Structure Wind Load Simulation Study using *RWIND Simulation* and *TCFD*

The following study is aimed at the simulations of wind-induced loads on general structures. Particularly, at a comparison of two software pieces with two different simulation strategies. RWIND Simulation software is a specialized tool for rapid CFD simulations of wind load on a large variety of structures. TCFD is a general hi-tech CFD software, designed for a wide range of industrial applications. The test case used in this study is the Burj Al Arab, a luxury hotel located in Dubai, United Arab Emirates.

Keywords

CFD, SIMULATION, WIND LOAD, STRUCTURES, BENCHMARK, RWIND Simulation, TCFD, AERODYNAMICS, AIR, AIRFLOW, STEADY-STATE, RANS, AUTOMATION

Benchmark Parameters

- Typical flow speed: *25 m/s*
- Flow model: in*compressible*
- FVM Mesh size: *0.5-5.0M cells*
- Medium: *air*
- Reference pressure: *1 atm*
- Reference density: *1.2 kg/m³*
- Dynamic viscosity: *1.8×10^{-5} Pa·s*
- Turb. Model: *RANS*

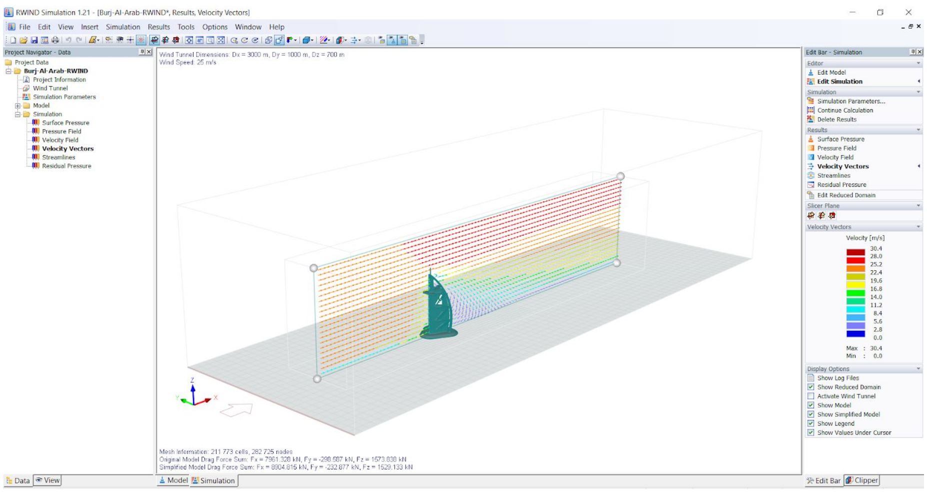
Test Case Description

The test case used in this study is the Burj Al Arab, a luxury hotel located in Dubai, United Arab Emirates. Of the tallest hotels in the world, it is the seventh tallest (2019), although 39% of its total height is made up of non-occupiable space. Burj Al Arab stands on an artificial island 280 m (920 ft) from Jumeirah Beach and is connected to the mainland by a private curving bridge. The shape of the structure is designed to resemble the sail of a ship. It has a helipad near the roof at a height of 210 m (689 ft) above ground[1].

Particularly for this study, it has been considered that the wind speed is 25 m/s. The wind attacks the building symmetrically from the seaside and the wind velocity profile is considered to be constant in time and space. The reference density is 1.2 kg/m^3. Dynamic viscosity is 1.8×10^{-5} Pa·s. The main goal of the numerical analysis is evaluating the forces acting on the building surface.

RWIND Simulation Case Setup

RWIND Simulation software, developed by companies PC-PROGRESS and DLUBAL, was designed as a specialized tool for rapid CFD simulations of wind load on a large variety of structures. RWIND Simulation works as standalone software, or it can be directly connected with structural design software RFEM or RSTAB.

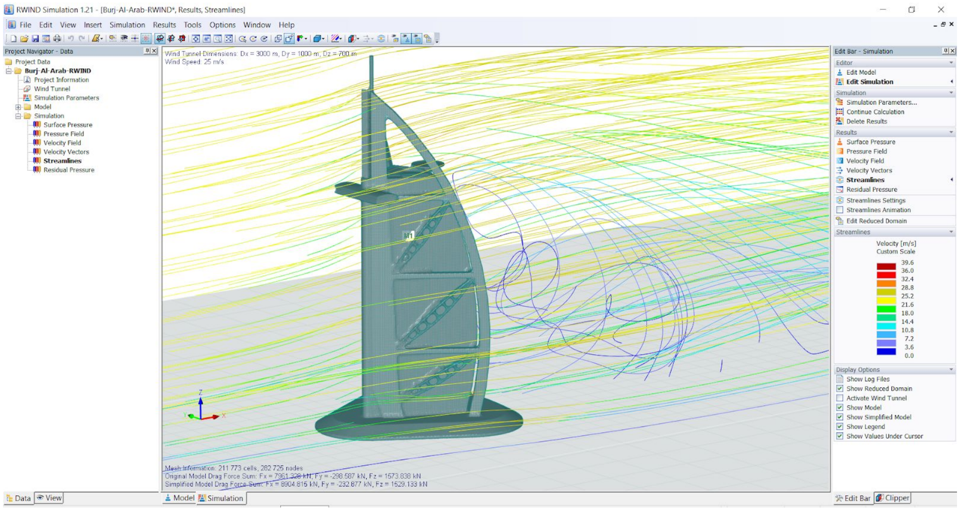


RWIND Simulation user interface is super ease of use with minimal necessary settings and user skills. The workflow is very simple. The input for RWIND Simulation is the surface model of the structure (CAD, .STL). The virtual wind tunnel is created around the structure. Wind speed is set. The rest of the parameters are not mandatory, but available.

RWIND Simulation is extremely robust on model complexity because it uses the so-called *simplified model*. Details that are not relevant to the given simulation and could cause simulation instability due to insufficiently fine discretization are simplified. The simplified model actually represents a special mesh by *shrink-wrapping* the original model. When simplifying the model, it is possible to specify the level of detail as well as the maximum size of holes to be closed. Such a mesh is topologically correct and can, therefore, be used as a model boundary for the generation of a 3D finite volume mesh. The simplified model will

[1] https://en.wikipedia.org/wiki/Burj_Al_Arab 31.8.2019

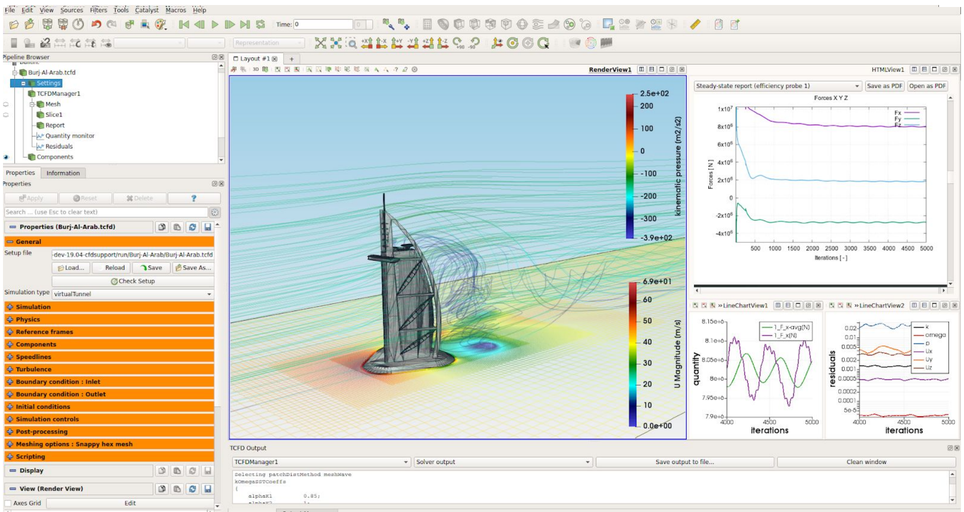
automatically correct most of the problems that would otherwise have to be corrected manually. RWIND Simulation supports parallel simulations on multiple CPU cores.



RWIND Simulation numerical model is based on Finite-Volume code *OpenFOAM*. The final volume mesh is created via application *snappyHexMesh*. The turbulence model is RANS k–ε. A typical simulation time with RWIND Simulation is a few minutes. The size of the finite volume mesh significantly determines the total simulation time. In this particular project, the virtual tunnel is 3000m long, 1000m wide, and 700m high. The basic unrefined mesh cell is a cube 60m wide. That leads to the finite volume mesh of about 200,000 cells. Simulation based on such a mesh converges in about three minutes on six CPU cores.

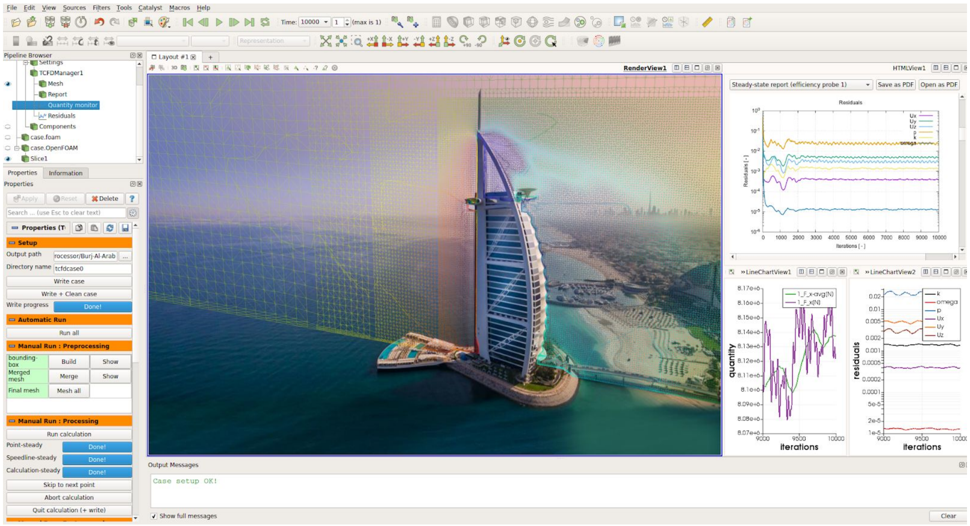
TCFD Case Setup

[TCFD](), developed by company [CFDSUPPORT](), is a general hi-tech CFD software, designed for a wide range of industrial applications. It successfully merges the benefits of both open-source and commercial code.



TCFD numerical solver is based on OpenFOAM. TCFD shows great performance at the external aerodynamics of various structures. TCFD is natively compiled for Windows and Linux operating systems. No need of any virtual systems. The engineering workflow is exactly the same on both systems. TCFD includes an automated meshing system. Based on snappyHexMesh. And also, the external meshes created in another software can be loaded. TCFD has strong postprocessing. Every simulation that is executed in TCFD, has its own HTML report with the results. Volume fields are visualized in ParaView. Integral results are saved in CSV files.

In this particular case, the simulation type is *Virtual Tunnel*, especially designed for simulations external aerodynamics. The turbulence model is RANS k-omega SST. The finite volume mesh is created using snappyHexMesh. The virtual tunnel is 3000m long, 1000m wide, and 700m high. The mesh is well refined close to the model, including the inflation layers, to model the flow *boundary layer* as accurate as possible. That leads to the finite volume mesh of about 5,000,000 cells. Simulation based on such a mesh converges in eleven hours on six CPU cores.



Results

Exactly the same building model was simulated both in RWIND Simulation and TCFD. Following table shows the main results, forces acting on the building surface in the x (from the sea side), y, and z directions.

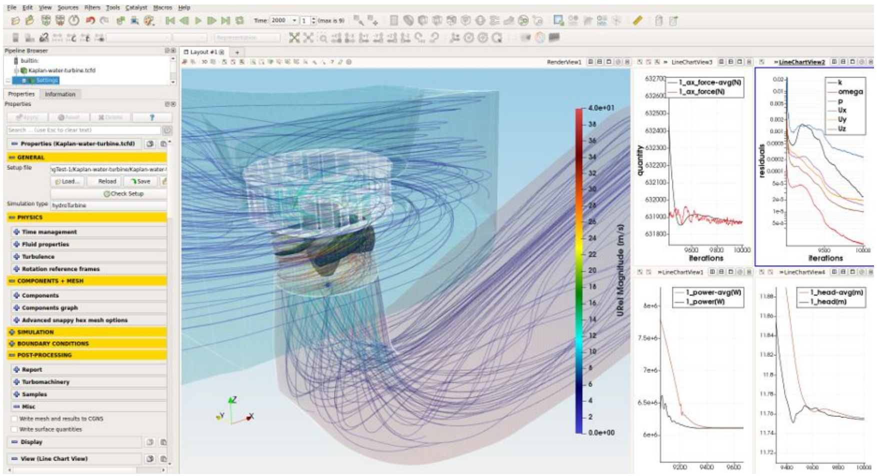
	RWIND Simulation	**TCFD**
Fx [MN]	7.96132	8.1376
Fy [MN]	-0.29858	-0.1525
Fz [MN]	1.57383	1.8624
Simulation time [h:m:s]	0:2:42	10:45:02
Simulation time ratio	239	1

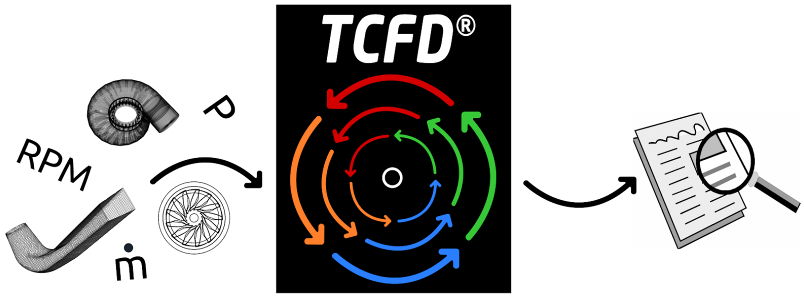
Conclusion

Both RWIND Simulation and TCFD show reasonably good agreement of the obtained results. The difference in their predictions, at the main force acting (Fx) on the building, is about **2.2%**. Both software tools show their strengths. RWIND Simulation is very easy to use. It is superquick, and still gives very reasonable results. TCFD is fully sophisticated, hi-tech CFD software where all the options are open and provide a wide range of methods for detailed CFD analysis. Potential questions regarding TCFD will be answered on request at *info@cfdsupport.com*. Potential questions regarding RWIND Simulation will be answered on request at *support@pc-progress.cz*.

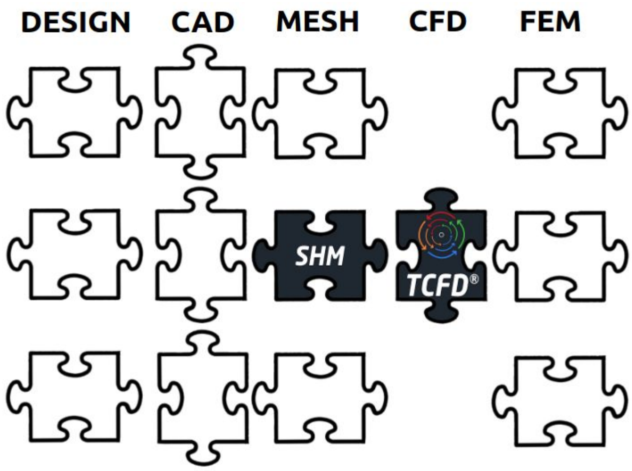
About TCFD®

CFD SUPPORT LTD. creates a new generation of CFD simulation tools. [TCFD®](https://www.cfdsupport.com/tcfd.html) massively increases productivity in CFD simulations. It successfully merges the benefits of both **open-source** and **commercial** code: due to its open-source nature, TCFD® is perpetual for an unlimited number of users, jobs, and cores, and it is further customizable. Due to its commercial nature, TCFD® is professionally supported, well tested, ready for the industry, robust, accurate, automated, offers a graphical user interface (GUI), documentation and much more. Since TCFD® is unlimited, it is well-suited to demanding workflows like optimization, transient or aeroacoustics.





TCFD® scales CFD simulations to the available hardware resource. TCFD® is fully automated and flexible. Its beauty is, that it is the user who decides how deeply to dive into CFD or not at all. TCFD® can be used as a black box (data in - data out) or as a fully sophisticated CFD code where all the options remain open at the same time. TCFD® was originally designed for simulations of rotating machinery like Pumps, Fans, Compressors, Turbines, etc. TCFD® proved to be so effective, that it was later extended with many other applications to cover even a wider range of CFD field. TCFD® shows great performance at the external aerodynamics of various objects. TCFD® numerical solver is open-source. TCFD® is not dependent on other software but it is fully compatible with standard other software packages. TCFD® is fully automated and can run an entire workflow with a single command: from data input, the new case is written down, the mesh is created, the case is set up and simulated, the results are evaluated and the results are written down into a report It operates both in graphical interface (GUI) and in batch mode. TCFD® is from the beginning developed to fit any existing CAE workflow. TCFD® has a modular (plugin) character. Any part of it can be used within other applications. It has strong interfaces to cover a wide range of input and output data. Technical support of TCFD® is unlimited.



www.cfdsupport.com | +420 212 243 883 | info@cfdsupport.com
https://www.cfdsupport.com/tcfd.html