Centrifugal Fan CFD Validation Benchmark using TCFD®

The following article is based on the public part of the report on the benchmark validation of a CFD simulation of a centrifugal fan using TCFD®. This project was undertaken jointly by ZVVZ MACHINERY, a.s. in collaboration with CFDSUPPORT. The aim of the benchmark validation was to evaluate the TCFD®, computational fluid dynamics (CFD) software and compare its results with the measurement data. The secondary aim of this project was to investigate the difference between steady-state and transient simulation results.

Keywords

CFD, VALIDATION, BENCHMARK, TCFD, SIMULATION, CENTRIFUGAL, RADIAL FAN, INCOMPRESSIBLE, AIR, TURBOMACHINERY, STEADY-STATE, TRANSIENT, LOW PRESSURE, AIR FLOW, AUTOMATION, SNAPPYHEXMESH

Benchmark Parameters

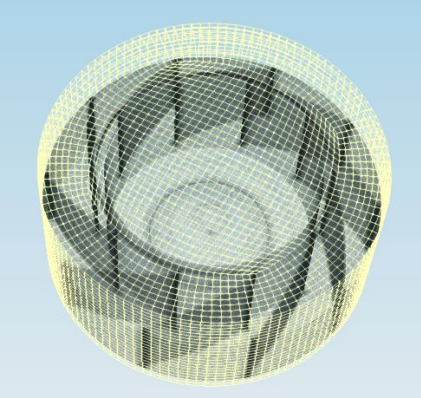
- Typical flow speed: *60 m/s*
- Flow model: *incompressible*
- Rotation RPM: *1200*
- Mesh size: *2.8M cells*
- Impeller diameter: *1.0 m*
- Medium: *air*
- Reference pressure: *1 atm*
- Typical Pressure Ratio: *1.03*
- Reference density: *1.2 kg/m*^{3}
- Dynamic viscosity: 1.8×10^{-5} *Pa · s*
- CPU Time Steady: *6 core.hours/point*
- CPU Time Trans.: *20 core.hours/point*

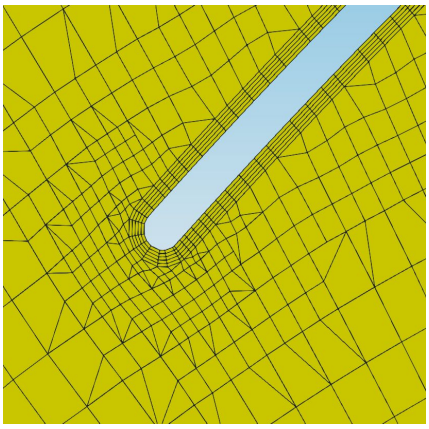
Preprocessing

In this particular project, the mesh for the CFD simulation created from the surface model via an automated process (alternatively, the external mesh can be loaded directly). The original CAD model of the centrifugal fan was in the STEP file format. Original STEP files are usually too complex for comprehensive CFD simulations, so certain preprocessing CAD work is generally required. While the original CAD model for this project was simplified and cleaned using FreeCAD open-source software, any other standard CAD system can be used instead. The principle is always the same: the surface model has to be created; all the tiny, unimportant, and problematic model parts must be removed, and all the holes must be sealed up. This model is split reasonably into individual waterproof components. Then, the final simulation-ready model, the 3D surface in STL file format, has to be refined to a reasonable level. This preprocessing phase of the workflow is extremely important because it determines the simulation potential and limits the CFD results.

Mesh

The computational mesh was created in an automated workflow using *snappyHexMesh*. A cylindrical mesh was used for the initial background mesh of the fan impeller. The mesh refinement levels can be easily changed, to obtain the coarser or finer mesh to better handle the mesh size. Then, a boundary layer mesh of 5 inflation layers was created on the wall surfaces. Alternatively, an external mesh can be loaded.



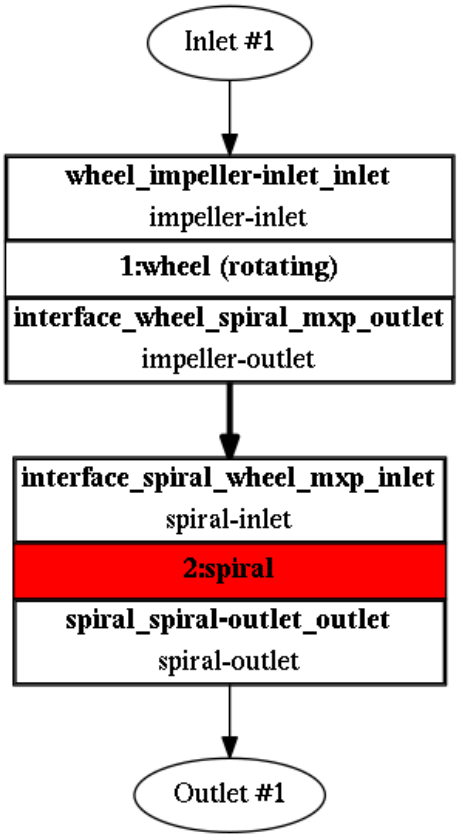


Mesh stats	Impeller	Spiral
points	2542103	3074101
faces	6881199	7920747
internal faces	6541317	7271691
cells	2174548	2433864
faces per cell	6.172554	6.242106
hexahedra	1906820	2117826
prisms	68676	67025
tet wedges	842	607
tetrahedra	35	8
polyhedra	198175	248398

TCFD® Case Setup

Basic TCFD® settings:

- Machine type: *fan*
- Both *Steady-state* and *Transient* calculation
- *Incompressible* fluid flow
- RANS turbulence modeling with *k-ω SST model*
- Number of components: *2*
- Mesh Mesh size: *4.6M cells*
- Inlet: *Flow rate*
- Outlet: *Average static pressure*
- Interface into Impeller: *AMI*
- Interface out of Impeller: *Mixing Plane (10 planes)*

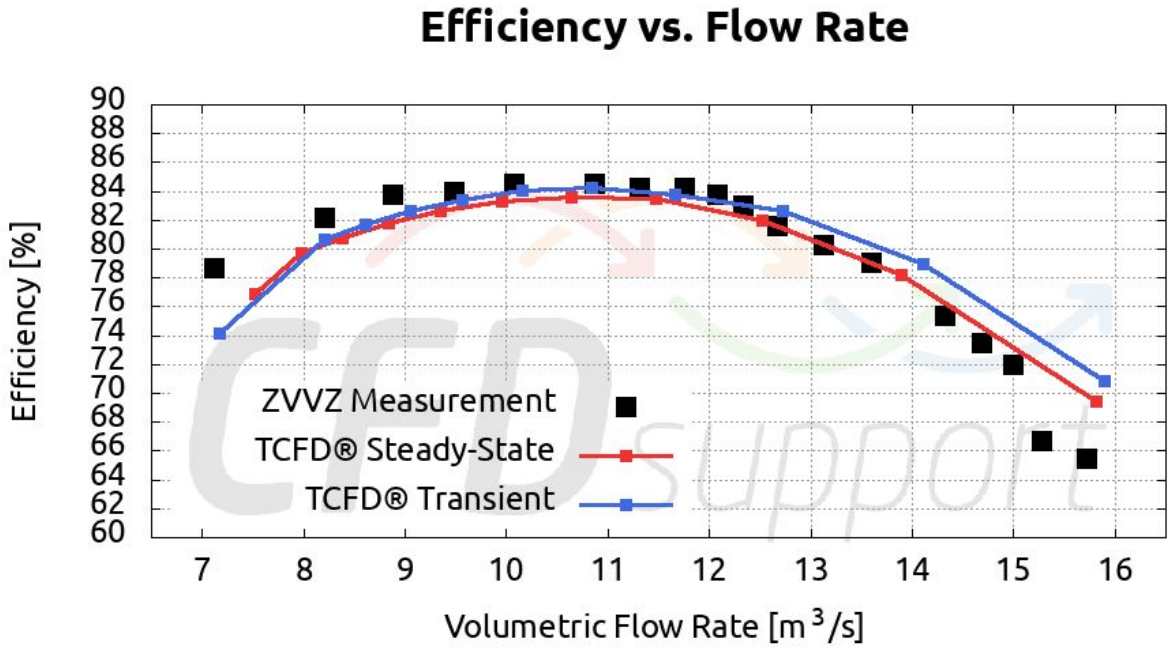


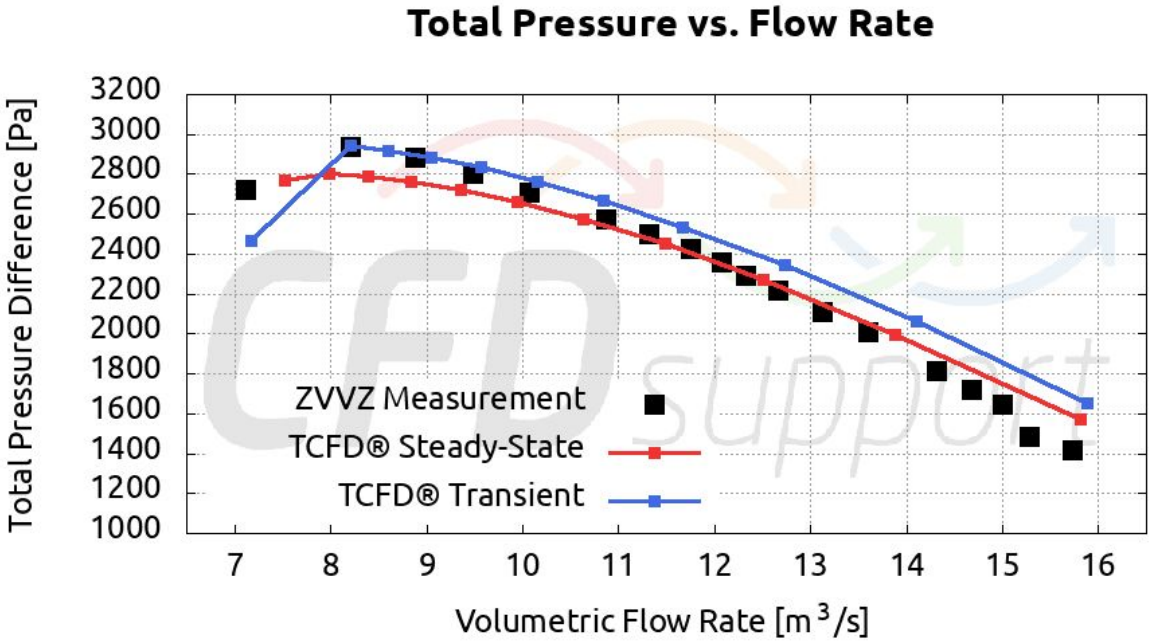
The simulation was executed in the automated workflow in both steady-state and transient modes. The single speed-line of 11 points at 1200 RMP was simulated. TCFD® is able to write the results down at any time during the simulation. The convergence of any quantity is monitored during the simulation. When any simulation point converges sufficiently, the simulation can move onto the next simulation point.

Post-processing

TCFD® includes a built-in post-processing module which automatically evaluates the required quantities, such as efficiency, torque, flow rates, and forces and moments. All these quantities are evaluated throughout the simulation run, and all the important data is summarized in an HTML report, which can be updated anytime during the simulation for every run. Furthermore, visual postprocessing of the volume fields can be done with ParaView.

Results - Comparison with the Measurement Data



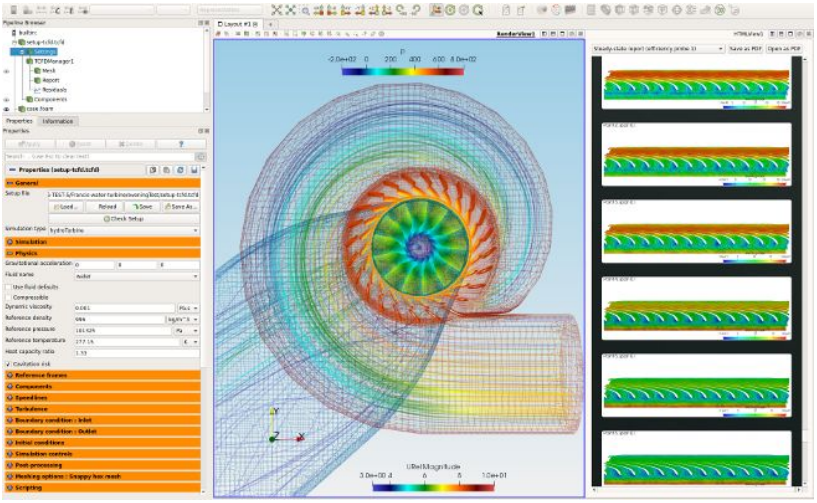


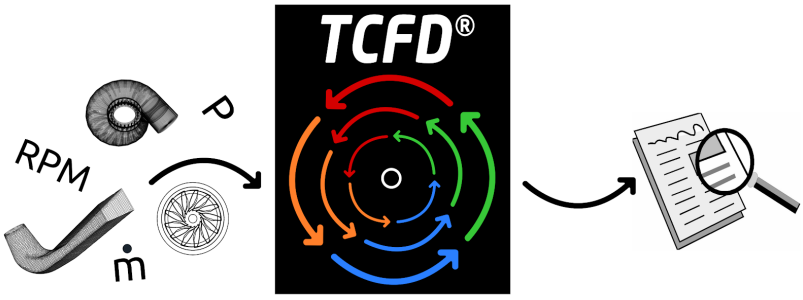
Conclusion

The complex CFD analysis of a centrifugal fan using TCFD® provides very good agreement with the measurement data, as has been demonstrated. Overall, the transient simulation's prediction is slightly closer to the measurement data than the steady-state simulation's.

About TCFD®

CFD SUPPORT LTD. creates a new generation of CFD simulation tools. TCFD® 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. It scales CFD simulations to the available hardware resource and 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 based on OpenFOAM®. TCFD® is not dependent on other software but it is fully compatible with standard OpenFOAM® and 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 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.