Francis Turbine CFD Benchmark using TCFD

This report presents the CFD benchmark validation of Francis hydro turbine. This project was made together with Hidroenergia, a hydro turbine manufacturing company. This project is unique, because the real existing hydro power turbine was physically measured and the measurement data were compared with the data from the CFD simulation by TCFD. The particular goal of this benchmark is to compare the turbine efficiency and power of the TCFD simulation results with the real experimental measurement data measured by Hidroenergia.

Keywords

CFD, VALIDATION, BENCHMARK, TCFD, SIMULATION, TURBOMACHINERY, HYDROPOWER, FRANCIS, HYDRO, TURBINE, INCOMPRESSIBLE, RANS, WATER, FLOW, STEADY-STATE, AUTOMATION

Benchmark Parameters

- Typical flow speed: *10 m/s*
- Rotation speed: *600 RPM*
- Flow model: in*compressible*
- Mesh size: *5M cells*
- Medium: *water*
- Components: *4*
- Turbine blades: *13*
- Turbine power: 3000 kW
- Reference density: *996 kg/m^3*
- Dynamic viscosity: *1.0×10^{-3} Pa · s*
- CPU Time: *30 core.hours*
- Turb. Model: *realizable k-epsilon*
- Simulation type: *HydroTurbine*
- Turbulence intensity: 5%

Francis Hydro Turbine - HPP FORTUNA II

Francis turbines are the most common water turbine in use today. This particular Francis turbine which was researched in this benchmark is called HPP FORTUNA II, and it was installed in Minas Gerais, Brazil. Hydro power plant has three Francis turbine units, the images on this page show the installation process. The Francis turbine is an inward-flow reaction turbine that combines radial and axial flow concepts. Reaction turbine, a category of turbine in which the working fluid comes to the turbine under immense pressure and the energy is extracted by the turbine blades from the working fluid. A part of the energy is given up by the fluid because of pressure changes occurring in the blades of the turbine, quantified by the expression of degree of reaction, while the remaining part of the energy is extracted by the volute casing of the turbine. At the exit, water acts on the spinning cup-shaped runner features, leaving at low velocity and low swirl with very little kinetic or potential energy left. The turbine's exit tube is shaped to help decelerate the water flow and recover the pressure.

A Francis turbine consists of the following main parts:
Spiral casing: The spiral casing around the runner of the turbine is known as the volute casing or scroll case. Throughout its length, it has numerous openings at regular intervals to allow the working fluid to impinge on the blades of the runner. These openings convert the pressure energy of the fluid into kinetic energy just before the fluid impinges on the blades. This maintains a constant velocity despite the fact that numerous openings have been provided for the fluid to enter the blades, as the cross-sectional area of this casing decreases uniformly along the circumference.

Guide and stay vanes: The primary function of the guide and stay vanes is to convert the pressure energy of the fluid into kinetic energy. It also serves to direct the flow at design angles to the runner blades.

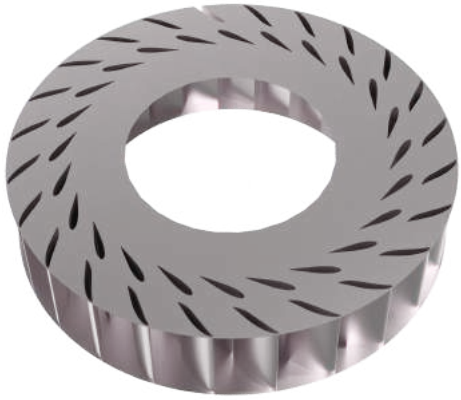
Runner blades (Impeller): Runner blades are the heart of any turbine. These are the centers where the fluid strikes and the tangential force of the impact causes the shaft of the turbine to rotate, producing torque. Close attention to design of blade angles at inlet and outlet is necessary, as these are major parameters affecting power production.

Draft tube: The draft tube is a conduit that connects the runner exit to the tail race where the water is discharged from the turbine. Its primary function is to reduce the velocity of discharged water to minimize the loss of kinetic energy at the outlet. This permits the turbine to be set above the tail water without appreciable drop of available head.

Preprocessing

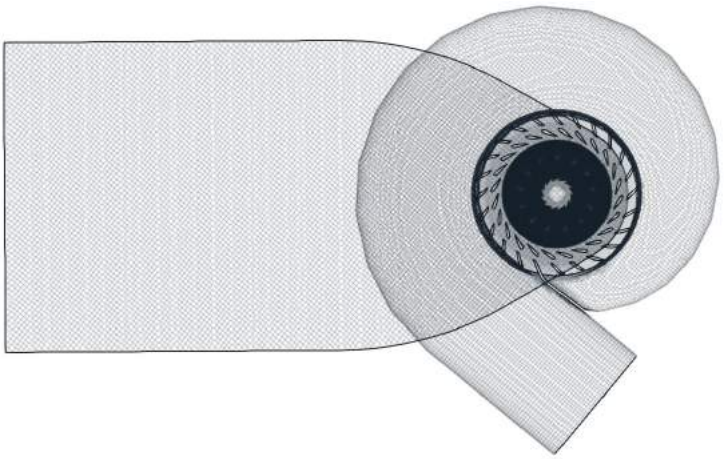
The CAD model of the Francis turbine was received in the STEP file format. Original STEP files are usually too complex for comprehensive CFD simulations, so certain preprocessing (cleaning) CAD work is generally required. While the original CAD model for this project was simplified and cleaned using 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, irrelevant, and problematic model parts must be removed, and all the holes must be sealed up (watertight surface model is required). This Francis turbine CAD model is reasonably simple. The final surface model in the STL format is created as input for the meshing phase. This preprocessing phase of the workflow is extremely important because it determines the simulation potential and limits the CFD results.

In this Francis turbine project the CAD model was split into four logical parts: **Spiral casing, Stay+Guide vanes, Impeller, and Draft tube**. Each turbine part is watertight and includes inlet interface, outlet interface and corresponding walls (wall, blade, hub, shroud, stay vanes, guide blades). For each individual turbine part has been created its own volume mesh, and during the simulation the meshes are merged by the TCFD solver.

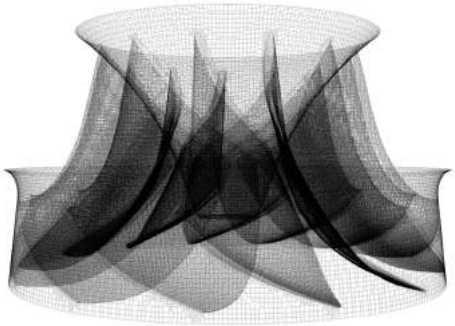


Mesh

The computational mesh was created in an automated workflow using *snappyHexMesh* application. For each turbine part (Spiral casing, Stay+Guide vanes, Impeller, and Draft tube), a cartesian block mesh was used as an initial background mesh, that is further refined. The whole turbine model is 6237mm long, 3364mm high, and 3473mm wide. Basic mesh cell size is a cube of about 100mm edge.
The mesh is gradually refined to the wall. The mesh refinement levels can be easily changed, to obtain the coarser or finer mesh, to better handle the mesh size. Inflation layers can be easily handled.

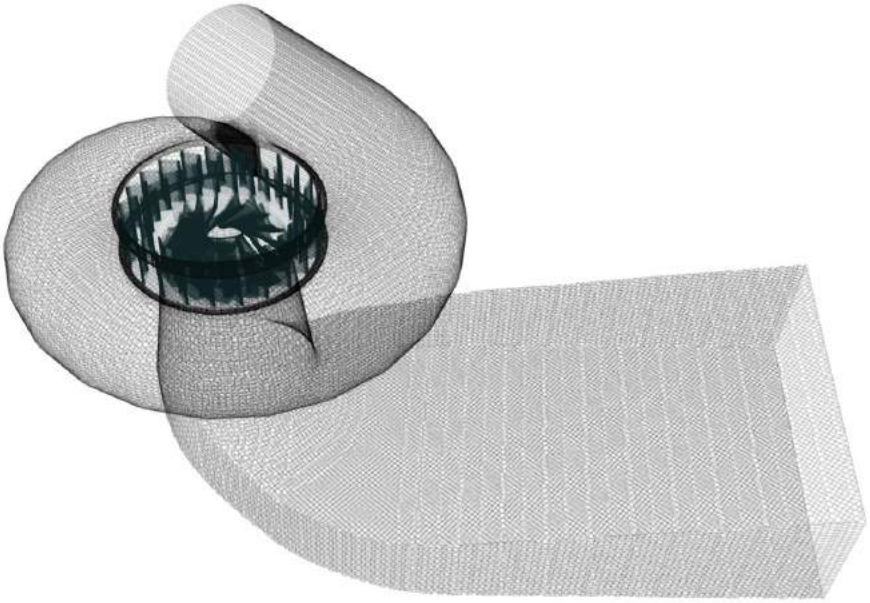


The final mesh used for the simulation has in total 4,937,072 cells and consists mostly of hexahedrons. (Spiral casing - 268751 , Stay+Guide vanes - 3256434, Impeller - 1253858, and Draft tube - 158029).



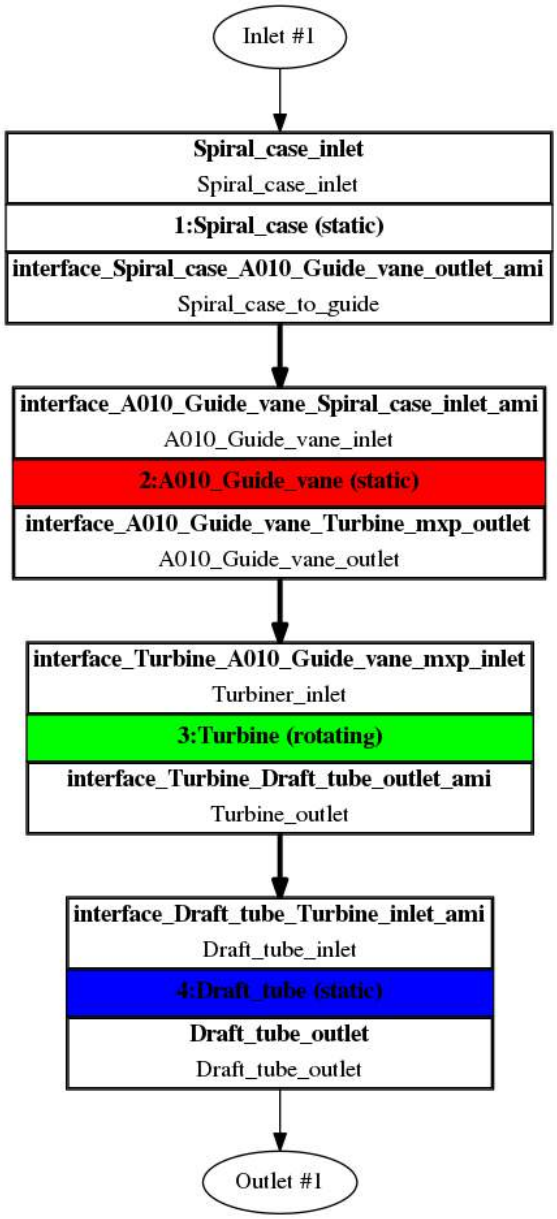
The snappyHexMesh is not a compulsory meshing tool for TCFD at all. In case of need, any other external mesh can be loaded in TCFD directly in *MSH*, *CGNS* or *OpenFOAM* format.

Mesh Elements	points	faces	internal faces	faces per cell	hexahedra
#	*6319570*	*16075879*	*15063115*	*6.307178*	*3605554*
Mesh Elements	prisms	pyramids	tet wedges	polyhedra	**cells**
#	*263951*	*0*	*6574*	*1060934*	***4,937,072***



TCFD Francis Turbine Case Setup

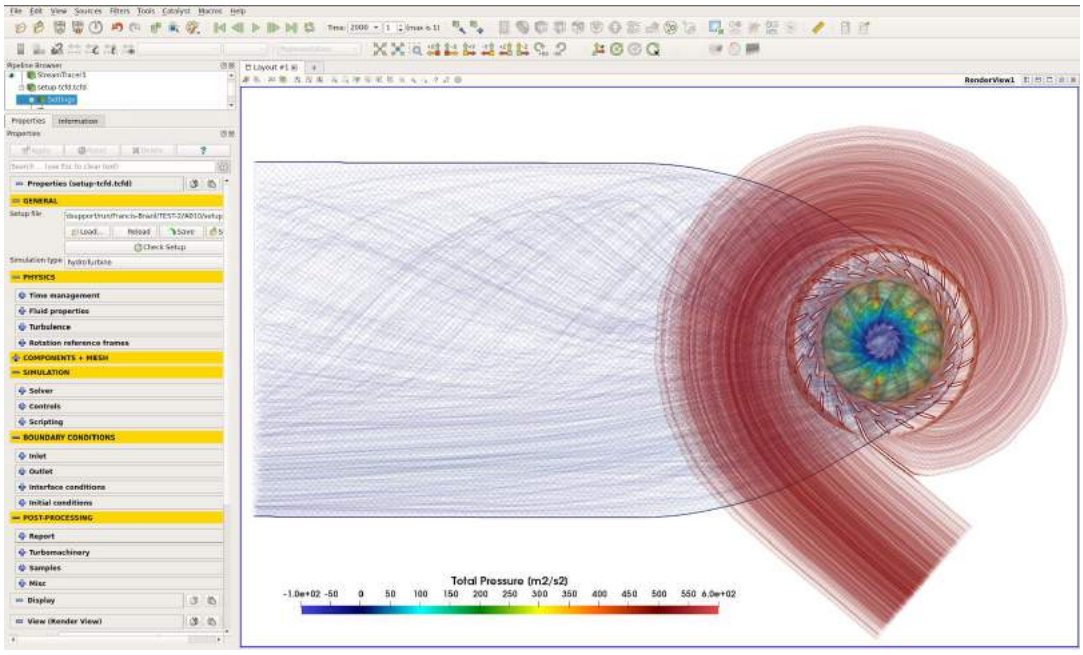
Complete CFD simulation setup is done in TCFD GUI in ParaView. Turbine has four components which are connected via interfaces. At the interfaces it is possible to choose the *frozen rotor* (direct connection) or *mixing plane* (averaging) boundary condition.



- Simulation type: *Hydro Turbine*
- Time management: *steady-state*
- Physical model: *Incompressible*
- Number of components: *4 [-]*
- Mesh Mesh size: *5M [cells]*
- Inlet: *Flow Rate 6 [m3/s]*
- Outlet: *Static pressure [m2/s2]*
- Impeller Inlet Interface: *mixing plane*
- Impeller Outlet Interface: *frozen rotor*
- Turbulence: *RANS*
- Turbulence model: *realizable k-epsilon*
- Wall treatment: *Wall functions*
- Turbulence intensity: 5%
- Rotation speed: *600 [RPM]*
- Speedlines: *1 [-]*
- Simulation points: *10 [-]*
- Fluid: *Water*
- Reference pressure: *1 [atm]*
- Reference density: *996 [kg/m3]*
- Dynamic viscosity: 1.0×10^{-3} *[Pa · s]*

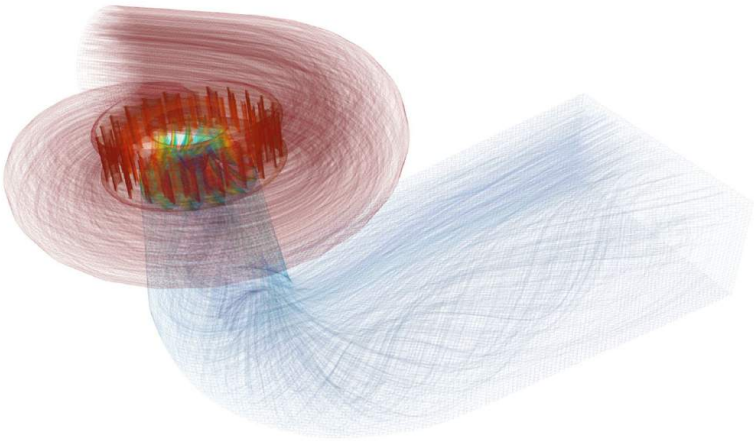
Simulation run

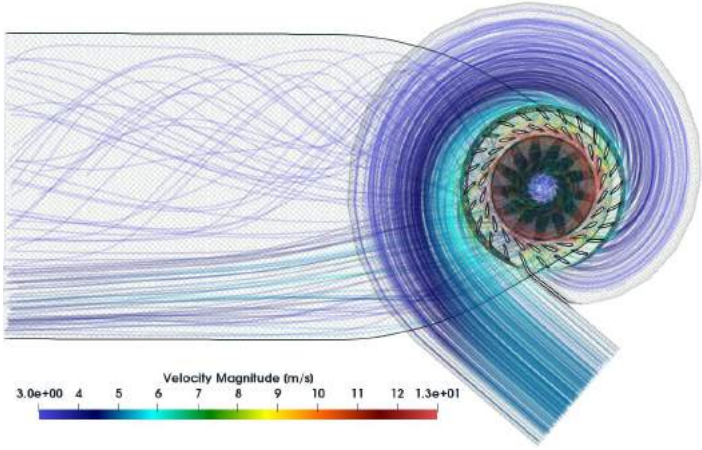
The simulation was executed in the automated workflow in steady-state mode, for 10 different flow rates, at 10 corresponding guide vanes openings. TCFD is capable of writing the results down at any time during the simulation. The convergence of any quantity is monitored during the simulation. When any simulation point converges sufficiently enough, the simulation can move onto the next simulation point. Single one point simulation takes about 30 core*hours.

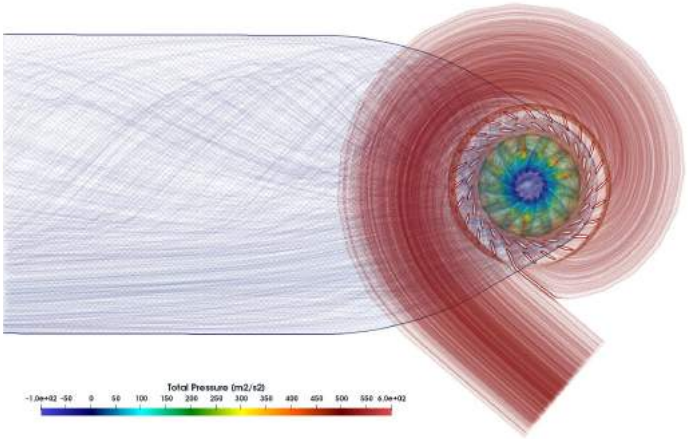


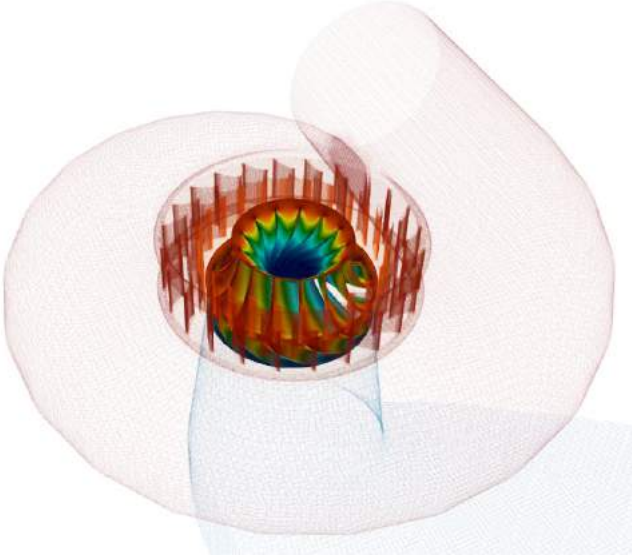
Results #1 - Flow Field Visualization

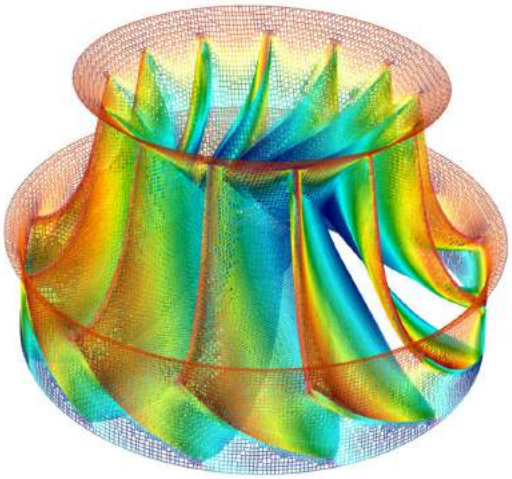
TCFD includes a built-in post-processing module that automatically evaluates all the required quantities, such as efficiency, torque, forces, force coefficients, flow rates, pressure, velocity, and much more. All these quantities are evaluated throughout the simulation run, and all the important data is summarized in an HTML report, which can be updated anytime during the simulation, for every run. All the simulation data are also saved in tabulated .csv files for further evaluation. Furthermore, visual postprocessing of the volume fields can be done with ParaView. ParaView is a very powerful open-source CFD postprocessing tool, and includes a large variety of useful visualization possibilities, as well as data operation possibilities.









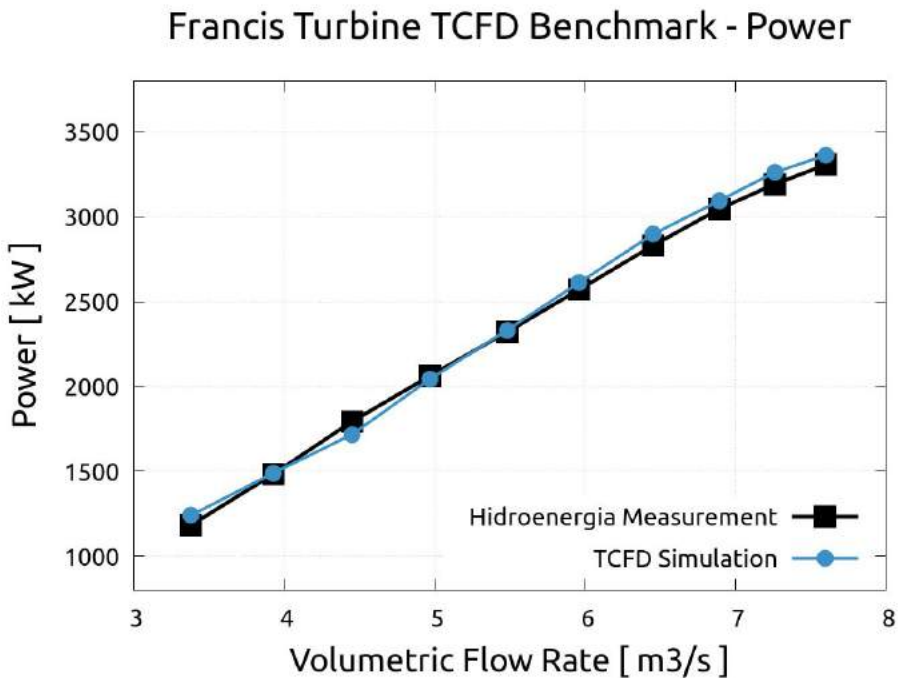


Results #2 - Turbine Power Comparison

A very important result of hydro turbine simulation is turbine torque, which is a result of fluid flow pressure and viscous forces acting on the impeller surface. The resulting turbine power is calculated from a simple formula

$$P = T \cdot \omega$$

Where P is power [W], T is torque [N.m], and ω is angular velocity [rad/s]. The image on the right hand side shows the comparison of turbine power from CFD simulation compared to the turbine power gained from the experimental measurement. There were investigated 10 turbine modes (flow rates), corresponding to the 10 guide vanes openings.

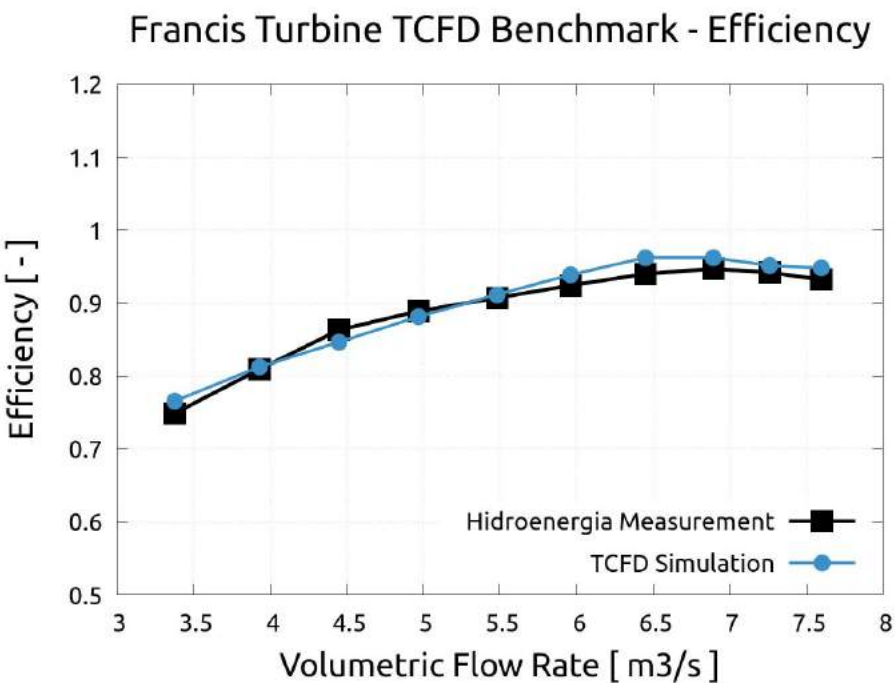


Results #3 - Turbine Efficiency Comparison

Another very important quantity at hydro turbine is turbine efficiency. Turbine efficiency is a ratio of the gained energy and water column energy potential and can be expressed as:

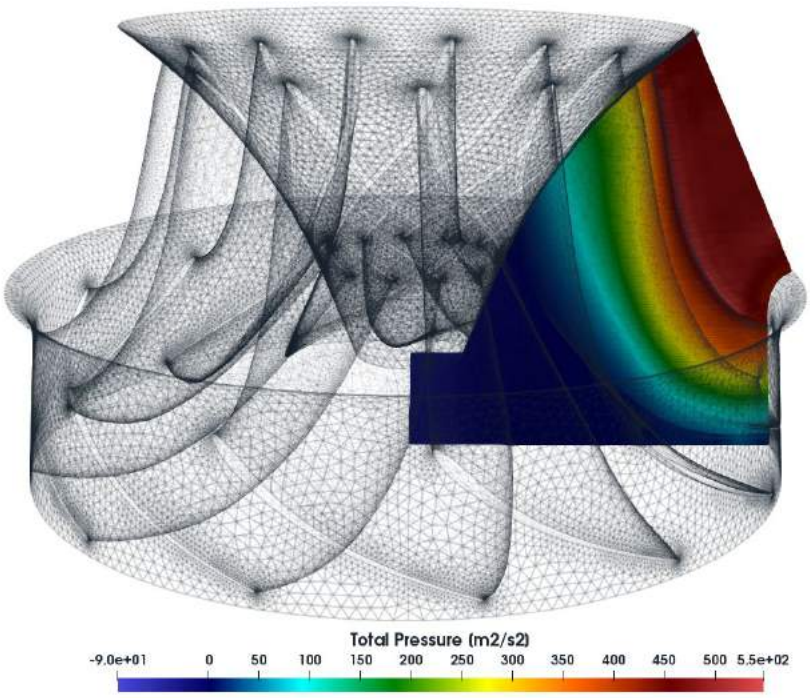
$$\eta = \frac{P}{m \cdot g \cdot \varrho \cdot h}$$

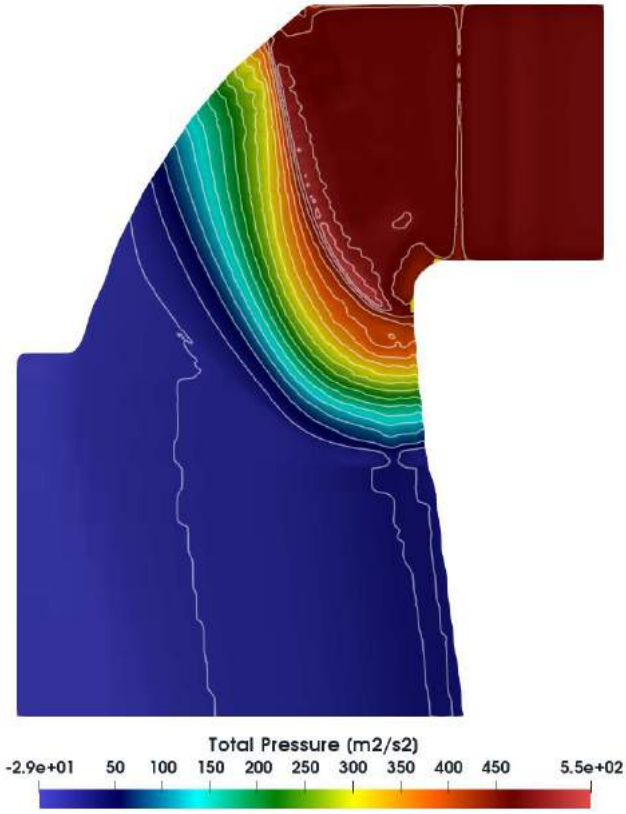
Where η is efficiency [-], P is power [W], m is flow rate [m3/s], g is gravitational acceleration [m/s2], ϱ is density [kg/m3], and h is net head, which is the difference between water levels (in the simulation results, the head is obtained from the total pressure difference between the turbine pressure at the inlet and the total pressure at the outlet). Because of the measurement method, the reference head of 47.91 m is considered for all the simulation modes. The image on the right hand side shows the comparison of turbine efficiency from CFD simulation compared to the turbine efficiency gained from the experimental measurement. There were investigated 10 turbine modes (flow rates), corresponding to the 10 guide vanes openings.



Results #4 - Meridional Average

Another important hydro turbine simulation results evaluation is a visualization of a Meridional Average of simulated quantities. This visualization gives the user valuable information about how, for example, the total pressure is spent in the flow passage of the turbine. A special TCFD filter Meridional Average, created for ParaView, is applied and creates a geometrical slice (a plane), containing the rotation axis and the circumferential averages of all the field data projected onto this slice. The Meridional Average method ignores, for example, blades or other obstacles, and as a result the resulting averaging plane has a shape of a compact flow passage.

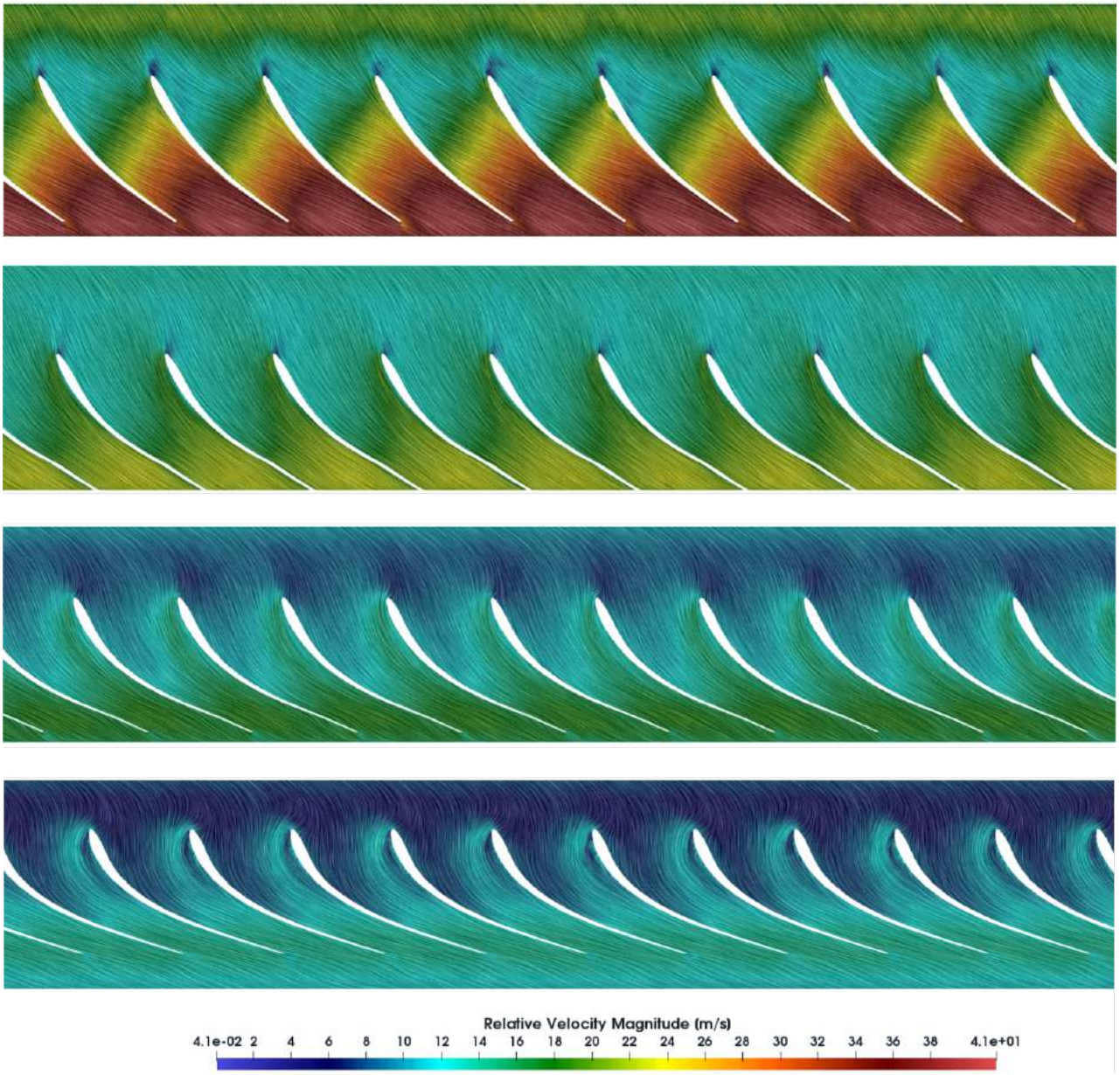




Results #5 - Blade to Blade View

Another important hydro turbine simulation results evaluation is a blade-to-blade view. The blade-to-blade view offers a unique perspective for an inspection of the flow field properties between the blades, at a fixed relative distance between the hub and shroud boundary surfaces. In TCFD, blade-to-blade view can be generated in two steps: First, the cylindrical mesh of the rotating zone needs to be transformed (unwrapped) into a normalized rectangular block (1x1x2Pi). Second, the unwrapped block is to be cut at the prefered normal distance (0-1), between hub and shroud.
A typically desired field view is, for example, streamtraces of the flow field or relative velocity at the impeller. The user can observe how smoothly the fluid flows and how effectively the fluid attacks the leading edges of the blades.

The following images show the streamtraces projected onto blade-to-blade planes in the form of LIC (Line Integral Convolution), colored by the flow field relative velocity. For the specific opening corresponding to the flow rate 5.48 m3/s. There are four planes at relative height 0.99, 0.8, 0.6, and 0.4 between hub and shroud.



Conclusion

The complex CFD analysis of the real existing Francis hydro turbine, using TCFD, was performed successfully. It has been shown that the TCFD simulation prediction gives a very good agreement with the measurement data. The Francis turbine power and efficiency are in good agreement with the measurement data available.

TCFD showed it is a very effective tool for a comprehensive CFD simulation analysis of hydro turbines, at all stages of the design and validation process.

More information about TCFD can be found on TCFD website:
https://www.cfdsupport.com/tcfd.html

Potential questions will be answered on request at *info@cfdsupport.com*

References

[1] TCFD Manual https://www.cfdsupport.com/download/TCFD-manual-v19.10.pdf
[2] TCFD Training https://www.cfdsupport.com/download/TCFD-Training-19.10.pdf
[3] Wikipedia https://en.wikipedia.org/wiki/Francis_turbine