Asim, Taimoor, Mishra, Rakesh, Kaysthagir, Sree and Aboufares, Ghada

Performance Comparison of a Vertical Axis Wind Turbine using Commercial and Open Source Computational Fluid Dynamics based Codes

Original Citation

Asim, Taimoor, Mishra, Rakesh, Kaysthagir, Sree and Aboufares, Ghada (2015) Performance Comparison of a Vertical Axis Wind Turbine using Commercial and Open Source Computational Fluid Dynamics based Codes. In: The International Conference on Jets, Wakes and Separated Flows, 16-18 June 2015, Stockholm, Sweden. (Submitted)

Performance Comparison of a Vertical Axis Wind Turbine using Commercial and Open Source Computational Fluid Dynamics based Codes

Taimoor Asim[1], Rakesh Mishra[1], Sree Nirjhor Kaysthagir[1], Ghada Aboufares[1]

[1] University of Huddersfield, Huddersfield, UK

{t.asim, r.mishra, sree.kaysthagir, ghada.aboufares}@hud.ac.uk

Abstract Computational Fluid Dynamics (CFD) is a very effective tool to analyse the flow characteristics within and around different mechanical artifacts such as automotive vehicles, turbomachines etc. CFD based analysis provides beneficial information about the flow behavior within a component, hence reducing the costs involved in the design process of that component. This has led the researchers to develop an Open Source CFD code, commonly known as Open Field Operation and Manipulation, or OpenFOAM. The development of OpenFOAM has made scientific research in the field of fluid dynamics available for almost no cost. Designers who rely on commercial CFD packages are hesitant on using Open Source CFD codes because of accuracy. They need to be convinced by proving its simulation capability as compared to any commercial CFD package. Hence, this study investigate the flow phenomenon in the vicinity of a Vertical Axis Wind Turbine (VAWT), using such open source code combinations that yields reasonably accurate results, which can be compared against any commercial CFD code.

1. Introduction

Computational Fluid Dynamics based commercial codes are in practice all over the world, and are helping the designers of turbomachines in accurate performance prediction of such machines. With special regards to Vertical Axis Wind Turbines (VAWT), [1] shows that the commercial CFD based code ANSYS-FLUENT is capable of predicting the performance output of a VAWT with very high level of accuracy. Similar trends have been reported in [2] where the experimental results match closely with the predicted numerical results from commercial CFD based codes. Further studies [3], [4], [5] all have reported that the use of commercial CFD based codes are capable of predicting the variations in the flow phenomena in the vicinity of a VAWT with reasonable accuracy.

More recently, various studies have been conducted on the performance evaluation and flow field prediction around a VAWT using open source CFD codes. Reference [6] has reported that the flow field prediction capability of the open source code (OpenFOAM) matches very closely with that of ANSYS-FLUENT. The study also reports that the libraries of OpenFOAM can be modified in order to suit specific applications. Hence, OpenFOAM provides the flexibility of altering the source code to suit various applications, which is not easily possible in case of commercial CFD based codes, like ANSYS-FLUENT, because the source code is not provided with the software.

Reference [7] compares the flow field prediction capability of OpenFOAM with commercial CFD package for flow around an aeroplane. It has been reported that OpenFOAM predicts the flow field that matches closely with that obtained through the use of commercial CFD package. However, it has been concluded that it is necessary to prepare templates for standard cases because of different requirements for different applications. Hence, this study has been carried out to develop a standardised code combination for simulating and analysing a Vertical Axis Wind Turbine.

2. NUMERICAL MODELING

A three dimensional Vertical Axis Wind Turbine model, similar to [8] comprising of 12 inner rotor blades and 12 outer stator blades has been numerically modeled in both ANSYS-FLUENT and OpenFOAM. The VAWT model is shown in Figure 1. The height of the VAWT is 1m, and its flow domain's dimensions are 11m, 9m and 3m for length, width and height respectively.

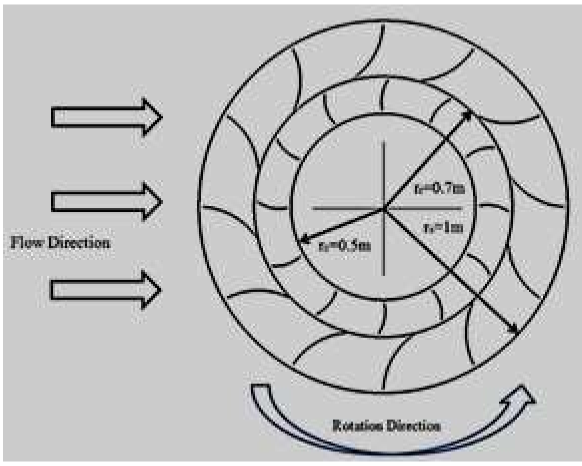


Fig.1.Geometrical Characteristics of the VAWT

The meshing techniques used by commercial CFD packages and OpenFOAM are very different to each other. Default ANSYS Meshing is controlled by the geometrical characteristics of the model and dominated by tetrahedral mesh elements. Hence, the geometrical integrity of the model is not violated even if the mesh quality is not good. However, in OpenFOAM, meshing is a two-step process, where a dominant hexahedral base mesh is initially generated for the flow domain. A special meshing tool, called Snappy Hex Mesh, is used afterwards to connect the geometrical model to the base mesh. Therefore, if the mesh quality is not good, the structural integrity of the model is violated. Hence, it is necessary to have an appropriate base mesh sizing for accurate geometrical interaction with the flow field.

Table 1. Mesh quality comparison between ANSYS Fluent and OpenFOAM

Type of Cells	ANSYS	OpenFOAM
Mixed Cells	67762	216661
Hexahedral Cells	103752	1046568
Tetrahedral Cells	1182046	7
Total	1,353,560	1,263,236

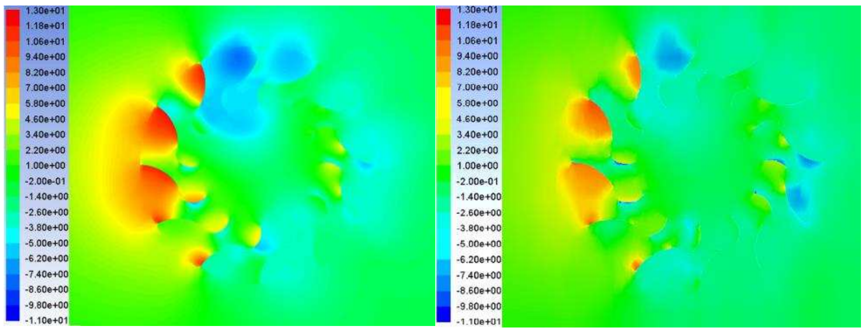
For effective comparison between the flow field predictions from ANSYS-FLUENT and OpenFOAM, the total numbers of mesh elements are maintained between 1.2 and 1.3 million. Table 1 summarises the type and number of mesh elements generated in both ANSYS and OpenFOAM. ANSYS mesh contains approximately 87% of tetrahedral elements from a total number of 1.3 million mesh elements. On the hand, the percentage of mesh dominated by the hexahedral elements is about 83% in case of OpenFOAM.

The hexahedral elements are preferred over tetrahedral mesh elements due to their lesser numerical diffusion. The maximum aspect ratio for OpenFOAM mesh is 10.74% less than ANSYS mesh, which shows that the overall mesh quality in OpenFOAM is better. Due to superiority in accurately modeling the wake regions and extreme pressure gradients, SST k-ω model with Multiple Reference Frame (MRF) for rotating the rotor zone has been chosen for both solvers. Default gradient and interpolation schemes have been used in the present study. Most recent studies [9], [10], [11] also show that k-ω turbulence model predicts the changes in the flow parameters in the vicinity of a VAWT with reasonable accuracy.

In the present study, the density of air is assumed to be constant. Since the flow of air in the present study is at low speeds (4m/sec), pressure based solver has been incorporated in the numerical modeling. It should be noted that OpenFOAM uses kinematic pressure as its primary pressure parameter and ANSYS-FLUENT uses static pressure as its primary pressure parameter. Hence, for comparison purposes, static pressure in ANSYS-FLUENT has been customised to represent kinematic pressure by dividing it with the density of air. For boundary condition, inlet velocity of 4m/sec (representing average wind speed in Huddersfield), Tip Speed Ratio of 0.2 (representing maximum torque production point), and an outlet boundary condition of atmospheric pressure (referring to zero gauge static pressure) has been used throughout this study [1]. No-slip condition is applied for all the walls.

3. RESULT AND DISCUSSION

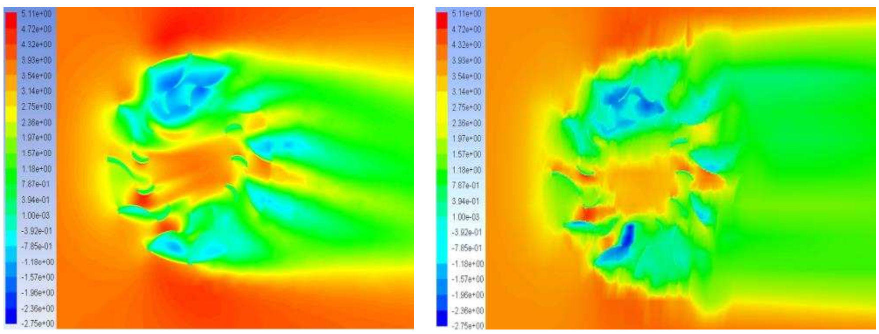
Figures 2 and 3 depict the variations in the kinematic pressure and axial velocity respectively from both ANSYS-FLUENT and OpenFOAM. These contours have been drawn on the same scale for effective comparison.



(a) (b)

Fig. 2.Variations in Kinematic Pressure (a) ANSYS-FLUENT, (b) OpenFOAM

The irregular axial velocity distribution observed in Figure 3(b) is because of the rendering of the post-processing tool and has no effect on the accuracy of the solver. Nevertheless, the magnitudes the parameters vary to some extent in both these figures. A detailed qualitative analysis is required in order to make these differences prominent.

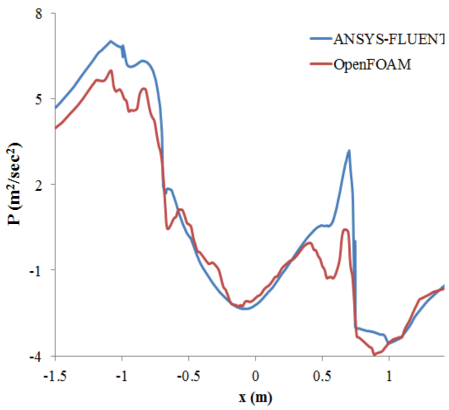


(a) (b)

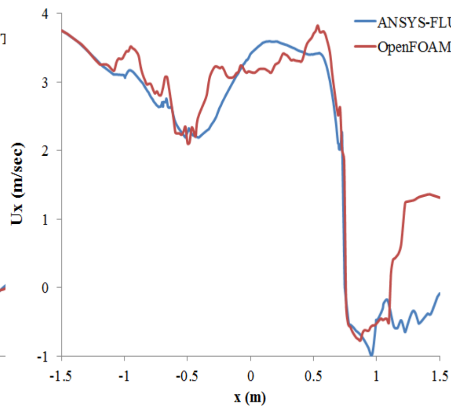
Fig. 3.Variations in Axial Velocity (a) ANSYS-FLUENT, (b) OpenFOAM

A straight horizontal line, termed as analysis line, has been generated in both the solvers that pass through the center of the VAWT. Flow field analysis has been carried out on this line for kinematic pressure and axial velocity variations in the vicinity of the VAWT.

Figures 4(a) and (b) depict the variations in the kinematic pressure and the axial velocity on the analysis line obtained from both the solvers. It can be seen in both the figures that both the CFD solvers under consideration predict the flow features in the vicinity of the VAWT with reasonable accuracy. The trends in kinematic pressure and the axial velocity profiles remains the same in both these solvers. Hence, OpenFOAM has the ability to predict the flow behaviour in turbomachines with reasonable accuracy, and that it has the potential to replace commercial CFD packages for the numerical analysis of such machines.



(a)



(b)

Fig.4. **Comparison between (a) Kinematic Pressure and (b) Axial Velocity on the analysis line from ANSYS-FLUENT and OpenFOAM.**

From Figure 4(b), it can be seen that apart from the wake region (x>1m), the axial velocities predicted by OpenFOAM matches closely with that obtained from ANSYS-FLUENT.

VAWTs are used to harness energy from the wind, hence the torque output of the VAWT is considered to be the primary performance parameter. The computed torque outputs from ANSYS-FLUENT and OpenFOAM are 7.74 and 7.68Nm respectively, having less than 1% difference between them. Hence, OpenFOAM not only predicts the local flow features with reasonable accuracy, but has the tendency to predict the global performance parameters of turbomachines as well.

4. CONCLUSION

A study has been presented here on the effectiveness of OpenFOAM in predicting the local flow features and global performance parameters of a Vertical Axis Wind

Turbine. The results have been compared against a well-known and globally used commercial CFD package called ANSYS-FLUENT. The mesh size and the modeling techniques used in both these solvers has been kept the same for a realistic comparison.

The comparative study shows close matching between the two solvers. It has been shown that OpenFOAM not only predicts the local flow features, but also the global performance parameters of a VAWT with reasonable accuracy. For this purpose, kinematic pressure and axial velocity have been chosen for analysis between the two solvers. Hence, it can be concluded that OpenFOAM can be used as an effective tool for the analysis of turbomachines.

Reference

[1] K. Park, "Optimal Design of a Micro Vertical Axis Wind Turbine for Sustainable Urban Environment," Ph.D. Thesis, University of Huddersfield, Huddersfield, U.K., 2013.

[2] A. Shahzad, T. Asim, R. Mishra, and A. Paris, "Performance of a Vertical Axis Wind Turbine under Accelerating and Decelerating Flows," Procedia CIRP, vol. 11, pp. 311–316, 2013.

[3] K. Park, T. Asim, and R. Mishra, "Computational Fluid Dynamics based Fault Simulations of a Vertical Axis Wind Turbine," J. Physics: Conference Series, vol: 364, no. 012138

[4] G. Colley, "Design, Operation and Diagnostics of a Vertical Axis Wind Turbine," Ph.D. Thesis, University of Huddersfield, Huddersfield, U.K., 2013.

[5] F. Mohammed, K. Park, S. Pradhan, R. Mishra, K. Zala, T. Asim, and A. Al-Obaidi, "The Effect of Blade Angles of the Vertical Axis Wind Turbine on the Output Performance," 27th Int. Cong. of Condition Monitoring and Diagnostic Engineering, Brisbane, Australia, 2014.

[6] F. Ambrosino, and A. Funel, "OpenFOAM and Fluent Features in CFD Simulations on CRESCO High Power Computing System," Final Workshop of Grid Projects, PON RICERCA, 2006.

[7] A. Kosik, "The CFD Simulation of the Flow around the Aircraft using OpenFOAM and ANSA," 5th ANSA and μETA Int. Conf., 2006.

[8] K. Park, T. Asim, and R. Mishra, "Effect of Blade Faults on the Performance Characteristics of a Vertical Axis Wind Turbine," 26th Int. Cong. on Condition Monitoring and Diagnostic Engineering Management, Helsinki, Finland, 2013.

[9] K. Park, T. Asim, and R. Mishra, "Simulation Based Approach to Predict Vertical Axis Wind Turbine Faults using Computational Fluid Dynamics," Int. Conf. on Through–Life Engineering Services, Cranfield, U.K., 2012.

[10] K. Park, T. Asim, and R. Mishra, "Condition Based Monitoring of Vertical Axis Wind Turbines Using Computational Fluid Dynamics," 39th National Conf. on Fluid Mechanics and Fluid Power, Surat, India, 2012.

[11] 11. A. Shahzad, T. Asim, K. Park, S. Pradhan, and R. Mishra, "Numerical Simulations of Effects of Faults in a Vertical Axis Wind Turbine's Performance," 2nd Int. Workshop and Cong. on eMaintenance, Lulea, Sweden, 2012.

[12] J. Luo, R. Issa, and A. Gosman, "Prediction of Impeller-Induced Flows in Mixing Vessels Using Multiple Frames of Reference," IChemE Symposium Series, no. 136, pp. 549-556, 1994