

Differences between TCFD and OpenFOAM

OpenFOAM is a set of tools that is free.

TCFD is a complex workflow for real engineering work. Focused. Robust. Tested. Validated. Best Practice. Tech support. Documentation. TCFD includes 20 man-years of focused development.

GUI

- GUI in ParaView
- 100% workflow in GUI
- Case settings
- Simulation run
- Live results monitoring
- PostProcessing
- Reporting
- User & Advanced mode

Boundary conditions

- Mixing Plane
- Periodic AMI
- Directed Flow Rate
- Directed Total Pressure
- Directed Total Temperature
- Hydrostatic Total Pressure
- Outlet Vent
- Fixed Mean Value

CFD Processor

- Full Automation
- Simulation type - focus
- Single configuration file .tcfD
- Dimension Units
- MSH Reader
- Check set-up
- Bind to Core
- Convergence Check
- Reference Frames

Post Processing

- Blade to Blade View
- Meridional Average

Real Tutorials

Tutorials are based on real industrial projects and existing machines that have been tested. Including best practice settings.

Operation system - native compilation

- Linux - native compilation in box
- Windows - native compilation in box

Solvers

- blueSolver - incompressible - robust convergence for limiting quantities
- redSolver - compressible - robust convergence for limiting quantities
- greenSolver - cavitation - robust convergence for limiting quantities

Reporting

- Every simulation has its report
- External Data
- Compare Report

- [Axial Pump](#)
- [Centrifugal Pump](#)
- [Axial Fan](#)
- [Centrifugal Fan](#)
- [Axial Compressor](#)
- [Centrifugal Compressor](#)
- [Axial Turbine Stage](#)
- [Radial Turbine](#)
- [Francis Turbine](#)
- [Kaplan Turbine](#)
- [Wind Turbine](#)
- [Spitfire](#)
- [Hydraulic Valve](#)
- [Double Rotor Fan](#)
- [DrivAer Car Model](#)