

# Centrifugal Compressor 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 compressor using TCFD®. This project was undertaken jointly by [CZ 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.

## Keywords

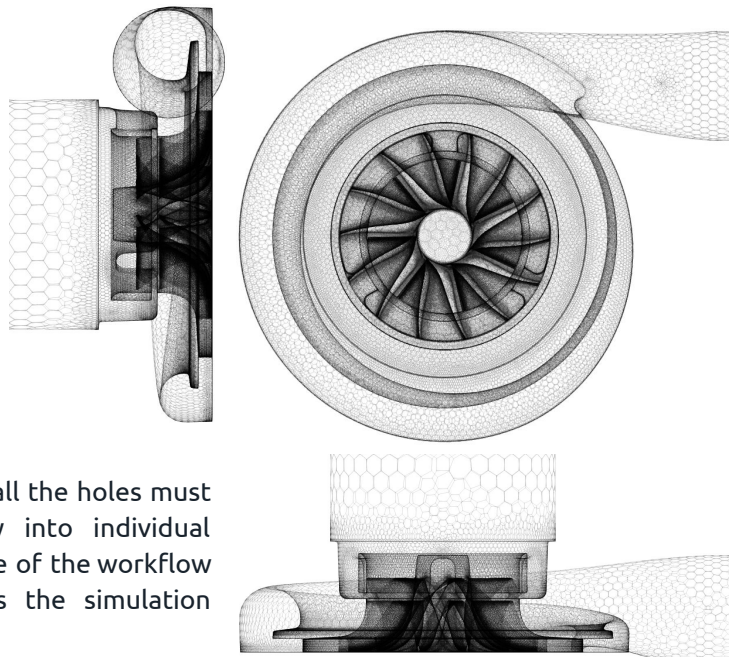
CFD, VALIDATION, BENCHMARK, TCFD, SIMULATION, CENTRIFUGAL, RADIAL, COMPRESSOR, COMPRESSIBLE, AIR, TURBOMACHINERY, STEADY-STATE, TRANSIENT, TURBOCHARGER, AIR FLOW, AUTOMATION

## Benchmark Parameters

- Typical flow speed: 50 m/s
- Flow model: compressible
- Rotation RPM: 90000
- Mesh size: 0.8M cells
- Impeller diameter: 110 mm
- Medium: air
- Reference pressure: 1 atm
- Typical Pressure Ratio: 4.0
- Reference density: 1.2 kg/m<sup>3</sup>
- Dynamic viscosity: 1.8 × 10<sup>-5</sup> Pa·s
- CPU Time: 19 core.hours/point
- Turb. Model: k-omega SST

## Preprocessing

The original CAD model of the centrifugal compressor was in the STEP file format. Original STEP files are usually too complex for comprehensive CFD simulations, so certain preprocessing (cleaning) CAD work is generally required. While the original CAD model for this project was simplified and cleaned using [Salome](#) 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. This preprocessing phase of the workflow is extremely important because it determines the simulation potential and limits the CFD results.



## Mesh

In this particular project, the volume mesh for the CFD simulation created in [Salome](#) from the surface model via NetGen (tetrahedral) and later simplified to polyhedral mesh using special OpenFOAM utility *polyDualMesh*. Alternatively, another external mesh can be loaded directly in *MSH*, *CGNS* or *OpenFOAM* format. Or, the mesh can be created using an automated meshing workflow based on *snappyHexMesh*.

## TCFD® Case Setup

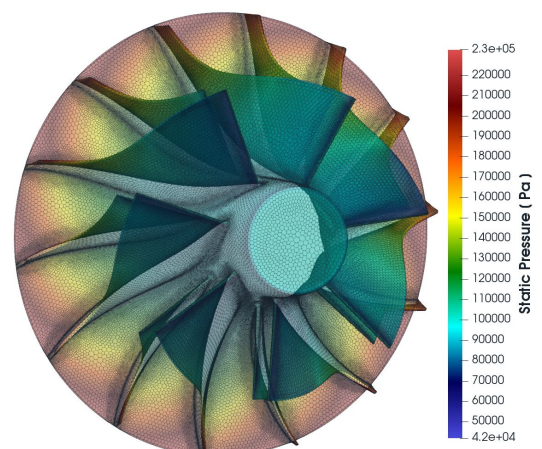
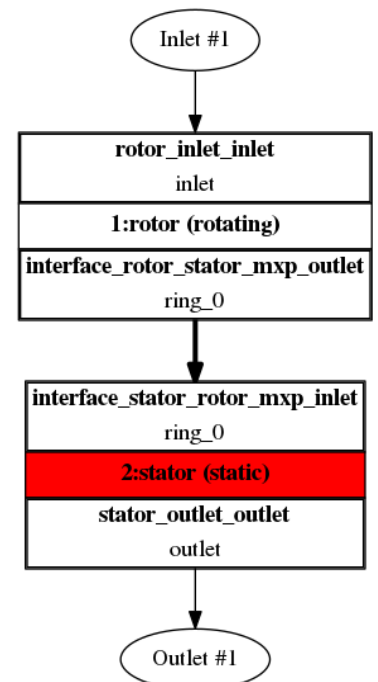
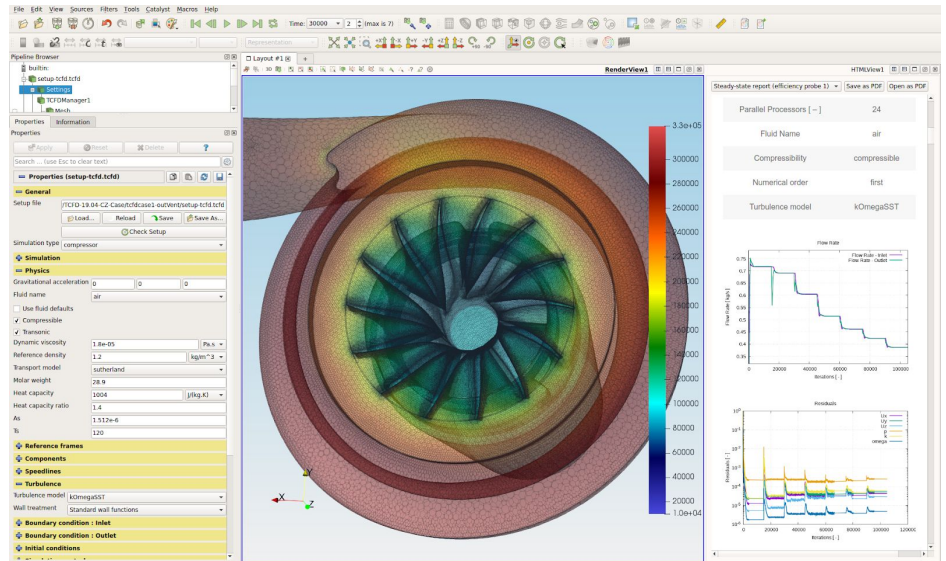
Basic TCFD® settings:

- Machine type: *compressor*
- Both *Steady-state* calculation
- *Compressible* fluid flow
- RANS turbulence modeling with *k- $\omega$  SST model*
- Number of components: 2
- Mesh Mesh size: *0.8M cells*
- Inlet: *Total Pressure*
- Outlet: *OutletVent*
- Interface into Impeller: *AMI*
- Interface out of Impeller: *Mixing Plane (10 planes)*
- Speed-lines: *60, 80, 90 kRPM*
- Simulation points: 33

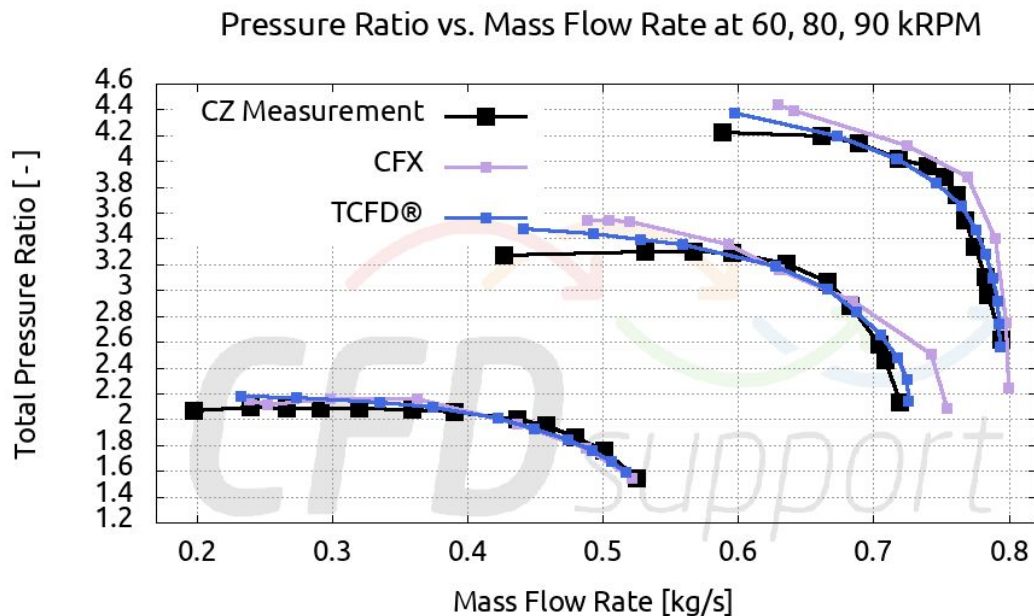
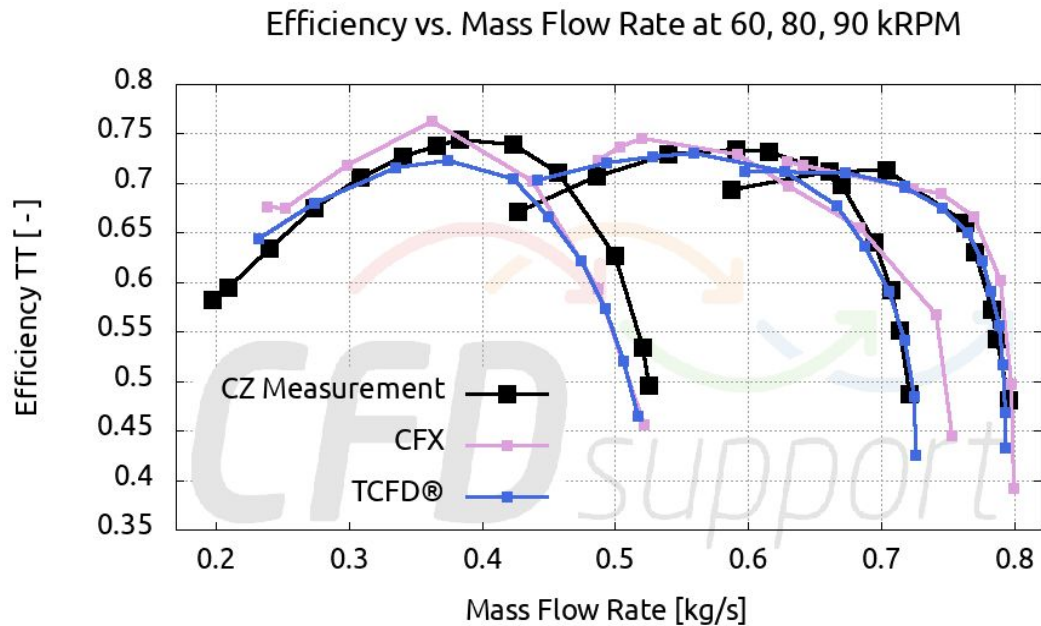
The simulation was executed in the automated workflow in steady-state mode. The three speed-lines at 60, 80, and 90 thousand RMP were simulated. TCFD® is able to write the results down at any time during the simulation. The convergence of any quantity is monitored during the simulation. When any simulation 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, pressure ratio, 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



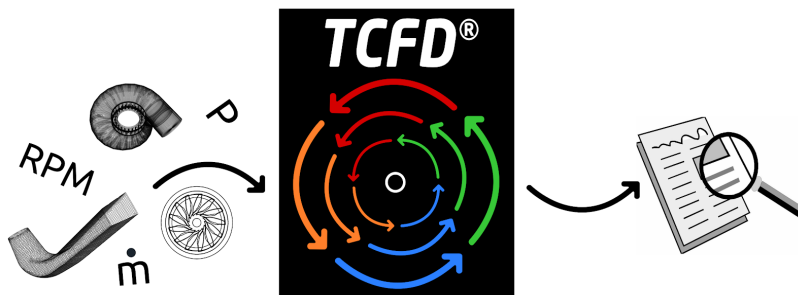
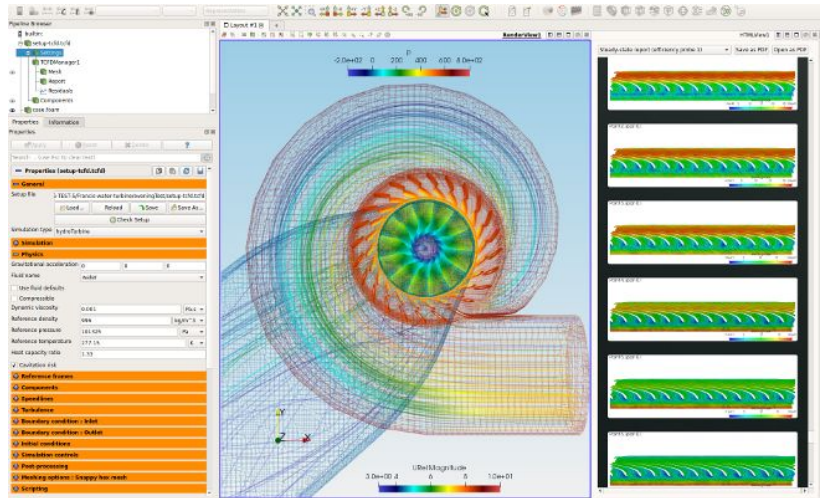
## Conclusion

The complex CFD analysis of a centrifugal compressor using TCFD® provides very good agreement with the measurement data and other CFD code, as has been demonstrated. Potential questions will be answered on request at [info@cfdsupport.com](mailto:info@cfdsupport.com).



## 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.

