An Open Source CFD Study of Air Flow over Complex Terrain

S. M. Fabre[1], T. J. Scanlon[1], M. T. Stickland[1] and A. B. Oldroyd[2]

[1]Department of Mechanical Engineering, University of Strathclyde, Glasgow, UK

[2]Oldbaum Services Limited, Stirling, UK

Abstract

This paper presents an open source computational fluid dynamics (CFD) study of air flow over a complex terrain. The open source C++ toolbox OpenFOAM has been used for the CFD analysis and the terrain considered is a scale model of Berlengas Island, which lies close to the Portuguese coast. In order to validate the CFD model, experimental work has been carried out in an open-section wind tunnel using hot-wire anemometry to measure the wind profiles above the island. In the majority of cases, the OpenFOAM CFD solutions show very good agreement with the experimental wind profile data, confirming that open source CFD solutions are possible for environmental flows over complex terrain.

Keywords: CFD, open source, validation, experimental, environmental, wind, complex terrain.

1. Introduction

As the deadline for the EU's promised 20% reduction in carbon emissions by 2020 fast approaches, wind energy is a key area of expansion for EU member states in order to meet their renewable energy obligations [1]. Technologies for wind flow analysis are required to facilitate better project planning, accurate yield prediction, and a fundamentally better understanding of the local climate conditions. However, there remain significant challenges ahead, not the least of which is the ability of such technologies to accurately assess the wind resource potential in on-shore locations that exhibit significant variations in terrain profile and consist of complex topographies.

Wind resource assessment relies on three main approaches: (i) wind tunnel testing, (ii) field experiments, and (iii) modelling and simulation. The wind tunnel is the primary design tool, providing significantly more data than the other techniques combined. However, tests are costly and time-consuming and usually do not fully replicate real operating and flow conditions. Field experiments which measure wind speed and turbulence data using meteorological masts with cup anemometers [2] and non-intrusive technologies such as LiDAR [3] and SoDAR [4] deliver authentic data relating to real atmospheric conditions, however, the capture of good data is difficult, and tests require meticulous and time-consuming planning, at considerable expense.

The most promising route for yield prediction in future wind renewable developments is through modelling and simulation, and the use of Computational Fluid Dynamics (CFD). The automotive and aircraft industries have already replaced the majority of their wind tunnel tests with CFD and the aero-space industry is fast following suit. While CFD has the potential to be very useful for the study of the environmental flows encountered in the wind industry — because it can deliver data that is difficult to measure or observe, under climate conditions we cannot reproduce in a laboratory — it still faces major challenges, especially if there are significant variations in the land topography.

1.1 Numerical modelling approaches for wind analysis

Several numerical simulation techniques exist for wind flow analysis, ranging in levels of complexity from simple linear solvers to direct numerical simulation. The principal analysis techniques are described below:

a) **Linear models:** These solve a set of linearized flow equations which contain simplified turbulence and roughness models. The models attempt to correct existing long-term physical data to account for several different effects including object blockage, terrain classification, and land topology.

b) **RANS: (Reynolds Averaged Navier-Stokes):** This CFD technique involves the solution of the time-averaged Navier-Stokes equations with the relevant scales of turbulence being modelled. It is the most well-known and widely-adopted method for practical engineering applications.

c) **Large Eddy Simulation (LES):** Another CFD approach in which the larger scales of turbulence, which contain most of the energy, are directly resolved while the smaller scales, below a certain filter level, are modelled.

d) **Detached Eddy Simulations (DES):** Is a mixture RANS and LES, where RANS model is employed in user-specified regions and LES in others. This hybrid modelling technique affords the user greater flexibility in the computational approach.

e) **Direct Numerical simulations (DNS):** This involves the direct numerical solution of the instantaneous equations that govern fluid flow (the unsteady Navier-Stokes-Fourier equations) using the appropriate length and time scales.

Over recent years the dominant computational method for modelling wind flow has been the linear model or wind atlas technique [5]. In simple terms, this method uses linearized flow equations to correct existing long-term measurements for various different effects including sheltering objects, terrain classification, and domain contours. The advantage of this method is that it is well established and relatively straightforward to apply. The most widely used application of the technique is the WAsP computer code developed by the RISØ National Laboratory in Denmark. WAsP has enjoyed such widespread adoption because the use of linearized flow equations make it able to predict the wind resource with sufficient accuracy and efficiency when the terrain is smooth enough to ensure that the flow remains attached. However, WAsP does have limitations and generates poor predictions when flow separation and recirculation are evident [6]. In an attempt to address this issue, a site ruggedness index (RIX) was proposed as a crude measure of the terrain complexity and hence the extent of flow separation [7]. The RIX is defined as "the fractional extent of the surrounding terrain which is steeper than a certain critical slope". However, despite corrections using the RIX, many researchers have concluded that it is not generally advisable to apply WAsP in complex terrain [7-10]. These conclusions, combined with the observation that the increase in wind power production has led to sites being selected with increasingly complex terrain [8], mean that alternative computational methods need to be established.

The choice of computational model requires the user to strike a delicate balance between required accuracy and the computational resources available. The range of length and timescales involved in DNS means that significant computational resources are required and the technique is currently impractical for real-world engineering problems. Employed correctly, LES-based modelling is likely to predict results with a higher degree of accuracy compared with RANS models, however, for the large, 3D, complex geometry problems normally encountered in the wind industry, the computational resources for a LES-based solution are currently beyond the reach of the general wind-energy community. Therefore, the current basis for the modelling and simulation of environmental flows in complex terrain is dominated by the use of linear and RANS-type models.

Given the limitations in the range of topologies that linearized models can handle, CFD is the evident choice as an alternative to WAsP and other linearized approaches, with the RANS approach the most likely choice given the computational restrictions. However, despite the impact of CFD techniques in many areas, such as automotive or aeronautical engineering, it has not yet become common in wind energy engineering [8]. Challenges remain in the numerical modelling of turbulence for atmospheric flows, particularly in complex terrain, and in CFD representations of atmospheric boundary layers [11, 12].

1.2 Open source CFD for complex terrain

The work presented in this paper uses the open source CFD toolbox OpenFOAM [13]. OpenFOAM is a flexible set of efficient, object-oriented C++ modules for solving complex fluid flows. It is freely available and open source under the GNU general public licence and runs under the linux operating system. The open source philosophy behind OpenFOAM means that commercial CFD licensing costs are eliminated, source code access gives significant flexibility in the development of additional modules and the unlimited parallel processing capacity means that practical engineering problems may be tackled within realistic time scales and on modest hardware budgets.

OpenFOAM has been applied previously to atmospheric flows in complex terrain. Risø DTU, the National Laboratory for Sustainable Energy of Denmark, recently organized a blind comparison of flow models for the evaluation of wind over complex terrain [14] based on a new dataset of measurements collected on the isolated presqu'ile of Bolund [15]. One of the main objectives of this exercise was to assess the competency of current wind resource assessment techniques in complex terrain. This flow case met the requirement of “complex” as, for primary wind directions, the orography resembles a forward-facing step and, with respect to the shallow boundary layer developed over open water, the step height was relatively large. Participation was open to all and, although the boundary conditions were tightly specified for consistency, modelers were free to select their simulation technique of choice to calculate the flow field. The two OpenFOAM simulations submitted [16] were ranked first and fifth with overall mean errors in predicted velocity of 13% and 14%. These results highlighted the capabilities of open source CFD and helped establish OpenFOAM as a credible alternative to commercial CFD packages in the wind energy industry. Tapia [17] has used OpenFOAM to develop roughness models and applied the code to flow situations over terrain of a limited degree of complexity. Comparisons of velocity profiles with those predicted by the industrial CFD code Fluent showed an excellent level of agreement and served to validate the open source code against one of its commercial equivalents.

This paper considers an experimental and numerical analysis of air flow over a complex terrain in order to further assess the capabilities of open source CFD in such situations. The terrain considered is a scale model of Berlengas Island, which lies close to Lisbon on the Portuguese coast, as shown in Figure 1. The work has been carried out as part of the EU Norsewind project [18] whose primary goal is to create an offshore wind atlas of European waters for use in wind exploitation. In order to validate our OpenFOAM CFD model, experimental work has been carried out in an open-section wind tunnel using hot-wire anemometry to measure the wind profiles above the island. The results build upon previous open source CFD work [16, 17] in that the level of topological complexity has been increased to include valleys, escarpments, steep cliffs and rocky outcrops. This paper makes a novel contribution to knowledge in the field of open source CFD applied to complex terrain and provides experimental data to validate the numerical results.

Figure 1 Berlengas Island with insert showing approximate geographic location

2. Experimental work

3. Numerical analysis

The computational work was carried out within the framework of the open source CFD package OpenFOAM [13]. In order to mesh the volume around the island, the OpenFOAM utility *snappyHexMesh* was used. The meshing process begins with a CAD model of the Berlengas island in stereolithographic (*.stl*) format as shown in figure 2. The next stage is to generate a mesh block, covering the island and its surroundings, that consists solely of hexahedral cells as shown in figure 3. This mesh block represents the extent of the computational domain within which the flow will be resolved. The *snappyHexMesh* utility was then employed to snap the hexahedral mesh onto the surface of the island, resulting in the surface mesh shown in figure 4. Separate meshes were created for each of the 12 flow directions considered by rotating the island to the appropriate angle of incidence and re-meshing.

The OpenFOAM solver *simpleFoam* was employed to resolve the flow field using the k-ω SST [19] RANS model to represent turbulence and employing the SIMPLE [19] algorithm for pressure-velocity coupling. Inlet turbulence conditions were based on a wind tunnel turbulence intensity of 1% and 2nd order convection discretisation was adopted. Finally, standard wall functions were used to model turbulence skin friction.

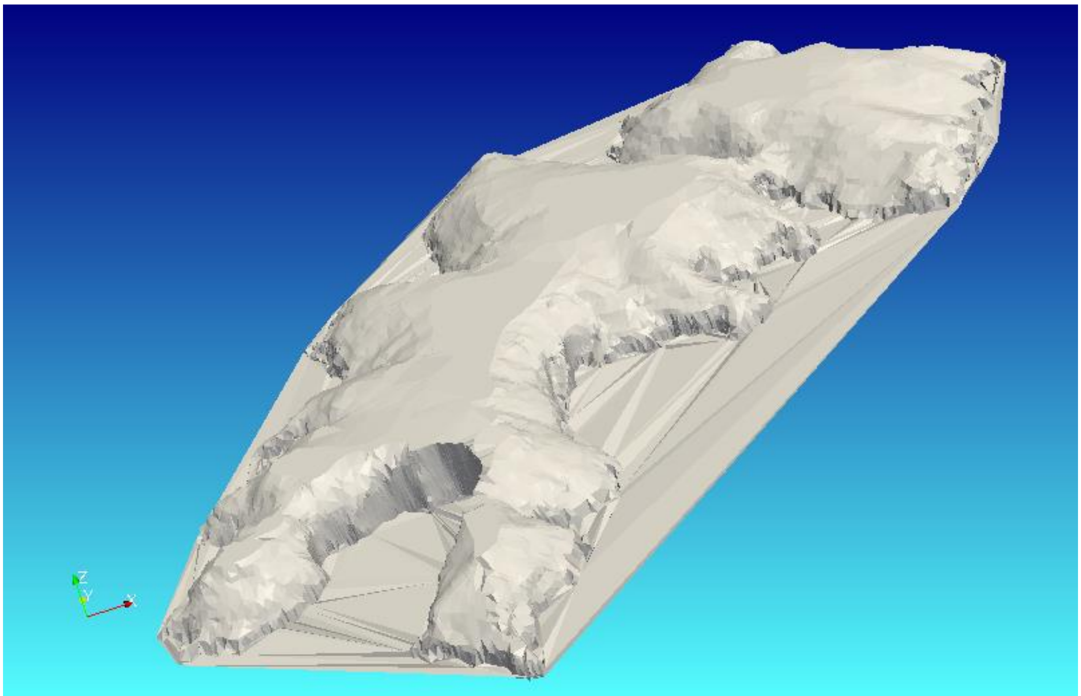

Figure 2 CAD model of Berlengas island in .stl format

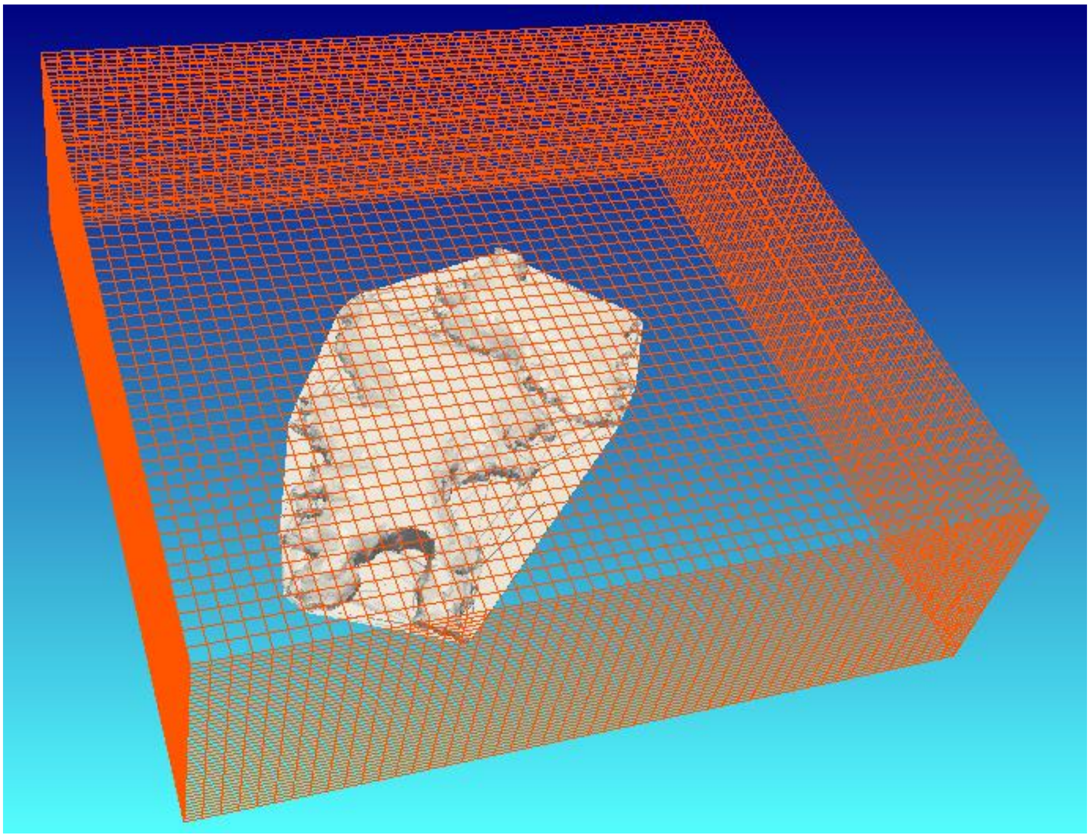

Figure 3 Hexahedral mesh block around the island

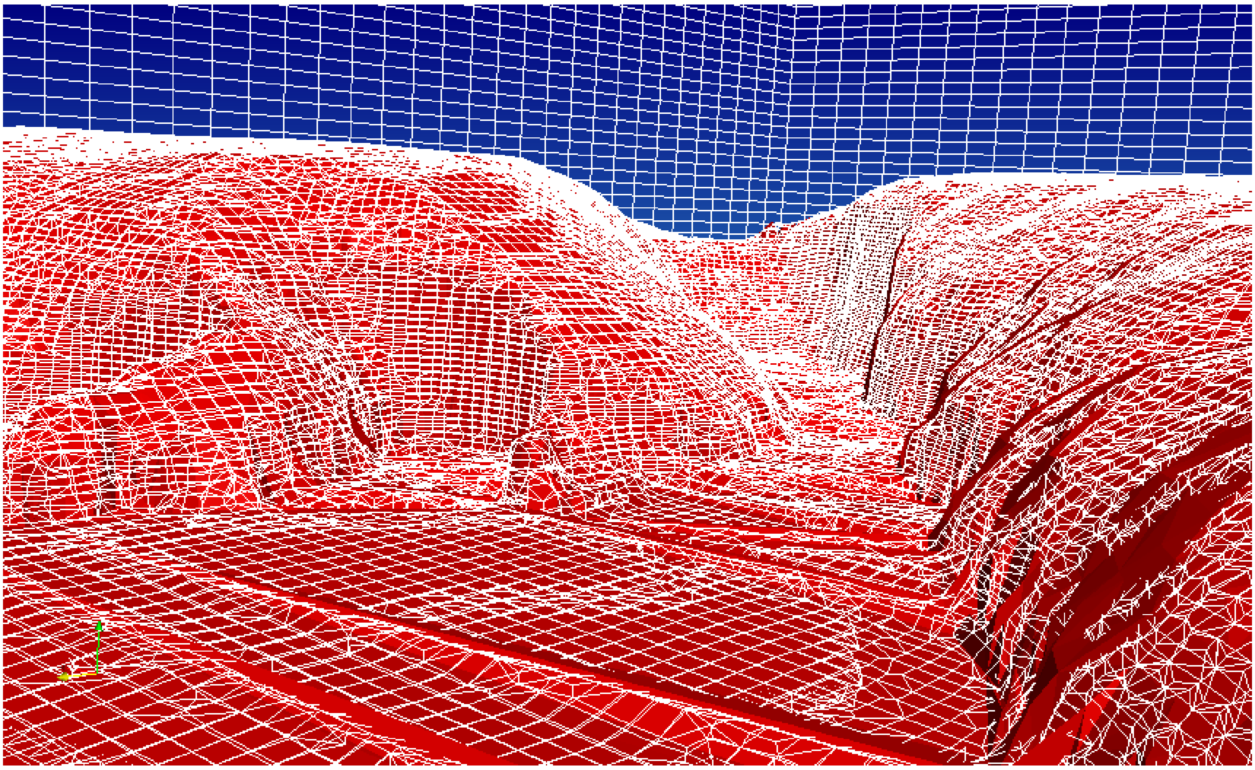

Figure 4 Snapped surface mesh on the island

The CFD velocity profiles were taken at the same measurement points as the wind tunnel for the vertical line described in section 2. Mesh sensitivity studies were carried out in order to assess the influence of mesh size on the predicted flow field. Figure 5 shows the velocity profiles for the 210^{o} flow angle case. The results show that there was little change between the smallest grid (0.7 million cells) and the largest (2 million cells) and a mesh size of approximately 2 million cells was employed for each flow angle. In order for the turbulent wall functions to operate correctly, the value of the turbulence parameter y+ should be in the range $30 < y+ < 300$. This was verified for each of the twelve flow angle cases considered. The cases were considered to be fully converged when the global values of the residuals were of the order 1×10^{-5}.

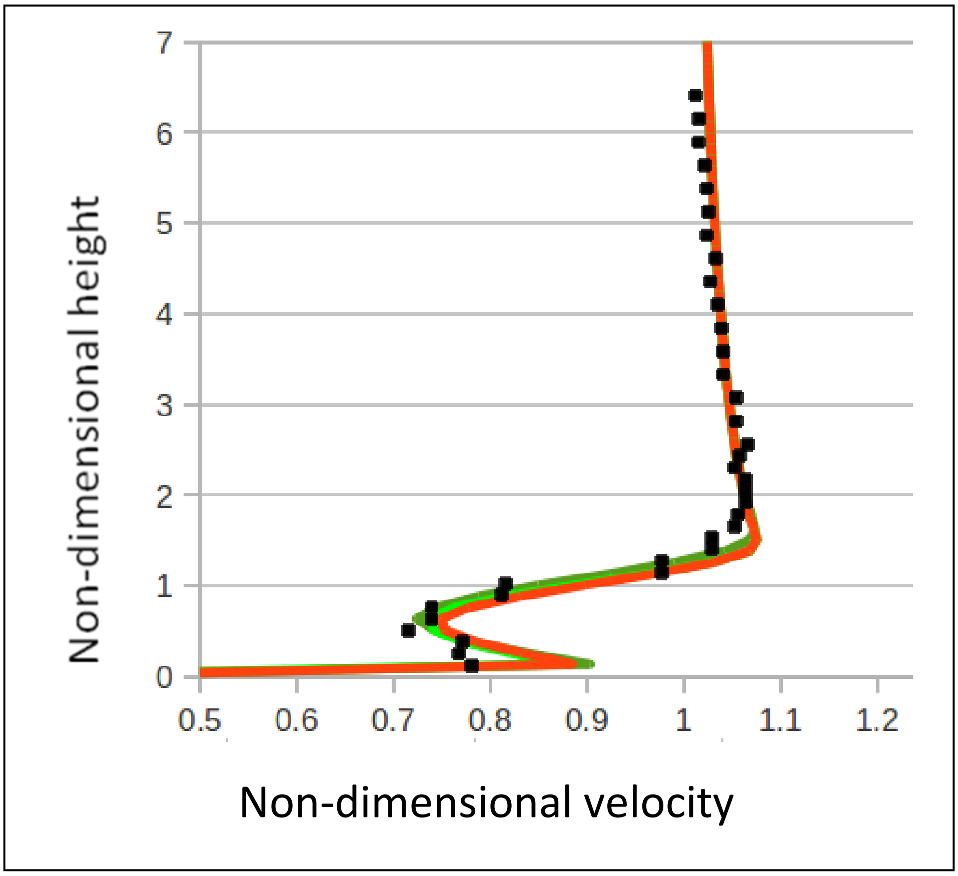

Figure 5 Mesh sensitivity analysis for the 210° flow angle case. Lines – OpenFOAM; symbols – wind tunnel.

dark green – 0.7 million cells; light green - 1.2 million cells; red – 2 milllion cells

4. Results

Figures 6 and 7 show the OpenFOAM-predicted velocity profiles compared with the wind-tunnel results for the 12 flow angles considered in the study. The height has been non-dimensionalised with respect to the island surface height at the location where the velocities were measured. In general, the results show a very good concurrence between the experimental wind tunnel measurements and the OpenFOAM CFD results. Each flow angle represented a different challenge to the CFD study as the topology of the island coastline upstream of the measurement point varied greatly, including steep cliffs, rocky outcrops and escarpments. However, the general flow features of flow acceleration and retardation above the island appear to have been captured very well by the CFD study. Figure 8 shows some of the flow separation and recirculation features, captured by the CFD study, in the canyons around the island coastline.

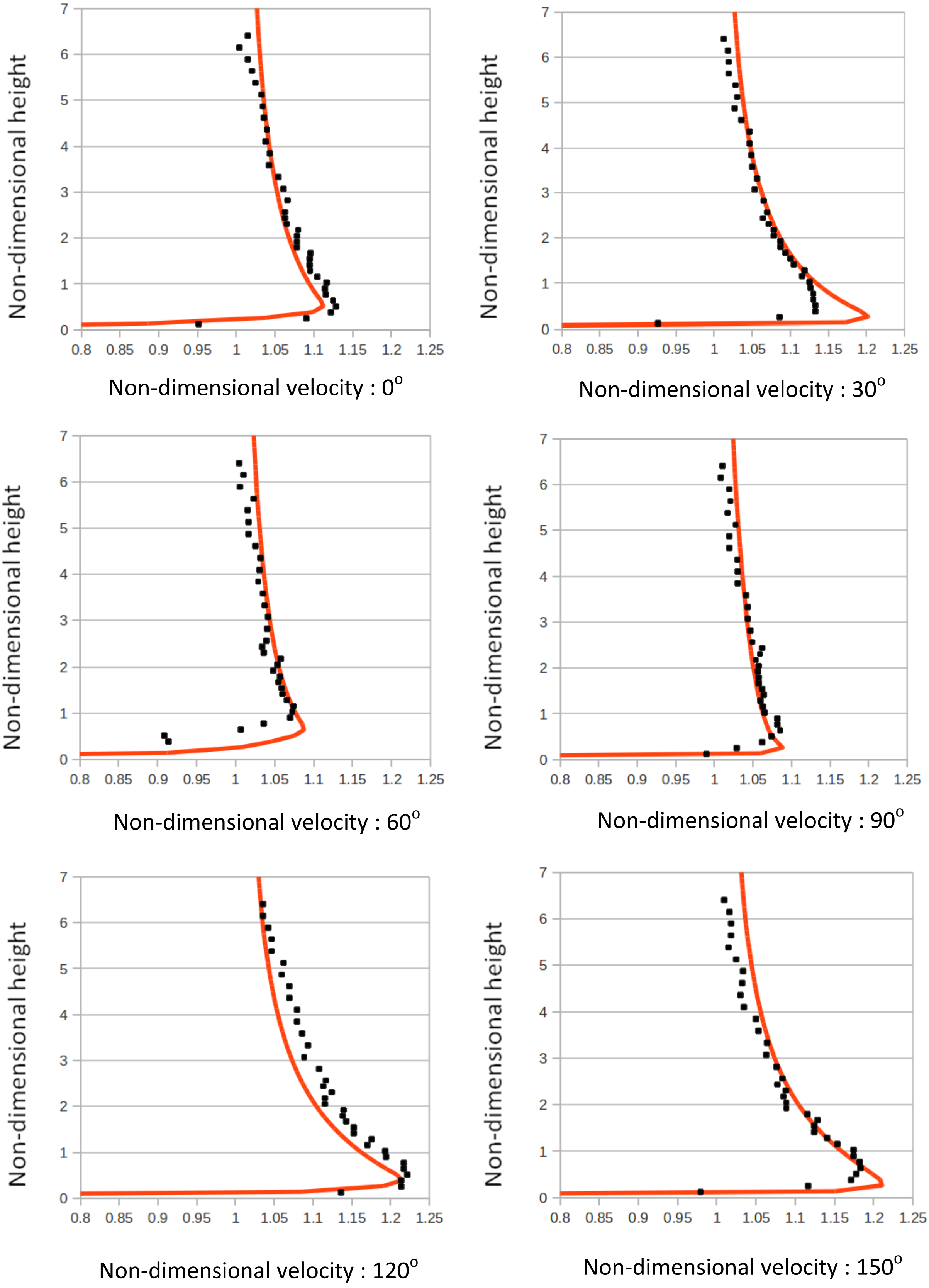

Figure 6 Velocity profiles for flow angles 0° to 150°. Lines - OpenFOAM, symbols – wind tunnel.

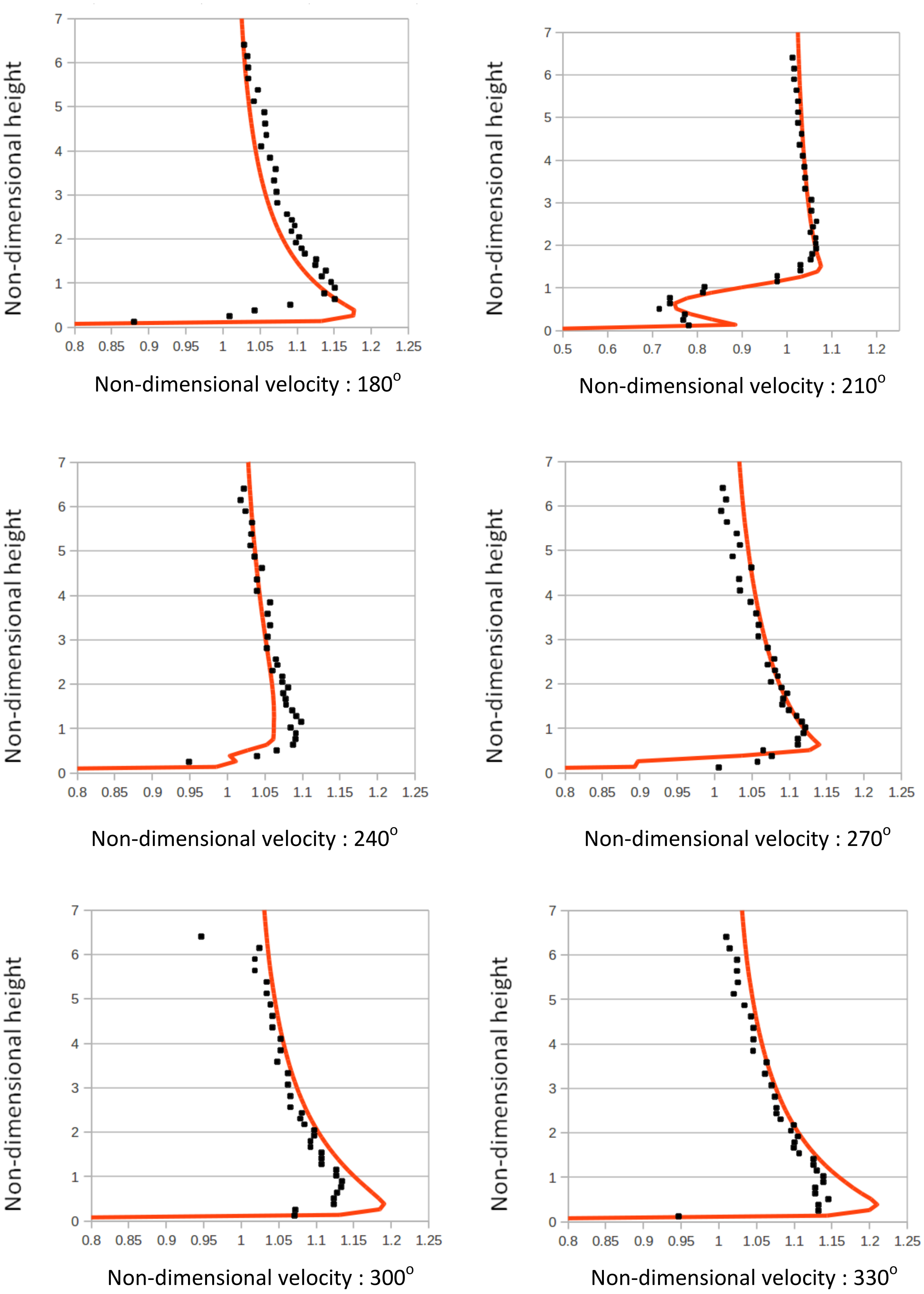

Figure 7 Velocity profiles for flow angles 180° to 330°. Lines - OpenFOAM, symbols – wind tunnel.

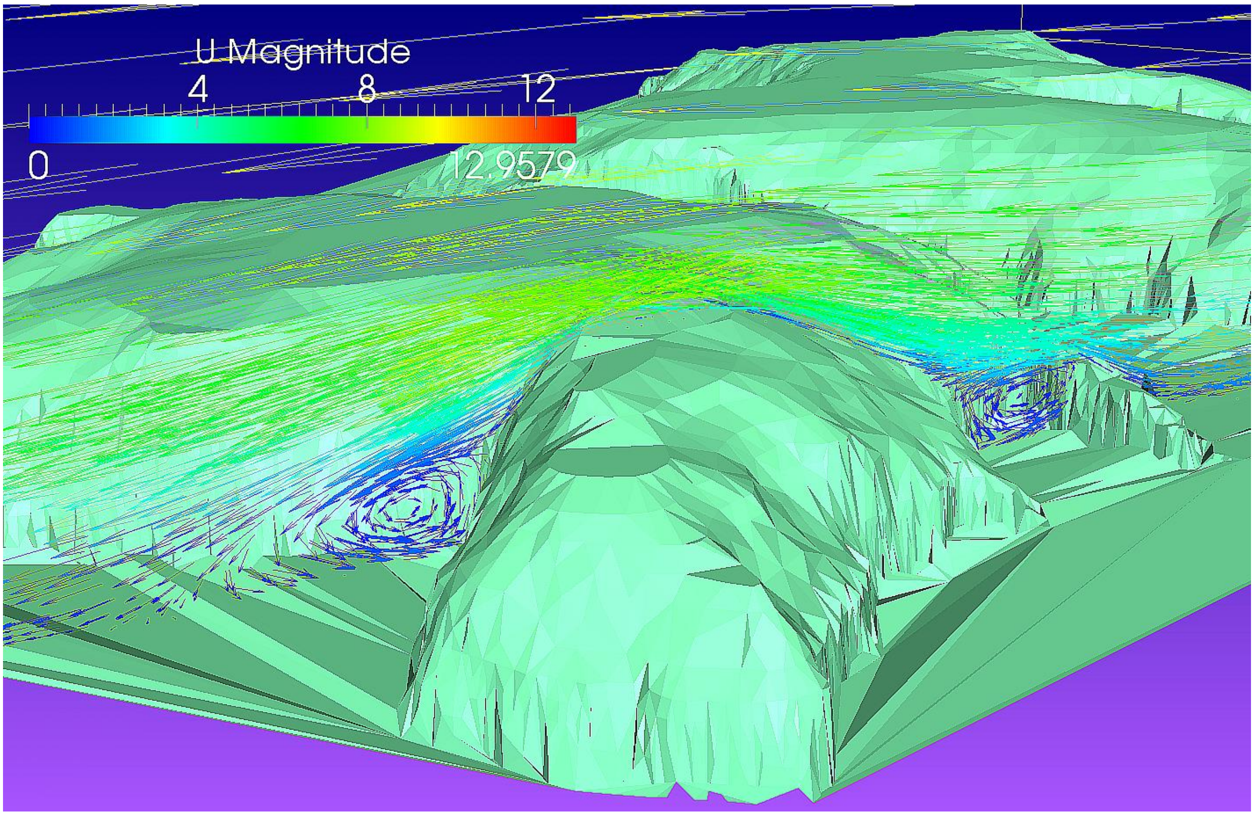

Figure 8 Velocity vectors showing flow separation and recirculation in the island canyons

5. Conclusions

The open source CFD code OpenFOAM has been used in a study of air flow over a complex terrain. Comparisons of OpenFOAM and wind tunnel studies show very good agreement for wind speed measurements above the island and flow separation and recirculation features in island canyons have been captured successfully. These results confirm that open source CFD solutions on a modest hardware budget are feasible for environmental flows over complex terrain. Finally, the cost benefits and open source nature of the OpenFOAM code mean that it has the potential reach a wider audience within the current wind energy analysis community.

References

1. The Accountability of European Renewable Energy and Climate Policy, CE Delft, Publication number: 11.3320.24 (2011).

2. Barthelmie R. J. *et al* Modelling and measurements of wakes in large wind farms *J. Phys.: Conf. Series*, 75, (2007).

3. Courtney, M., Wagner, R. and Lindelow, P. Testing and comparison of LiDARs for profile and turbulence measurements in wind energy, *14th International Symposium for the Advancement of Boundary Layer Remote Sensing, IOP Conference Series, Earth and Environmental Science*, 1, 012021, (2008).

4. Coulter, R. L. and Kallistratova, M. A., Two decades of progress in sodar techniques: a review of 11 ISARS proceedings 2003 Meteorology and Atmospheric Physics Volume 85,1-3, pp. 3-19 (2003).

5. Landberg, L., Myllerup, L., Rathmann, O., Petersen, E. L., Jørgensen, B. H., Badger, J. and Mortensen, N. G., Wind resource estimation - an overview, *Wind Energy* 6 (3), pp. 261–271 (2003).

6. Bowen, A. J. and Mortensen, N. G., Exploring the limits of WAsP: the wind atlas analysis and application program, *European Wind Energy Conference and Exhibition*, pp. 584–587. Gothenborg, Sweden (1996).

7. Mortensen, N. G., Bowen, A. J. and Antoniou, I., Improving WAsP predictions in (too) complex terrain. *European Wind Energy Conference and Exhibition*, Athens, Greece (2006).

8. Palma, J. M. L. M., Castro, F. A., Ribeiro, L. F., Rodrigues, A. H. and Pinto, A. P., Linear and nonlinear models in wind resource assessment and wind turbine micro-siting in complex terrain. *J. of Wind Engineering and Industrial Aerodynamics*, 96 (12), pp. 2308–2326, (2008).

9. Perivolaris, Y. G., Vougiouka, A. N., Alafouzos, V. V., Mourikis, D. G., Zagorakis, V. P., Rados, K. G., Barkouta, D. S., Zervos, A. and Wang, Q., 2006 Coupling of a mesoscale atmospheric prediction system with a CFD microclimatic model for production forecasting of wind farms in complex terrain: Test case in the island of Evia. *European Wind Energy Conference and Exhibition*, Athens, Greece (2006).

10. Wood, N., Wind flow over complex terrain: A historical perspective and the prospect for large-eddy modelling. *Boundary-Layer Meteorology*, 96 (1), pp. 11–32, (2000).

11. Undheim, O., Andersson, H. and Berge, E., Non-linear, microscale modelling of the flow over Askervein Hill, *Boundary-Layer Meteorology*, 120 (3), 477–495, (2006).

12. Hargreaves, D. M. and Wright, N. G., On the use of the k-ε model in commercial CFD software to model the neutral atmospheric boundary layer, *J. of Wind Engineering and Industrial Aerodynamics,* 95 (5), 355–369, (2007).

13. http://www.openfoam.com

14. Risø DTU: The Bolund Experiment: Blind comparison of CFD codes – Wind in complex terrain, http://www.risoe.dk/da/Research/sustainable energy/wind energy/projects/VEA Bolund/Bolund Blind Comparison.aspx?sc lang=en. (2009).

15. Bechmann, A., Berg, J., Courtney, M., Jørgensen, H., Mann, J. and Sørensen, N.: The Bolund Experiment: Overview and Background, *Technical Report R-1658(EN)*, Risø DTU, (2009).

16. Sumner, J., Masson, C.,Odemark, Y. and Cehlin, M. OpenFOAM simulations of atmospheric flow over complex terrain, *5th OpenFOAM Workshop*, Chalmers, Gothenburg, Sweden, June 21-24, (2010).

17. Tapia, X. P., Modelling of wind flow over complex terrain using OpenFoam, *Master's Thesis*, University of Gävle, Sweden, June (2009).

18. http://norsewind.eu

19. Versteeg, H.K. and Malalasekera, W., An introduction to Computational Fluid Dynamics: the Finite-Volume Method (2nd edition), Pearson, (2007)