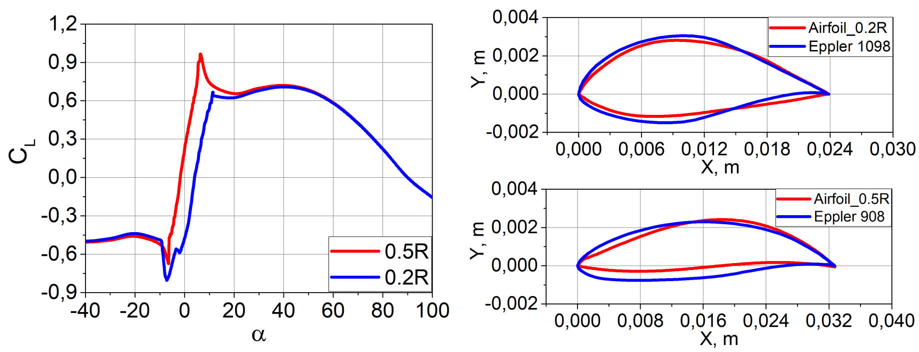
Fig. 2: Computational domain scheme with initial and boundary conditions

Number of actuator line elements is 16. One of the main complexities of ALM applying is the determination of the aerodynamic coefficients of an element. The most accurate method to solve the problem is to capture the foils by cylindrical intersections with subsequent numerical simulation, but this approach is too time consuming. An alternative to this method is using a foil database, such as airfoiltools.com [36], where are drag and lift coefficients for airfoils that are geometrically similar to those obtained by blade model dissecting. Disadvantage of the foil database resources is the lack of data for high angles of attack. For eliminate this problem it's proposed to use the Viterna method of extrapolation [37]. The main time costs are associated with the preparation of data for each

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_23

airfoil. Further, they can be reduced by using simpler methods for aerodynamic characteristics determination [38].

An example of the lift coefficient dependence from angle of attack obtained using Viterna extrapolation method and a comparison of the real foil and a similar one from airfoiltools.com are presented on Fig. 3.



(a) (b)

Fig. 3: Blade intersection: a) lift coefficient from angle of attack b) comparison between real airfoil and similar from airfoiltools.com

The main goal of the actuator line is to replace real screw with the mass force in the right part of Navier-Stocks equation. For the AMReX and Nek5000 simulation the set of 16 pointwise forces per blade obtained from the OpenFOAM solution are used. To convert them into the mass forces they are smoothed by gaussian bell by formula:

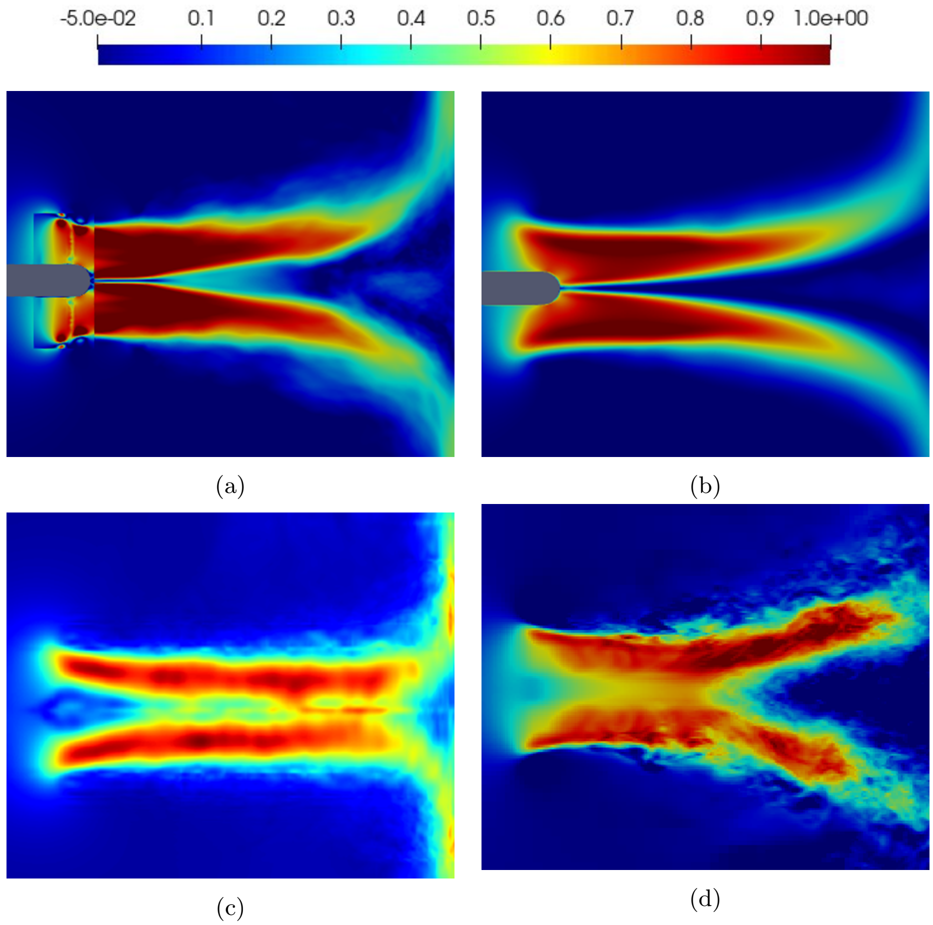
$$f(x,y,z) = \sum_i f_i N(\epsilon) exp\left(-\frac{(x-x_i)^2+(y-y_i)^2+(z-z_i)^2}{2\epsilon^2}\right), \qquad (11)$$

where $N(\epsilon)$ is a norm of this bell. Here (x_i, y_i, z_i) is a point where the current poinwise force is applied. Hence the screw rotates, these points are also rotate around it's center. Additionally, we must sum the forces by both points and blades. Finally, this force is added into Navier-Stocks equation.

4 Results and discussions

Figures 4 and 5 show mean velocity flow structures behind the propeller for different distances. Comparison of the results obtain from different CFD-toolboxes with the implementation of hybrid models and the arbitrary mesh interface method [35] is presented on Fig. 6-7.

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_23



(a) (b) (c) (d)

Fig. 4: Mean velocity field on 3D: a) OUC numerical simulation;OpenFOAM; c) Nek5000; d) AMReX

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_23

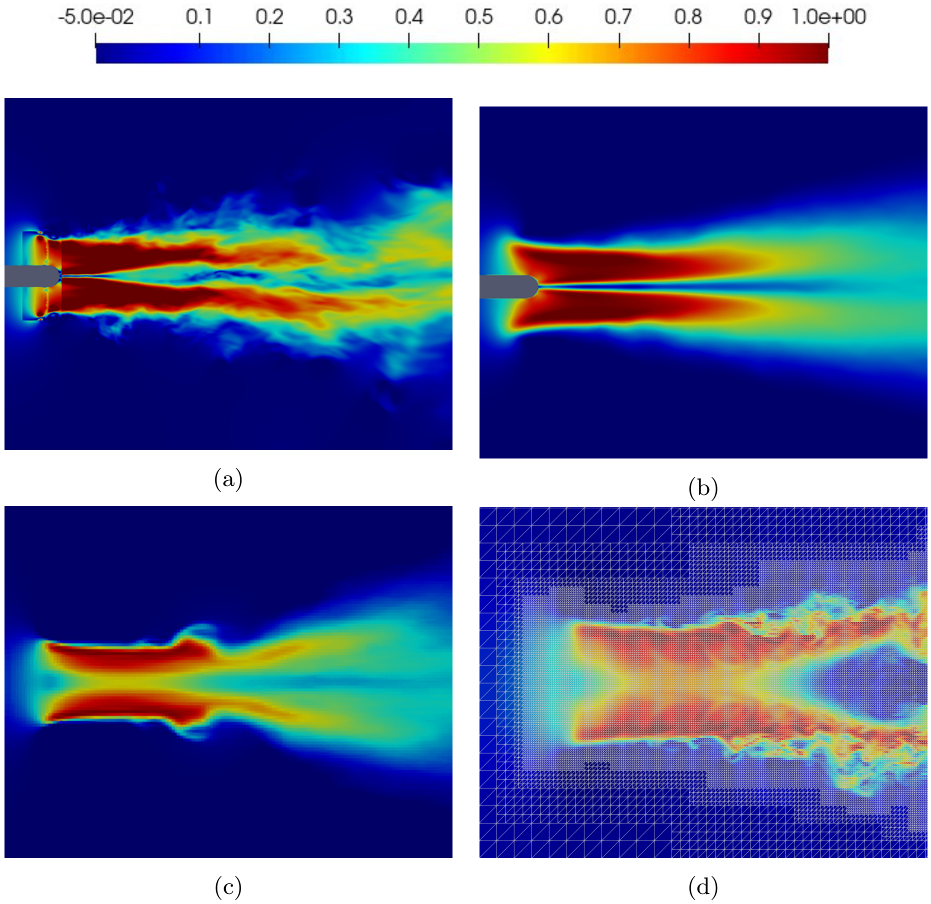
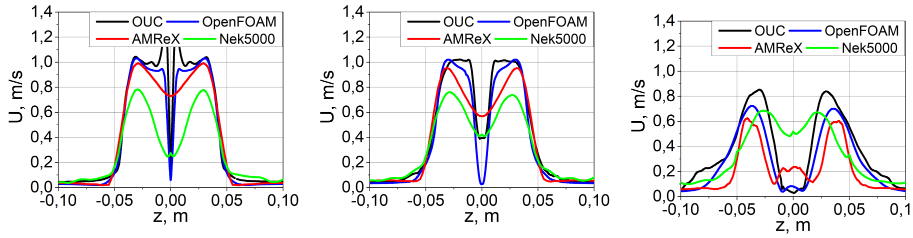


Fig. 5: Mean velocity field on 10D: a) OUC numerical simulation; b) OpenFOAM; c) AMReX; d) Adaptive mesh refinement

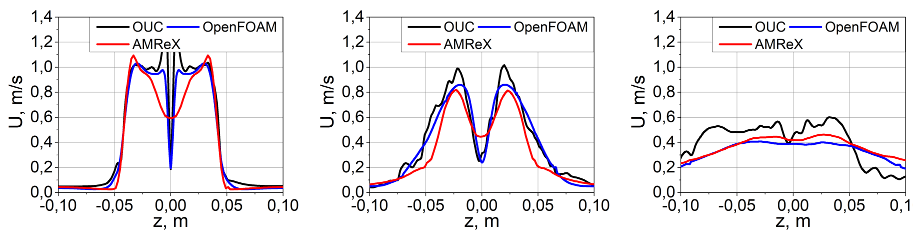
Flow structure generated by propeller has four areas: near-wake, transition, far-wake regions and near-wall depending from degree of destabilization.The near-wake region is close to the propeller disk (0.5D), there is an area with maximum values of velocity. Beside, this is the region, where hub‘s vortexes starting to affect on the flow. It should be noted that the actuator model cannot reproduce the effect of a rigid body on the flow, so propeller hub must be included in the grid. The area where the vortexes lose their original morphology and interact with outer flow is the transition area (1D). Downstream, velocity profile lose its charge, vortexes are broken down into small-scale, disordered turbulence in the far-wake region (2D and more). For cases with 3D distance between propeller and wall, far wake region transforms into the near-wall region, where propellers wake vortex structure interacting the wall.

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_23



(a) (b) (c)

Fig. 6: Comparison of mean velocity at distance: a) 0.5D; b) 1D; c) 2D from propeller



(a) (b) (c)

Fig. 7: Comparison of mean velocity at distance: a) 0.5D; b) 1D; c) 2D from propeller

The obtained results show that the actuator line model represents propeller hydrodynamics correctly. The main difference between maximal velocities values can be caused by inaccuracy in initial data determination. Also, the difference in the flow structure can be related to the small field averaging time in [35].

Different simulation approaches show the similarity flow structures. For all calculations 36 cores are used. Table 1 includes main calculation characteristics.

Table 1: Calculation characteristics.

	AMR-Wind	turbinesFOAM	Nek5000
t_{iter}, sec	6	15	3.5
number of cores	36	36	36
number of cells	15 000 000	10 000 000	10 000

To compare the efficiency of CFD software it's necessary to convert obtained calculation characteristics into the calculation time per core per computational cell. Thus, applying of actuator line model on block-structured mesh in AMReX

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_23

software led to a reduction of computational time by about 4 times compared with the unstructured mesh used in OpenFOAM. Comparison of hybrid models implementation in OpenFOAM and Nek5000 show that spectrum-element based method with actuator surface model and 8-order scheme increases the computational time by approximately 235 times. Despite the lower performance (in comparison with finite volume methods) Nek5000 carries out the calculations with much higher order and represent the solution as the set of basis polynomials rather than the set of point values, allowing to reproduce small-scale turbulence that can be investigated in post-processing after being interpolated on a fine grid. Such small vortices cannot be detected by finite volume methods because they are limited by their mesh step. Another advantage of Nek5000 is the ability of accurate calculation of spatial derivative, which can be necessary for further problem development (e.g., sedimentation problem). The main cause of the divergence between wake structure obtained via OpenFOAM and Nek5000 must be insufficient grid resolution in vertical direction. That's why only in a horizontal one wakes are similar. Mesh refining in this direction must yield a more proper results, but requires additional computational resources.

5 Conclusion

Investigation of applying actuator line model for numerical simulation of propeller Ka4-70 are carried out. The finite volume method (OpenFOAM), finite volume method on block-adapted mesh (AMReX) and spectrum-element based method (Nek5000) are compared. The results show that the application of actuator line model saves computing resources and reproduces the characteristics and hydrodynamics of the propeller with sufficient accuracy. For modelling of instantaneous characteristics in the far field it's advisable to use adaptive mesh approach. However, for detailed representation of near wake Nek5000 using is more efficiency, while for determining mean fields and force and moment characteristics it's preferably to use OpenFOAM.

Acknowledgements The reported study was funded by Russian Foundation for Basic Research (RFBR, Proj. No. 21-57-53019) and National Natural Science Foundation of China (NSFC, Proj. No. 52111530047).

References

1. Bachler, G., Schiffermüller, H., Bregant, A.: A parallel fully implicit sliding mesh method for industrial cfd applications. In: Jenssen, C.B., Andersson, H.I., Ecer, A., Satofuka, N., Kvamsdal, T., Pettersen, B., Periaux, J., Fox, P. (eds.) Parallel Computational Fluid Dynamics 2000, pp. 501–508. North-Holland, Amsterdam (2001). https://doi.org/10.1016/B978-044450673-3/50129-9
2. Paulo A.S.F. Silva, Tsoutsanis, P., Antoniadis, A.F.: Simple multiple reference frame for high-order solution of hovering rotors with and without ground effect. Aerospace Science and Technology **111**, 106518 (2021). https://doi.org/10.1016/j.ast.2021.106518

3. Rai, M.M.: A conservative treatment of zonal boundaries for euler equation calculations. Journal of Computational Physics **62**(2), 472–503 (1986). https://doi.org/10.1016/0021-9991(86)90141-5
4. Steijl, R., Barakos, G.: Sliding mesh algorithm for cfd analysis of helicopter rotor–fuselage aerodynamics. International Journal for Numerical Methods in Fluids **58**(5), 527–549 (2008). https://doi.org/10.1002/fld.1757
5. McNaughton, J., Afgan, I., Apsley, D.D., Rolfo, S., Stallard, T., Stansby, P.K.: A simple sliding-mesh interface procedure and its application to the cfd simulation of a tidal-stream turbine. International Journal for Numerical Methods in Fluids **74**(4), 250–269 (2014). https://doi.org/10.1002/fld.3849
6. Ramírez, L., Foulquié, C., Nogueira, X., Khelladi, S., Chassaing, J.-C., Colominas, I.: New high-resolution-preserving sliding mesh techniques for higher-order finite volume schemes. Computers Fluids **118**, 114–130 (2015). https://doi.org/10.1016/j.compfluid.2015.06.008
7. Jasak, H., Jemcov, A., Tukovic, Z.: OpenFOAM: A c++ library for complex physics simulations (2013)
8. Chandar, D., Gopalan, H.: Comparative analysis of the arbitrary mesh interface(AMI) and overset methods for dynamic body motions in OpenFOAM. (2016). https://doi.org/10.2514/6.2016-3324
9. Vilfayeau, S., Pesci, C., Ferraris, S., Heather, A., Roesler, F.: Improvement of arbitrary mesh interface (AMI) algorithm for external aerodynamic simulation with rotating wheels. Fourth international conference in numerical and experimental aerodynamics of, road vehicles and trains (Aerovehicles 4),Berlin, Germany, August 23-25, 2021
10. Nuernberg, M., Tao, L.: Three dimensional tidal turbine array simulations using OpenFOAM with dynamic mesh. Ocean Engineering **147**, 629–646 (2018). https://doi.org/10.1016/j.oceaneng.2017.10.053
11. Daaou Nedjari, H., Guerri, O., Saighi, M.: Full rotor modelling and generalized actuator disc for wind turbine wake investigation. Energy Reports **6**, 232–255 (2020). https://doi.org/10.1016/j.egyr.2019.10.041. Technologies and Materials for Renewable Energy, Environment and Sustainability
12. Baltazar, J.M., Rijpkema, D., Falcão de Campos, J., Bosschers, J.: Prediction of the open-water performance of ducted propellers with a panel method. Journal of Marine Science and Engineering **6**(1) (2018). https://doi.org/10.3390/jmse6010027
13. Vermeer, L.J., Sørensen, J.N., Crespo, A.: Wind turbine wake aerodynamics. Progress in Aerospace Sciences **39**(6), 467–510 (2003). https://doi.org/10.1016/S0376-0421(03)00078-2
14. Amer, E., Dobrev, I., Massouh, F.: Determination of wind turbine far wake using actuator disk, (2014)
15. Sørensen, J.N., Myken, A.: Unsteady actuator disc model for horizontal axis wind turbines. Journal of Wind Engineering and Industrial Aerodynamics **39**(1), 139–149 (1992). https://doi.org/10.1016/0167-6105(92)90540-Q
16. Martínez Tossas, L., Leonardi, S., Churchfield, M., Moriarty, P.: A comparison of actuator disk and actuator line wind turbine models and best practices for their use. (2012). https://doi.org/10.2514/6.2012-900
17. Ammara, I., Leclerc, C., Masson, C.: A viscous three-dimensional differential/actuator-disk method for the aerodynamic analysis of wind farms. Journal of Solar Energy Engineering-transactions of The Asme - J SOL ENERGY ENG **124** (2002). https://doi.org/10.1115/1.1510870
18. Sorensen, J., Shen, W.Z.: Numerical modeling of wind turbine wakes. Journal of Fluids Engineering **124**, 393 (2002). https://doi.org/10.1115/1.1471361

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_23

19. Troldborg, N., Sørensen, J., Mikkelsen, R.: Numerical simulations of wake characteristics of a wind turbine in uniform flow. Wind Energy **13**, 86–99 (2010). https://doi.org/10.1002/we.345
20. Lynch, C.E., Prosser, D.T., Smith, M.J.: An efficient actuating blade model for unsteady rotating system wake simulations. Computers Fluids **92**, 138–150 (2014). https://doi.org/10.1016/j.compfluid.2013.12.014
21. Dobrev, I., Massouh, F., Rapin, M.: Actuator surface hybrid model. Journal of Physics: Conference Series **75**(1), 012019 (2007). https://doi.org/10.1088/1742-6596/75/1/012019
22. Shen, W.Z., Zhang, J.: The actuator surface model: A new navier–stokes based model for rotor computations. Journal of Solar Energy Engineering-transactions of The Asme - J SOL ENERGY ENG **131** (2009). https://doi.org/10.1115/1.3027502
23. Yang, X., Sotiropoulos, F.: A new class of actuator surface models for wind turbines (2018)
24. Bachant, P., Goude, A., Wosnik, M.: Actuator line modeling of vertical-axis turbines. arXiv preprint 1605.01449 (2016)
25. Troldborg, N.: Actuator line modeling of wind turbine wakes. (2009)
26. turbinesFoam library. https://github.com/turbinesFoam/turbinesFoam. Last accessed: 17 Apr 2023
27. Gritskevich, M.S., Garbaruk, A., Schütze, J., Menter, F.R.: Development of ddes and iddes formulations for the k-ω shear stress transport model. Flow, Turbulence and Combustion **88**, 431–449 (2012)
28. Spalart, P., Allmaras, S.: A one-equation turbulence model for aerodynamic flows. AIAA **439** (1992). https://doi.org/10.2514/6.1992-439
29. Epikhin, A.S.: Numerical schemes and hybrid approach for the simulation of unsteady turbulent flows. Mathematical Models and Computer Simulations **11**(6), 1019–1031 (2019). https://doi.org/10.1134/S2070048219060024
30. AMR-Wind solver. https://github.com/Exawind/amr-wind. Last accessed: 14 Apr 2023
31. Zhang, W., Almgren, A., Beckner, V., Bell, J., Blaschke, J., Chan, C., Day, M., Friesen, B., Gott, K., Graves, D., Katz, M., Myers, A., Nguyen, T., Nonaka, A., Rosso, M., Williams, S., Zingale, M.: Amrex: a framework for block-structured adaptive mesh refinement. Journal of Open Source Software **4**, 1370 (2019). https://doi.org/10.21105/joss.01370
32. AMReX software. https://github.com/AMReX-Codes/amrex. Last accessed: 01 Apr 2023
33. Nek5000 software. https://github.com/Nek5000. Last accessed: 31 Jan 2023
34. Kuiper, G.: The Wageningen Propeller Series. MARIN publication. Maritime Research Institute, Netherlands (1992)
35. Wang, M., QingXu, Zhang, Q., Epikhin, A., Liang, B.: Comparative analysis of non/ductedpropeller under the influence of vertical wall. In: 2022 Ivannikov Ispras Open Conference (ISPRAS), pp. 124–129 (2022). https://doi.org/10.1109/ISPRAS57371.2022.10076863
36. Airfoiltools Homepage. http://airfoiltools.com/. Last accessed: 04 Mar 2023
37. Viterna, L., Janetzke, D.: Theoretical and experimental power from large horizontal-axis wind turbines. NASA Technical Memorandum (1982)
38. Petrov, A.G., Sukhov, A.D., Sibgatullin, I.N., Britov, A.D.: Analytical and numerical methods for zhukovsky airfoils aerodynamics coefficients. In: 2022 Ivannikov Ispras Open Conference (ISPRAS), pp. 62–64 (2022). https://doi.org/10.1109/ISPRAS57371.2022.10076854

ICCS Camera Ready Version 2023
To cite this paper please use the final published version:
DOI: 10.1007/978-3-031-36030-5_23