Actual Total Cost reduction of commercial CFD modelling tools for Wind Resource Assessment in complex terrain

S Barber[1], A Schubiger[1], S Koller[2], A Rumpf[3], H Knaus[3] and H Nordborg[1]

[1]University of Applied Sciences Rapperswil, Oberseestrasse 10, 8640 Rapperswil, Switzerland
[2]Meteotest AG, Fabrikstrasse 14, 3012 Bern, Switzerland
[3]Hochschule Esslingen, Kanalstrasse 33, 73728 Esslingen, Germany

E-mail: `sarah.barber@hsr.ch`

Abstract. In this paper, new automated processes for applying the commercial Computational Fluid Dynamics (CFD) tools *ANSYS Fluent* and *ANSYS CFX* to wind modelling in complex terrain are developed with the goal of decreasing the Actual Total Costs (ATC) related to planning wind energy projects. Simulations are carried out at the complex terrain site Stötten in southern Germany using *ANSYS Fluent* and *ANSYS CFX*, and the ATCs related to the simulations estimated. The simulation set-up and post-processing effort are identified as having the highest effect on the ATC, and therefore the automated processes focus on reducing the effort of these tasks. Simulations of the same test site are carried out with the new automated processes, and compared to the manual processes as well as to an industry-standard tool, *WindSim*. The new automated tools are found to reduce the ATC of this case by a factor of 12 for *Fluent* and seven for *CFX*, to approximately half the value of *WindSim*. All three simulations show similar deviations compared to measurements and therefore these results are comparable. It should be noted that these results are highly specific to this case, and the absolute cost-saving values cannot be directly transferred to other cases. Nevertheless, it can be concluded that these new processes have significantly reduced the Actual Total Cost and are likely to have a large effect on the choice of the most cost-effective model for a given wind energy project. On-going work involves quantifying the effect of these cost savings on the choice of most appropriate model for this simulation site and further comparing the results to measurements.

Keywords: Wind resource assessment, complex terrain, CFD, cost reduction, comparison metrics

1. Introduction

For a given wind resource assessment (WRA) project, the modeller is faced with a difficult choice of simulation tools with varying accuracies, complexities and costs. In

order to help modellers choose the most suitable tool for a given project, a new method for estimating the accuracy, or skill, and cost of each tool was developed in previous work [1]. This involves the modeller estimating scores for a range of weighted parameters relating to the skill and cost of the tool, using a pre-defined template. An initial study applying this new method to a range of simulation tools for the Bolund Hill experiment [2] has shown that it works well for modelling microscale effects [1]. One of the most interesting findings was that commercial Computational Fluid Dynamics (CFD) flow modelling tools that do not have in-built automated processes especially designed for wind energy, such as *ANSYS Fluent*, lost out significantly compared to specific wind energy tools such as *ZephyCFD*, even though the simulation accuracy (in terms of the average Root Mean Square Error compared to measurements) was slightly higher.

Some organisations carrying out WRAs may prefer the flexibility offered by commercial CFD tools. Therefore, in order to reduce the overall costs related to applying commercial CFD modelling tools to WRA - defined in this paper by the Actual Total Cost (ATC) - new automated processes for applying *ANSYS Fluent* and *ANSYS CFX* to wind modelling in complex terrain have been developed in this present work, and the relative ATCs compared to the wind energy tool *WindSim*. This was achieved by firstly defining an appropriate test site and simulation case, then setting up and carrying out initial simulations, as described in Section 2. Next, the main cost-saving potential was identified and automated processes were developed, and the relative ATCs for each automated simulation compared to the initial simulations using the manual processes for the same test site, as described in Section 3.

2. Simulation set-up

2.1. Test site

The test site chosen for this work was a complex terrain, partly-forested site close to Stötten in southern Germany, whose central feature is a steep incline above 30% and a main wind direction of 270º–300º. Wind speed and direction data was available from a met mast located about 1 km away from the incline, as well as a lidar, as described in [3]. In this paper, the met mast data from four cup anemometers with an accuracy 1% and three 3D sonic anemometers with an accuracy of 1.5% were used for calibration, and the wind speed data recorded using the SWE-Scanner, a fast pulsed lidar wind scanner based on a Leosphere Windcube V1 system with an adapted scanner unit, were used for validation. For the measurement campaign used in this work, the lidar was positioned approximately 300 m west of the met mast for approximately one year (March 2015 - February 2016). An overview of the site and the wind rose from the met mast at 98 m over the entire measurement period of 16 months (March 2015 - July 2016) are shown in Figure 1.

In this work, a geostrophic wind speed of 10 m/s at a height of 500 m and a logarithmic profile down to the ground was used, and simulations with the commercial
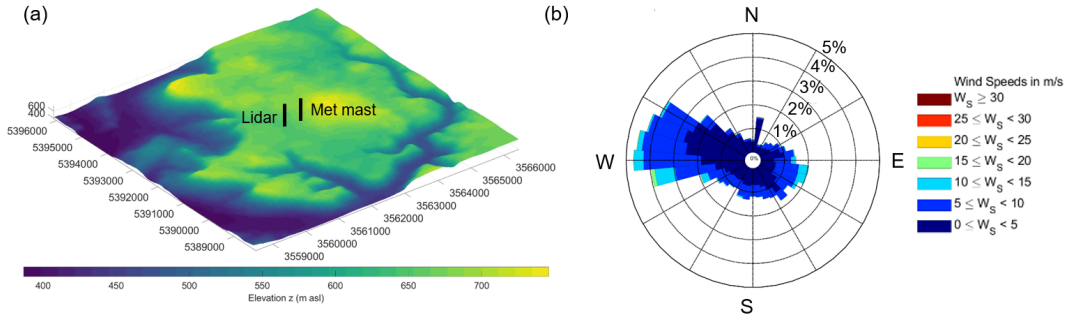
Figure 1. The test site Stötten: (a) Height contours with the met mast positions marked; (b) The wind rose of the met mast data over the entire measurement period.

CFD solvers *ANSYS Fluent* and *ANSYS CFX* as well as with a wind industry tool, *WindSim*, were carried out as described in the next sections. It should be noted that the *ANSYS* add-on *WindModeller* has not been considered here, as the focus was on entirely generic tools. For all the simulations, a digital elevation model and surface roughness from the Copernicus database [4] were used with a horizontal resolution of 25 m, and a neutral Atmospheric Boundary Layer (ABL) with an average turbulence intensity of 10% was assumed. The ground boundary conditions were set to no-slip and individual roughness lengths were attributed to each land use of the domain according to the work of Wieringa [5] and Silva et al. [6] for all simulations. The average wind profile over the entire measurement period from the met mast data was used to define the inflow wind profile, but because the met mast was not located at the inlet, the resulting wind speeds at the met mast deviated from the input profile. In order to re-create wind conditions equivalent to the average conditions over the entire measurement period, the ratio between the simulated wind speed at the met mast location for the 98 m height and the average wind speed at this height over the entire measurement period was calculated. This ratio was then used to linearly scale all of the simulated wind speeds in the entire domain. The lidar data were used to check the accuracy of the simulations by comparing the average RMSE between measurements and simulations (for overlapping time periods only). The set-ups are summarised in Table 1. Varying grid sizes and set-ups were chosen deliberately in order to be able to compare their effects on the total costs. The costs comparisons are not meant as direct comparisons of general tool quality.

2.2. WindSim

WindSim is a Computational Fluid Dynamics (CFD) model based on the PHOENICS code, a 3D Reynolds Averaged Navier Stokes (RANS) solver from the company CHAM. The RANS equations govern the transport of the averaged flow quantities, and can model the whole range of turbulence fluctuations using turbulence modelling. The

approach is very popular for engineering applications because it allows reasonably accurate modelling of a wide range of flow phenomena, but with significantly lower computational effort than other approaches such as Large Eddy Simulations (LES). In this work, the simulation of the Stötten site was conducted using the standard k-ϵ turbulence model [7]. Based on the experience of the user and the guidelines of the tool, an 8 km by 8 km by 5 km domain was created with about 4.5 million cells with a horizontal resolution of 25 m and a stretched grid on the vertical axis providing four layers in the lowest 100 m and then 31 more layers to the model roof. The boundary condition at the top was fixed pressure. The wind rose measured at the met mast was used as an input in order to simulate all wind sectors.

2.3. ANSYS Fluent

Fluent is a commercial CFD tool for modelling the flow in industrial applications and can be set up to solve the RANS equations, as well as for the Large Eddy Simulations (LES) or Detached Eddy Simulations (DES) approach. In this work, *Fluent* was first set up to solve the RANS equations with the SST k-ω turbulence model [8], and then the results were used to initialise DES, which was performed unsteadily with a time step of 1 ms. After an additional unsteady initialisation, the wind speeds were averaged over 10 minutes. Following a mesh sensitivity study, a 8 km by 8 km by 1.5 km structured mesh with a horizontal resolution of 20 m and 20 million cells was created. The first cell height was chosen to be 1 m and the average growth rate 1.2. The Delayed Detached Eddy Simulation (DDES) model was applied, which ensures that RANS is preserved, even in high-aspect ratio boundary layers. In order to introduce fluctuating velocities at the inlet, the *Fluent Synthetic Turbulence Generator* was used. The wind rose measured at the met mast was used to choose the three main wind directions, and only these directions were simulated. In order to obtain results for all wind sectors, the three main wind directions were scaled linearly according to their measured average wind speeds and frequency distributions.

2.4. ANSYS CFX

The all-purpose CFD software *ANSYS CFX* (version 19.2) was applied using a URANS (Unsteady RANS) approach in combination with the k-ϵ model [7]. The equation system was extended to an anelastic formulation whereby the density is influenced by buoyancy forces using the Boussinesq approximation [9]. The Earth's rotation was considered with additional terms in the momentum equation to describe the Coriolis force at a given angular velocity. Forested areas were considered through a canopy model [10]. Individual roughness lengths of the canopy were attributed to each land use and are chosen according to the work of Wieringa [5]. These methods have previously been applied successfully to another application [11]. A structured mesh of about 17 million cells was created over an area of 22 km by 20 km reaching an height over ground of 2 km. The first layer above the ground was adjusted to an height of 3 m. Further layers

up to an height of 60 m were dimensioned to a growth rate from 1.3 to 1.5, giving 36 cell layers in total. Static boundary conditions were applied around the resolved volume for the chosen wind direction and wind speed, forming a Taylor Spiral. Since the upper boundary condition of the domain is flat, but the ground surface is not, the mass flow rate was balanced at the top surface of the domain in order to prevent an acceleration due to mass conservation of the setup. The wind rose measured at the met mast was used to choose the three main wind directions, and only these directions were simulated. In order to obtain results for all wind sectors, the three main wind directions were scaled linearly according to their measured average wind speeds and frequency distributions.

Table 1. A summary of the simulation set-ups for all three tools applied in this work.

Set-up	WindSim	Fluent	CFX
Type	CFD RANS	CFD DDES	CFD URANS
Turbulence model	k-ϵ	SST k-ω	k-ϵ
Grid dimensions	8 km by 8 km by 5 km	8 km by 8 km by 1.5 km	20 km by 22 km by 2 km
First layer cell height	4.8 m	1 m	3 m
Total number of cells	4.5 million	20 million	17 million
Atmospheric model	None	None	Non-hydrostatic anelastic formulation
Earth's rotation	None	None	Additional term for Coriolis force
Forest model	None	None	Canopy model

3. Automated processes

After carrying out simulations with the three tools, the main cost-saving potential was assessed by estimating the Actual Total Costs (ATCs) related to carrying out the simulations. The ATC was calculated by splitting the costs up into the categories described below and adding up the totals. In order to compare the results with each other fairly, the results were normalised according to the following assumptions: (1) Number of years of usage of the software = 10; (2) Number of projects per year = 12; (3) Staff hourly rate = \$80/hour; (4) Computational cost per core per hour = \$0.04/hour/core; (5) Number of cores = 100; (6) Processor clock speed = 2 GHz.

- **Software costs**: estimated by dividing the total license and support costs by the number years of usage, and dividing this by the number of projects carried out per year;

- **Time to learn and training costs**: estimated by adding the staff costs for the time taken to learn how to use the tool to any training costs, and dividing this by

the number of projects carried out per year and the estimated number of years of usage;

- **Simulation set-up effort costs**: estimated by recording the number of hours required to set up the simulations and multiplying this by the hourly staff rate (for all calculated wind directions);
- **Simulation run time costs**: estimated by recording the run-time of the simulations and the number of cores that they were run on, and multiplying this with the computational cost per core per hour (for all calculated wind directions);
- **Post-processing effort costs**: estimated by recording the number of hours required to post-process the results and multiplying this by the hourly staff rate (for all calculated wind directions).

The results are shown in Table 2 in terms of the relative ATCs compared to the maximum ATC of all three tools. It can be seen that the costs for the simulation set-up effort and the post-processing effort far outweigh the other costs, especially for the *Fluent* and *CFX* simulations. The ATCs for the *Fluent* and *CFX* simulations are very similar, and approximately twice as high as the *WindSim* simulations. The main cost-saving potential was therefore identified to be in the terrain and mesh creation process, as well as in the post-processing. These areas were focused on for the development of the automated processes for *Fluent* and *CFX*. The resulting automated processes are described in the next sections.

Table 2. The relative ATCs of the initial simulations for all three tools applied in this work (without automated processes).

Cost category	WindSim	Fluent	CFX
Software costs	0.6%	3.9%	3.9%
Time to learn and training costs	0.3%	0.7%	0.9%
Simulation set-up effort	14.9%	55.9%	32.6%
Simulation run time	0.04%	2.4%	4.7%
Post-processing effort	5.6%	37.2%	55.9%
Relative Actual Total Cost (ATC)	**21.4%**	**100.0%**	**97.9%**

3.1. General process

In order to identify parts of the general simulation process that require automation, a general process was firstly generated as shown in Figure 2(a). In this process, the wind data, such as met mast measured wind speeds and directions, are first processed in order to create input wind profiles for the simulation set-up (step 5), for comparison data for the post-processing (step 7) as well as in order to create the 3D digital model at the required angle (step 3). Next, the site data, such as the terrain and land cover data, is

processed to produce raster data that can be used to create a 3D digital model in the next step. Then the mesh is created for the required wind directions and input into the simulation set-up space, which is combined with initial parameters as well as additional source terms, if being used. This produces a simulation file, which can then be used to actually carry out the simulations. The results are then combined with any available comparison data in a post-processing step, and the quality of the results are checked using additional comparisons with measurement data. Steps 3-5 are then repeated for all selected wind directions. The post-processing includes the scaling process discussed in Section 1 as well as the linear scaling of non-simulated wind speeds using the wind rose as discussed in Section 2.

3.2. Fluent

The automated steps implemented for *Fluent* in this work are shown in Figure 2(b). It consists of the following tools or scripts:

- **MATLAB pre-processing tool**: The wind measurement data is filtered in and a first automated data evaluation is performed and the required wind directions chosen manually. A wind profile is created for each wind direction, which serves *Fluent* as an inlet boundary condition. A tessellated surface model (.STL) is generated from the GIS terrain data, which is then cut and aligned depending on the chosen wind direction. Finally, a profile of the surface condition or roughness is generated.

- **Glyph2 meshing script**: The surface model is transferred to the meshing programme *Pointwise*, which automatically generates a structured mesh with the desired resolution using a Glyph2 script, which then projects it onto the surface model. A structured mesh up to the defined maximum height is then generated and optimised using internal functions. Boundary conditions such as inlet and outlet are assigned to the resulting 3D domain and exported as a *Fluent* setup file (.cas). The time needed for mesh generation can therefore be reduced to a minimum.

- **Fluent simulation script**: A scheme script merges the automatically generated inputs and defines important parameters such as time step length and turbulence model. Monitoring points and lines are created for the assessment of wind conditions at pre-defined locations. The script currently works in a way that in a first step a stationary RANS simulation is performed. This data is then used as initialisation for a transient DDES simulation.

- **MATLAB post-processing tool**: This tool is used to compare simulation results in a pre-defined manner. It automatically plots and compares the average wind profiles and directions with the measurement results, and calculates the Root Mean Square Error (RMSE) between simulations and measurements. Additionally, the data is stored in a pre-defined format and location.

3.3. CFX

The automated steps implemented for *CFX* in this work are shown in Figure 2(c). It consists of the following tools or scripts:

- **Python pre-processing tool**: A script was written in Python in order to filter measurement data and to classify the data in sectors of wind direction. The proportion of measured velocities for each wind direction delivers a priority list of to be simulated wind directions for the consecutive process of post-processing. The boundary conditions for the respective wind class are generated based on the Taylor spiral and the local ground elevation.

- **Boxmesher**: A fully automatized in-house mesh generator is used to generate a hexahedral mesh within the user-specified volume. The land cover as well as the digital elevation model are read in dxf- and grid-file format, respectively, and cut along the boundaries of the computational domain. The extent of the computational domain, the horizontal and vertical cell sizes are set in an input text file. The mesh resolution in the vertical direction, especially near the wall, can be well controlled by defining distances of grid lines or growth factors. For a defined ground layer thickness the cells are extruded perpendicular to the ground surface as far as possible. The tool reliably produces high quality meshes in CGNS-format within seconds.

- **CFX simulations**: In this step, the *ANSYS CFX5PRE* software is used for the build-up of the numerical set-up. Solver settings, roughness lengths and functions implementing the anelastic formulation, Coriolis force as well as the canopy model are defined for all the computed cases. They are used for all the cases computed and therefore only the boundary conditions coming from the Python pre-processing tools have to be initialised.

- **Python post-processing tool**: For each simulated wind direction, this tool carries out a check by giving the differences of each averaged velocity level for each wind direction sector at all reference positions provided.

3.4. Relative Actual Total Costs

The simulated wind profiles at the met mast and lidar positions are compared to measurements in Figure 3 for the 270º case. It can be seen that the simulated profiles match the measured profiles reasonably well. The average Root Mean Square Error (RMSE) over all heights between the simulations and the lidar measurements was 0.38 m/s for *WindSim*, 0.55 m/s for *Fluent* RANS, 0.55 m/s for *Fluent* DES, and 0.62 m/s for *CFX*. The reason for these discrepancies are being studied further.

The resulting relative ATCs estimated for the three tools are compared in Figure 4, for both the original (manual) *Fluent* and *CFX* simulations as well as the new automated processes. The dominating costs of the simulation set-up effort and post-processing
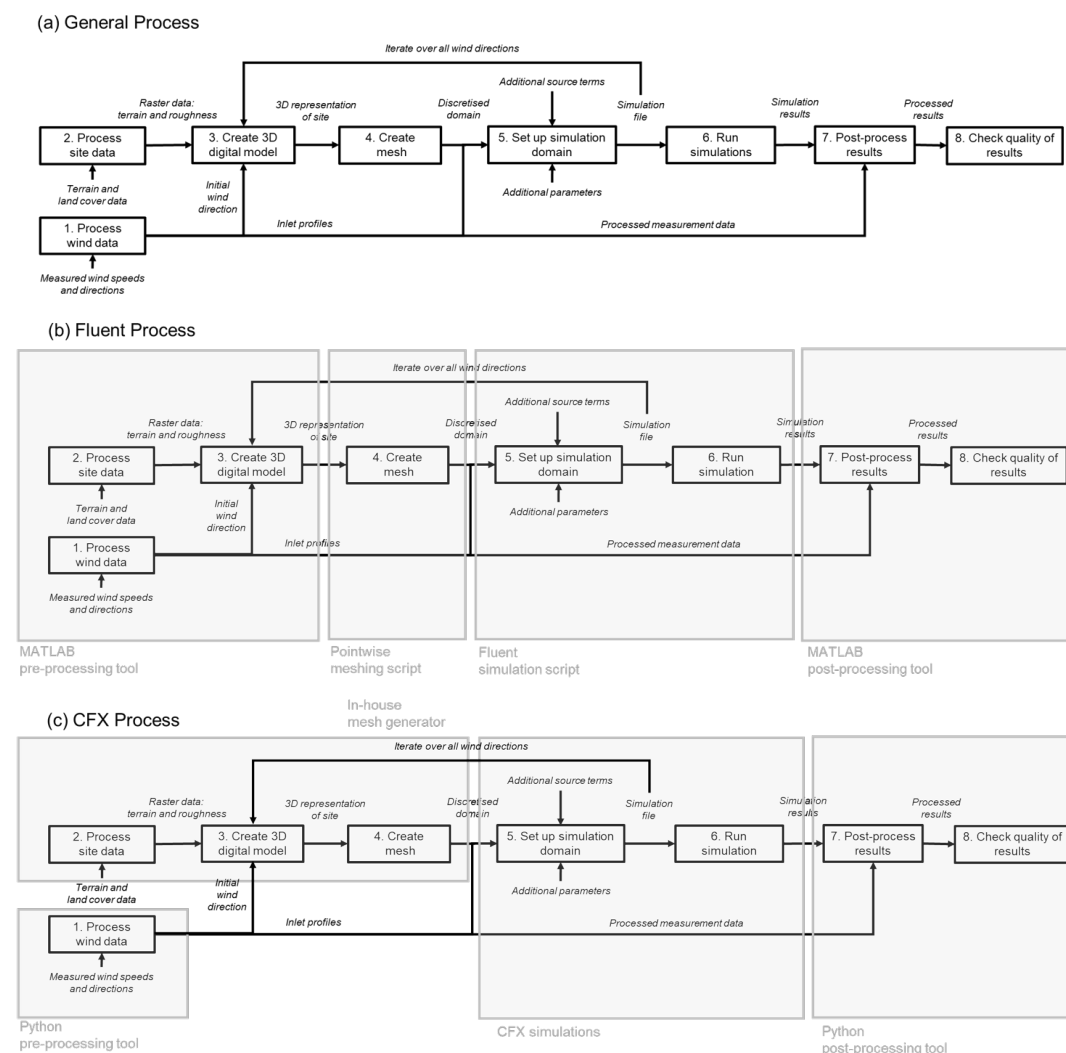
Figure 2. Schematic of the simulation processes: (a) General process; (b) Fluent process, (b) CFX process.

effort are clear to see, especially for the manual *Fluent* and *CFX* simulations. The new automated processes have lead to a significant drop in relative ATC (12 times cheaper for *Fluent* and 7 times cheaper for *CFX*). It is very interesting that these new processes have allowed the ATCs to be reduced even below those of *WindSim*, despite the higher software / license costs. This is because the *WindSim* simulations require a certain amount of set-up effort that the new automated processes do not.

As the different set-ups and user skills will significantly affect the costs, it is important to note that these results should not be used to directly compare the general costs of different tools with each other, but only to compare specific simulation set-ups with each other. This can be useful in helping modellers choose the most appropriate tool for a certain project, when combined with skill scores as described in the introduction. In other words, the results should not be interpreted as the costs of the tools themselves, but as a combination of the user, the tool and the set-up. This is a great strength
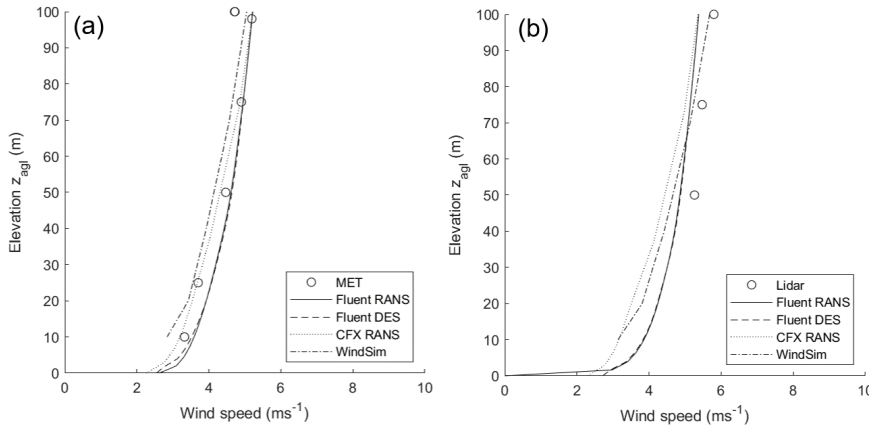
Figure 3. Comparison between the simulated and measured wind speeds at the (a) met mast, (b) lidar locations.
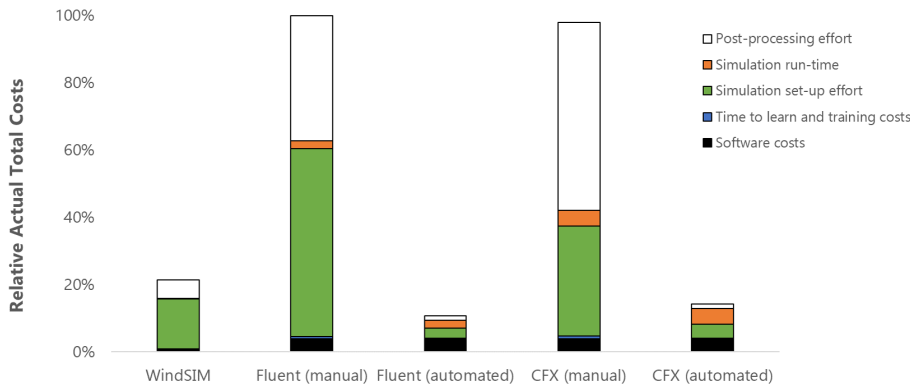

Figure 4. Relative ACTs of all three simulation tools as well as of the new automated processes.

of this method for real applications, because in reality the costs always result from the combination of user, tool and set-up. Nevertheless, the manual and automated processes can be directly compared with each other, as the set-up and user was the same, and therefore it can be concluded that these new processes have significantly reduced the total simulation costs and are likely to have a large effect on the choice of the most cost-effective model for a given wind energy project. On-going work involves comparing these costs to the actual simulation accuracies, as well as refining the skill score prediction process.

4. Conclusions

New automated processes for applying the commercial CFD tools *ANSYS Fluent* and *ANSYS CFX* to wind modelling in complex terrain have been developed with the goal of decreasing the Actual Total Costs (ATCs) related to planning wind energy projects.

Initial simulations were carried out at the complex terrain site St'otten in southern Germany using *ANSYS Fluent* and *ANSYS CFX*, and the total costs related to the simulations estimated. The simulation set-up and post-processing effort were identified as having the highest effect on the ATCs, and therefore the automated processes were focused on reducing the effort of these tasks. Simulations of the same test site were carried out with the new automated processes, and compared to the manual processes as well as to an industry-standard tool, *WindSim*. The new automated tools were found to reduce the ATCs of the simulation case by a factor of 12 for *Fluent* and 7 for *CFX*, to approximately half the value for *WindSim*. All three simulations showed similar deviations compared to measurements and these results are therefore comparable. Further work involves examining the relationship between simulation accuracy and costs.

5. References

[1] S. Barber, A. Schubiger, N. Wagenbrenner, N. Fatras, and H. Nordborg. A new method for the pragmatic choice of wind models for wind resource assessment in complex terrain. *Wind Energy Science*, (Discussion paper), 2020.

[2] J. Berg, J. Mann, A. Bechmann, M. S. Courtney, and H. E. Jorgensen. The bolund experiment, part i: Flow over a steep, three-dimensional hill. *Boundary-Layer Meteorology*, 141(2):219–243, 2011.

[3] C. Schulz, M. Hofsäß, J. Anger, A. Rautenberg, T. Lutz, P. Wen Cheng, and J. Bange. Comparison of different measurement techniques and a cfd simulation in complex terrain. *Journal of Physics: Conference Series*, 753:082017, 2016.

[4] Copernicus. https://land.copernicus.eu/.

[5] J. Wieringa. Representative roughness parameters for homogeneous terrain. *Boundary-Layer Meteorology*, (63(4)):323–363, 1993.

[6] J. Silva, C. Ribeiro, R. Guedes, M. Rua, and F. Ulrich. Roughness length classification of corine land cover classes. In *Proceedings of EWEC*. 2007.

[7] B. E. Launder and D. B. Spalding. The numerical computation of turbulent flows. In *Numerical prediction of flow, heat transfer, turbulence and combustion*, pages 96–116. Elsevier, 1983.

[8] F. R. Menter. Zonal two equation k-omega turbulence models for aerodynamic flows. *AIAA*, (Paper 93-2906), 1993.

[9] C. Montavon. Validation of a non-hydrostatic numerical model to simulate stratified wind fields over complex topography. *J. Wind Eng. Industrial Aerodynamics*, (74–76):273–282, 1998.

[10] J. Liu, J. M. Chen, T. A. Black, and M. D. Novak. k-epsilon modelling of turbulent air ow downwind of a model forest edge. *Boundary-Layer Meteorology*, (77(1)):21–44, 1996.

[11] H. Knaus, A. Rautenberg, and J. Bange. Model comparison of two dierent non-hydrostatic formulations for the navier-stokes equations simulating wind flow in complex terrain. *Journal of Wind Engineering and Industrial Aerodynamics*, (169):290–307, 2017.