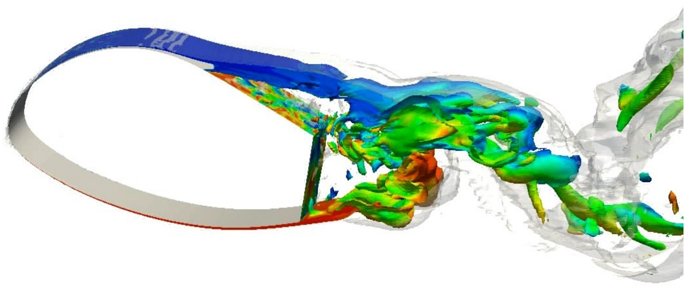
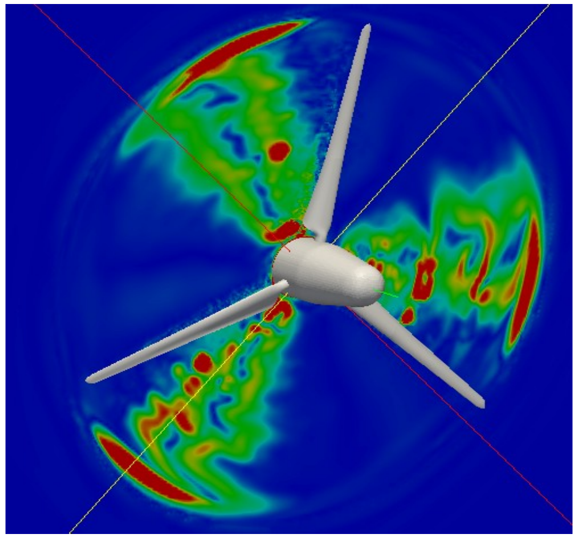
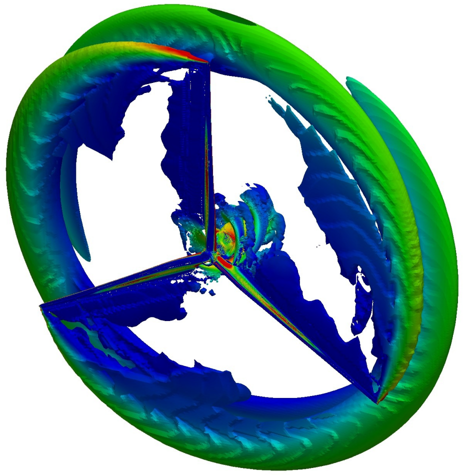
Computational Modeling of Wind Turbines in OpenFOAM







Hamid Rahimi
hamid.rahimi@uni-oldenburg.de

ForWind - Center for Wind Energy Research
Institute of Physics,
University of Oldenburg, Germany

Outline

- Computational Fluid Dynamics (CFD)
- A CFD library: Introduction to OpenFOAM
- CFD aerodynamic research
  - 2D → Airfoils
  - 3D → Rotor blades
- Summary

ForWind
Center for Wind Energy Research

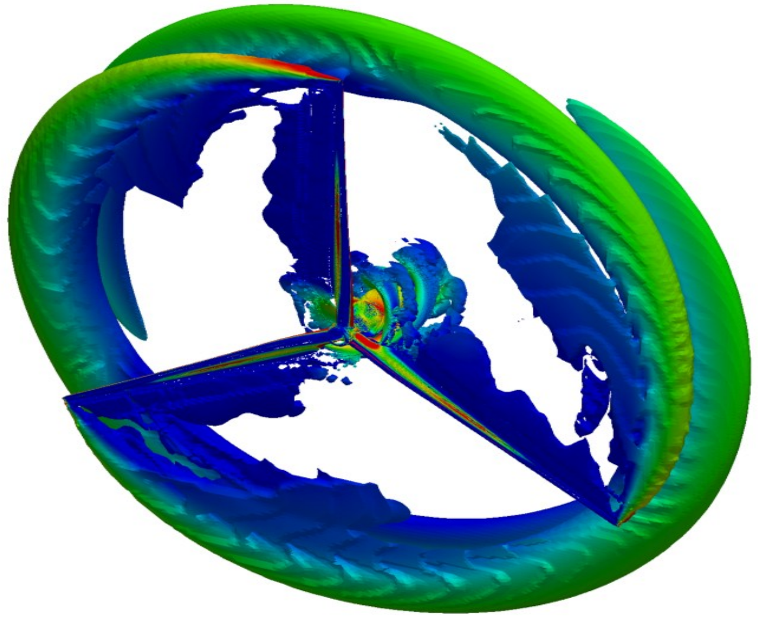
- More than 20 institutes of the Universities:
- Oldenburg, Bremen, Hannover (500 employees)
- Fundamental research in every field of wind energy

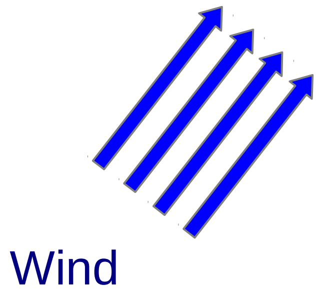
Fraunhofer IWES

- Member of Fraunhofer society
- Fraunhofer IWES with 500 employees
- Applied research in every field of wind energy

Computational Fluid Dynamics

- Navier-Stokes equations (NSE)
- Numerical modelling of NSE
  - Can be cheaper than Experiment
  - Can be fast
  - Gain detailed insight into entire flow field
  - Reproducible
  - A better understanding of flow phenomena leads to more control over them





What is OpenFOAM?

- Open Field Operation and Manipulation – software
- Huge Library of field calculation tools
- Mainly used for CFD calculations
- OpenSource – with different branches

OpenFOAM

What is OpenFOAM?

Open∇FOAM

Advantages

- Open Code – you can change it
- Very powerful toolbox for own development
- Once you know it – you know what it does!
- Free
- There is now a large community using and improving it

What is OpenFOAM?

OpenFOAM

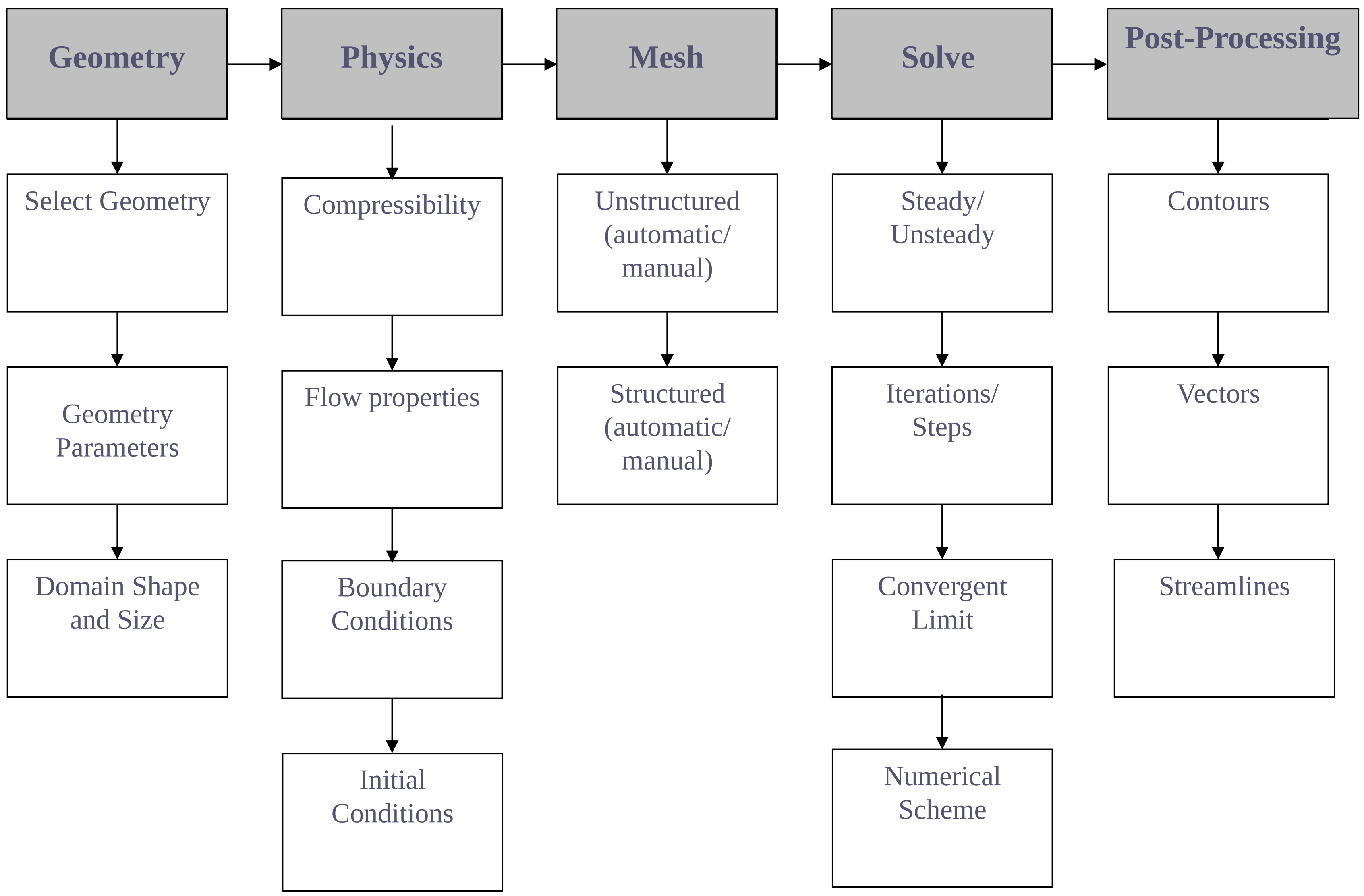
Advantages

- Open Code – you can change it
- Very powerful toolbox for own development
- Once you know it – you know what it does!
- Free
- There is now a large community using and improving it

Disadvantages

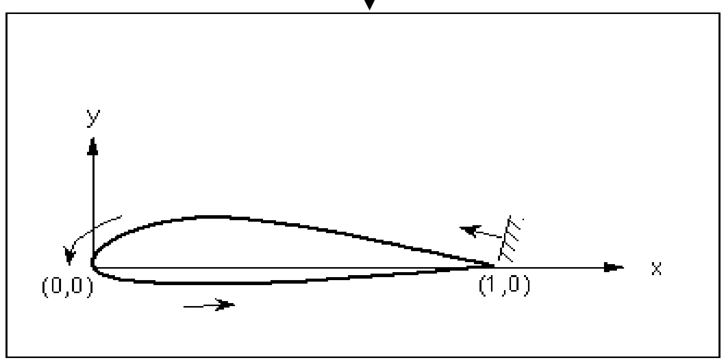
- Open Code – you can do what you want – is not always correct
- Steep learning curve
- Documentation is severe issue

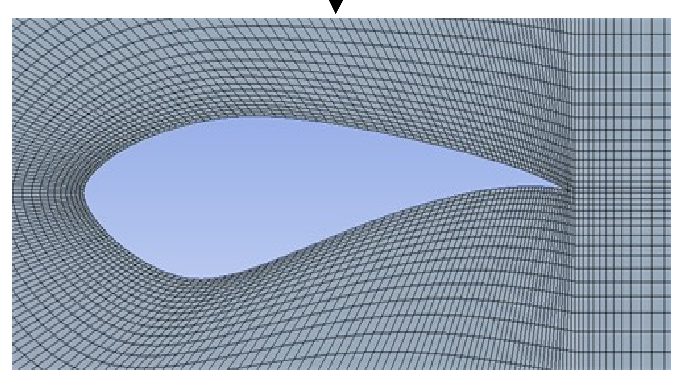
CFD Process

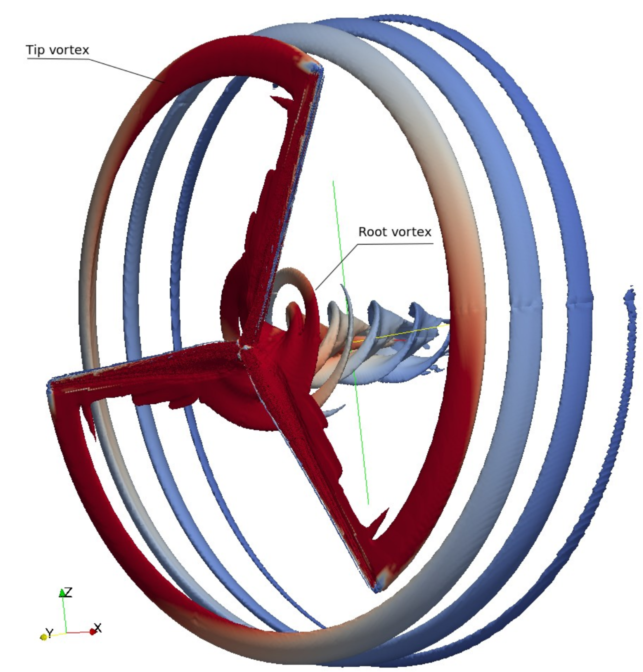

Geometry
Physics
Mesh
Solve
Post-Processing
Select Geometry
Geometry Parameters
Domain Shape and Size
Compressibility
Flow properties
Boundary Conditions
Initial Conditions
Unstructured (automatic/ manual)
Structured (automatic/ manual)
Steady/ Unsteady
Iterations/ Steps
Convergent Limit
Numerical Scheme
Contours
Vectors
Streamlines

CFD Process

Geometry → Physics → Mesh → Solve → Post-Processing





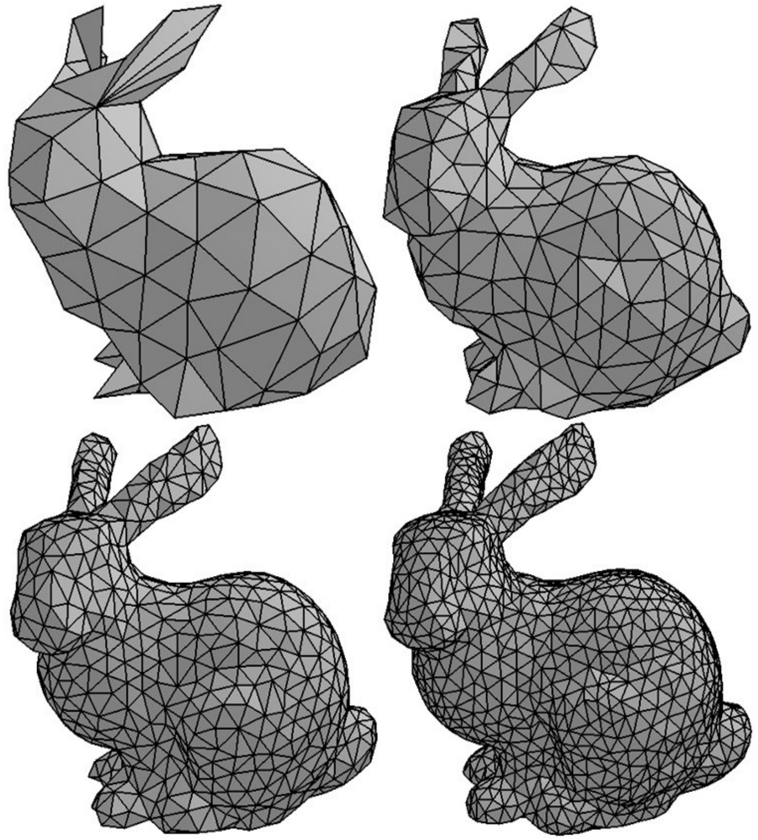


CFD Process

Geometry → Physics → Mesh → Solve → Post-Processing

Meshing

- CFD requires discretization
- Size & Quality impact:
  - Solution
  - Computation time
  - Convergence

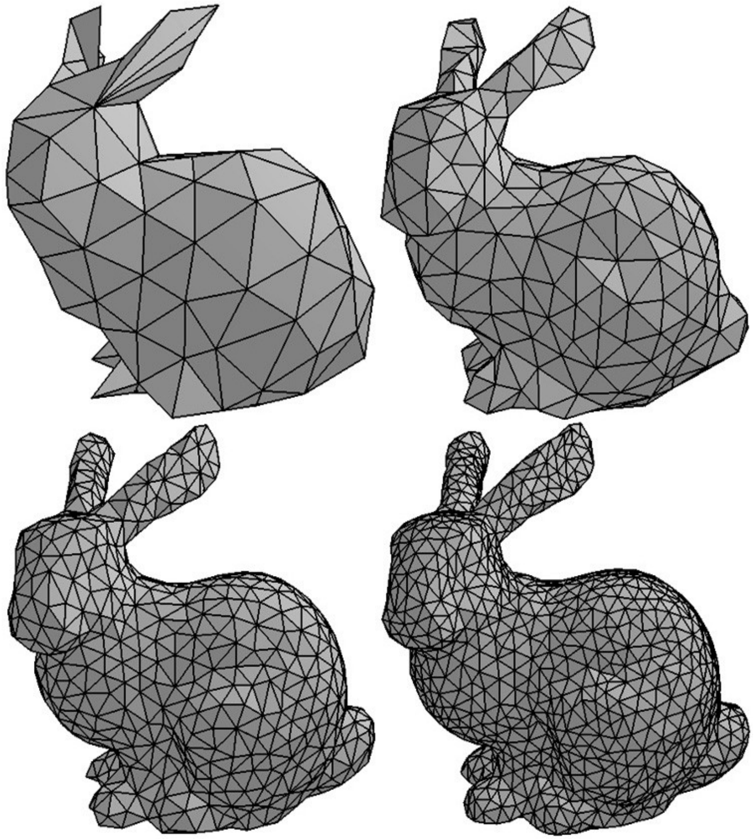


Ref: www.cmap.polytechnique.fr

Meshing

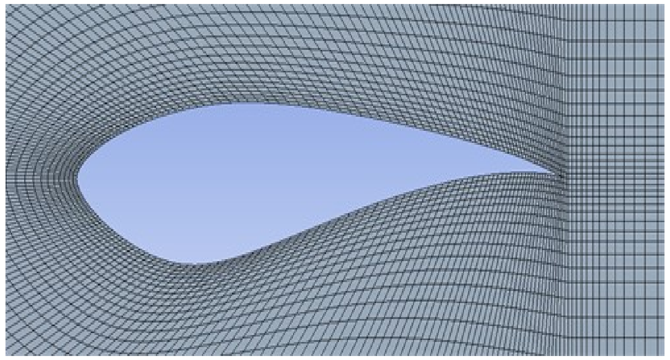
- CFD requires discretization
- Size & Quality impact:
  - Solution
  - Computation time

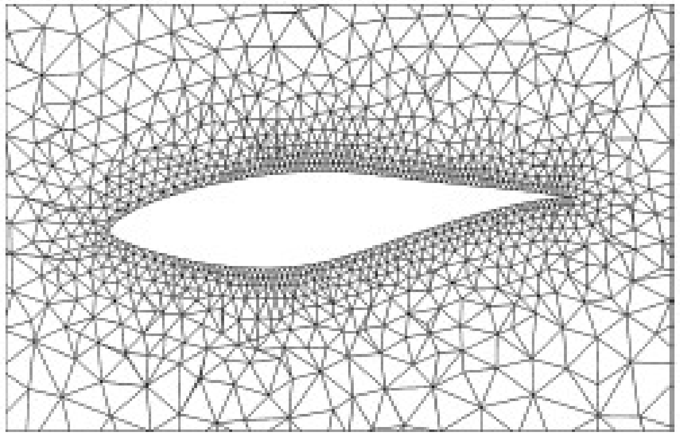
Highly important, non trivial, most time consuming step in preprocessing

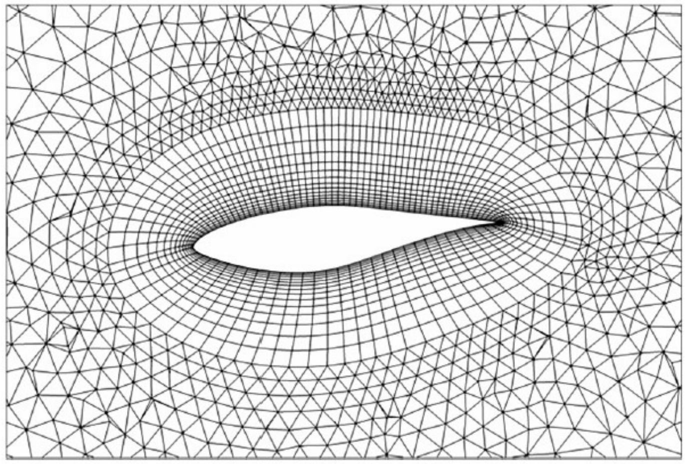


Ref: www.cmap.polytechnique.fr

Mesh classification



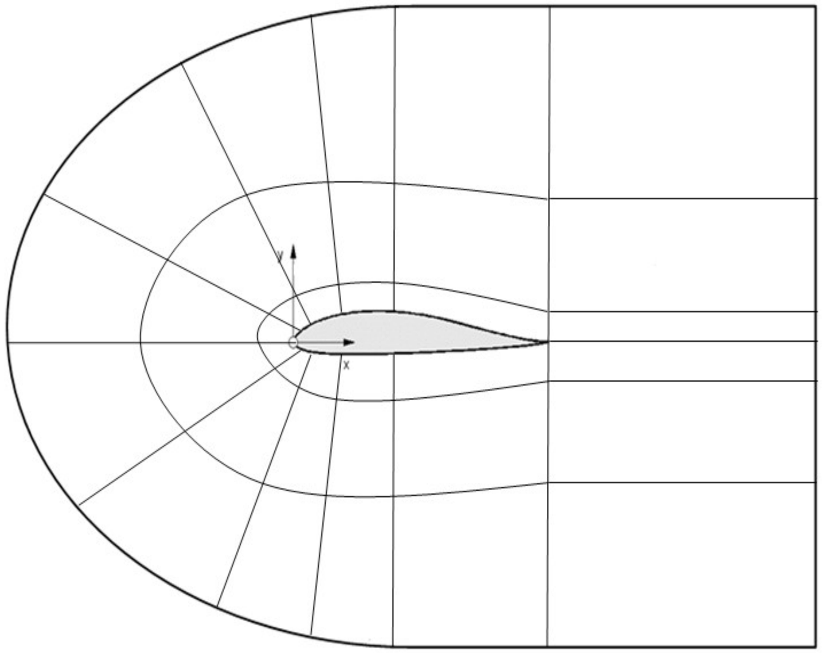




- Structured:
  - Identified by regular connectivity
  - Hexahedral in 3D
  - Can effect on efficiency and convergence
- Unstructured:
  - Identified by irregular connectivity
  - Tetrahedral in 3D
  - Faster to create
- Hybrid grids

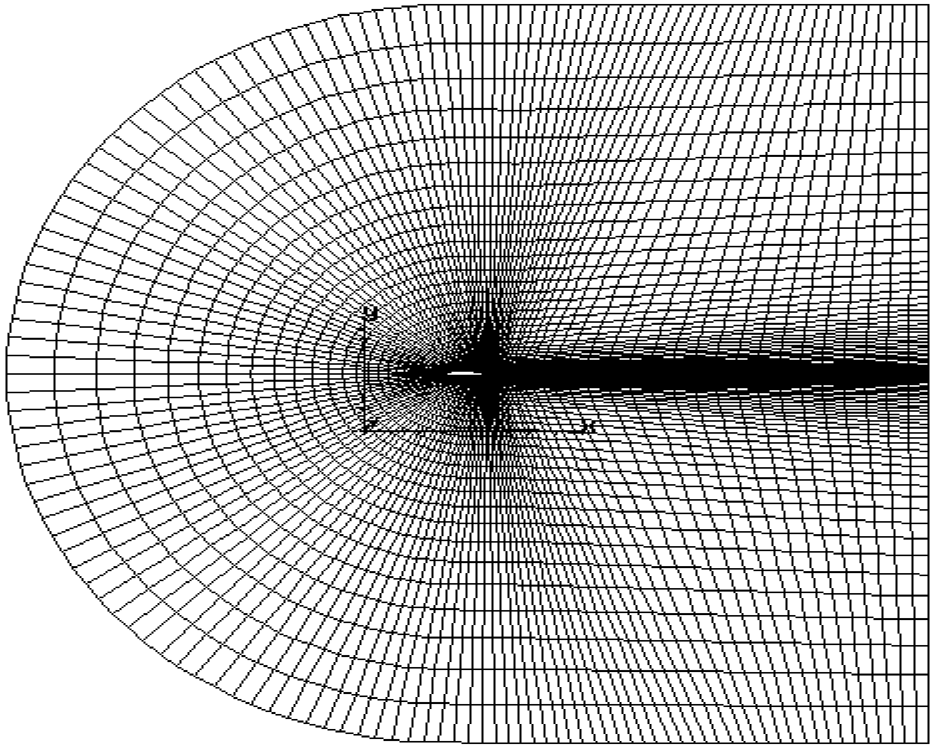
Meshing with OpenFOAM

- blockMesh
  - Structured mesh
  - Block decomposition of the computational domain
  - Simple geometries
  - Time consuming procedure



Meshing with OpenFOAM

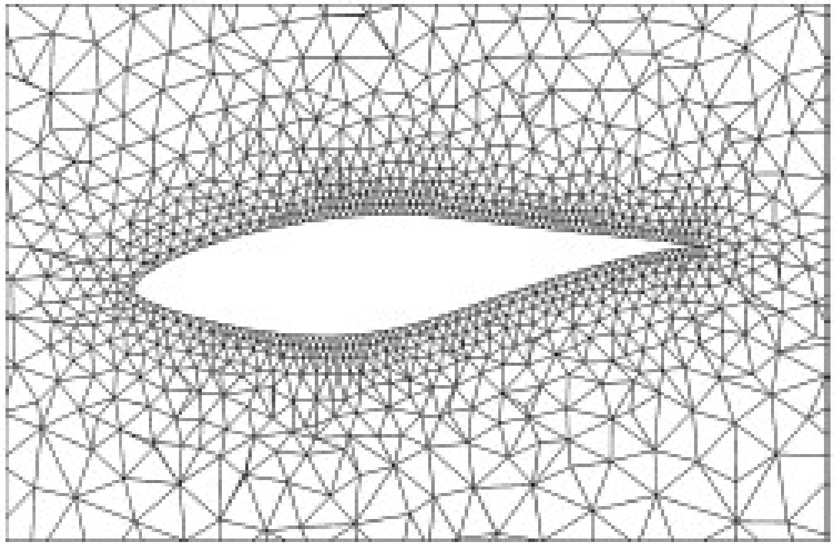
- blockMesh
  - Structured mesh
  - Block decomposition of the computational domain
  - Simple geometries
  - Time consuming procedure



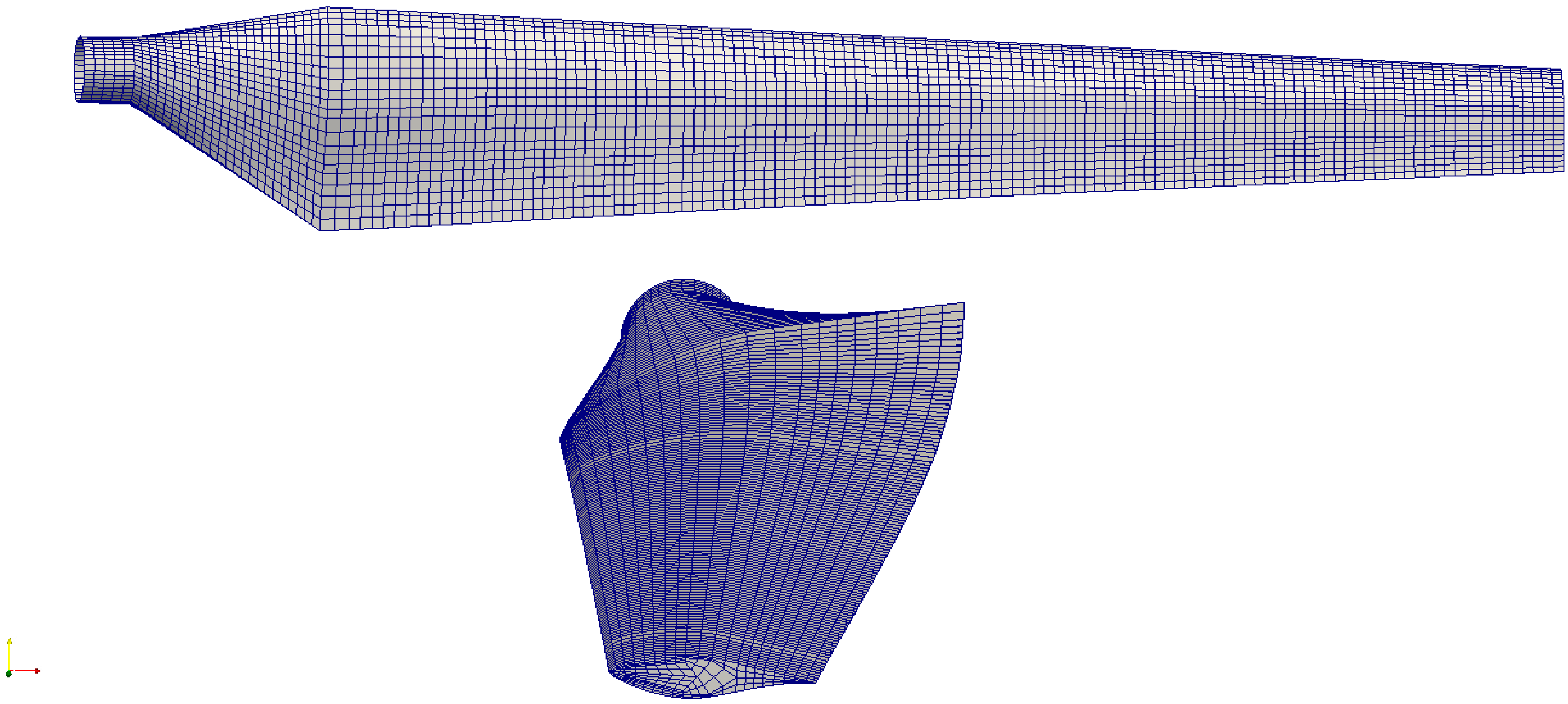
Meshing with OpenFOAM

- snappyHexMesh
  - Unstructured mesh
  - Meshes directly to surfaces from CAD file
  - Can be a time consuming procedure
  - Problem with sharp edges:

    eg:Trailing edge of blades, can not represent the geometry well



Mesh generation tool



CFD Process

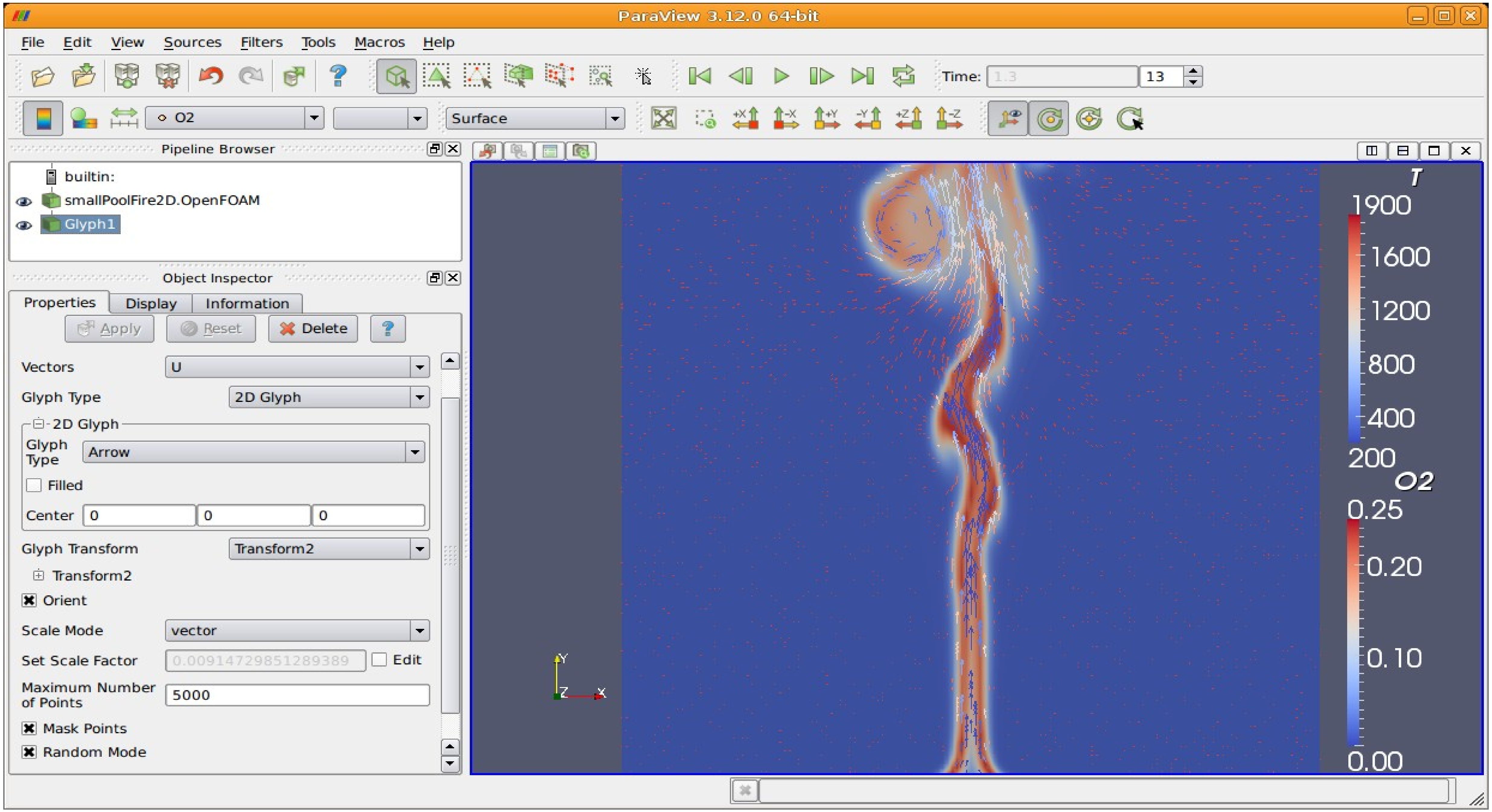
Geometry → Physics → Mesh → **Solve** → Post-Processing

CFD Process

CFD Process

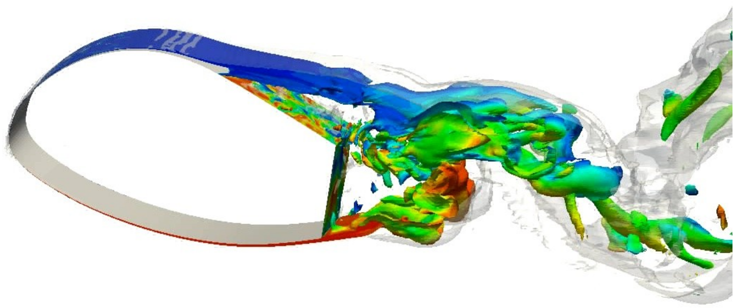
Geometry → Physics → Mesh → Solve → **Post-Processing**

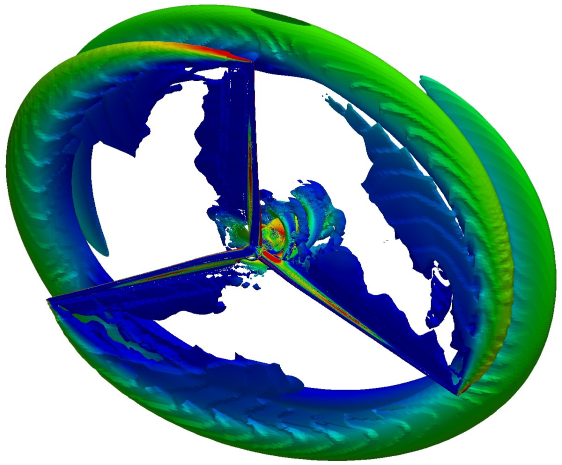
CFD Process


ParaView 3.12.0 64-bit
File Edit View Sources Filters Tools Macros Help
Time: 13
O2
Surface
Pipeline Browser
builtin:
smallPoolFire2D.OpenFOAM
Glyph1
Object Inspector
Properties Display Information
Apply Reset Delete
Vectors U
Glyph Type 2D Glyph
2D Glyph
Glyph Type Arrow
Filled
Center 0 0 0
Glyph Transform Transform2
Transform2
Orient
Scale Mode vector
Set Scale Factor 0.00914729851289389 Edit
Maximum Number of Points 5000
Mask Points
Random Mode
T
1900
1600
1200
800
400
200
O2
0.25
0.20
0.10
0.00

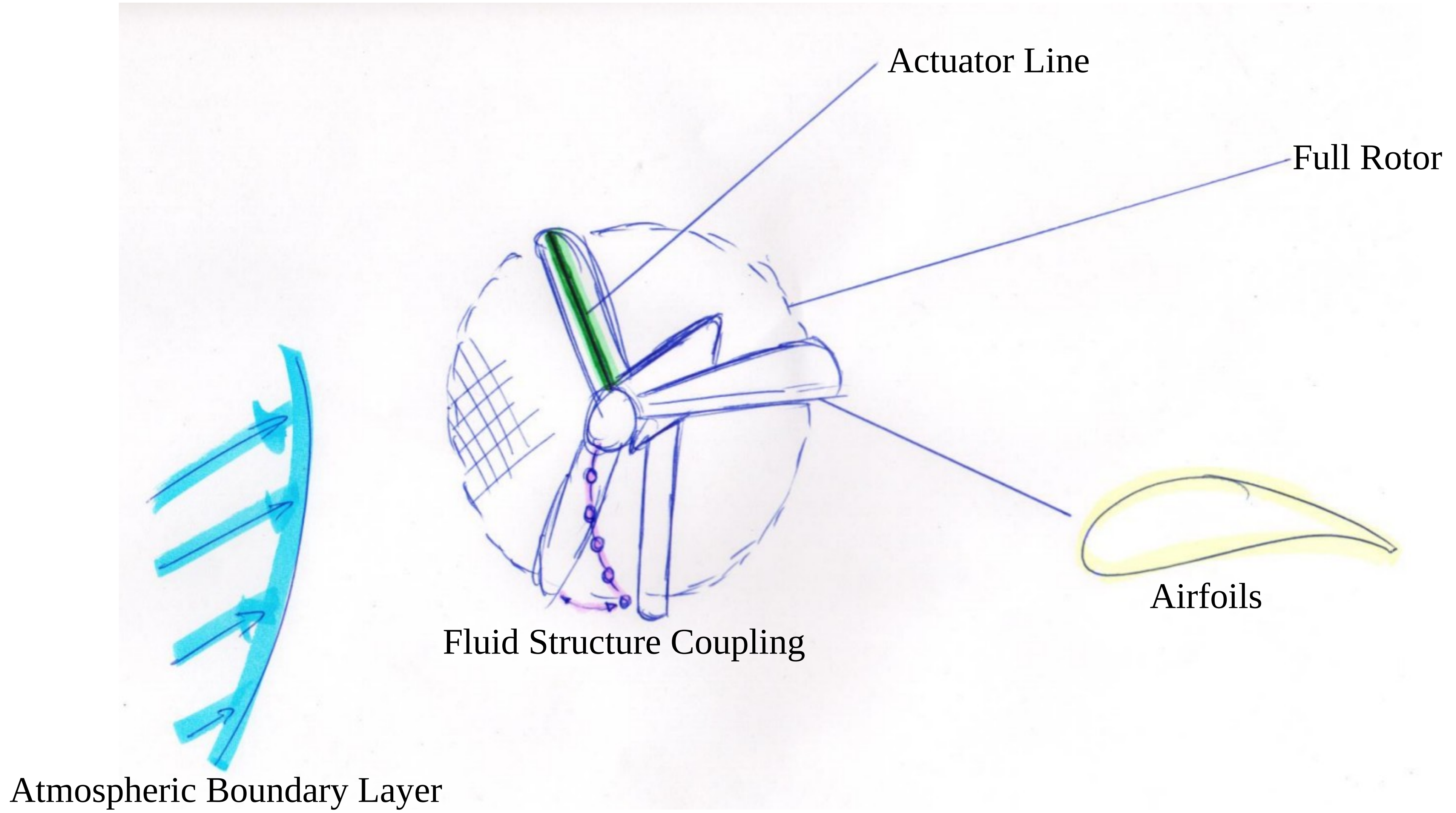
Why should we use CFD methods in wind energy?

- Load calculations based on 2D models with limited accuracy
- Especially in non-standard load cases models show problems (e.g. yawed inflow)
- In non-standard cases for atmospheric flows (complex terrain, water, ...)
- Detailed aerodynamics only with measurements or CFD



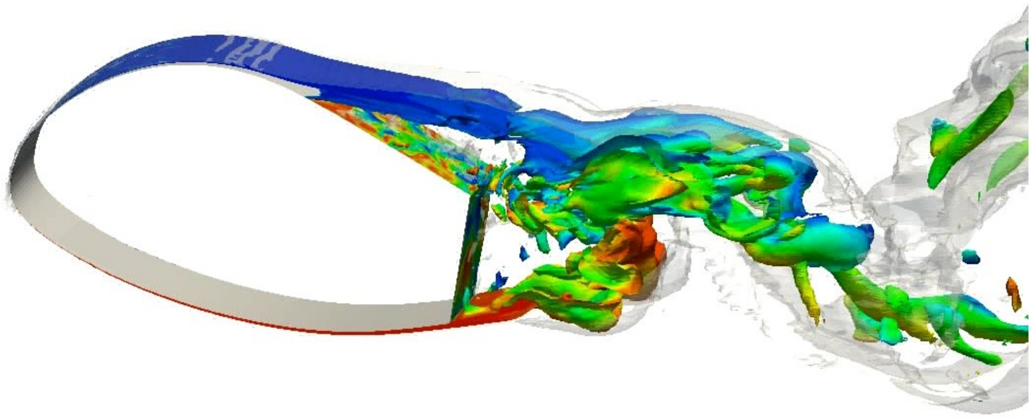


Why should we use CFD methods in wind energy?



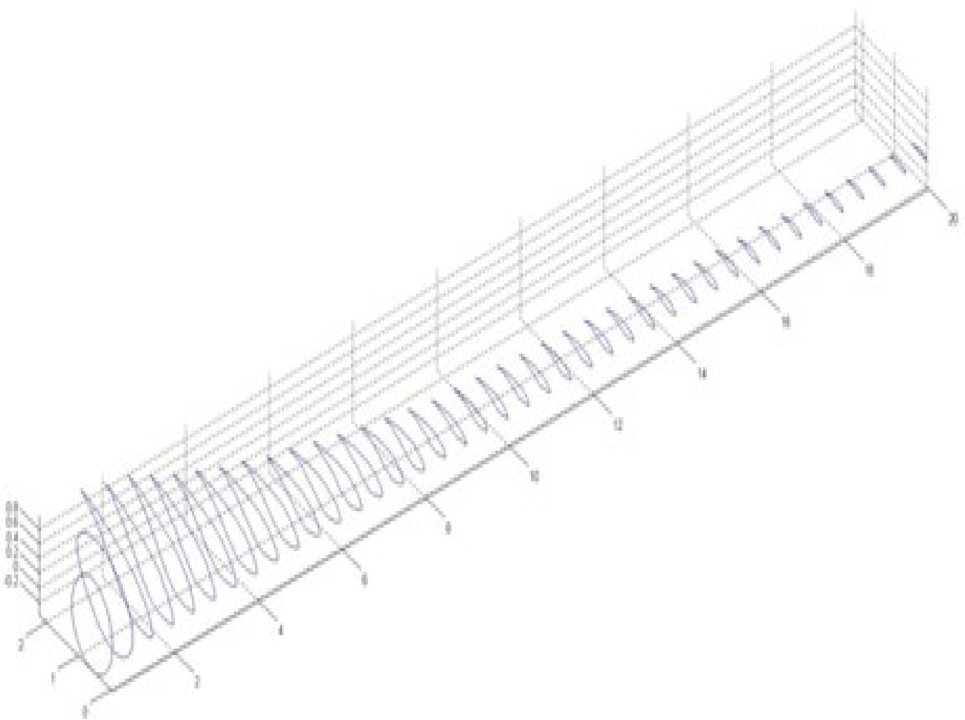
CFD Simulations: Airfoil

- 2D Airfoil Characteristics needed for:
  - Airfoil Design
  - Blade Design
  - Load Calculations



CFD Simulations: Airfoil

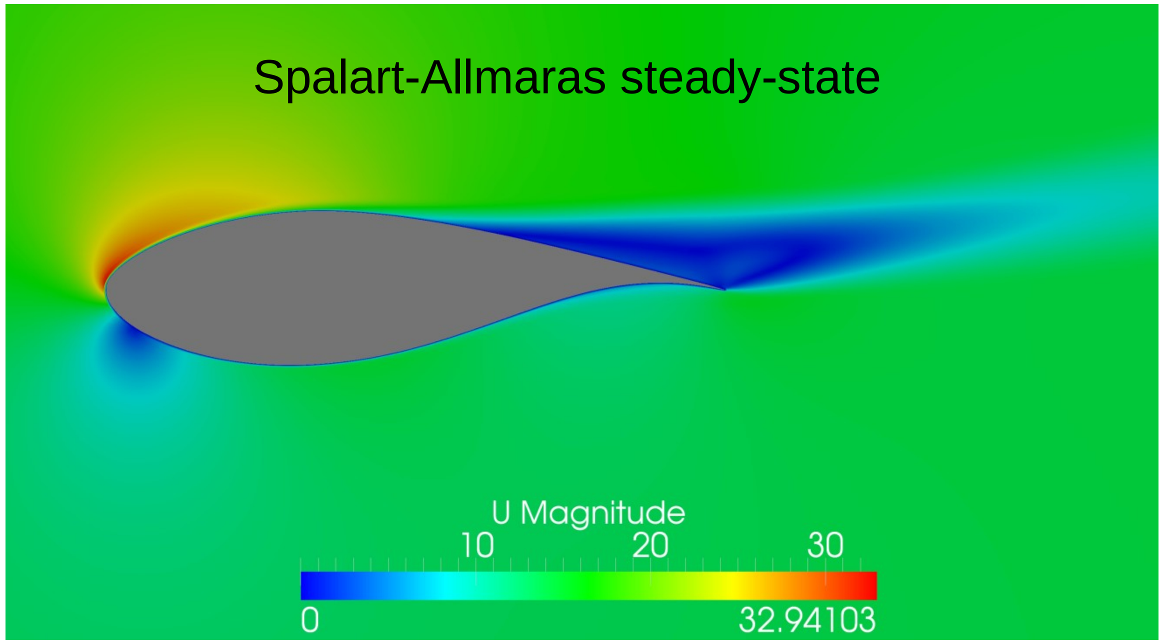
- 2D Airfoil Characteristics needed for:
  - Airfoil Design
  - Blade Design
  - Load Calculations



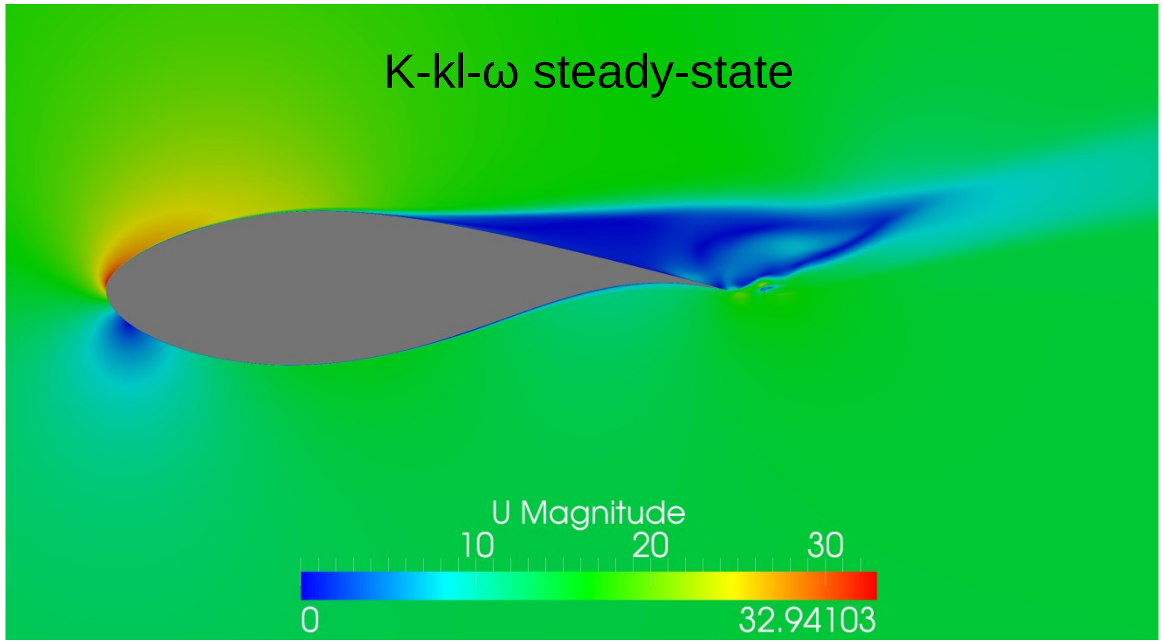
Ref: [www.siemens.com]

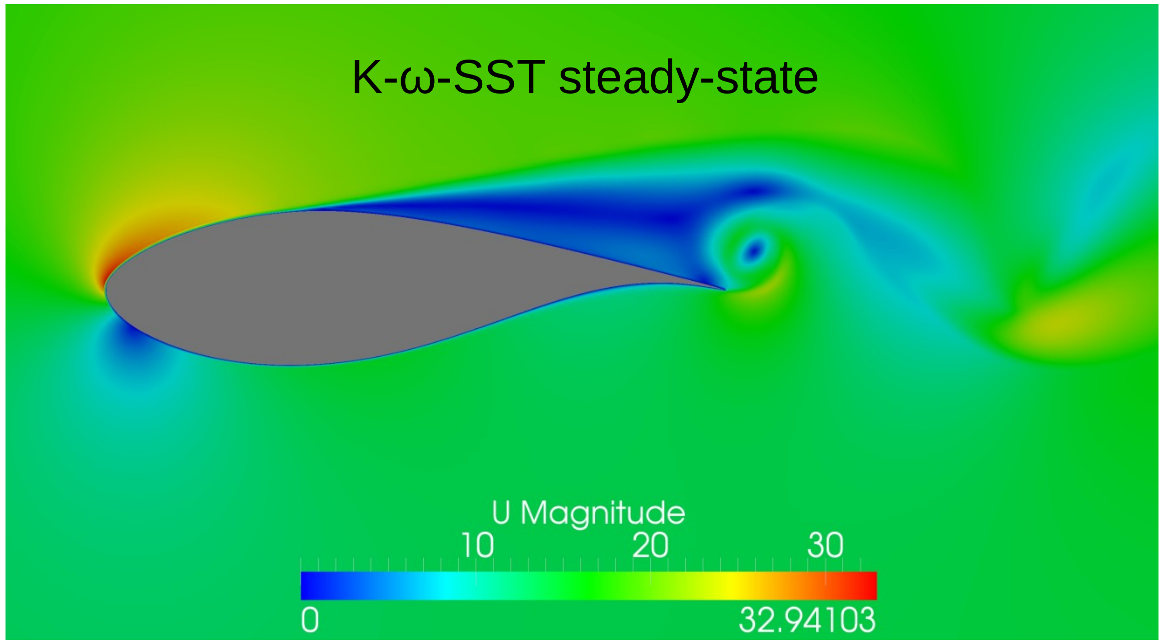
DU-91-W2-250

Velocity Distribution – Re = 1E6, α = 15.19°



Know your model!

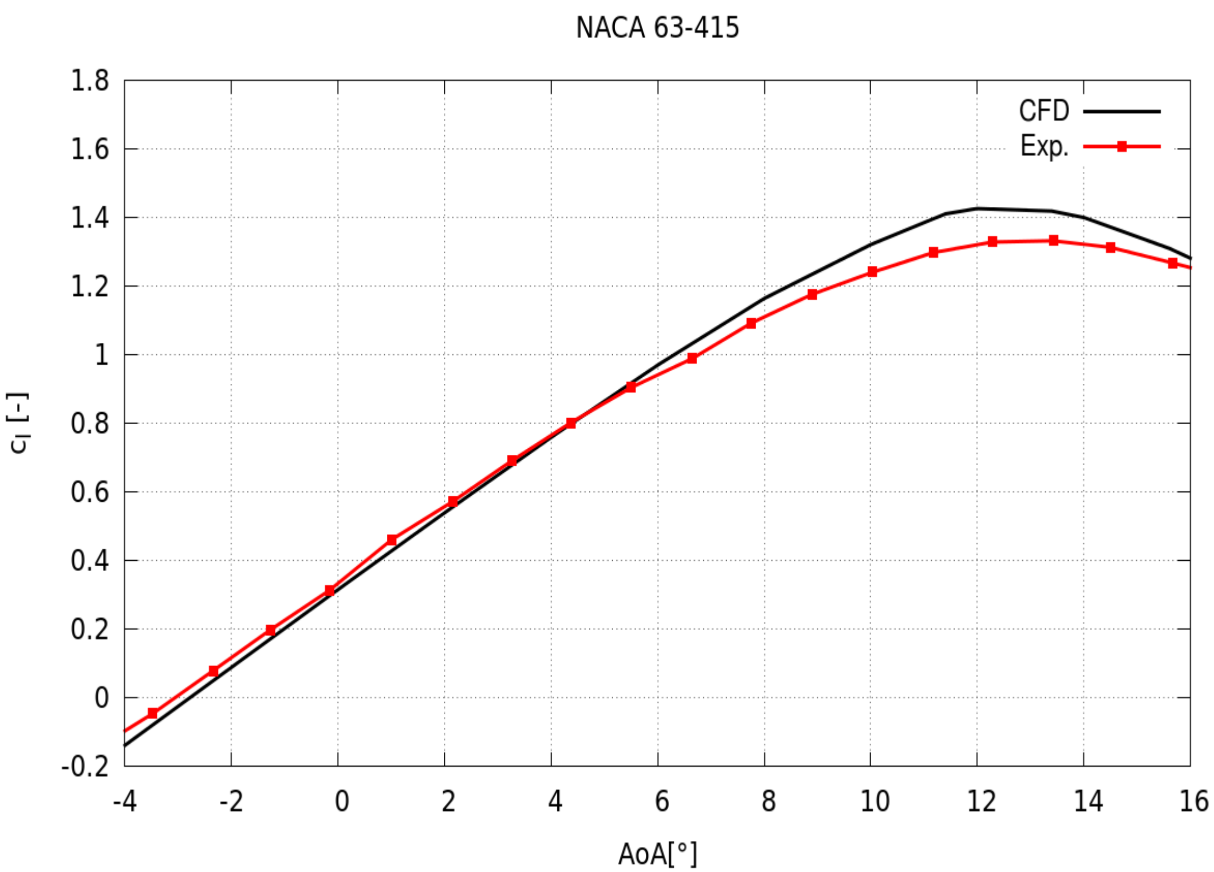


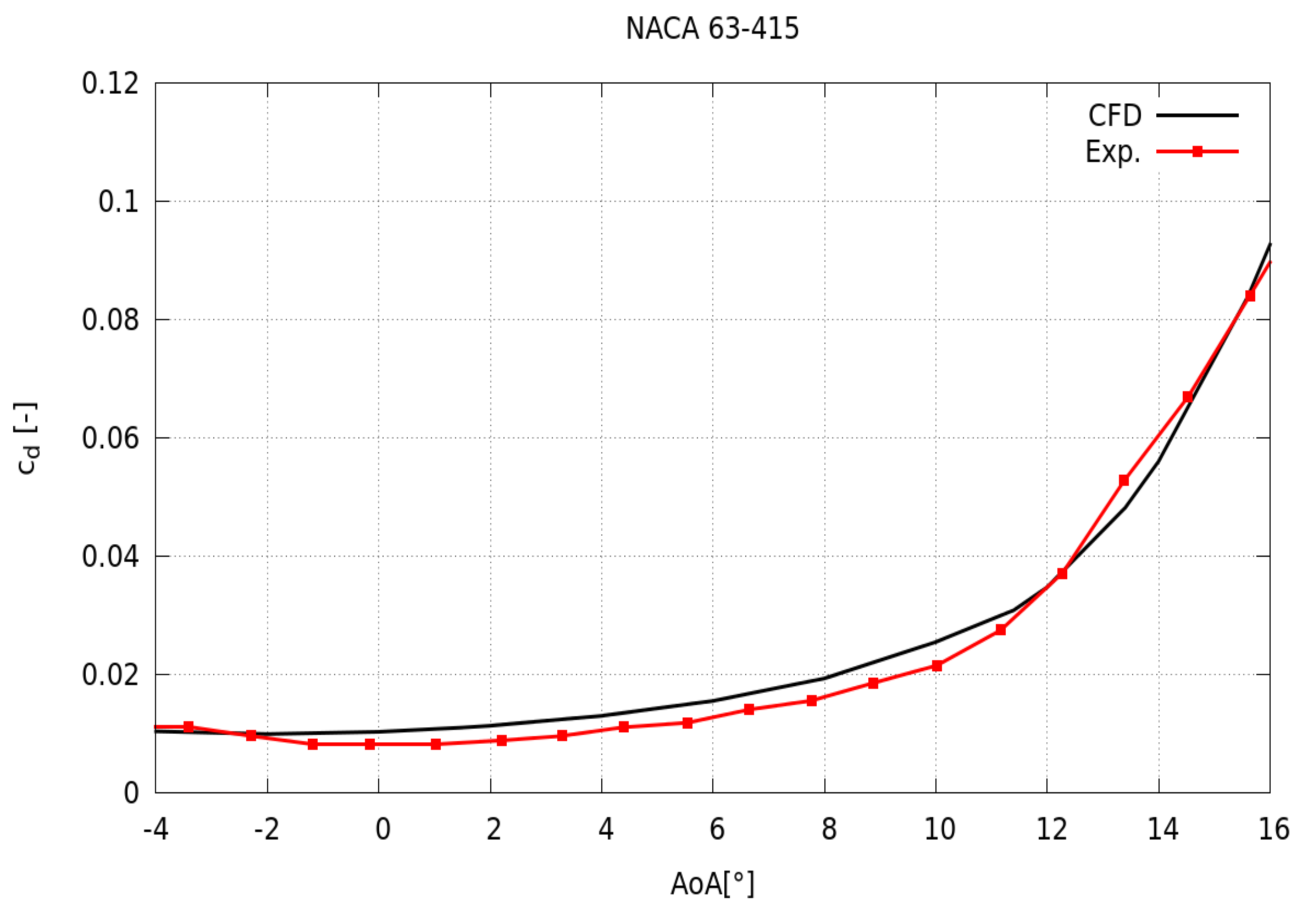


Ref: Rahimi.H ForWind Center for Wind Energy Research, Institute of Physics, University of Oldenburg, Germany

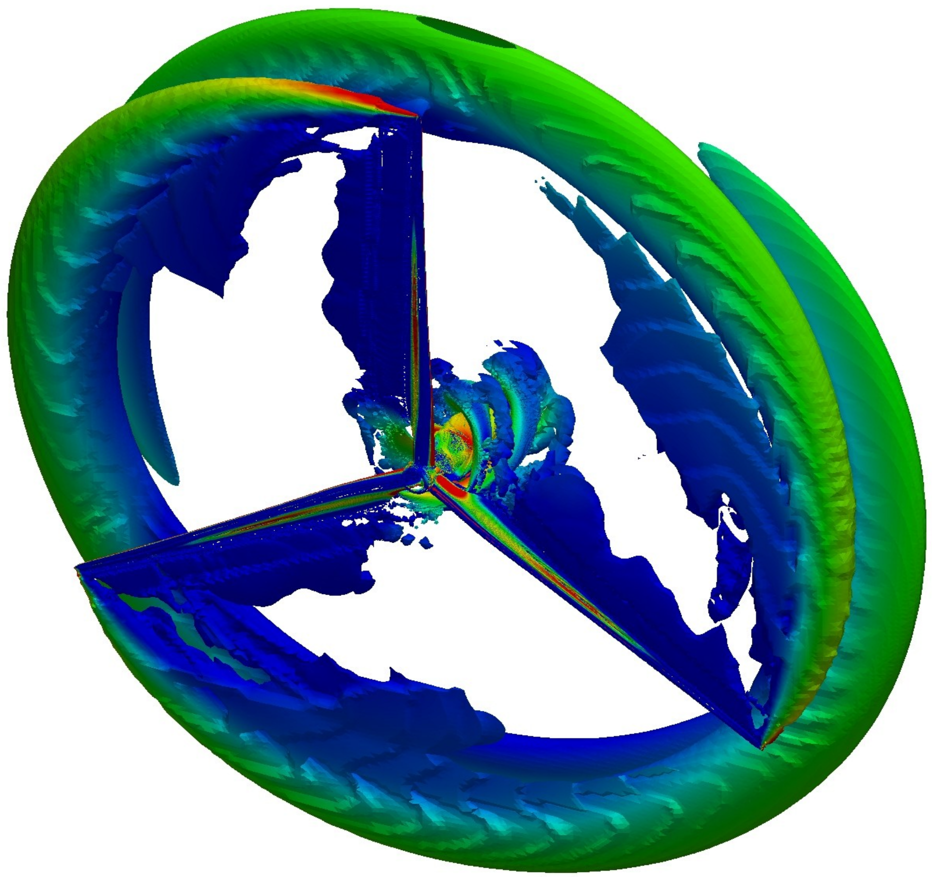
Airfoil Simulations

NACA-63-415
Lift and Drag Coefficient





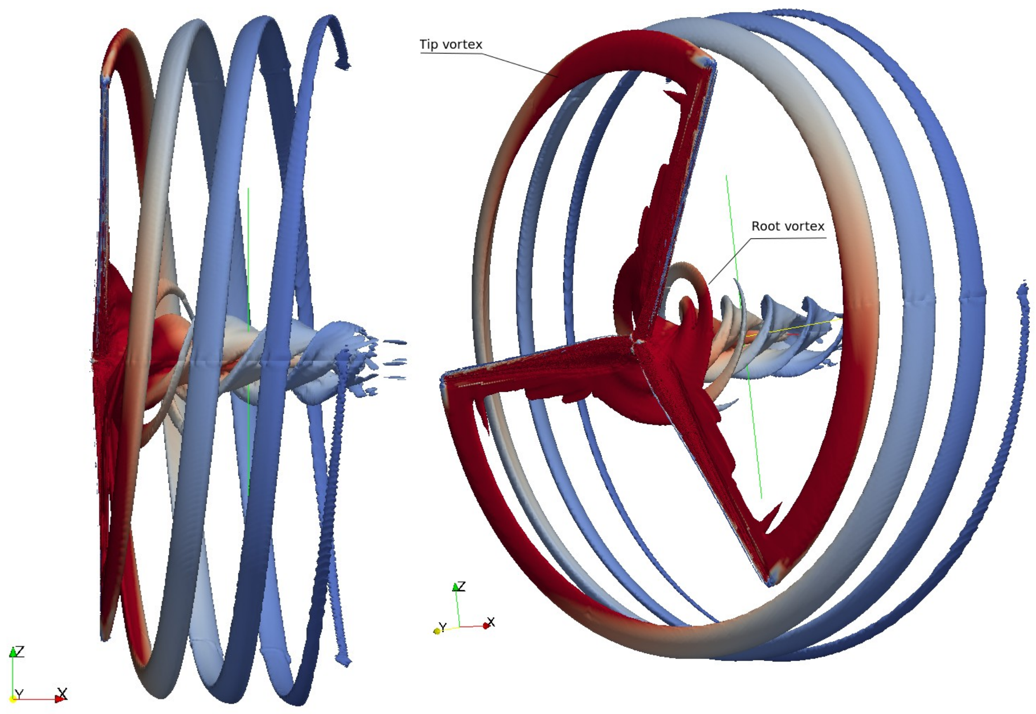
Wind Turbine Simulations



Ref: Schramm.M ,fraunhofer IWES, Oldenburg, Germany

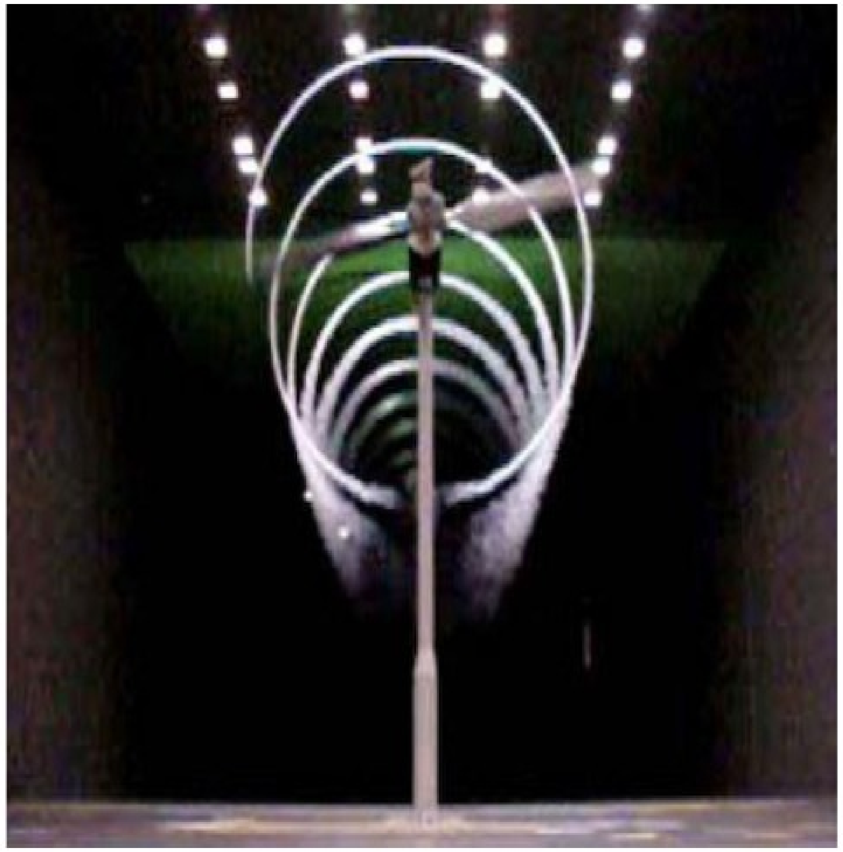
Motivation

- Get knowledge about full rotor aerodynamics
- Investigation of flow pattern
- Rotor & tower interaction
- Get knowledge about 3D effects
- Wake investigation

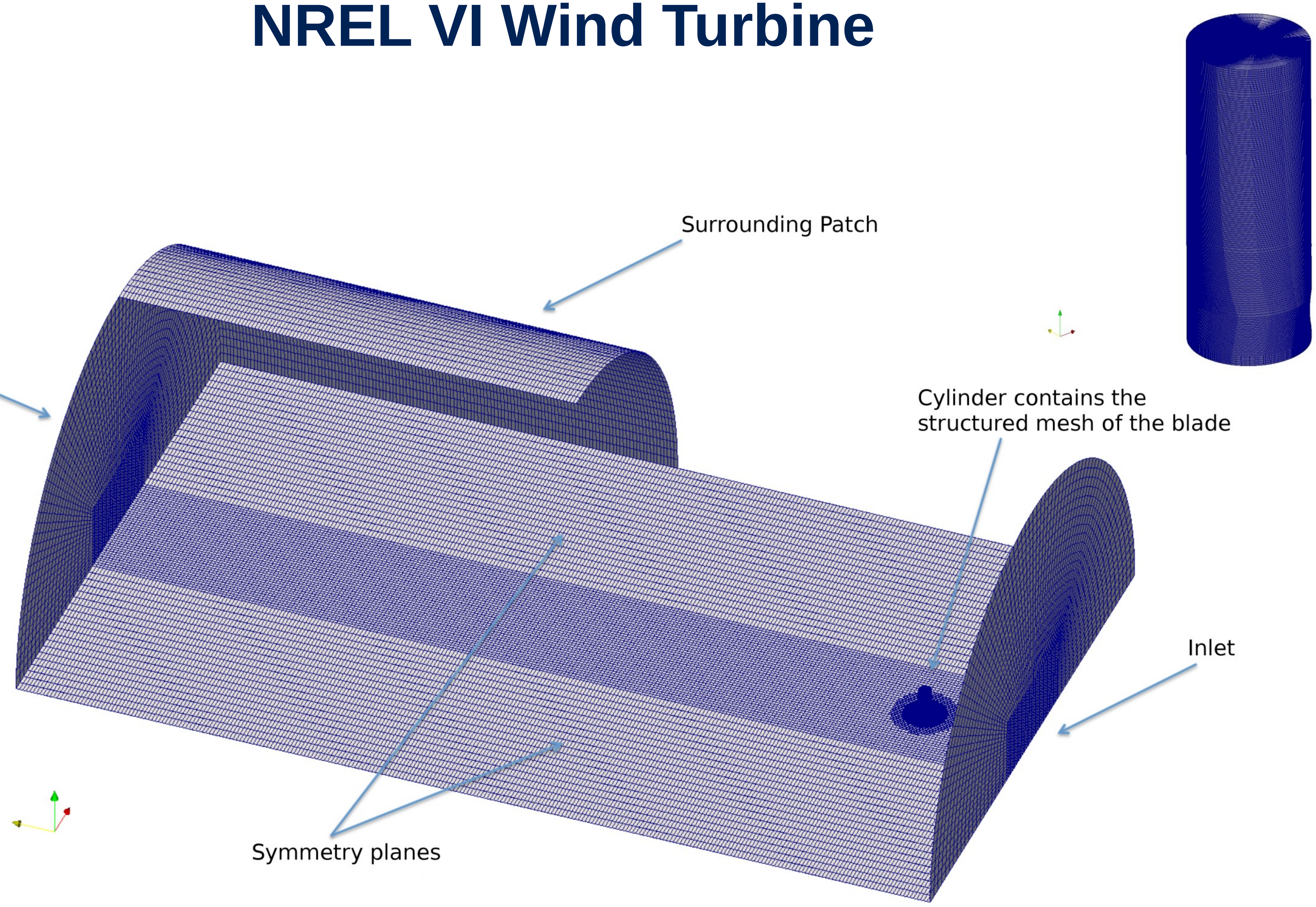


CFD Simulation: NREL VI Wind Turbine

- 10 m rotor diameter
- Measurements in NASA wind tunnel
- Pressure, load as experimental data available



NREL VI Wind Turbine



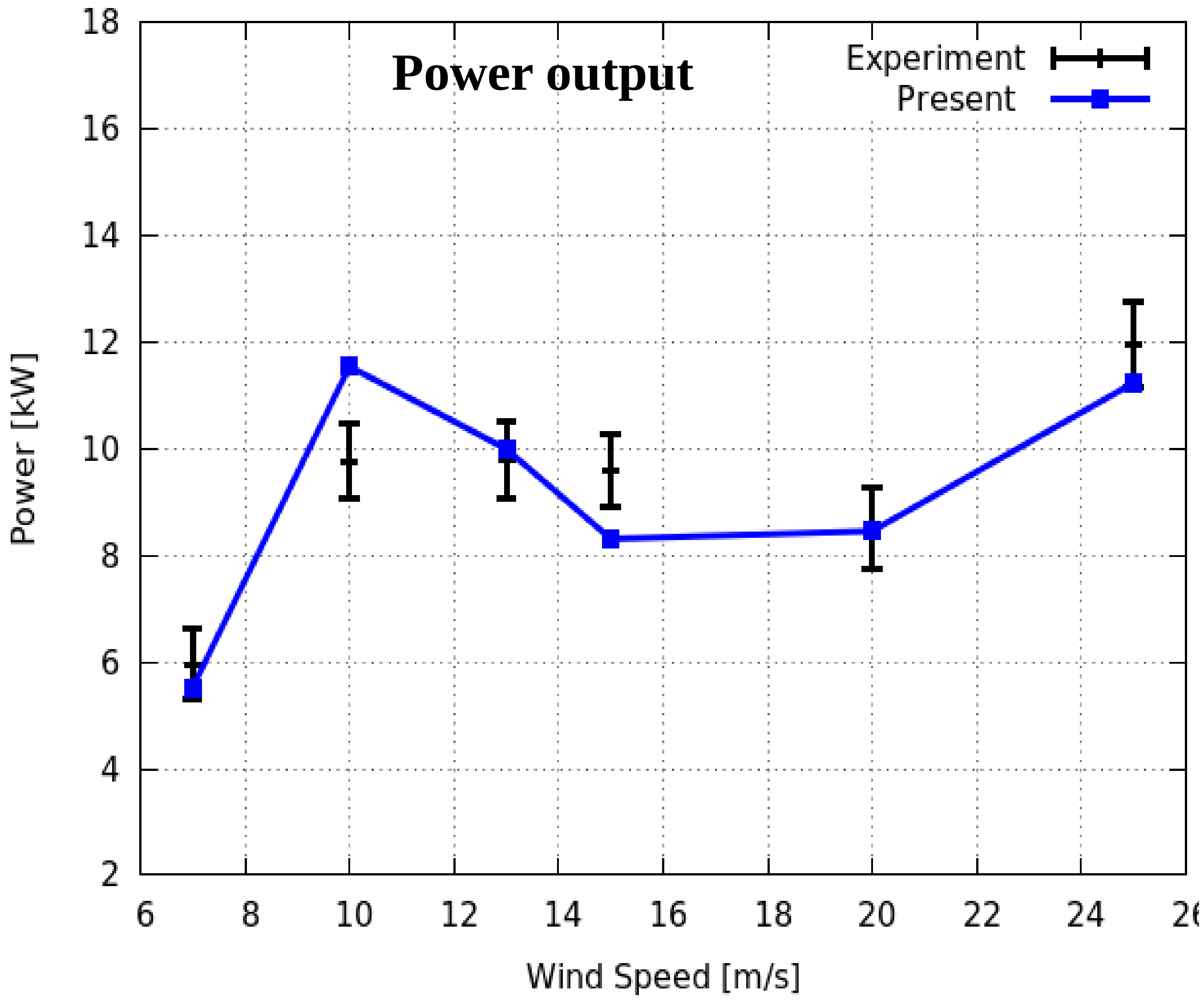
Ref: Rahimi.H ForWind Center for Wind Energy Research, Institute of Physics, University of Oldenburg, Germany

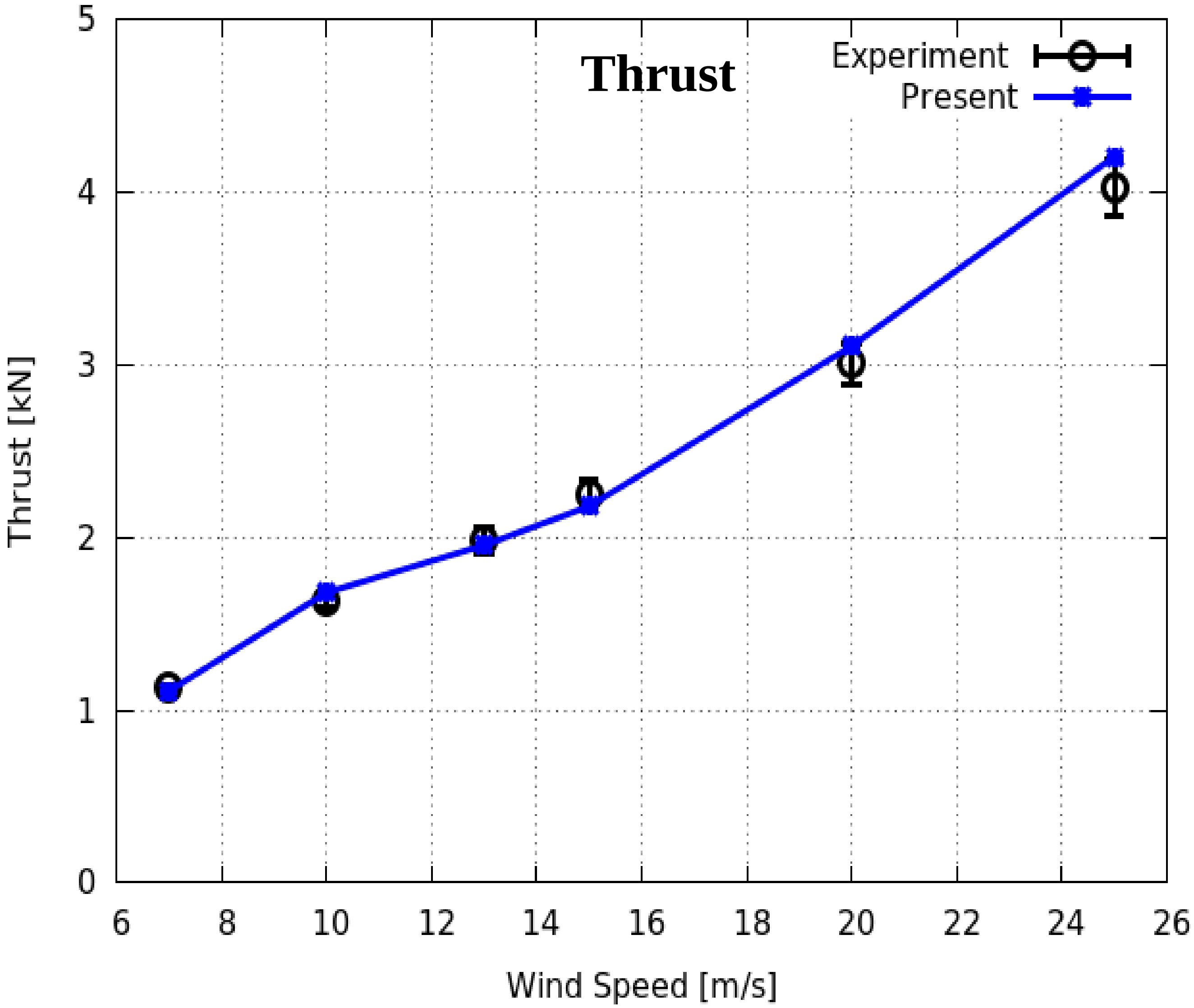
NREL VI Wind Turbine

- Simulation conducted on the FLOW * cluster, with 92 CPU cores
- Steady-State simulation
- Total grid size: 7 Million
- k-ω SST turbulence model
- Convergence achieved within 5 hours CPU time

* The Facility for Large-Scale cOmputations in Wind energy research (FLOW)

NREL VI Wind Turbine

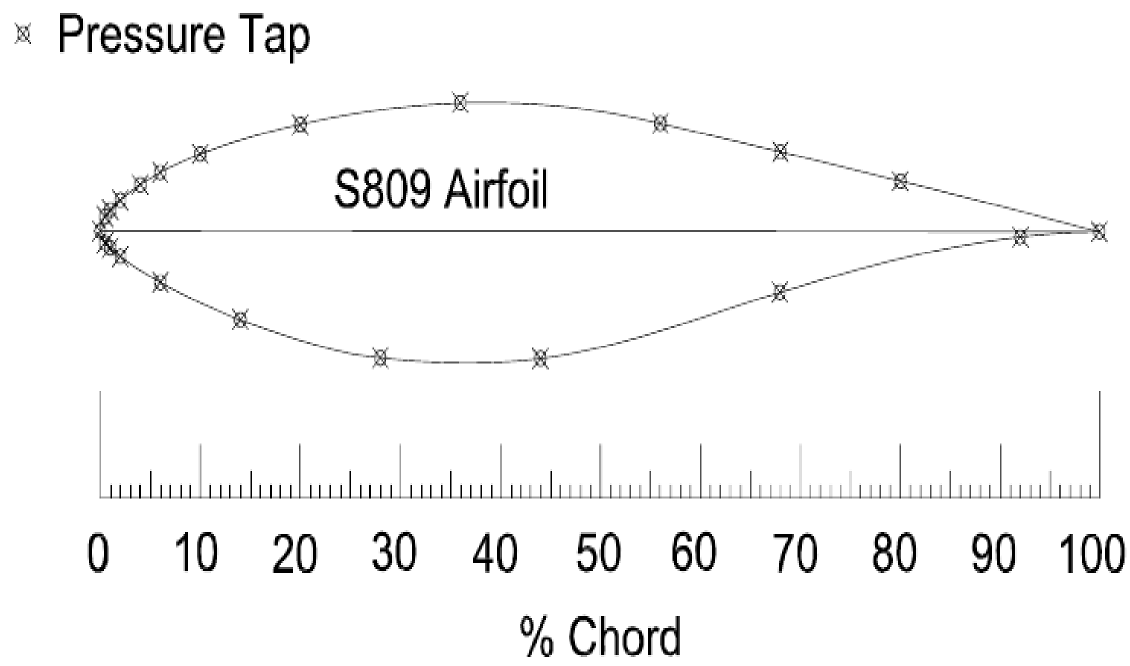


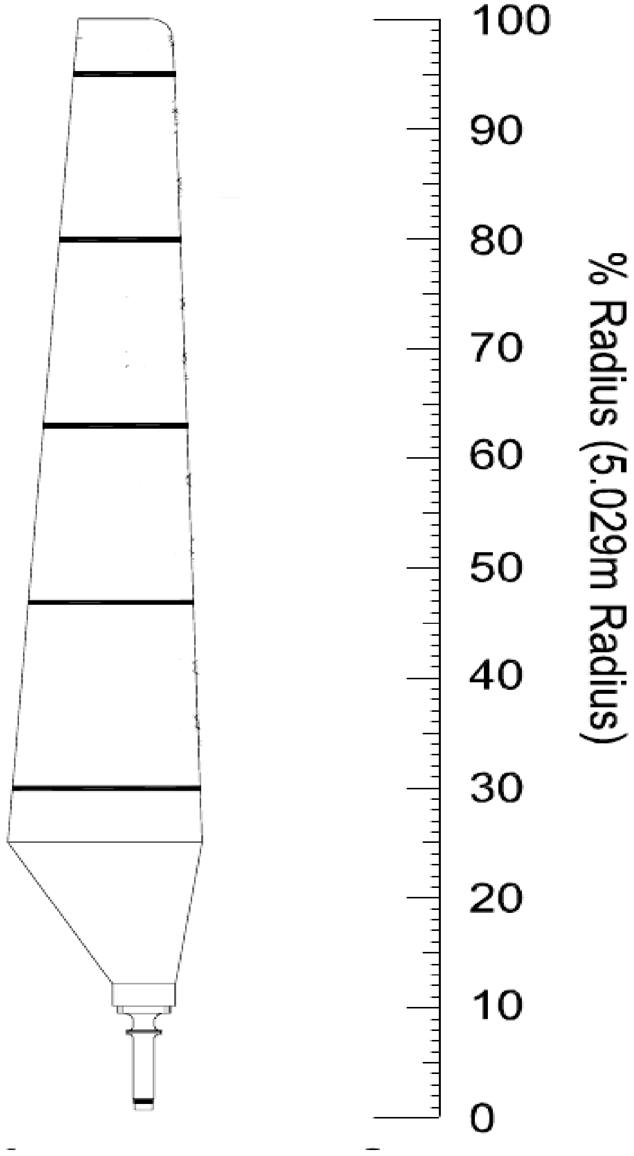


Ref: Rahimi.H ForWind Center for Wind Energy Research, Institute of Physics, University of Oldenburg, Germany

NREL VI Wind Turbine

Pressure distribution



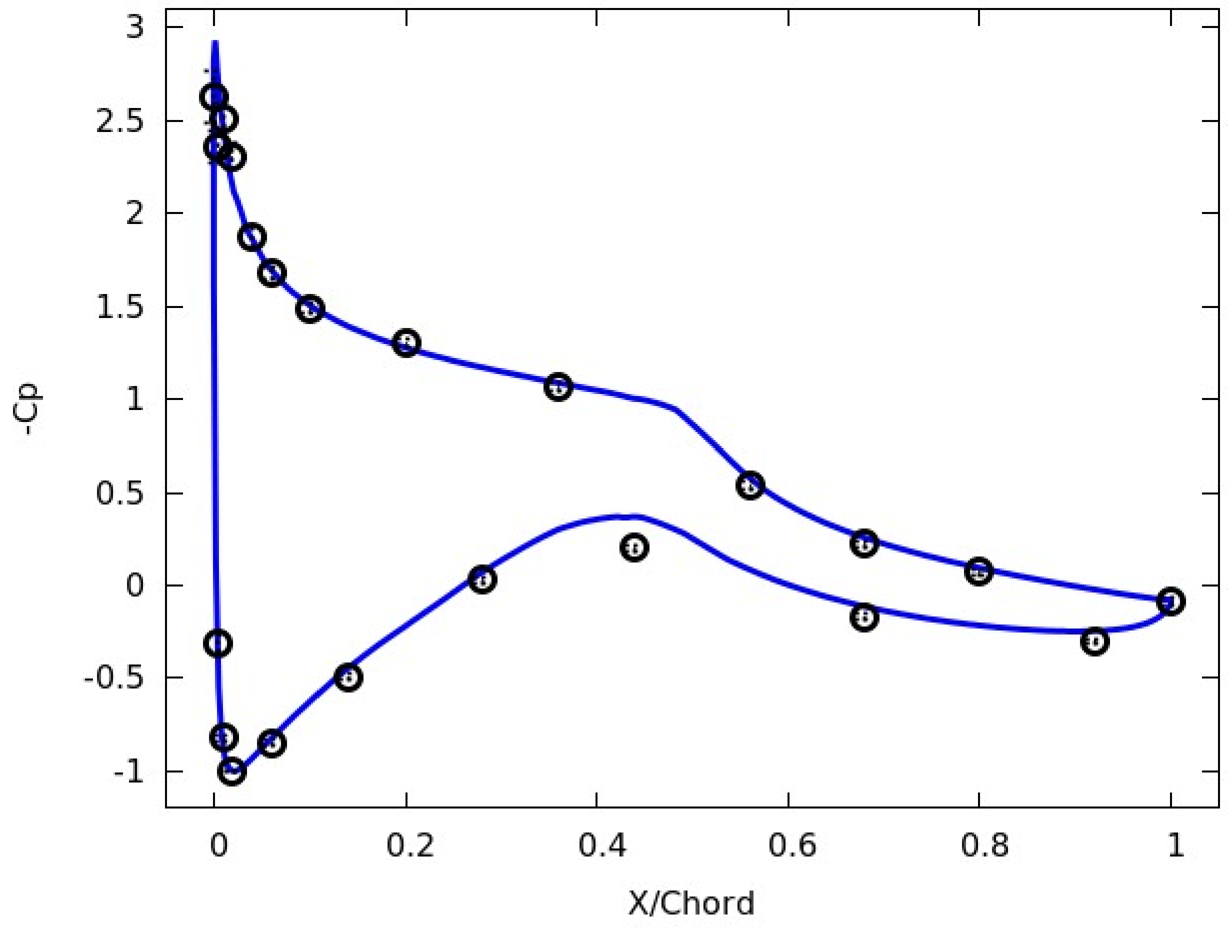


Ref: http://www.nrel.gov/wind/

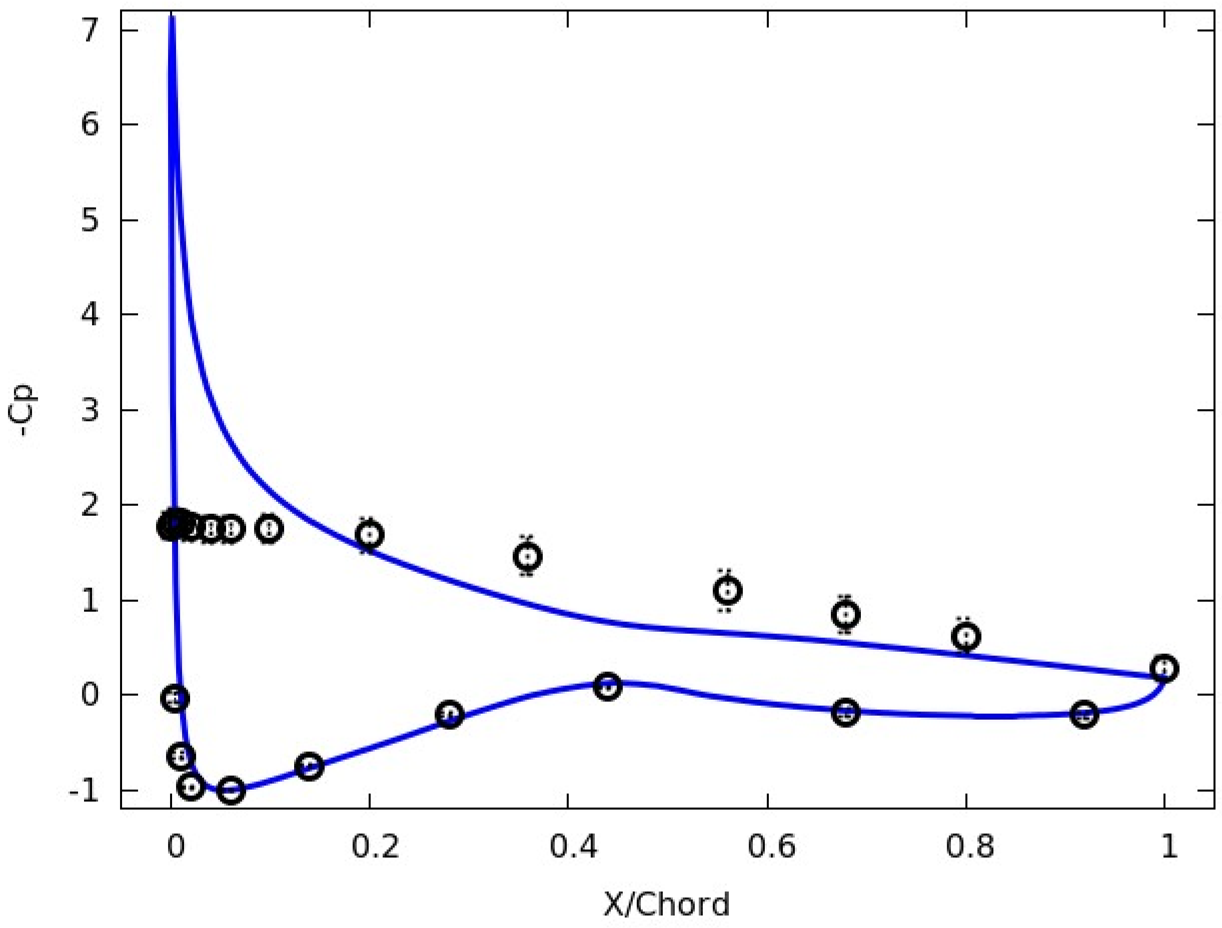
NREL VI Wind Turbine

Pressure distribution

7 m/s

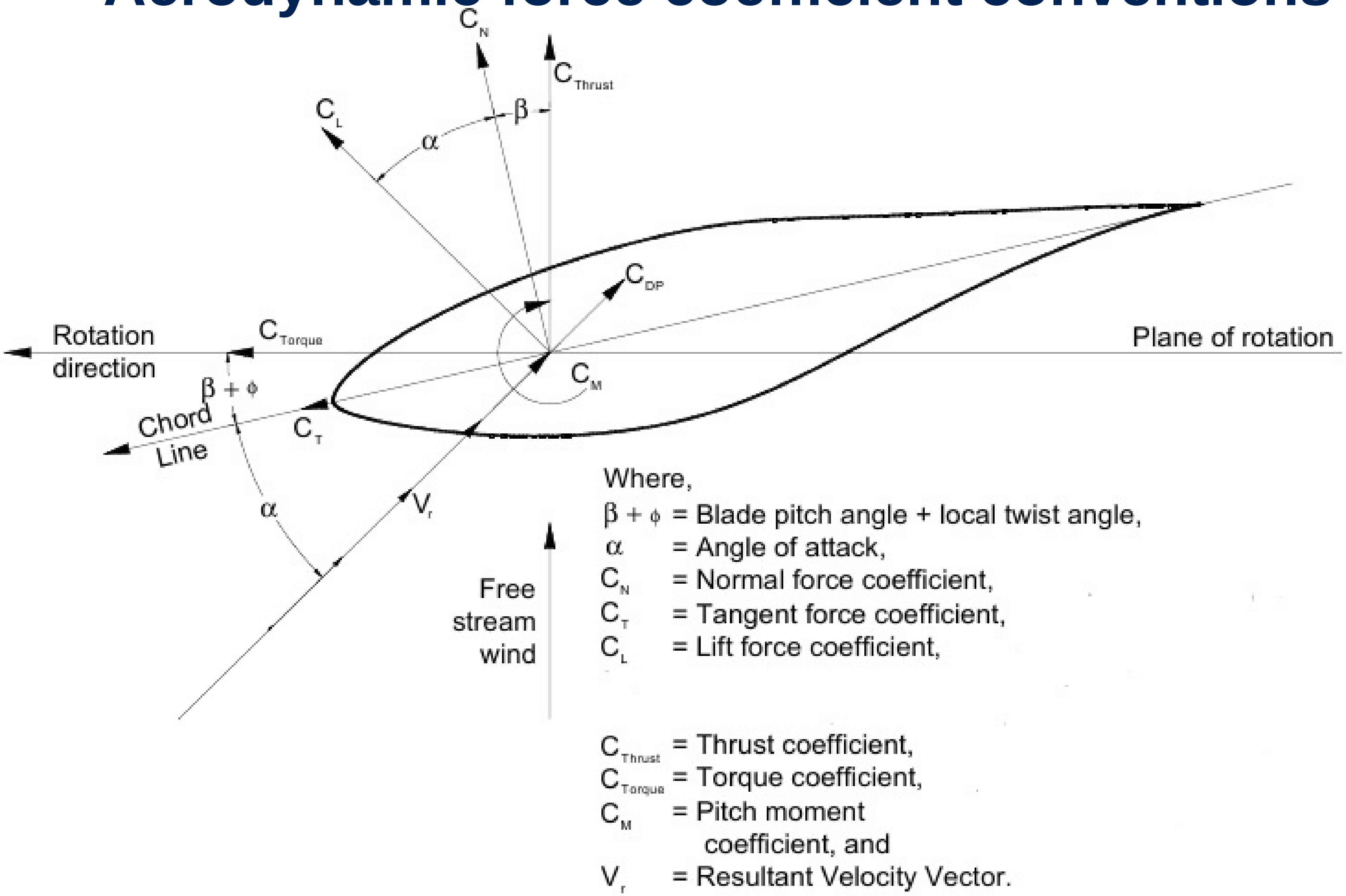


10 m/s



Ref: Rahimi.H ForWind Center for Wind Energy Research, Institute of Physics, University of Oldenburg, Germany

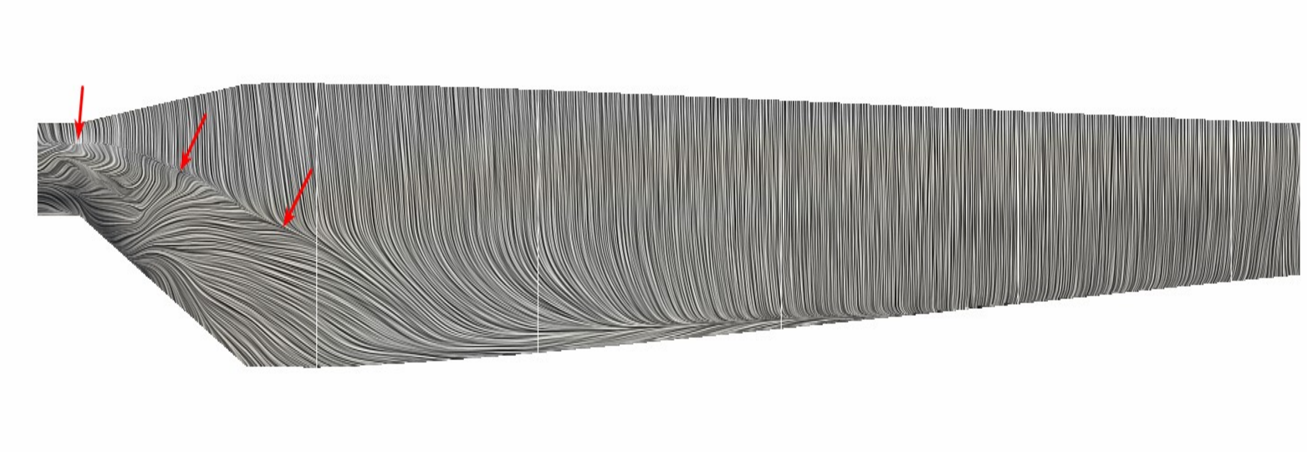
Aerodynamic force coefficient conventions

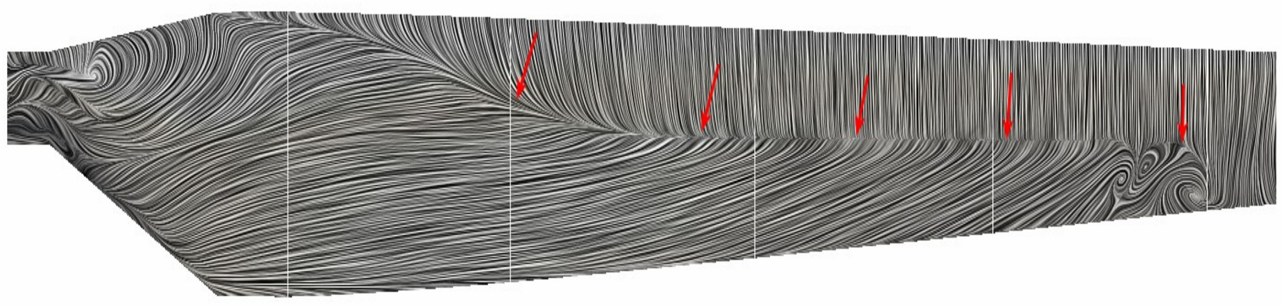


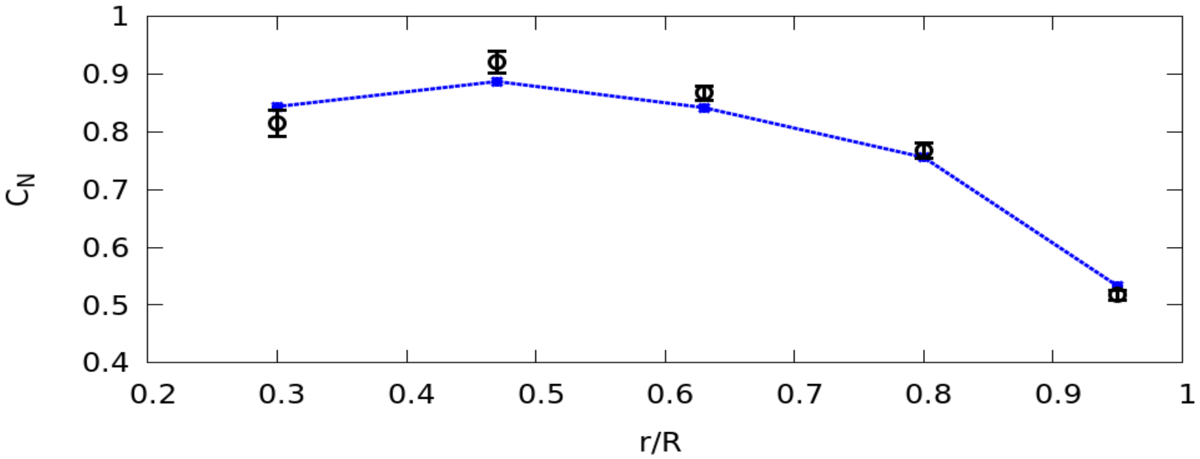
Ref: http://www.nrel.gov/wind/

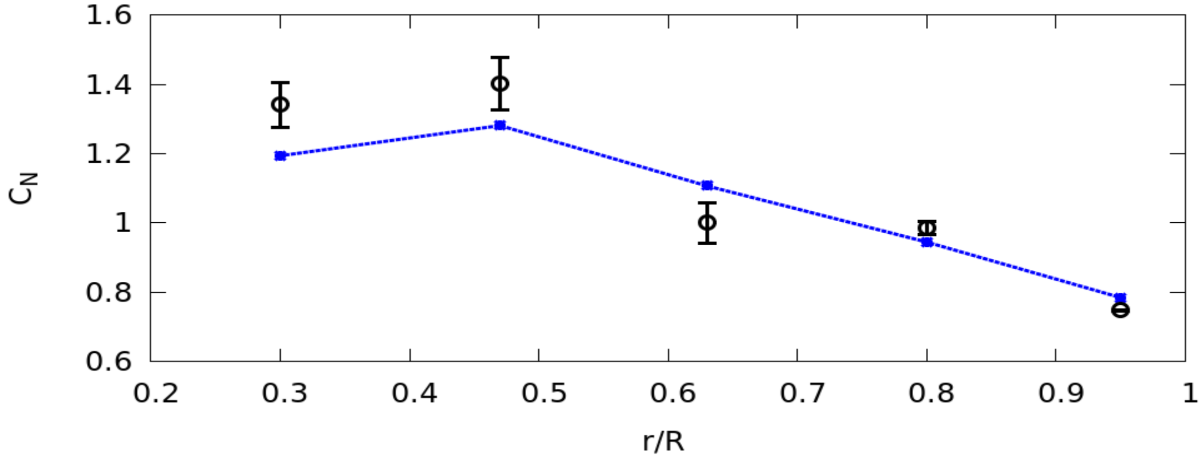
7 m/s

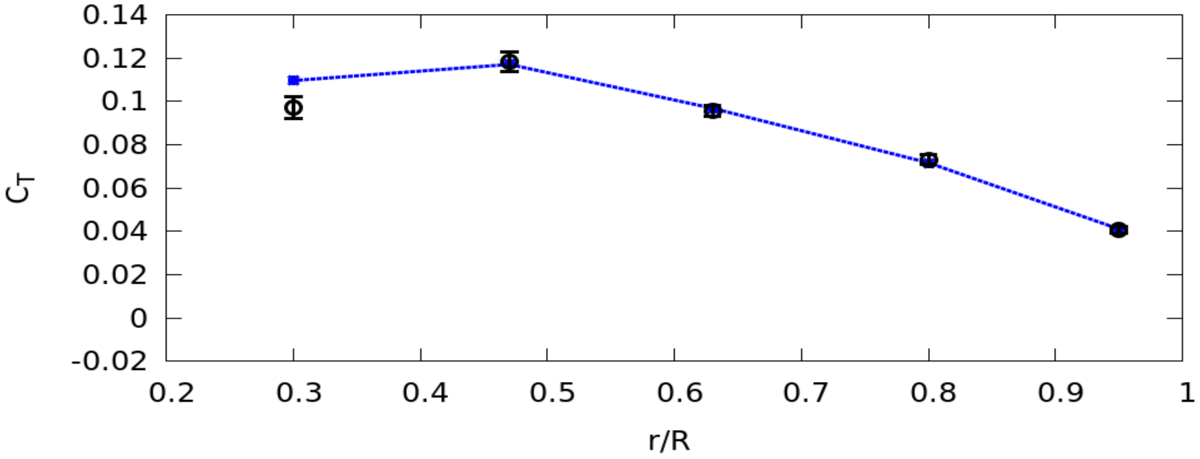
10 m/s

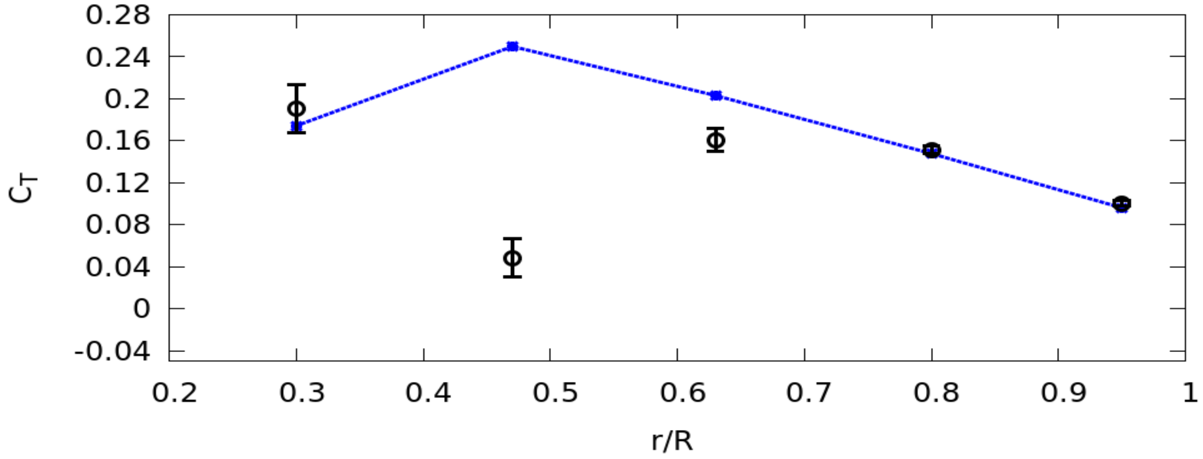








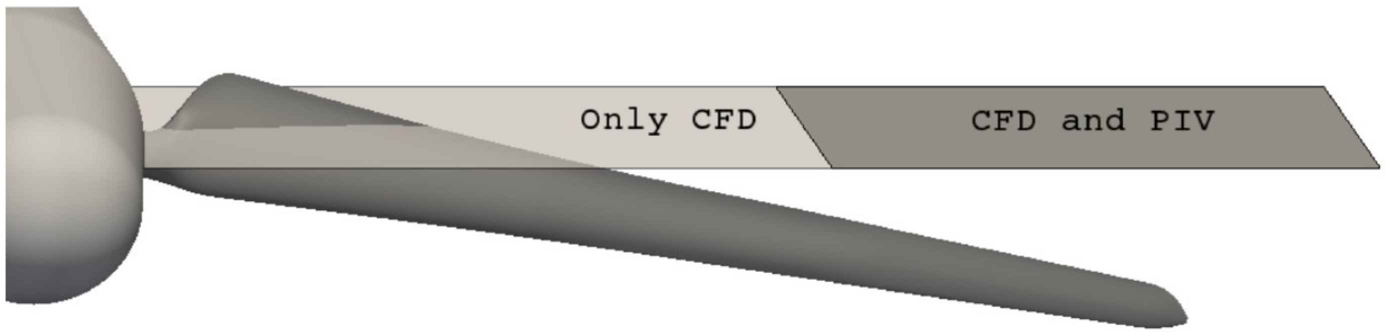


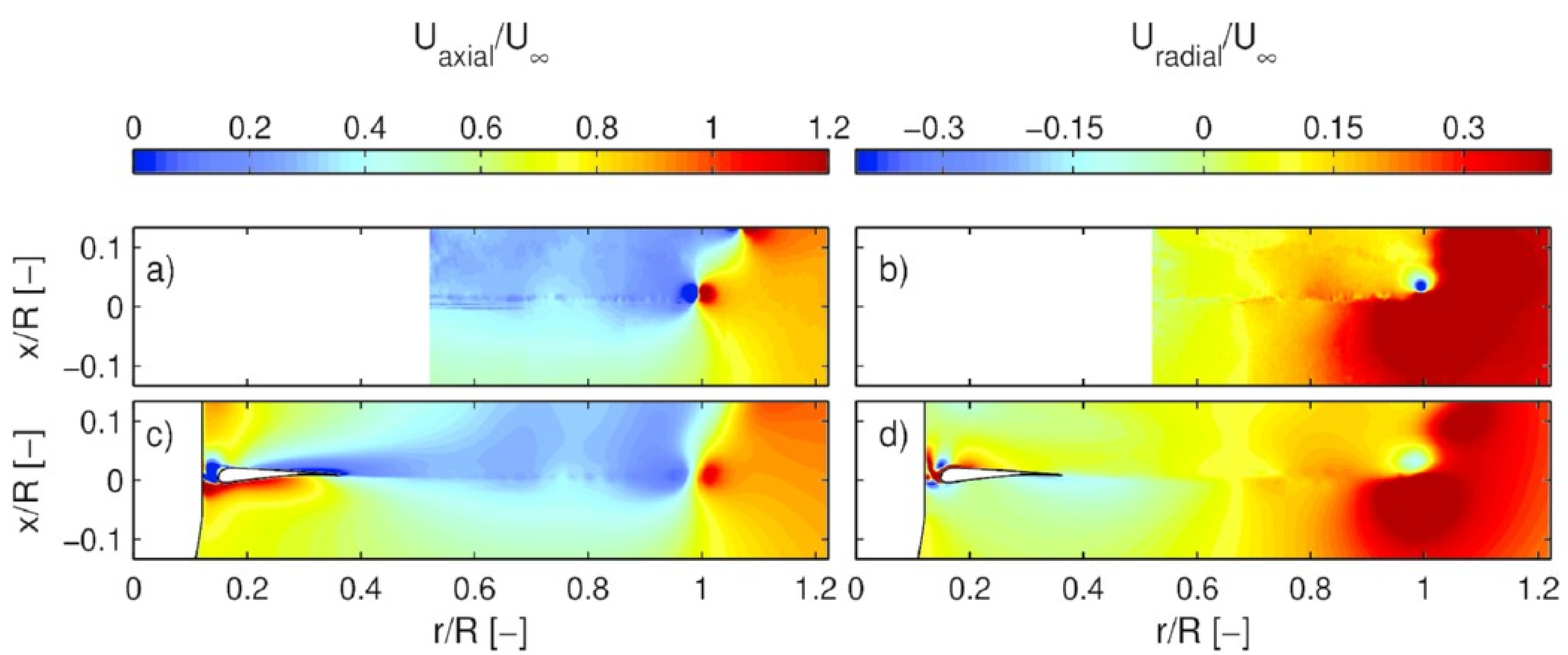


CFD Simulation: Mexico Wind Turbine

- 4.5 m rotor diameter
- Measurements in 9x9 m² open section wind tunnel
- Pressure, load and PIV experimental data available
- Considered cases: axial inflow with 10, 15, 19, 24 and 30 m/s

Mexico Wind Turbine





U=10 m/s
Linear range

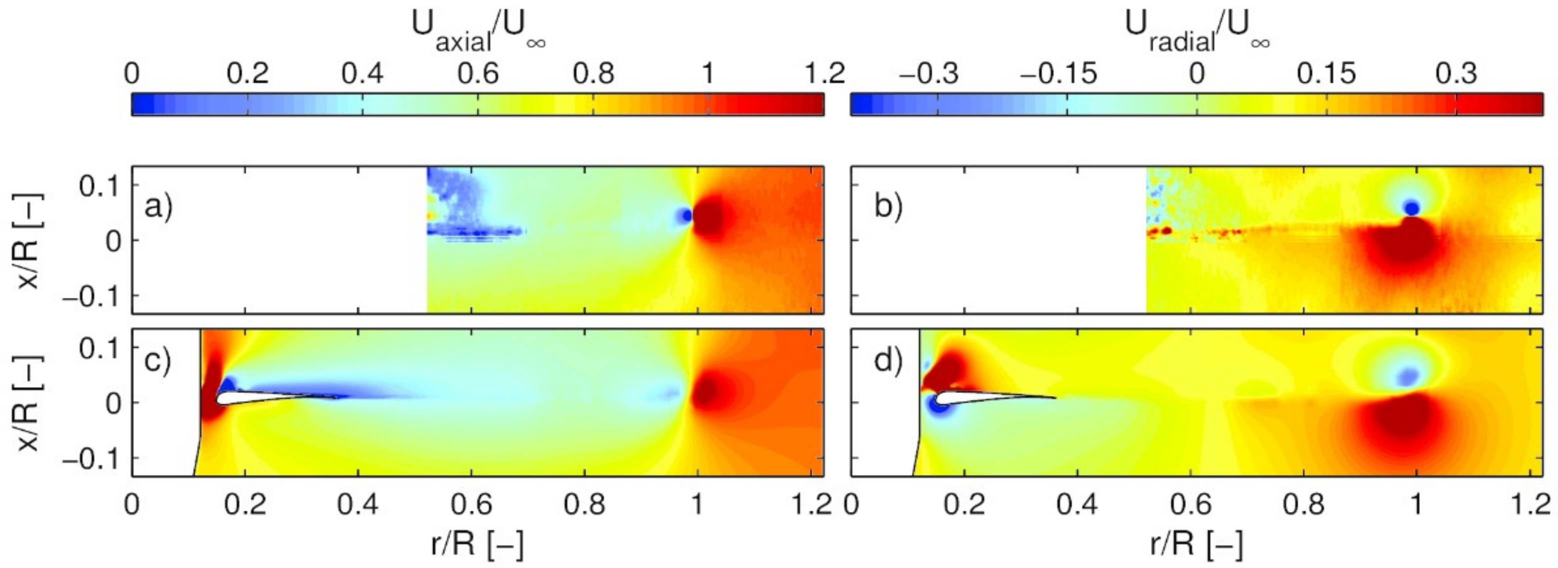
Ref: Herráez, I.; Stoevesandt, B.; Peinke, J. Insight into Rotational Effects on a Wind Turbine Blade Using Navier–Stokes Computations. Energies 2014, 7, 6798-6822.

ForWind Center for Wind Energy Research Bremen Hannover Oldenburg

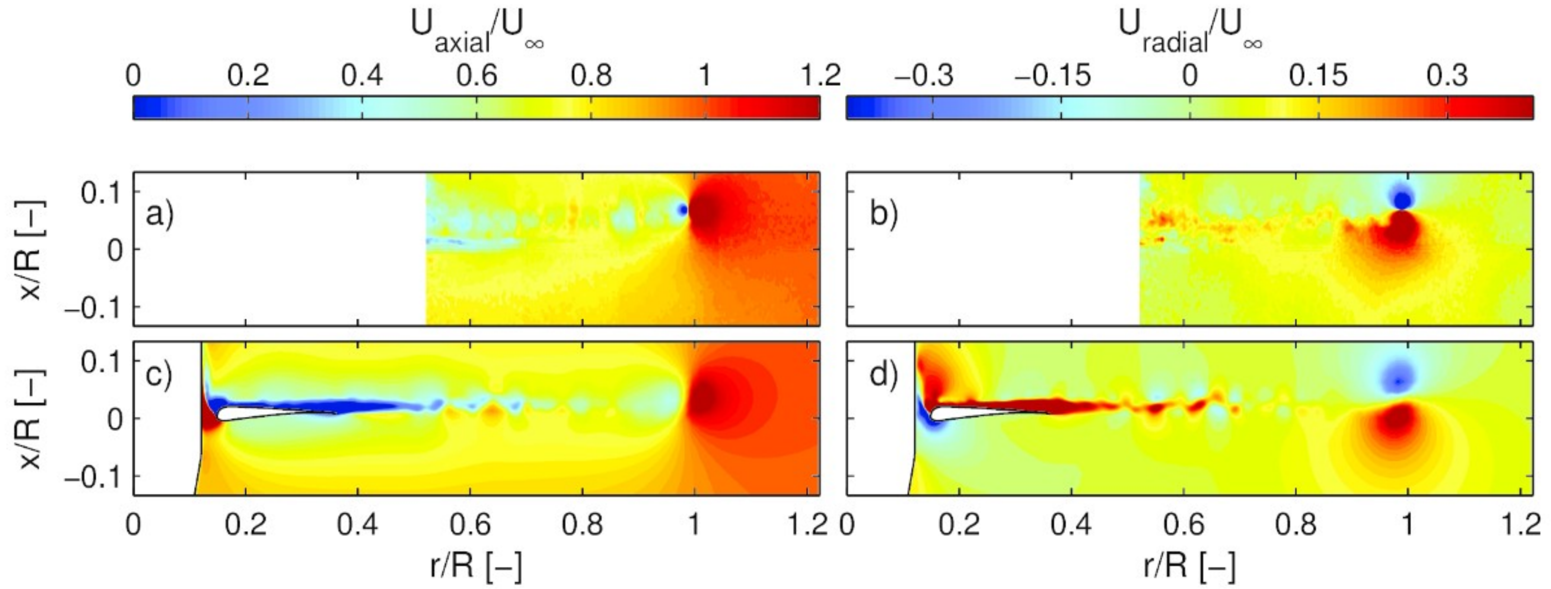
TWiSt Turbulence ▸ Wind Energy ▸ Stochastics

Mexico Wind Turbine

U=15 m/s
Design conditions



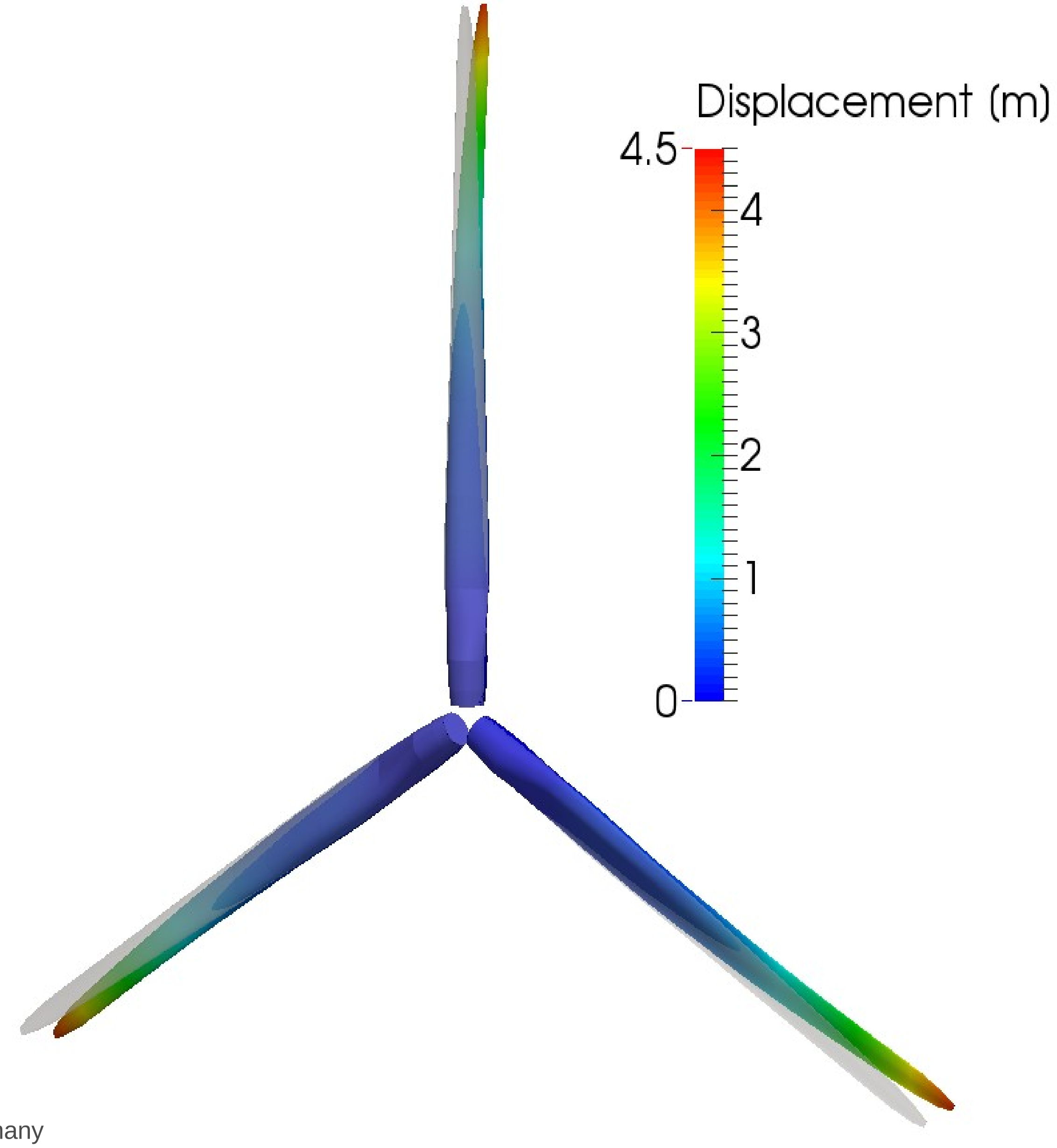
U=24 m/s
Stall conditions



Ref: Herráez, I.; Stoevesandt, B.; Peinke, J. Insight into Rotational Effects on a Wind Turbine Blade Using Navier–Stokes Computations. Energies 2014, 7, 6798-6822.

Fluid-Structure Coupling

- Large, flexible rotor blades deform
- Conventional CFD assumes stiff blades
- CFD+FEA improves accuracy
- (FEA = Finite Element Analysis)



Ref: Dose.B ForWind Center for Wind Energy Research, Institute of Physics, University of Oldenburg, Germany

Summary

- OpenFOAM is a very useful tool for simulating wind energy applications (2D, 3D, steady state, transient)
- Open source concept
- Shows good results for wind energy applications
- Join the community!

Thank you for your attention

hamid.rahimi@uni-oldenburg.de

Smart Blades Project

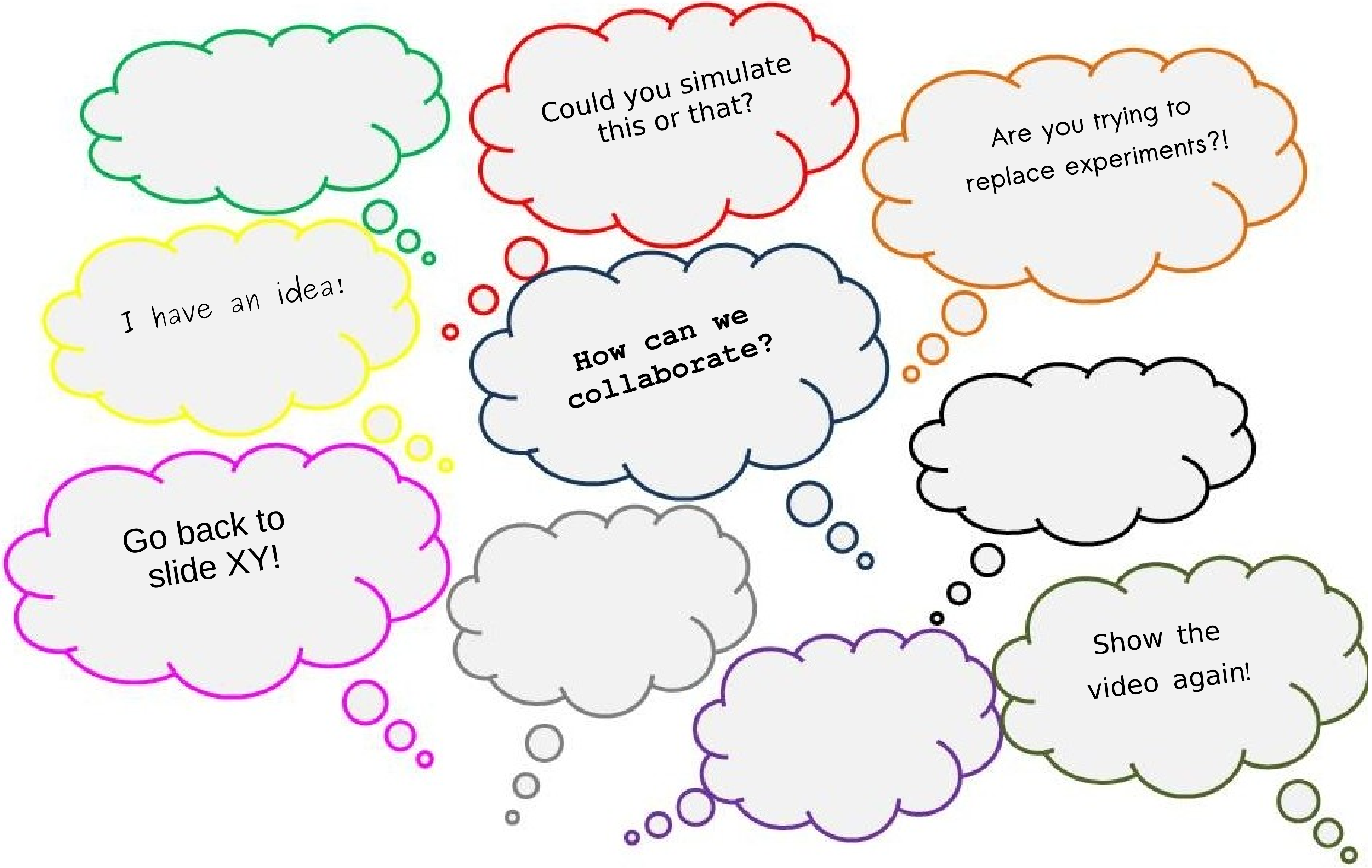
- Development and Design of Intelligent Rotor Blades -

A joint research project of DLR, ForWind and Fraunhofer - IWES

Federal Ministry for the Environment, Nature Conservation and Nuclear Safety

* The Facility for Large-Scale cOmputations in Wind energy research (FLOW)

Discussion


Could you simulate this or that?
Are you trying to replace experiments?!
I have an idea!
How can we collaborate?
Go back to slide XY!
Show the video again!