Towards an Automatic Block-Structured Hexahedral Meshing Tool for a Complete Wind Turbine Rotor

Cherif Mihoubi, Jonas Schmidt and Bernhard Stoevesandt

Fraunhofer Institute for Wind Energy and Energy System IWES
Ammerländer Heerstr. 136, Oldenburg, Germany
cherif.mihoubi@iwes.fraunhofer.de

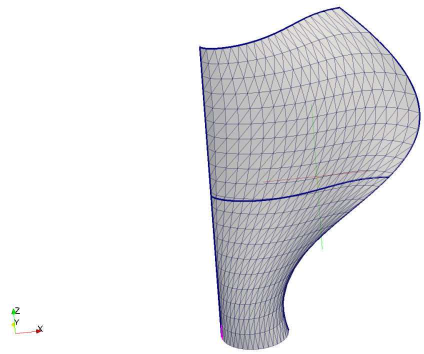
OpenFOAM has become an attractive Computational Fluid Dynamics (CFD) tool for scientific and industrial research institutions, due to its great degree of flexibility and extensibility for the creation of modeling tools for different CFD applications. Several substantial research efforts have been devoted to extend OpenFOAM for different applications (for more information please refer to [1], and [2]).
In our research group, we develop in-house software and tools for wind turbine design and blade aerodynamics. OpenFOAM is used consistently to provide highly accurate numerical solutions based on full 3D simulations. The mesh quality is critical, it has an important impact on efficiency and accuracy of computation especially when we use higher order schemes for compressible flow calculations. For large scale simulations, the mesh generation tools must be robust and effective enough to be able to handle large meshes. Therefore, the present work was carried out to provide a mesh generator, which is designed to enable the users to use OpenFOAM effectively to perform CFD simulations of complete wind turbine rotors.

The tool described in this paper generates 3D structural hexahedral mesh. All the settings must be specified within configuration files, called dictionaries. These specify the overall control parameters. The tool starts by creating the surface mesh of one blade and the third of the nacelle, with exporting capability of the surface mesh as standard CAD file. The blade is created by a set of airfoil input profiles situated at different locations measured from the blade root (flange), and different span, chord and twist angle depending on the user given parameters.The contour surface is built as parametric splines along the blade span, using [3], as illustrated in figure 1a. Here we mention that in our implementation we distinguish between lower and upper surfaces of the airfoils data which gives us more control of the blade curvature.

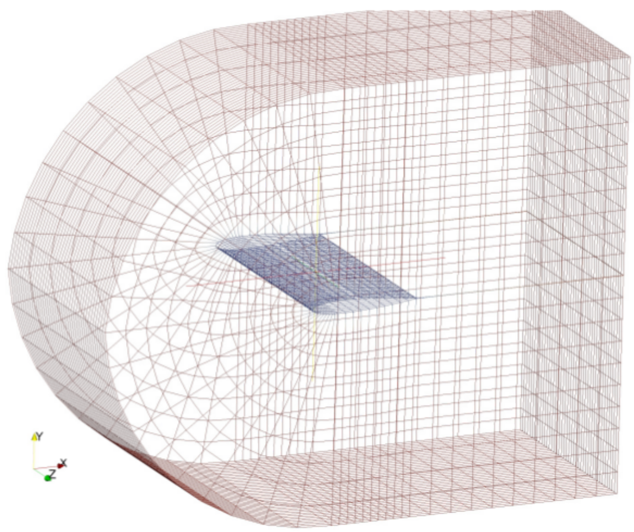
The next step is to create the 3D structural hexahedral mesh of one blade of type C-Mesh family (see figure 1b). For more effective processing we take advantage of the 120° symmetrical geometry of the rotor. We generate first the mesh around one blade and the third part of the nacelle. Then, we assembly the rotor by rotating the resulting mesh by an angle of 120°.

The output of the tool is OpenFOAM blockMesh dictionary file which contains list or set of hexahedral blocks, the vertices and edges of the blocks, as well as the boundary specifications block. This gives sufficient information for OpenFOAM's blockMesh utility to generate the full mesh data for the CFD application.

The meshing tool described in this paper can be used in combination with OpenFOAM as an analyzing tool for blade aerodynamics and blade load calculations, and for evaluation of the lift and drag characteristics for a given wind turbine. This is can be very useful especially in the preliminary development stages of a new wind turbine system.



(a) *detail surface mesh of root of blade*



(b) *3D structural hexahedral mesh around part of a blade*

References

[1] H. Jasak, A. Jemcov, and Z. Ťuković, "Openfoam: A c++ library for complex physics simulations," in *In Proceedings of the International Workshop on Coupled Methods in Numerical Dynamics*, sep 2007.

[2] "OpenFOAM finite volume programming environment for CFD." `http://www.openfoam.com/`.

[3] J. Schmidt, C. Peralta, and B. Stoevesandt, "Automated generation of structured meshes for wind energy applications," *Open Source CFD International Conference*, 2012.