3D flow simulations

Commercial codes VS OpenFOAM
Subcontracting or Investing ?

Alexis Lapouille, chairman
Yann Recoquillon, training manager

AERO CONCEPT ACE ENGINEERING ©

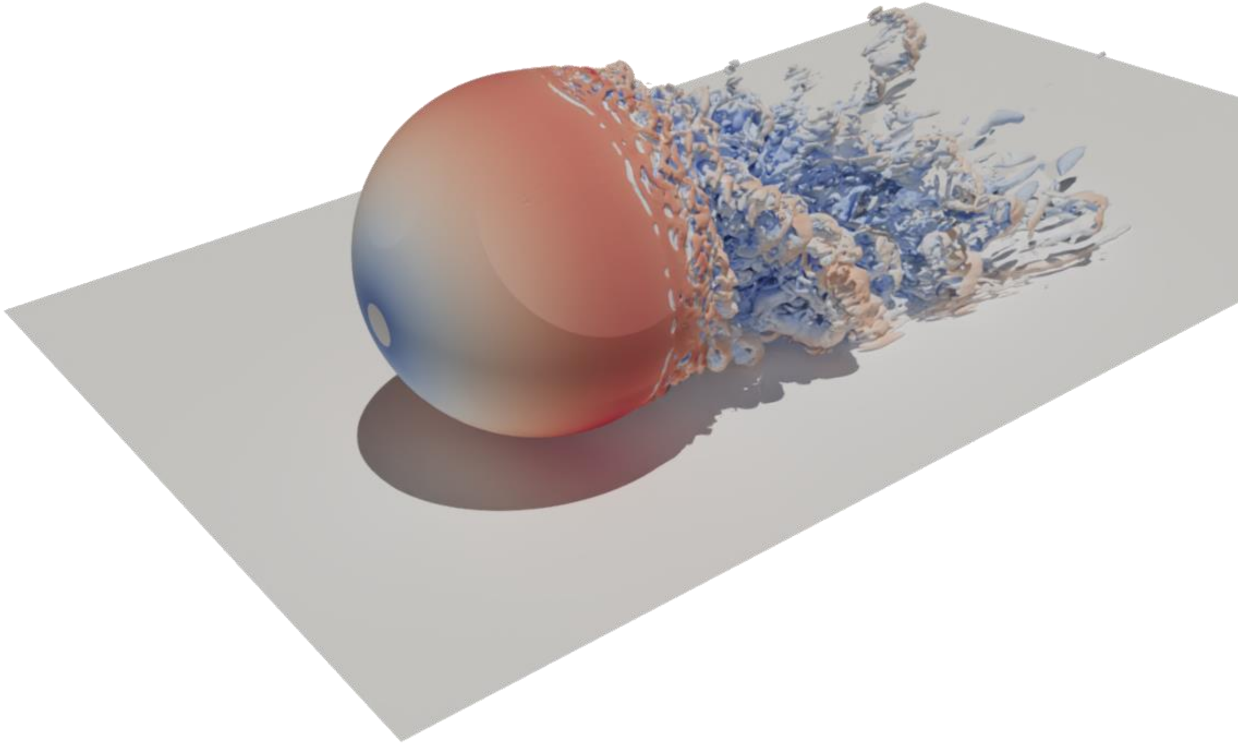
Introduction

- **Webinar outline**

  - Numerical flow simulation
    - Description
    - Solving methods
    - What to expect?
    - When to use it?

  - Simulation software
    - Which tools available on the market ?
    - How to chose the tool fitting my needs?

  - Subcontract or invest
    - Choice criteria
    - Costs: rough estimate



Introduction

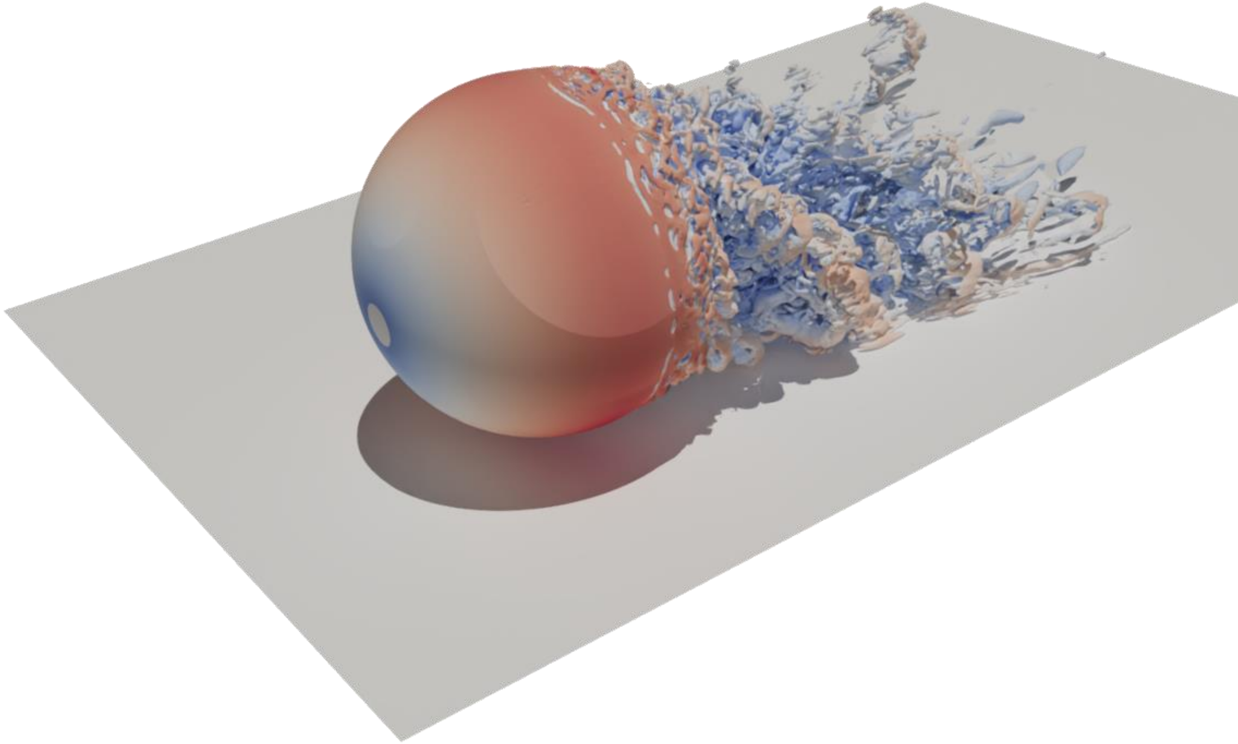
- **Speakers**

Alexis LAPOUILLE
- Founder and chairman of Aero Concept Engineering since 2002
- Experience:
  - ESTACA engineer
  - CFD engineer at Fluent Benelux
  - Aerodynamics simulations manager at Prost Grand Prix

Yann RECOQUILLON
- CFD engineer and training manager at Aero Concept Engineering since 2016
- Experience :
  - Mechanical engineer
  - PhD in fluid mechanics
  - PhD student at Renault
  - Research engineer at SATT Sud-Est

Aero Concept Engineering

- Presentation
- Wind tunnel specifications
- Flow simulation



Aero Concept Engineering

- **Presentation**

  - Created in 2002 in Magny-Cours, France
  - Wind tunnel (formerly Ligier and Prost Grand Prix)
  - 8 employees: engineers or PhDs
  - Services :

Simulation

- CFD consultancy with OpenFOAM
- OpenFOAM trainings
- ACE of Aircraft software

Design

- CAD with CATIA V5
  - Wind tunnel models
  - Measurement devices/sensors
  - Surface modeling

Experimental measurements

- Wind tunnel tests
- Production/Instrumentation of wind tunnel models

Aero Concept Engineering

Wind tunnel specifications

- Test section: 2.2 x 2.2m
- Max velocity: 40m/s (144km/h)
- Fixed or moving ground (treadmill)

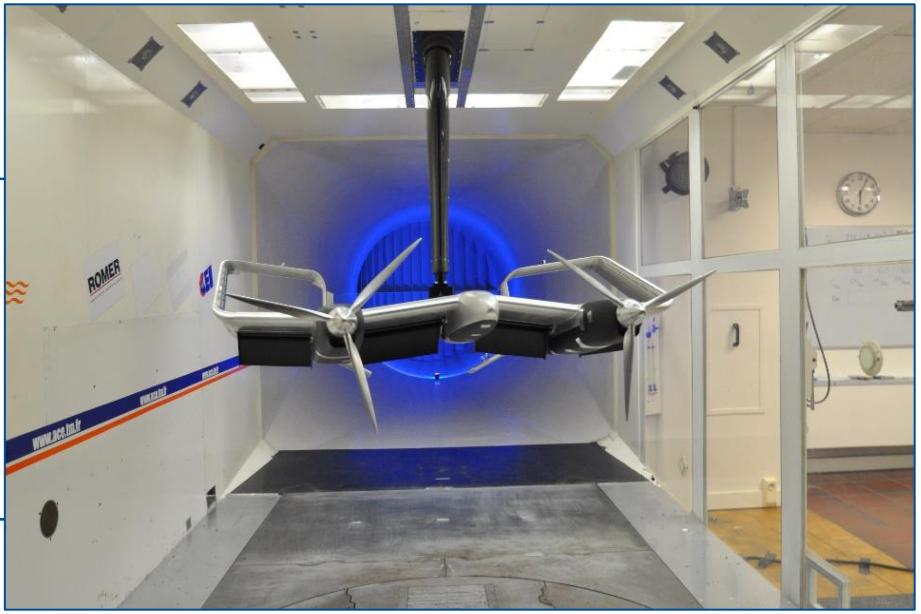
Half plane configuration

- Rotating table: +30 to -30°
- Half wingspan up to 1,5m
- Fx, Fz, My, engine thrust and torque

Aeronautics

Full plane configuration

- Ground effect or fixed ground
- 6 components
- Motorized pitch
- Manual yaw

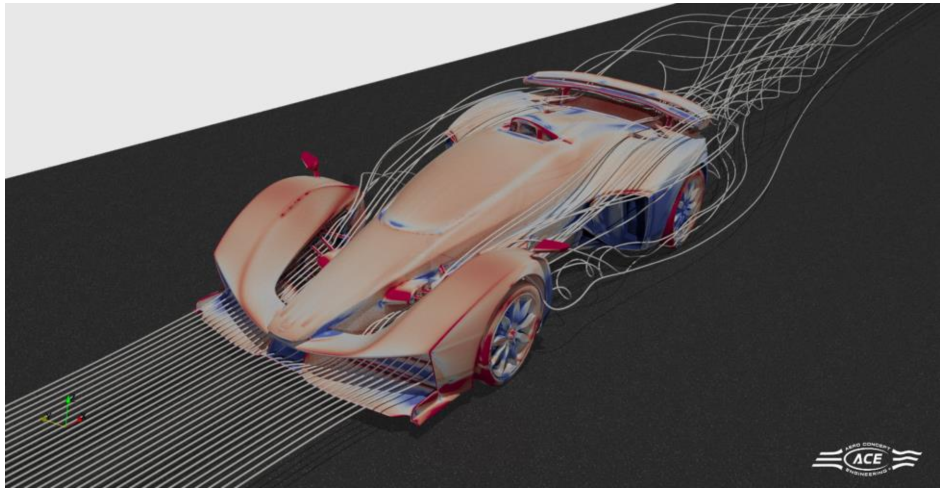


- Automotive

  - Model scale up to 40%
  - 6 components measurements
  - Fixed or moving ground
  - Model adjustments
    - Motorized pitch and roll (laser monitored)
    - Yaw: +/- 6°
    - Steering angle up to 7°
  - Wheels can be attached to the model or to supports for independent drag measurements

Aero Concept Engineering

- **Computational Fluid Dynamics (CFD)**
  - 4 engineers
  - In-house server: 200 cores and 2To RAM
  - Simulation code: OpenFOAM (open source)

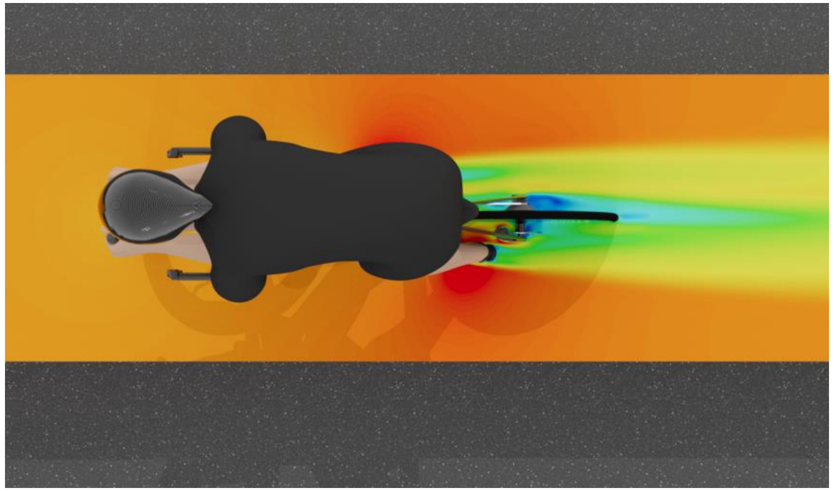


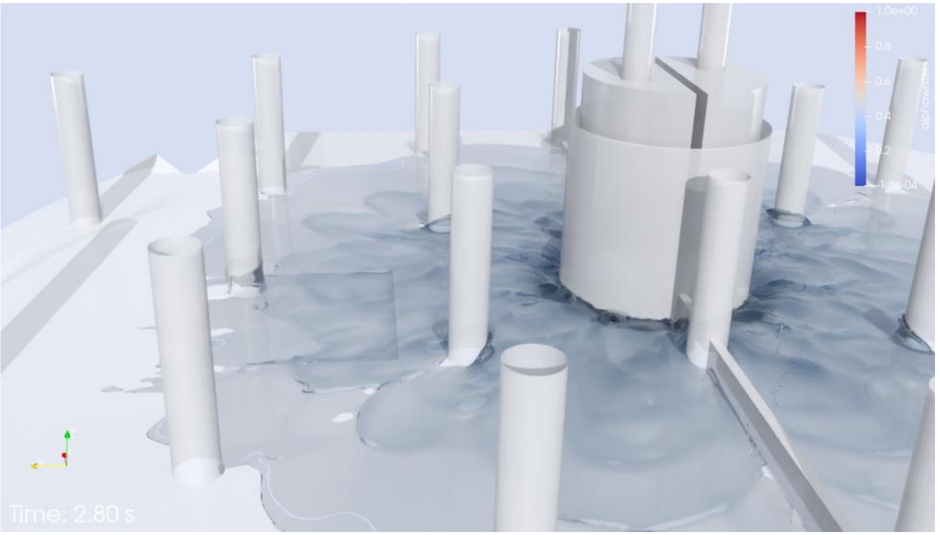
OpenFOAM trainings

- On-site personalized programs

Sectors

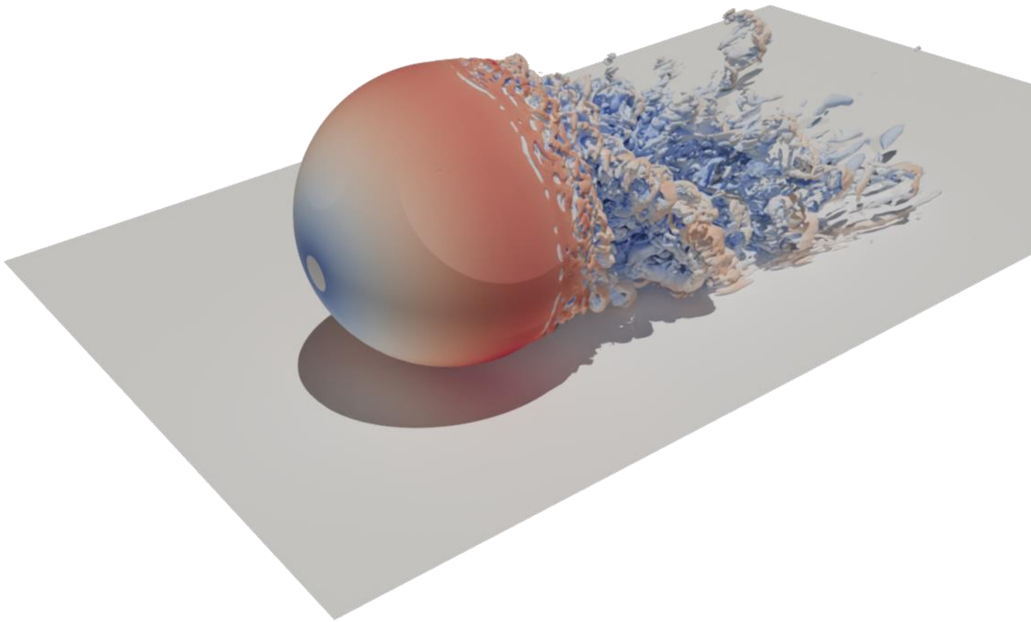
- Aeronautics
- Automotive
- Defense
- Industrial process
- Sport





Computational fluid dynamics (CFD)

- Historical landmarks
- Solving methods
- General workflow
- Navier-Stokes equations
- What to expect from simulation?
- When to use simulation?
- Accuracy and sources of errors
- Experiment or simulation?
- Order of magnitude



Computational Fluid Dynamics

- **Historical landmarks**
  - 40's-50's: first simulations using finite difference method on discretized domains
  - Early 80's: first commercial codes (PHOENICS, Fluent)
  - Since 80's: increasing number of simulation codes. The constant increase in computational power gives access to more complex methods.

- **Implementation in industry**
  - 70's-80's: aeronautics (airfoils, engines, full aircraft)
  - 80's-90's: automotive (from very basic shapes to more realistic geometries)
  - Since 90's: growing number of applications in marine, oil&gas, chemistry, industrial processes, construction and town planning

Computational Fluid Dynamics

- Solving method
  - Each method has pros and cons (accuracy, simulation time, type of physics)
  - The most widely used technique solves Navier-Stokes equations using finite volume method

VLM Vortex Lattice Method	Panel method	Navier-Stokes	SPH Smoothed Particles Hydrodynamics	LBM Lattice Boltzmann Method
• 40's	• 60's	• 60's-70's	• Late 70's	• Late 80s

Simple methods: interesting for pre-sizing or to have a rough estimate of aerodynamics performance

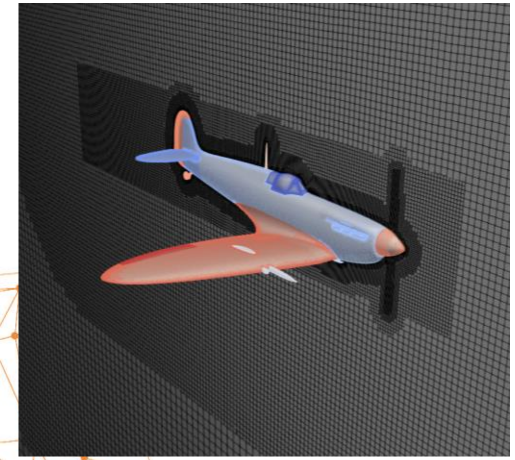
More computationally intensive methods: different types of physics can be simulated, in a more detailed and/or accurate fashion. Method should be chosen depending on the application

Computational Fluid Dynamics

- **General workflow for finite volume method (FVM)**

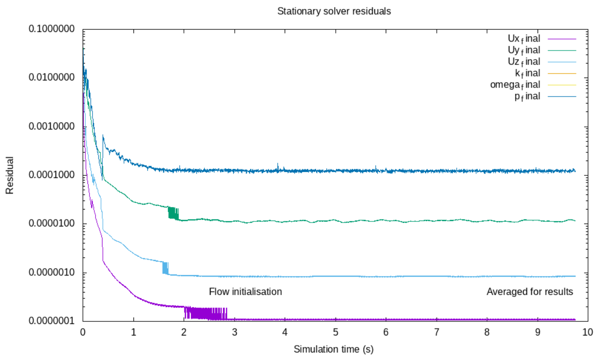
Pre-processing → Processing → Post-processing

CAD geometry import



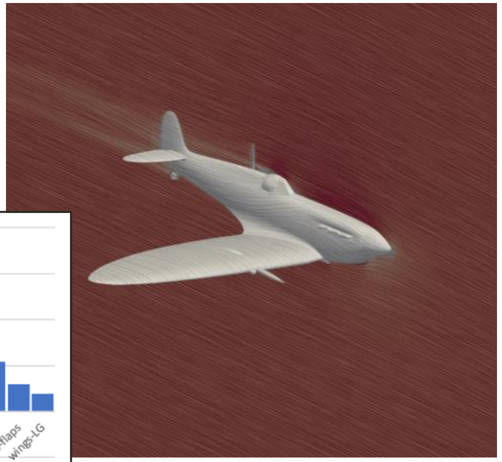
Meshing

Case setup

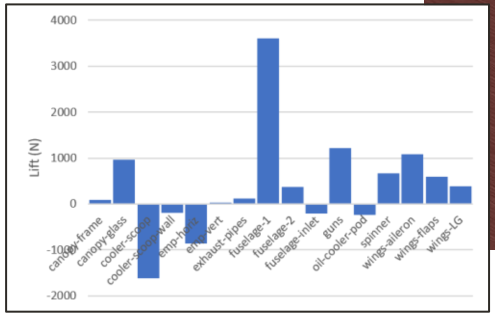


Simulation (X hours on N cores)

Flow visualization



Results analysis



Computational Fluid Dynamics

- **Navier-Stokes solving with Finite Volume Method**

Navier-Stokes

complexity (+ / -)

- DNS [1]
  Direct Numerical Simulation
- LES
  Large Eddy Simulation
- DES
  Detached Eddy Simulation
- RANS [2]
  Reynolds-Averaged Navier-Stokes

- Mesh
  - Cell count
  - Mesh refinement in areas of interest
  - Boundary layer resolution/modelling

- Numerical schemes
  - 1st /2nd order, space and time
  - Balance between stability and accuracy

- Turbulence model:
  - k-ε standard
  - k-ε realizable
  - k-ε RNG
  - LRR
  - Spalart-Almaras
  - k-ω standard
  - k-ω SST
  - k-ω Langtry-Menter SST
  - k-kL-ω
  - …

- Boundary conditions
  - Flow characteristics (velocity/flowrate, pressure, temperature, turbulence intensity, …)
  - Proper condition type depending on the physics involved

[1] Research applications only
[2] Industry standard

Computational Fluid Dynamics

- **Navier-Stokes resolution with Finite Volume Method**

Navier-Stokes

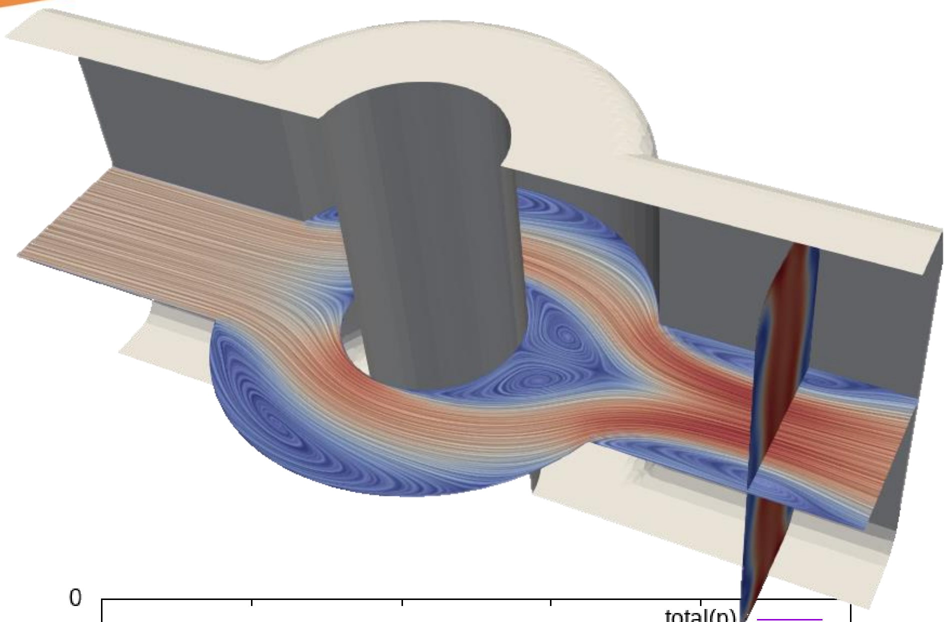
- DNS
  Direct Numerical Simulation
- LES
  Large Eddy Simulation
- DES
  Detached Eddy Simulation
- RANS
  Reynolds-Averaged Navier-Stokes

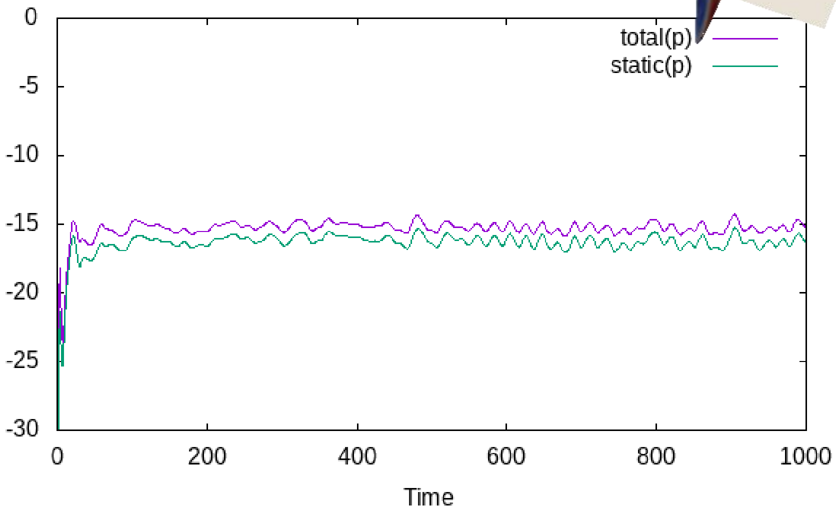
- Mesh
  - Cell count
  - Mesh refinement in areas of interest
  - Boundary layer resolution/modelling
- Numerical schemes
  - 1st /2nd order, space and time
  - Balance between stability and accuracy
- Turbulence model:
  - k-ε standard
  - k-ε realizable
  - k-ε RNG
  - LRR
  - Spalart-Almaras
  - k-ω standard
  - k-ω SST
  - k-ω Langtry-Menter SST
  - k-kL-ω
  - …
- Boundary conditions
  - Flow characteristics (velocity/flowrate, pressure, temperature, turbulence intensity, …)
  - Proper condition type depending on the physics involved

- Case setup relies on multiple parameters
- Each parameter choice has consequences on other parameters
- Requires a minimum of knowledge to setup at case
- Result is user-dependent

Computational Fluid Dynamics

- What to expect from a simulation?
  - Computed variables
    - Pressure/velocity, (turbulence, temperature, density, volume fractions, concentrations…)
  - Derived data
    - Forces, pressure loss (delta P), flowrates, heat fluxes, wall shear stress, …
    - Monitor variables in any location inside the simulation domain (point, surface, section, volume…)
  - Visualization :
    - 3D flow visualization
    - Allows to link results data to flow topology





Computational Fluid Dynamics

- **When to use simulation?**
  - Innovation :
    - Explore new ideas / concepts
    - Develop comprehension of the involved physics
  - Product design :
    - Explore several designs
    - Analyze flow behavior and identify potential improvement leads
    - Design pre-validation earlier in the project, before being able to produce first prototypes
  - Sizing
    - Product design is fixed, but sizing depends on client application → simulation can help to validate sizing relatively to the client specifications before starting production
  - Crisis management
    - Simulation can provide additional data to understand the causes of a malfunction and to give insights on potential solutions to solve the issue

Computational Fluid Dynamics

- Accuracy and sources of errors

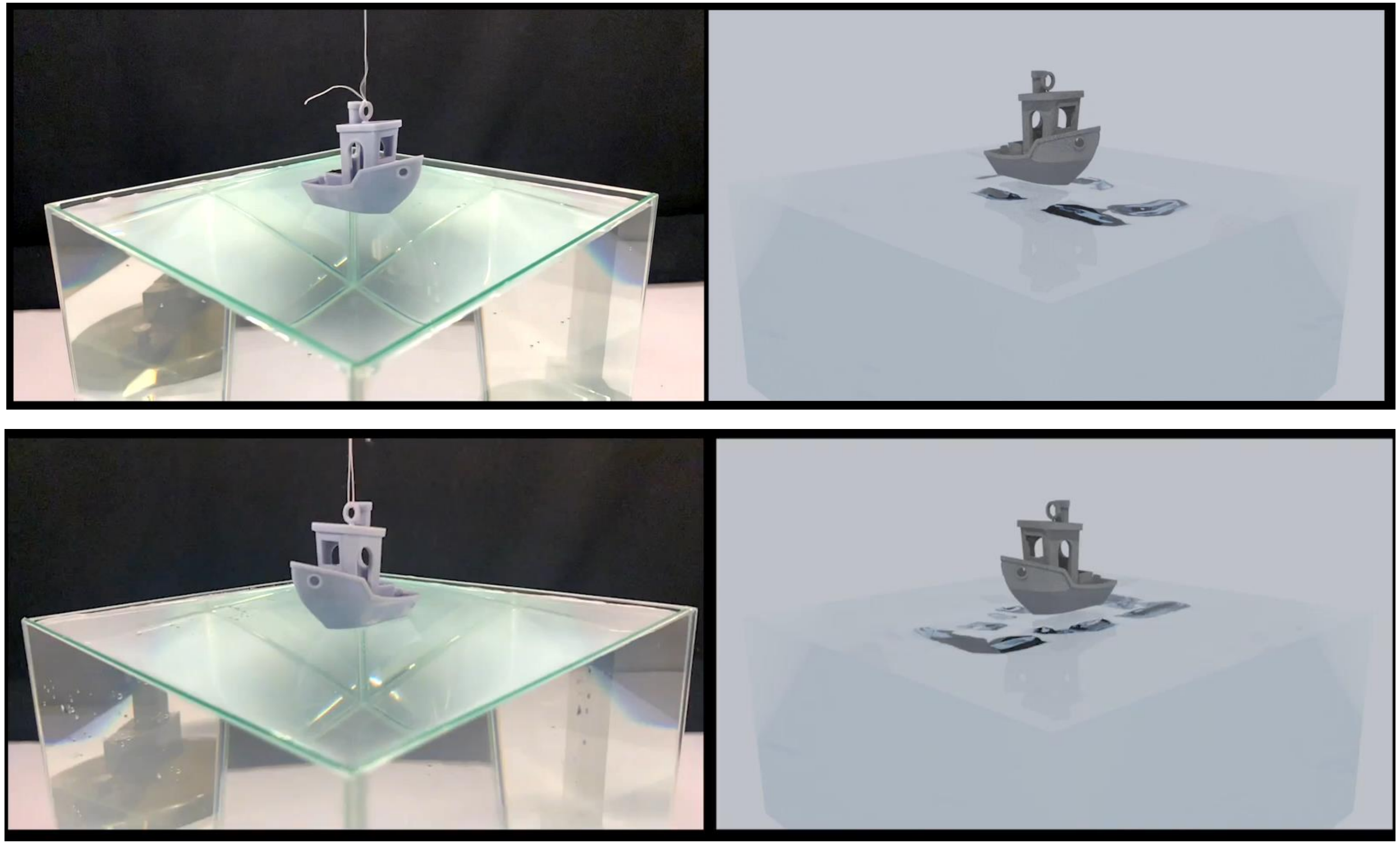
  - Total error is the sum of small errors related to different aspects of the simulation

    - CAD model error → simplified geometry, different from the real object (sealed watertight surface)
    - Mathematical model error → set of simplifying assumptions (incompressible, steady flow, viscosity model, …)
    - Boundary conditions error → simplification of the real world conditions
    - Discretization error → the real world problem (continuous) is discretized into space and time
    - Numerical error → rounding errors related to floating point precision
    - Human error → very common due to the complexity of case setup
    - Post-processing error → can arise from units mismatch, use of wrong quantities or mistakes in the calculations (e.g. area or flux weighted average)
    - X-factor error → small overlooked details or differences in a case can lead to big differences in results

Lubos Pirkl: *CFD is not a calculator*.

Computational Fluid Dynamics

- Accuracy and sources of errors



- Initial conditions
- Physical properties
  - Mass
  - Centre of gravity
  - Moment of inertia
- Numerical setup
- Others?

Accuracy and sources of errors

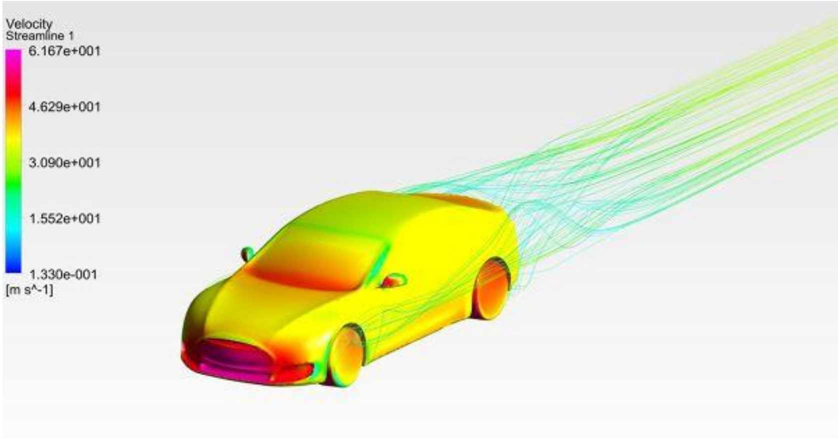
- « All models are wrong, but some are useful »
  - A simulation is always an approximation of reality
  - More or less simplifying assumptions are involved depending on the chosen models
  - More or less geometric simplifications
  - Less simplifications implies more expensive simulations, longer to compute

- « No one believes a simulation, except those who conducted it. Everyone believes an experiment, except those who conducted it. »
  - What is the geometry precision? Production tolerance, assembly, positioning precision, …
  - Do I use intrusive measurement techniques?
  - Am I quantifying the same data in the experiment and simulation?
  - How close is my simulation from the actual test conditions?

Computational Fluid Dynamics

- **Accuracy and sources of errors**

  - Examples of external aerodynamics
    - Car in wind tunnel ≠ Car in simulation ≠ Car in real driving conditions



    - Pressure measurement in wake → can (sometimes) influence the wake behavior

• Experiment or simulation?

- Both approaches are complementary
  - Exploring an operational range (velocity, flowrate, position…)
    → Advantage to experiment
  - Exploring geometric configurations
    → Advantage to simulation
  - Different cost and implementation time depending on the application

- Ideally
  - Experimental database on reference configurations
  - Setup simulations on these reference configurations
  - Once the simulation methodology is validated on reference cases, it can be applied to new cases
  - Correlation between tests and simulations is a good way to improve both simulation methodology and tests protocols

Computational Fluid Dynamics

• Rough order of magnitude

- Mesh
  - Small: few thousands to few millions cells
  - Intermediate : about ten millions cells
  - Big : hundreds of millions cells and more

- Hardware
  - Small : 8 core CPU, 32 Go RAM
  - Intermediate: 32 core CPU, 128 Go RAM
  - Big : 128 core CPU, 512 Go RAM → calculation server / cluster

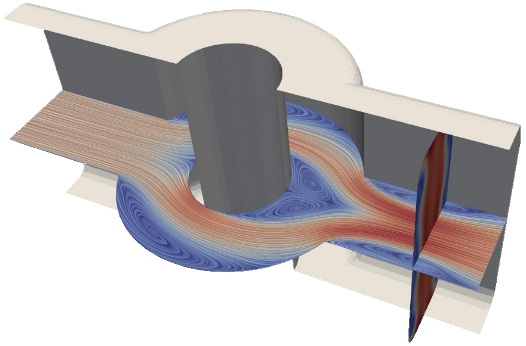
  - Storage: depending on the simulation, a case can range from few Go to several hundreds of Go

CPU	RAM
Acts on the simulation time, according to the number of cores and their frequency CFD does not profit from hyperthreading, only physical cores matter	Acts on the maximum admissible mesh size Rough estimation: about 2Go of RAM for 1 million cells

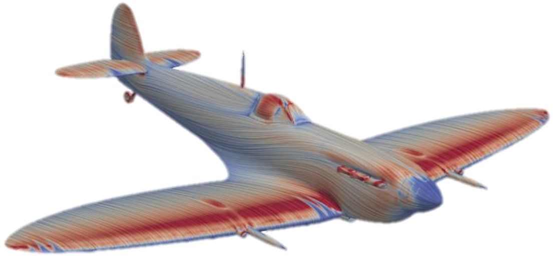
Computational Fluid Dynamics

- **Rough order of magnitude**

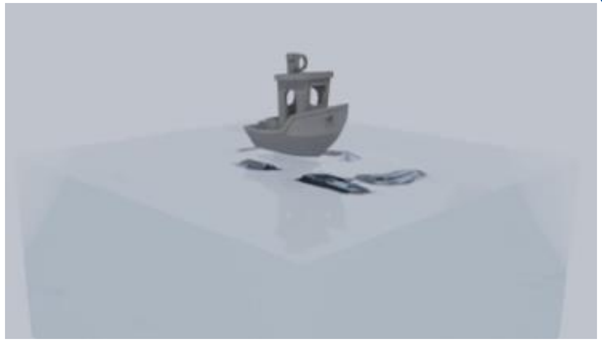
  - Simulation time
    - Few minutes to few days, weeks or even months depending on the mesh size, type of simulation and hardware specifications
    - Simulation time speedup achievable by increasing number of cores, up to a limit (time loss under 25 000 to 50 000 cells / core)



20min of simulation on 4 cores for 10s of physical time, 200 000 cells mesh (on a laptop)



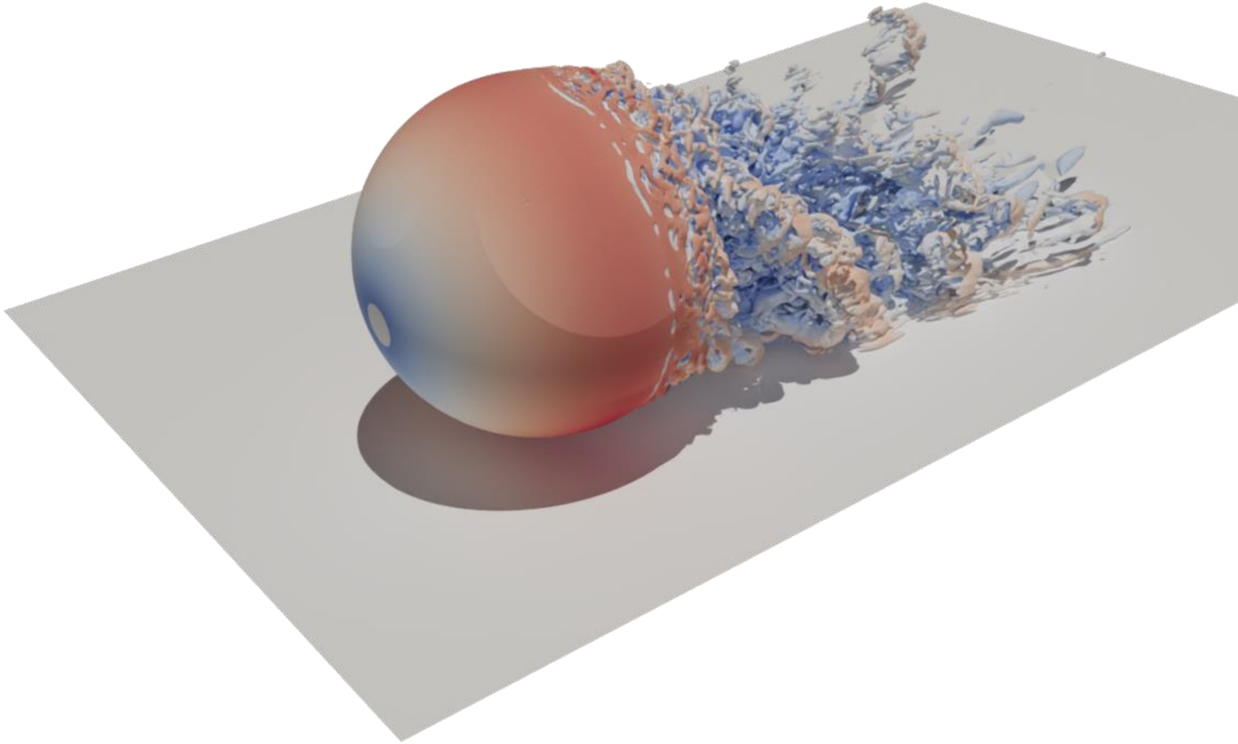
16h of simulation on 6 cores, 24 millions cells mesh. (steady state)



9 days of simulation on 24 cores for 2s of physical time, 2,3 millions of cells

Simulation software

- Market leaders on CFD software
- Choosing a software
    - Proprietary VS open source codes
    - Features / usage
    - Global cost
    - ACE experience



Simulation software

- Market leaders on CFD software

SIEMENS	Ansys	OpenFOAM
Simcenter Star-CCM+	CFX / Fluent	OpenFOAM Foundation / ESI-OpenCFD

Resolved Analytics: *CFD software user survey.* https://www.resolvedanalytics.com/cfd-user-survey-results

- Others

Proprietary	Open source
SIMSCALE, ProLB, COMSOL, DASSAULT SYSTEMES	Elmer, OpenLB, SU2 code, code_saturne

Simulation software

- Proprietary VS open source codes

SIEMENS	Ansys	OpenFOAM
Simcenter Star-CCM+	CFX / Fluent	OpenFOAM Foundation / ESI-OpenCFD

Proprietary software

- License costs: 20 to 50 k€/year/user according to the license type
- Included support
- Detailed documentation
- Graphical User Interface (GUI)

Open source software

- License cost: 0€
- No support
- No GUI
- Limited documentation

Simulation software

- **Proprietary VS open source codes**

  - Similar features are implemented on the 3 leading codes (DNS/LES/RANS, turbulence models, type of physics…)

Proprietary software

- Case setup through GUI
- GUI allows to orient user choices → easier user experience

- Can have a « black box » effect: it can be difficult to understand what is going on in the software due to closed sources

Open source software (OpenFOAM)

- Case setup through text files
- Considerable freedom in case setup, ability to tweak all details/parameters
- Great flexibility for automation

- This great freedom comes at the expense of a greater complexity which can be discouraging for beginners and new users

Simulation software

• Proprietary VS open source codes: work environment integration

Proprietary software

- Tends of gather all tools in a large software suite
- Allows to streamline user experience with a centralized workflow (from pre-processing to post-processing in a single place)
- Allows to access other tools if there is a need in the company

- For a small company, you pay 100% to use 10% of the software capacity
- Greater dependency on the software editor if the whole workflow depends on its software

Open source software (OpenFOAM)

- Integrated conversion tools to import/export data to other software
- Lets you choose what tools you want to use for pre/post-processing, whether it is proprietary or open source tools
- Ability to use third-party software to tailor to your needs (GUI, add-ons)

- Can require to learn how to use new tools if not already deployed in the company (CAD to prepare geometries, visualization tool for post-processing)

Simulation software

- **Proprietary VS open source codes: global cost**

Proprietary software

- Yearly large expense for the license renewal
- Easier to start using the software, thanks to the GUI, documentation and support

- Still requires a good knowledge in fluid mechanics and numerical simulation which can require time and/or training (additional costs)

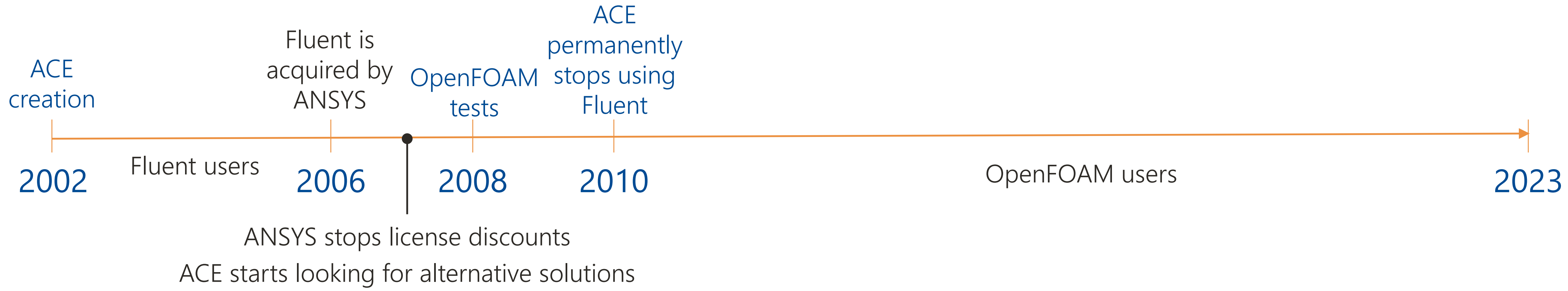
Open source software (OpenFOAM)

- No license cost: no investment required to start using the software
- Ability to use it on any hardware: laptop, workstation, calculation server, cloud, …

- BUT: larger investment required in time to get a grip of the software and develop skills
- Additional costs if you decide to buy training or support from the developers or third-party companies

Simulation software

- **Proprietary VS open source codes: ACE example**
  - Small company, expertise in fluid mechanics



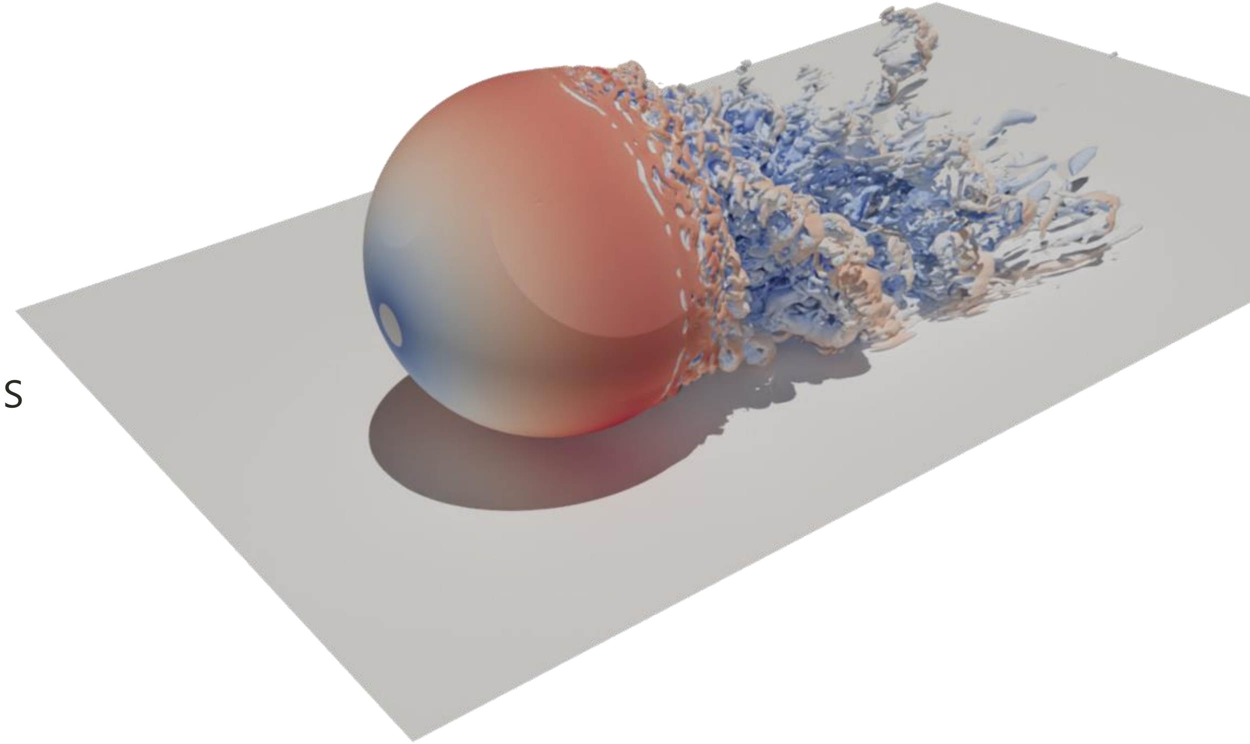
Cost
- Student hired as trainee during his gap year
- In-house training for the CFD team
- Fluent/OpenFOAM correlation tests

Today
- Independent learning to maintain and develop skills
- Investment in human resources and hardware rather than licenses

Service
- Training/support to help new users to get an easier start with OpenFOAM

Insourcing or outsourcing?

- Situational analysis: needs and resources
- Decision based on the workload
- Costs
- Intermediate choice: software package solutions



Insourcing or outsourcing?

- **Situational analysis: needs and resources**

  - Skills
    - Do I have in-house skills?
    - Do I need to train my team?
    - Do I have to hire?
  - Hardware
    - Do I have in-house resources? (workstation, calculation server)
    - Do I need to invest in hardware or cloud service?
  - Workload
    - What are my needs in CFD? Few days a year? Few months a year? Full time?

Insourcing or outsourcing?

- **Decision based on workload**

  - 2 to 10 simulations per year → outsourcing
    - It is difficult to maintain skills in-house if the workload is too low
    - A regular activity is required to be efficient

  - 2 to 10 simulations per month → outsourcing OR insourcing
    - Depends on the required investment (human and hardware) and the frequency (steady or occasional need throughout the year)
    - Start with outsourcing to assess the potential of CFD for my activity, then insourcing to reduce costs
    - 3rd option: software package tailored to your needs

  - 2 to 10 simulations per week → invest and insource
    - For a steady workload over the year, it is better to invest over several years to develop skills in-house. Cheaper on the long run, and the know-how stays in-house.
    - Software package can also be an option

Insourcing or outsourcing?

- Costs

  - Outsourcing: about 4500€/week
    - Be careful about what you are buying: do you expect raw results or a counselling service where the subcontractor will provide analysis and guidance?

  - Insourcing: licences costs (0 to 50k€/year/user) + optional additional investments
    - Evaluate hardware investment
    - Evaluate human investment
      - Senior engineer hire
      - Junior engineer hire + period to develop skills and experience
      - Team members training + period to develop skills and experience
    - Evaluate your needs in training/support

  - Time spent on a project: order of magnitude
    - First simulation: from 1 day to 3 weeks according to the simulation complexity
    - Next simulations: from 1 hour to 1 day

Insourcing or outsourcing?

- **Another option: software package (ACE)**
  - Aim
    - Create a tool dedicated to a specific application
    - Allow to restrict the usage perimeter to define a specific simulation methodology
  - Operation
    - User provides 3D geometry (STL/OBJ) and a list of parameters (operating point, ...)
    - The tool automatically creates mesh, runs simulation(s) and processes results
    - User gets formatted results (images, plots, data tables...)
  - Which usage?
    - Recurrent simulations (sizing or product development)
    - Can allow to deal with usual needs alongside a CFD activity for R&D (in-house or through subcontractor)

Insourcing or outsourcing?

- **Another option: software package (ACE)**

  - Pros
    - Simple usage, no need to have specific skills in numerical simulation
    - Adjusts to the profession (input/output, vocabulary, …)
    - Automates repetitive tasks which are prone to human error

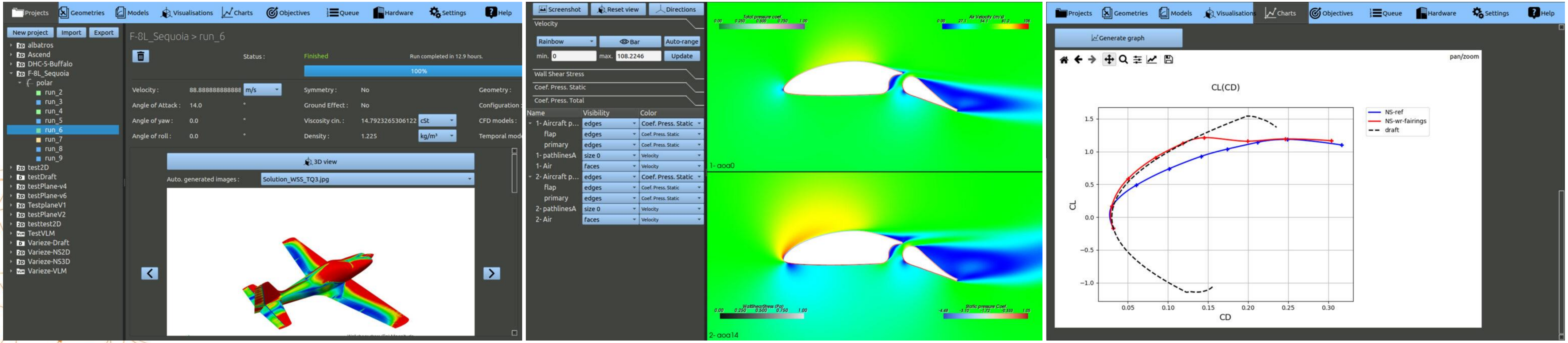
  - Cons
    - Limited to a predefined perimeter with limited options on the simulation parameters and expected outputs
    - Not as flexible as a full CFD software

  - Costs
    - Depends on the application and the expected level of usability (automated scripts or GUI)
    - Specific development or off-the-shelf package
    - For a specific development: 10 to 50k€. You own the software package.

Insourcing or outsourcing?

- **Another option: software package (ACE)**
  - Example : Ace Of Aircraft
    - Tool dedicated to general aviation
    - Evaluation of aircraft aerodynamics performances
    - Includes different tools, from pre-sizing to 3D CFD
    - Limited to subsonic flight on small aircraft (does not allow to simulate an Airbus A380 !)

Cost : 6 to 8k€/year depending on the license (no limitation on users or cores)



Insourcing or outsourcing?

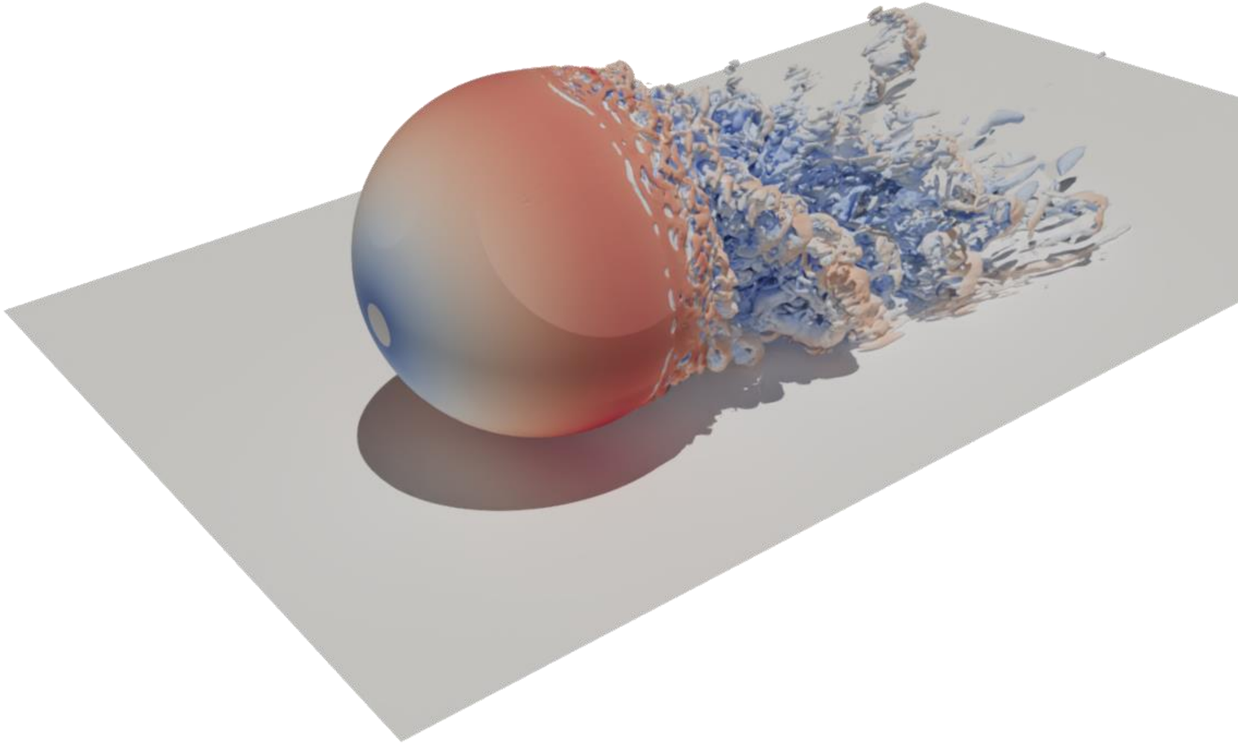
- **Another option: software package (ACE)**

  - Typical Ace Of Aircraft user:
    - Small company
    - Irregular needs for CFD
    - No budget for a full CFD software costing 50k€/year
    - To small to hire a CFD expert

User 1	User 2	User 3
• Simple user: the tools included in Ace Of Aircraft cover all the company needs regarding aerodynamics	• Regular use of Ace Of Aircraft • Occasional subcontracting projects for specific needs	• Subcontracting • Insourcing (training) • Ace of Aircraft for standard simulations + CFD expert

To dig deeper…

- Questions / Answers
- Contact



To dig deeper…

- Questions / Answers

CFD consultancy / Software packages / Wind tunnel
alexis.lapouille@aero-ce.com

03 86 58 01 65
www.aero-ce.com

CFD consultancy / OpenFOAM trainings
yann.recoquillon@aero-ce.com

AERO CONCEPT ACE ENGINEERING ©

Technopole, 58470 MAGNY-COURS
03 86 58 01 65

www.aero-ce.com

Chairman
Alexis LAPOUILLE
alexis.lapouille@aero-ce.com

Training manager
Yann RECOQUILLON
yann.recoquillon@aero-ce.com