CFD Course Formula Student Bizkaia 2017

Overview

BCAM - Basque Center for Applied Mathematics
UPV/EHU

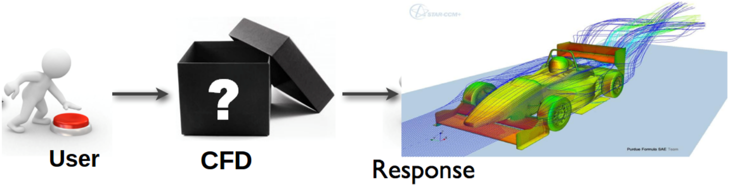
Fall 2017

1. Overview on Computational Fluid Dynamics (CFD)

What is **not** CFD?

What is not CFD?

- Although CFD software packages are getting more user friendly, we are **not (yet)** at the level of “black box use”
- Some software packages permit a *rough* estimation of flow: e.g. aeration inside building
- But if we need **accuracy** and **confidence**, we still need more knowledge of physics and engineering



Source: https://mdx.plm.automation.siemens.com/star-ccm-plus
FSAE How to make Videos for STAR-CCM+: https://cfd-online.com

What **is** CFD?

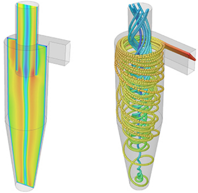
What is CFD?

- **Fluids:** mainly liquids and gases
- The governing equations are known, but not their analytical solution: thus, we **approximate** them
- By CFD we typically denote the set of **numerical techniques** used for the **approximate** solution (prevision) of the motion of fluids and the associated phenomena (heat exchange, combustion, fluid-structure interaction . . .)
- The solution of the governing differential (or integro-differential) equations is **approximated** by a **discretization** of space and time
- From the **continuum** we move to the **discrete** level
- The CFD is deeply connected to the improvement of **computers** in the last decades

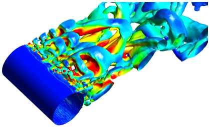
Applications of CFD I

Any field where the **fluid motion** plays a relevant role:

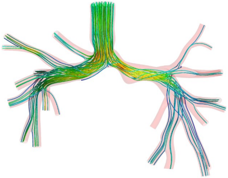
Industry



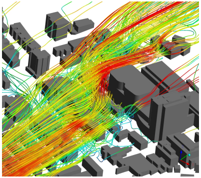
Physics



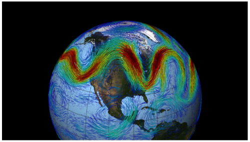
Medicine



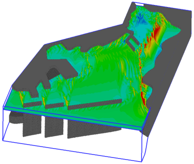
Architecture



Meteorology



Environment

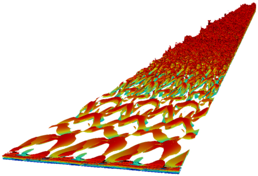


Source: (top-left) https://aryaalborz.com (top-center) https://computationalfluiddynamics.com.au (top-right) https://digitaleng.news/de (bottom-left) https://wildeanalysis.co.uk (bottom-center) https://summerofhpc.prace-ri.eu (bottom-right) Master thesis "Numerical simulation of the mixing characteristics in the bay of Muggia"

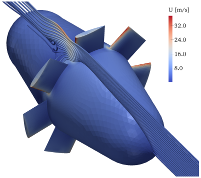
Applications of CFD II

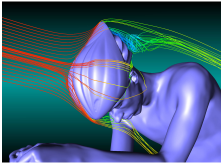
Moreover, the **research front** is particularly active:

Basic research on fluid mechanics (e.g. transition to turbulence)



Application oriented (e.g. renewable energy, competition ...)





New numerical methods

Source: (left) https://gauss-centre.eu (center-top) Paper "Numerical investigation of the aerodynamic performance for a Wells-type turbine in a wave energy converter" (center-bottom) https://commons.wikimedia.org (right) https://www.olcf.ornl.gov

CFD: limits and potential I

Method	Advantages	Disadvantages
Experimental	1. More realistic 2. Allows “complex” problems	1. Need for instrumentation 2. Scale effects 3. Difficulty in measurements & perturbations 4. Operational costs
Theoretical	1. Simple information 2. General validity 3. Understanding and interpretation of phenomena	1. Limited to simple cases 2. Typically linear problems
CFD	1. Not limited to linear cases 2. Allows “complex” problems 3. Stationary and non-stationary 4. Relatively affordable cost 5. Integration in the project chain	1. Errors: discretization, truncation 2. Difficulty in boundary conditions 3. Simplifications needed 4. Time for set-up & run 5. Time for post-processing 6. Difficult interpretation

CFD: limits and potential II

CFD is closer to experimental approach:

We need simplified analysis, heuristics, intuition, experience . . .
trial-and-error!

CFD cannot (at the present) substitute experiments:

- The equations are not rigorously exact (assumptions are made)
- The continuum eq. are discretized: we would ideally need an infinitely fine grid for perfect analogy
- There might be numerical phenomena that alter the equation's behaviour (e.g. numerical diffusivity)
- "Small" scale effects are typically modelled (e.g. turbulence, combustion . . .)

CFD: key ingredients

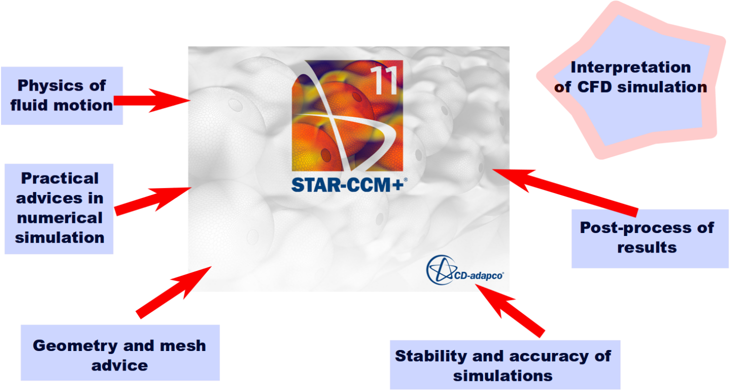
Behind the term “CFD” there is a whole WORLD

. . . and if we try to put it into one slide:

1. Target problem: physics, industrial or research, level of accuracy . . .
2. Mathematical model: 2D or 3D, stationary or not, compressible or not, required information, available resources . . . There is **not** a general purpose model!
3. Discretization method: Finite difference / volume / element, avail. software . . .
4. Computational mesh: Structured or not, curved . . .
5. Approximation techniques: Several choices balancing accuracy and efficiency
6. Solution method: Type of solver, direct, iterative . . .
7. Convergence & stability: Choice of the criteria based on accuracy and efficiency
8. Post processing & interpretation: From variables field calculate and visualize what needed: flow characteristics, lift/drag . . .

Aim of this course

- Introduction of key aspects in CFD ...
- ... and associate them to Star CCM whenever possible
- Be able to run your **own** simulations and **interpret** the results

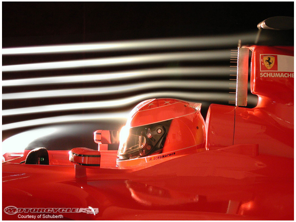


Source: https://mdx.plm.automation.siemens.com/star-ccm-plus

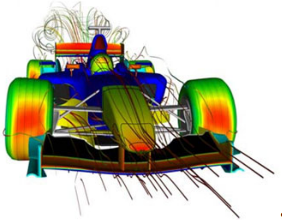
CFD in F1

- The current regulations enforce limits on both wind tunnel time (30 hours) and CFD computations (30 teraflops) ?
- Is CFD technology mature enough to obviate real-world wind tunnel testing?
- Ban of wind tunnel tests: Ferrari, Mercedes and Williams were especially skeptical, while McLaren, Red Bull and Force India are supportive
- Conclusion: (also) in F1, CFD is still in **development**

Wind tunnel



CFD



Source: (left) https://www.reddit.com (right) https://www.computationalfluiddynamics.com.au

2. References

References

- J. Blazek, *Computational Fluid Dynamics: Principles and Applications*, 2nd edition, Elsevier 2005 (CFD textbook)
- Lorena A. Barba, *CFD Python: 12 steps to Navier-Stokes*
  http://lorenabarba.com/blog/cfd-python-12-steps-to-navier-stokes/
  Check also her **videos** on YouTube:
  https://www.youtube.com/watch?v=cDy5XGOokBY