NASA Turbulent Flat Plate Validation Benchmark using TCFD®

The following article is aimed at the turbulence modeling tests on a flat plate, using TCFD®. This turbulent flat plate test case is well known and it has been well described on NASA website: https://turbmodels.larc.nasa.gov/flatplate.html The aim of this benchmark is the validation of the turbulence models in the TCFD®, computational fluid dynamics (CFD) software, and compare its results with the analytical and measurement data. The secondary aim of this test case is to compare and discuss two different wall treatments; The Low-Re modeling (resolved mesh) vs. Wall Functions modeling.

Keywords

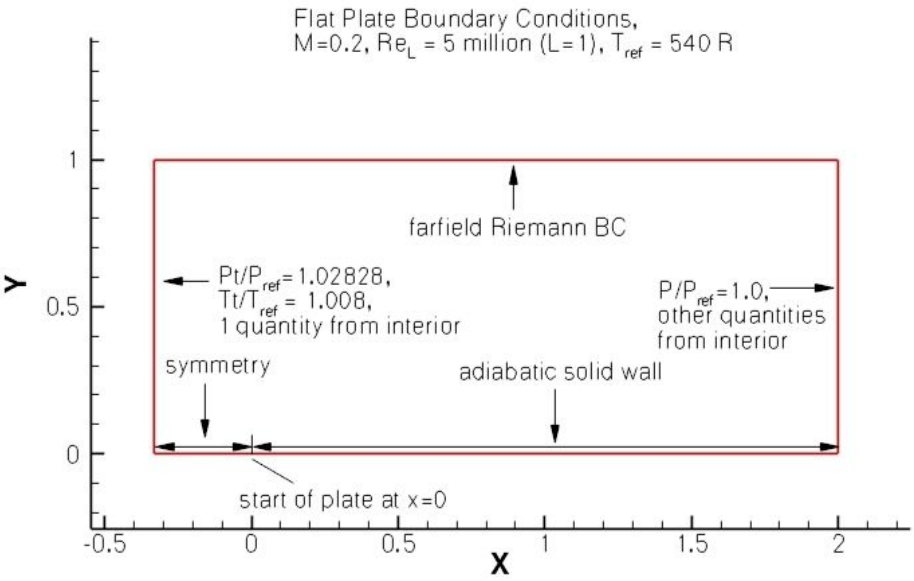
CFD, VALIDATION, BENCHMARK, TCFD, SIMULATION, FLATPLATE, AIR, STEADY-STATE, AIR FLOW, AUTOMATION

Benchmark Parameters

- Typical flow speed: *50 m/s*
- Flow model: in*compressible*
- Rotation RPM: -
- Mesh size: 0.01/*0.08M cells*
- Impeller diameter: - *mm*
- Medium: *air*
- Reference pressure: *1 atm*
- Typical Pressure Ratio: *0.0*
- Reference density: *1.2 kg/m³*
- Dynamic viscosity: *1.8 × 10⁻⁵ Pa·s*
- Turb. Model 1: *k-omega SST (WF)*
- Turb. Model 2: *v2f (Low-Re)*

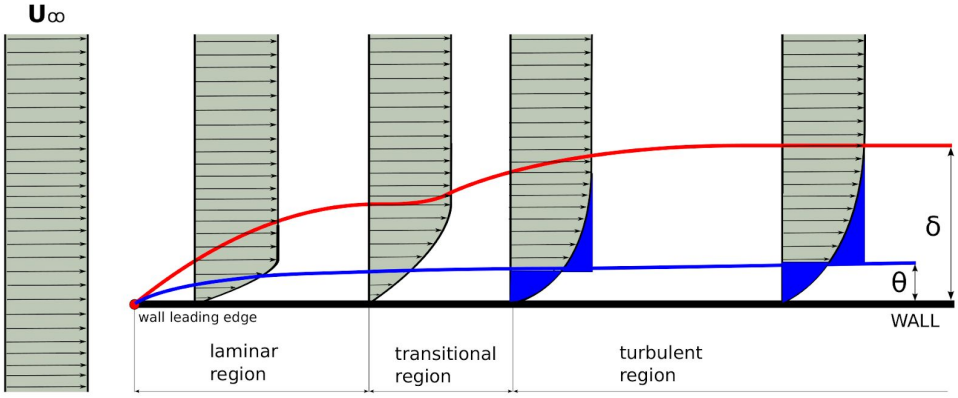
Test Case

The test case topology is well described on the NASA website. It is a two-dimensional simple case sometimes called *Zero Pressure Gradient Flat Plate*. The flat plate is a flat wall (a line in 2d) that is being attacked by a freestream flow of 50 [m/s]. The flat plate is 2 meters long, along the x-axis, and starts in the point [0;0]. This test case is very suitable for turbulence model testing because the test case is small enough to converge quickly, and a lot of analytical and measurement data for validation is available.



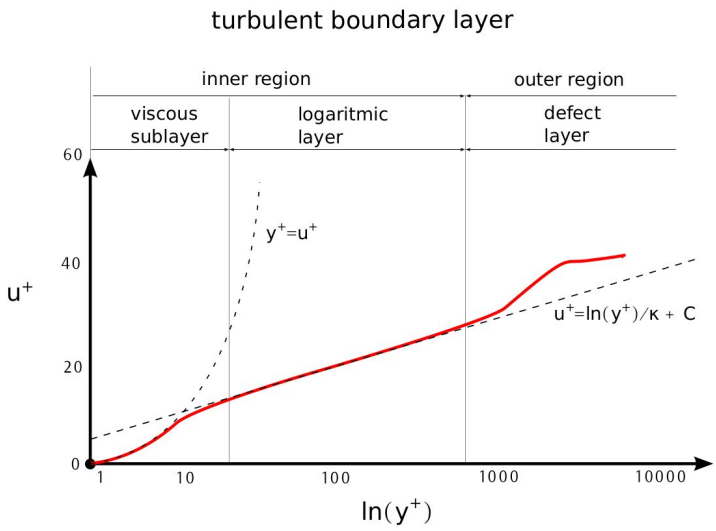
Turbulent Boundary Layer at Re_θ = 10,000

A very important characteristic of any turbulent flow is developing the turbulent boundary layer (BL). As the free stream attacks the flat plate (with zero angle of attack), the boundary layer develops along the flat plate. Three development regions can be observed; *laminar* region, *transitional* region, and *turbulent* region.



The *momentum thickness*, θ, is the distance by which a surface would have to be moved parallel to itself towards the reference plane in an inviscid fluid stream of velocity U_∞ to give the same total momentum as exists between the surface and the reference plane in a real fluid. The definition of the momentum thickness, θ, for incompressible flow is based on the mass flow rate:

$$\theta = \int_0^\infty \frac{u(y)}{u_0}\left(1 - \frac{u(y)}{u_0}\right) dy \quad \text{, then } Re_\theta = \frac{U\,\theta}{\nu}$$

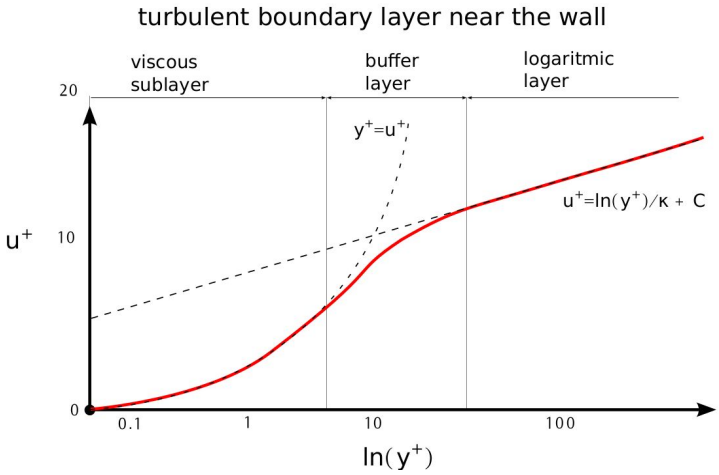


All turbulent boundary layers (BLs) on a flat plate have similar characteristics. Fully developed turbulent BL has three regions; *viscous sublayer*, *logarithmic layer*, and *defect layer*. To be able to compare different BLs, any BL can be displayed in dimension-less coordinates *u+* and *y+*

$$u^+ = \frac{U}{u_\tau}, \; y^+ = \frac{u_\tau y}{\nu}, \; u_\tau = \sqrt{\frac{\tau_w}{\rho}}$$

Then the important part of the BL is its logarithmic layer. In the logarithmic layer the *Law of the wall* is valid:

$$u^+ = \frac{1}{\kappa} ln(y^+) + C, \quad C = 5, \quad \kappa = 0.41$$



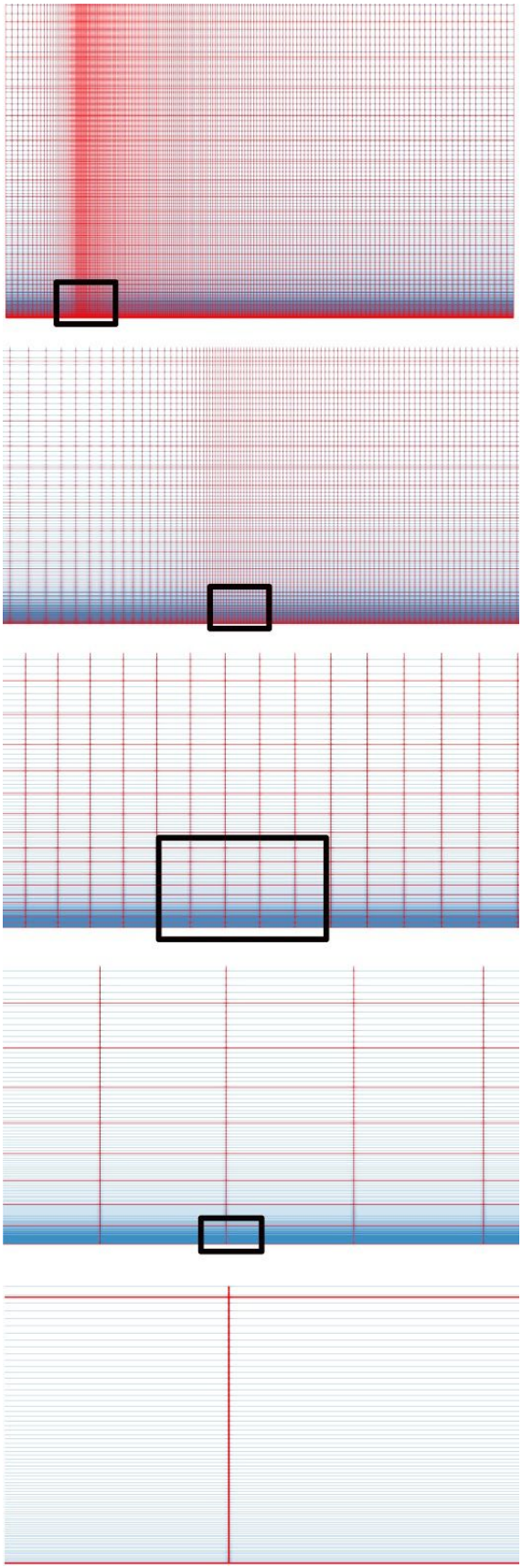
The fully developed boundary layer can be viewed as divided into four regions;

Viscous sublayer is a region closest to the wall. In this region, y+ = u+. The fluid flow is always laminar here. The Viscous sublayer region is typically where y + < 10. The *buffer layer* is a region, where no law holds and it is associated with a certain amount of uncertainty. The Buffer layer region is typically where 5 < y + < 30. *Logarithmic (Log-law) layer* is a region, where the Law of the wall holds. The Log-law layer region is typically where 30 < y+ < 300, and all the wall functions are trying to hit this region. *Outer (Defect) layer* is a region, where the free-stream is taking over. The Outer layer region is typically where 300 < y+.

Mesh

In this particular project, the volume meshes for the CFD simulation were created using *blockMesh* application. Alternatively, another external mesh can be loaded in TCFD directly in *MSH*, *CGNS*, or *OpenFOAM* format.
Each mesh consists of two simple blocks connected at coordinate X=0. Each mesh has a significant grading in the Y-direction and minor grading in the X-direction. All the cells are hexahedral and the test case is formally three-dimensional, but all the information in the Z-direction is completely irrelevant.
A simple *bash* script was used for mesh creation. For the final presentation, two meshes were created, for comparison of the wall functions method and the Low-Re method. The *rough mesh 200x50* has 200 cells along the X-axis and 50 cells along the Y-axis. The *fine mesh 200x400* has 200 cells along the X-axis and 400 cells along the Y-axis. These two meshes have exactly the same characteristics along the X-axis, but they differ only in the direction along Y-axis.
The table below shows some selected mesh statistics:

Mesh stats	**200x50**	**200x400**
X cells	200	200
X grading ratio	10	10
Y cells	50	400
Y grading ratio	1:500	1:50000
Approx Y+	100	0.3
Y min	4.8E-4	1.6E-6
hexahedra	100%	100%
polyhedra	0%	0%
Max. aspect ratio	82	18600
Total cells	10,000	80,000



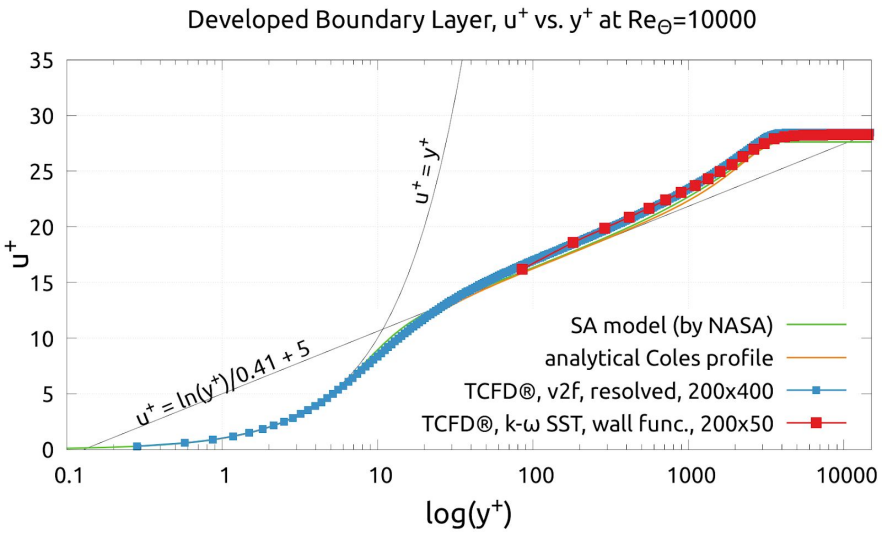
TCFD® Case Setup

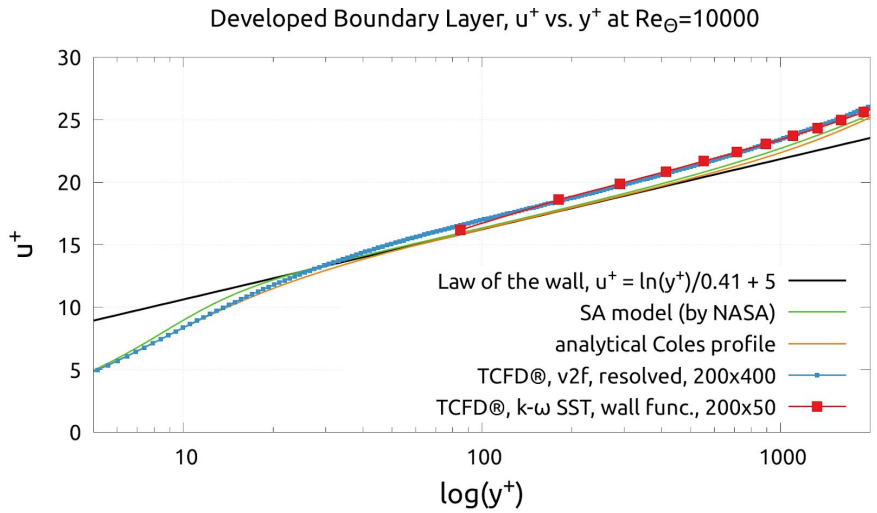
Basic TCFD® settings:

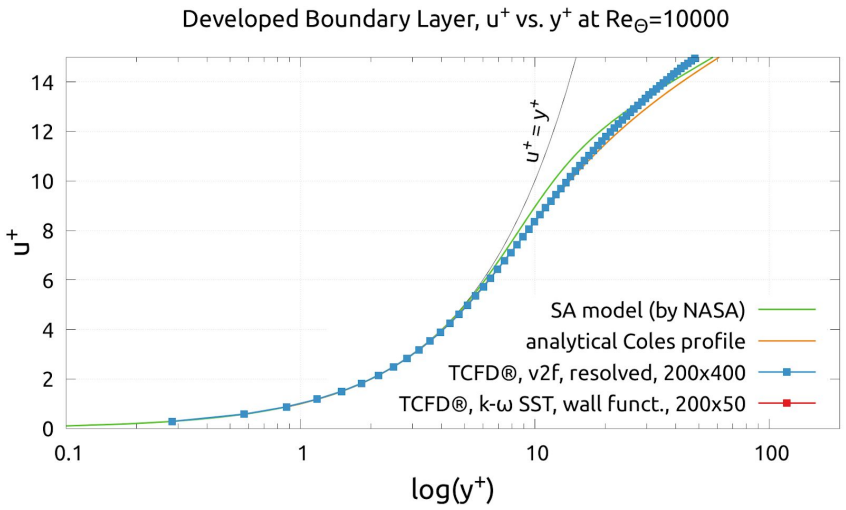
- Machine type: *stator*
- *Steady-state* calculation
- *Incompressible* fluid flow
- RANS turbulence
- Model *k-ω SST and v2f (scripted)*
- Number of components: *1*
- Mesh Mesh size: *0.8M cells*
- Inlet: *Velocity 50 [m/s]*
- Outlet: *Static Pressure*
- Speed-lines: *1*
- Simulation points: *1*

Results - Turbulent Boundary Layer at Re_θ = 10,000

For the final presentation, two simulations were performed in TCFD using two different methods:
Case 1. Rough mesh 200x50, wall functions, with *k-ω SST* turbulence model
Case 2. Fine mesh 200x400, Low-Re, with *v2f* turbulence model



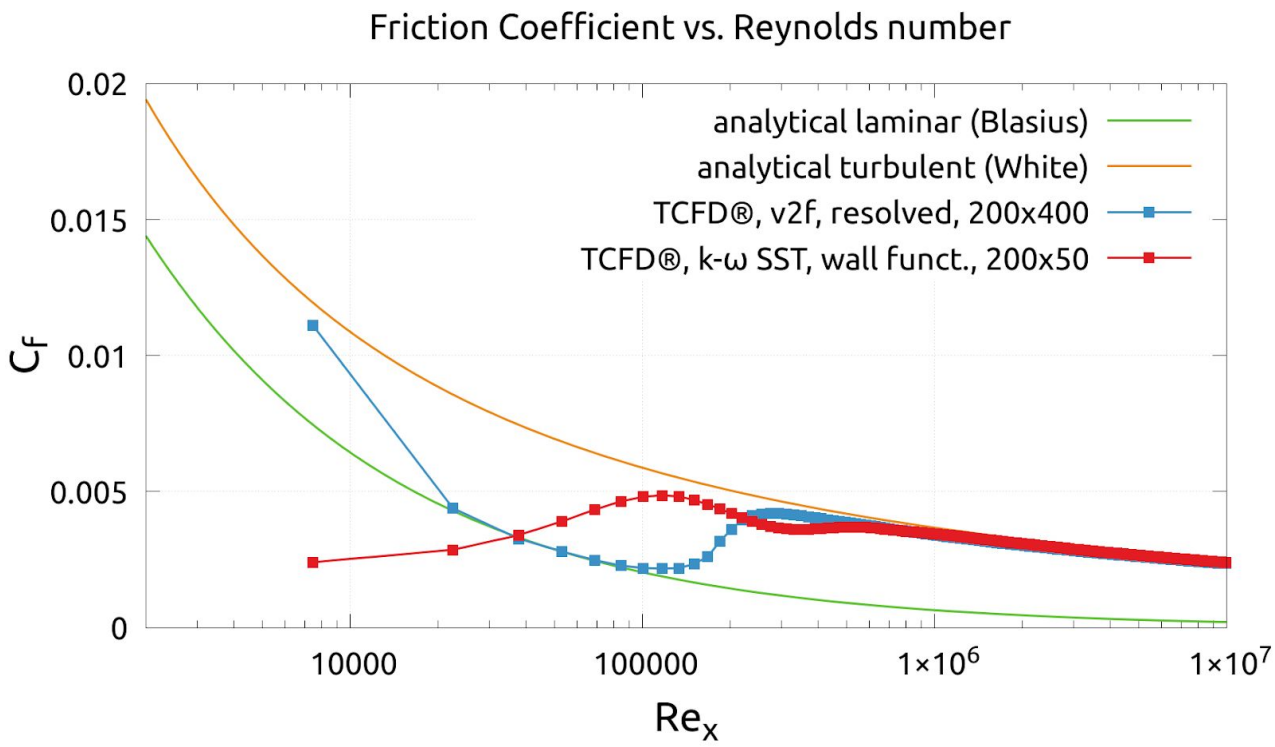




Results - Friction Coefficient vs. Re_X

Another important aerodynamic characteristic of viscous fluid flow at a flat plate is *Friction coefficient*, C_f. Generally speaking, it is the ratio of the weight of an object being moved along a surface and the force that maintains contact between the object and the surface. It is defined as: $C_f = \frac{2\tau_w}{\rho_\infty U_\infty^2}$. The friction coefficient can be evaluated along the flat plate, to give us valuable information about the flow behavior. According to H. Blasius, the friction coefficient of the incompressible laminar fluid flow can be derived as: $C_f = \frac{0.664}{\sqrt{Re_X}}$, where $Re_X = \frac{U\,X}{\nu}$. According to F. White, the friction coefficient of the incompressible turbulent fluid flow can be derived as: $C_f = \frac{0.445}{ln^2(0.06 \cdot Re_X)}$

For the final presentation, two simulations were performed in TCFD using two different methods:
Case 1. Rough mesh 200x50, wall functions, with *k-ω SST* turbulence model
Case 2. Fine mesh 200x400, Low-Re, with *v2f* turbulence model



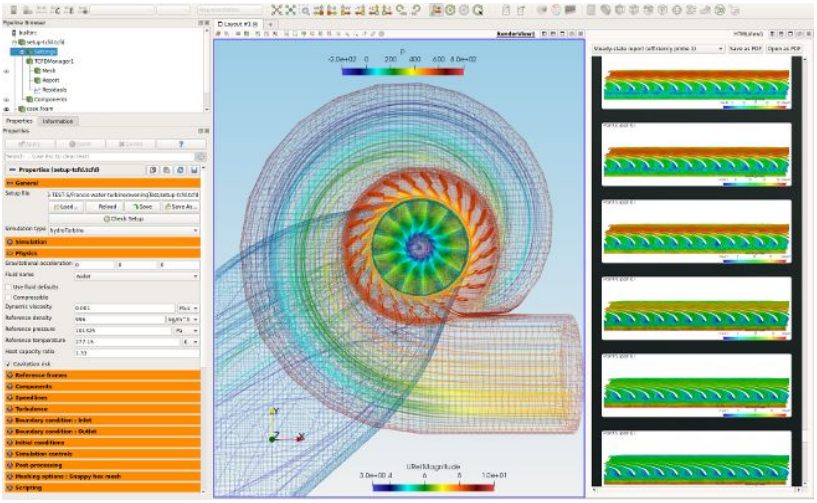
Conclusion

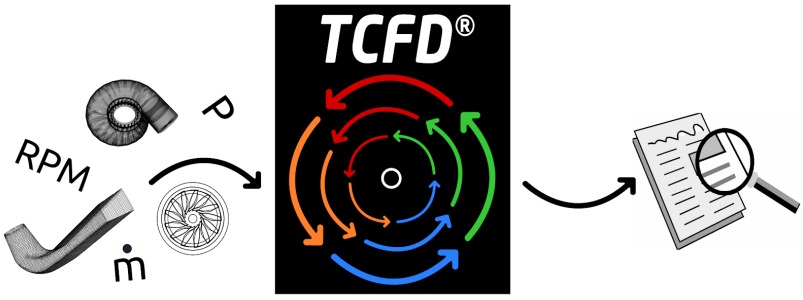
Turbulent flat plate benchmark is a very suitable test case for quick simulation tests - because, on the 2D mesh, the simulation generally converges very quickly. Case 1. shows very quick convergence (0.2 core*hours) and fully turbulent behavior everywhere along the flat plate. Case 2. shows significantly slower convergence (12 core*hours) and very high accuracy. All in all, the complex CFD analysis of fluid flow on the flat plate using TCFD® provides very good agreement with the analytical data, as has been demonstrated. Potential questions will be answered on request at *info@cfdsupport.com*.

[1] Wilcox D.C., Turbulence Modeling for CFD , DCW Industries
[2] Stephen B. Pope, Turbulent flows. SB (2000). Cambridge Univ. Press, Cambridge.
[3] H. Versteeg, W. Malalasekera, An Introduction to Computational Fluid Dynamics: The Finite Volume Method (2nd Edition)
[4] White, Frank (2011). Fluid Mechanics. New York City, NY: McGraw-Hill. pp. 477–478. ISBN 9780071311212
[5] Prandtl, L. (1925). "Bericht uber Untersuchungen zur ausgebildeten Turbulenz". Zeitschrift fur angew. Math. u. Mechanik
[6] https://en.wikipedia.org/wiki/Skin_friction_drag

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.