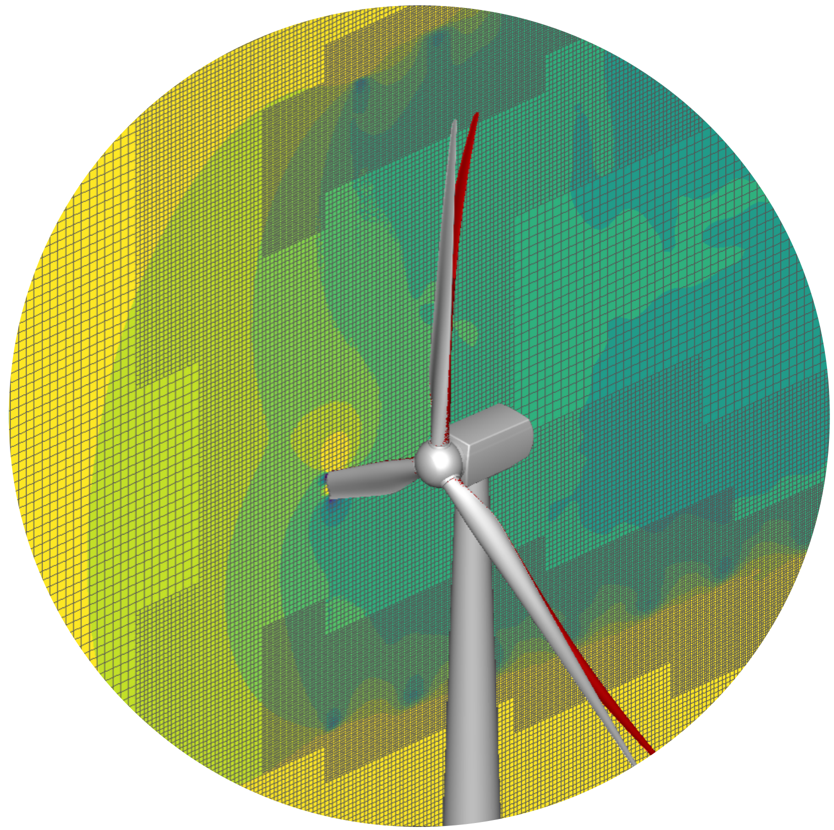
CFD for Wind Turbine Engineering Problems



Thorsten Lutz

G. Bangga, G. Guma, L. Klein, P. Letzgus,
C. Schulz, F. Seel, P. Weihing, A. Wolf, P. Zamre
and further former PhD students

lutz@iag.uni-stuttgart.de

WindForS
Windenergie Forschungscluster
Wind Energy Research Cluster

DAAD Deutscher Akademischer Austausch Dienst German Academic Exchange Service

KIT Karlsruhe Institute of Technology

UNIVERSITY OF PATRAS

Overview

Introduction to CFD

- CFD basics & mesh generation
- BEM vs. CFD
- Advanced CFD chain

Exemplary CFD Applications

- Noise prediction & reduction
- Dynamic stall & vortex generators
- Rotor & wake flow physics
- Aeroelasticity & low frequency noise
- Complex & urban terrain

Introduction to CFD

Introduction to CFD Motivation

- Why do we need CFD simulations in Wind Turbine Design?
  - Loads and power output are influenced by
    - 3D viscous effects, aeroelasticity
    - Blade-tower interaction, wake impact, unsteady aerodynamic effects
    - atmospheric boundary layer interaction, complex topography
  - Provides detailed information of the 3D flow field which can be used for an aerodynamic and aeroacoustic evaluation
    - In CFD calculation fewer simplifying assumptions are needed compared to traditional engineering models like BEM

⇨ **More accurate in prediction of turbine behavior**

Introduction to CFD The Navier-Stokes-Equations

$$\frac{\partial}{\partial t} \oiiint_V \underline{W}\, dV + \oiint_S (\underline{F}_c + \underline{F}_v)\, dS = \oiiint_V \underline{Q}\, dV$$

- This basic equation can be derived with the assumption of a finite control volume
  - In this control volume the change with respect to time of a flow variable φ is the sum of:
    - Change of φ by Convection into the control volume
    - Change of φ by Diffusion into the control volume
    - Production of φ within the control volume
- The Navier-Stokes Equation is the fundamental equation for viscous flow analysis

Introduction to CFD The Navier-Stokes-Equations

- There are different methods to solve the Navier-Stokes Equation and to simulate a (turbulent) flow around a rigid body
  - **DNS** **D**irect **N**umerical **S**imulation
    - The Navier Stokes Equation is solved directly
    - A large amount of grid points is needed
    - Flow simulations with DNS are only possible for low Reynolds Numbers

Introduction to CFD The Navier-Stokes-Equations

- **LES** **L**arge **E**ddy **S**imulation
  - Only large turbulence scales are resolved, therefore the range of length scales is limited (→ low pass filtering)
  - Small scales are described by a subgrid scale model
  - Less computational time is needed according to DNS

- **(U)RANS** (**U**nsteady) **R**eynolds-**A**veraged **N**avier-**S**tokes Equations
  - State of the art for complex turbulent flow simulations
  - A flow quantity is decomposed into a time-averaged and a fluctuating quantity
  - With the averaging, models for the description of turbulence are needed (closure problem)

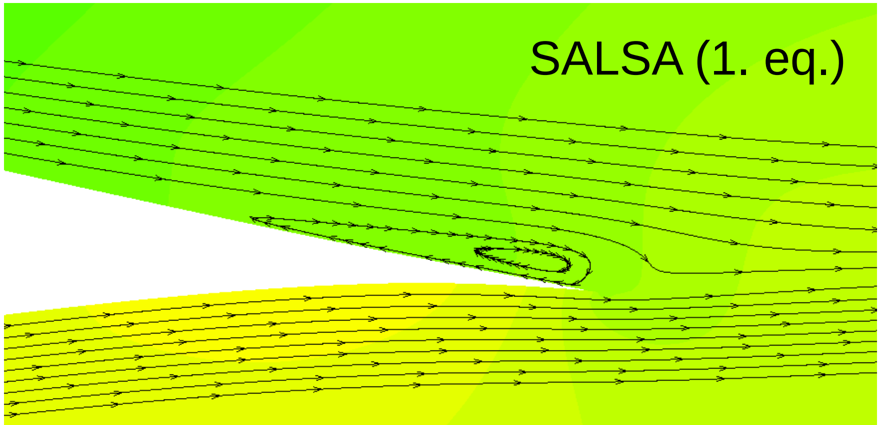
Introduction to CFD Turbulence Modelling in RANS

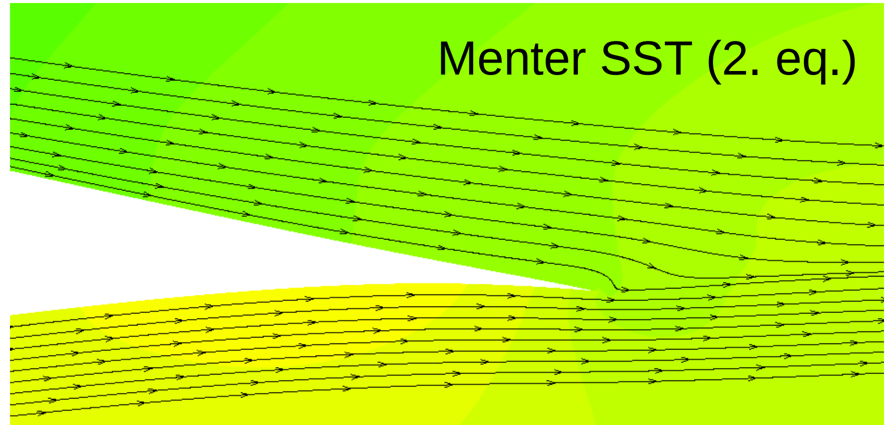
- Boussinesq introduced the concept of the eddy viscosity ν_t
  - Turbulent Stresses are related to the mean flow

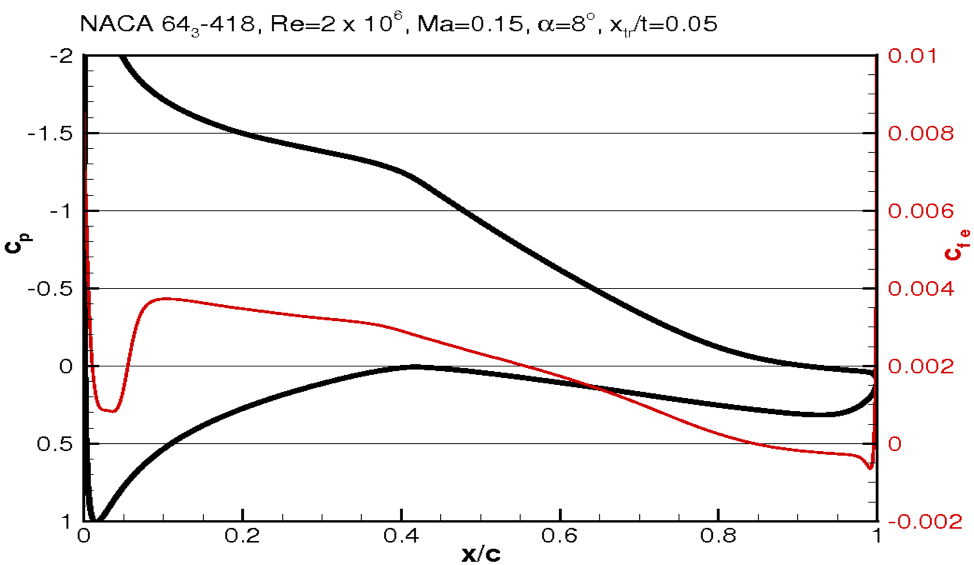
$$\tau_{ij}^{F} = -\rho \overline{u_i^{'} u_j^{'}} = 2\rho \nu_t \overline{S_{ij}} - \frac{2}{3} \rho k \delta_{ij}$$

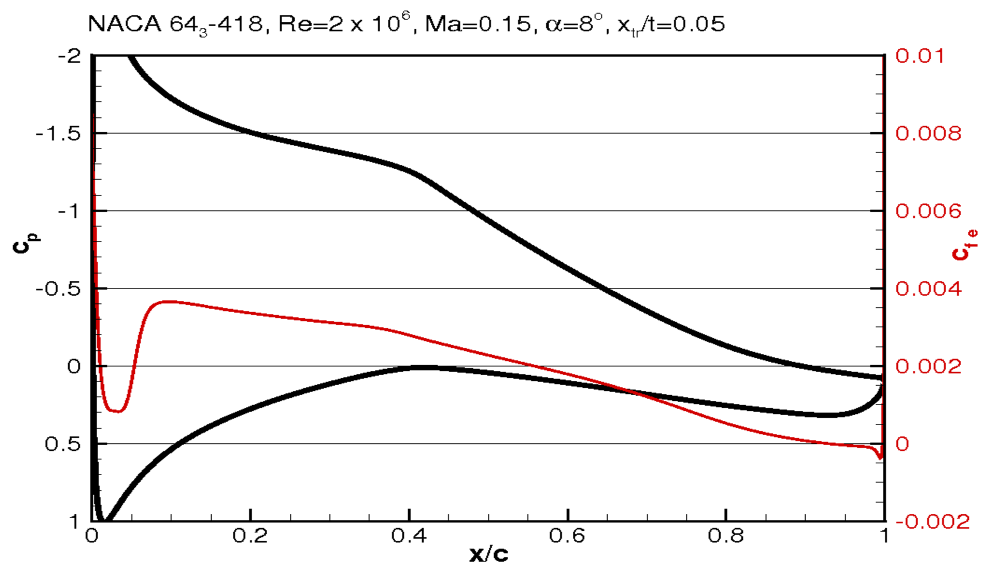
  - Many turbulence models are based on this assumption
    - Zero-Equation Models: Calculation of eddy viscosity by an algebraic equation
    - One-Equation Models: Solving one transport equation for the viscosity quantity
    - Two-Equation Models: Solving two transport equations e.g. for the turbulent kinetic energy **k** and the turbulent dissipation **ε**

Introduction to CFD Turbulence Modelling in RANS



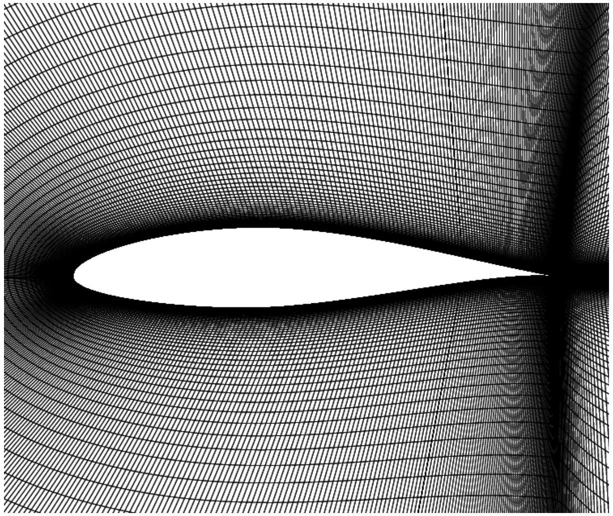






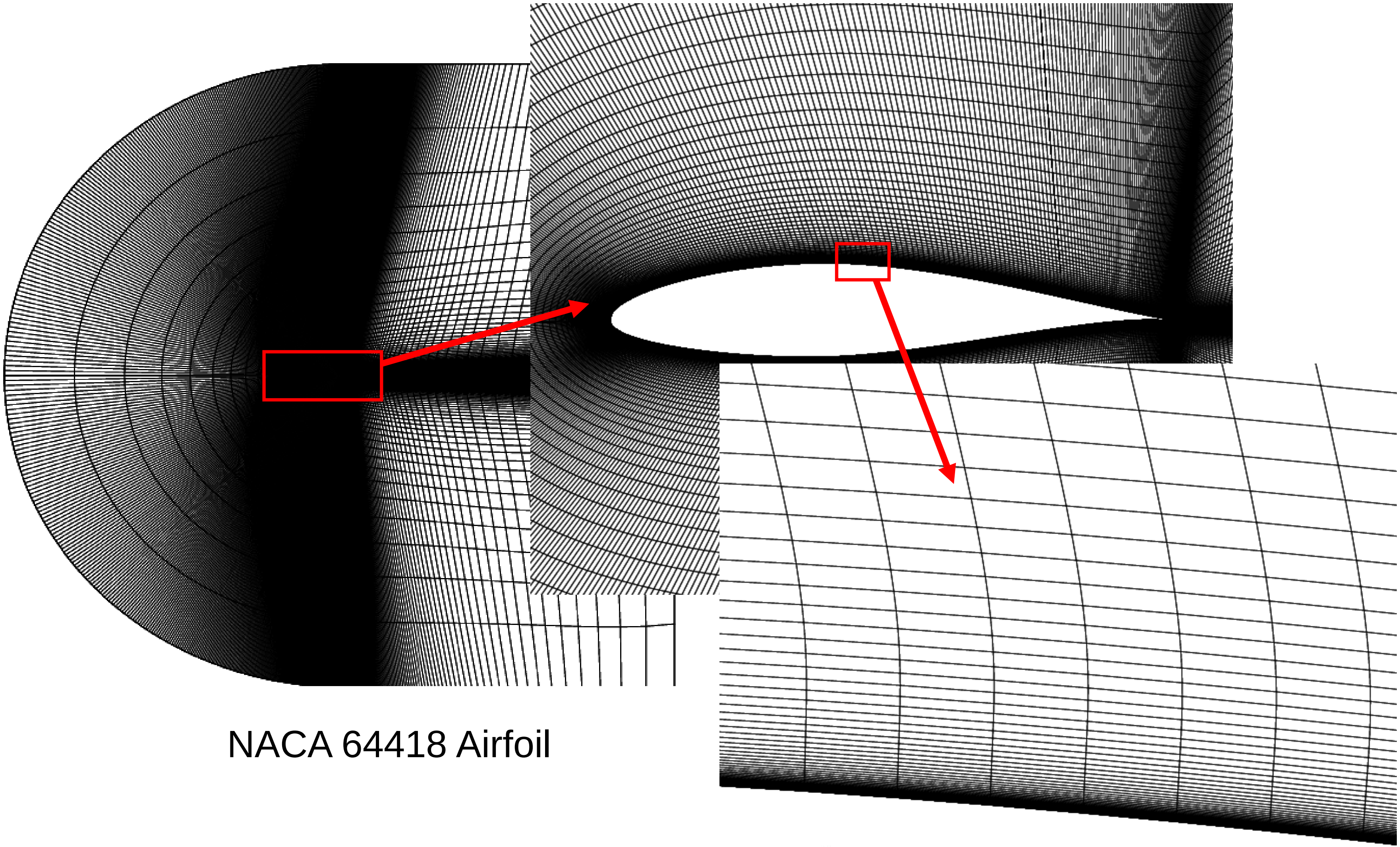
Th. Lutz, USTUTT

Introduction to CFD Grid Generation



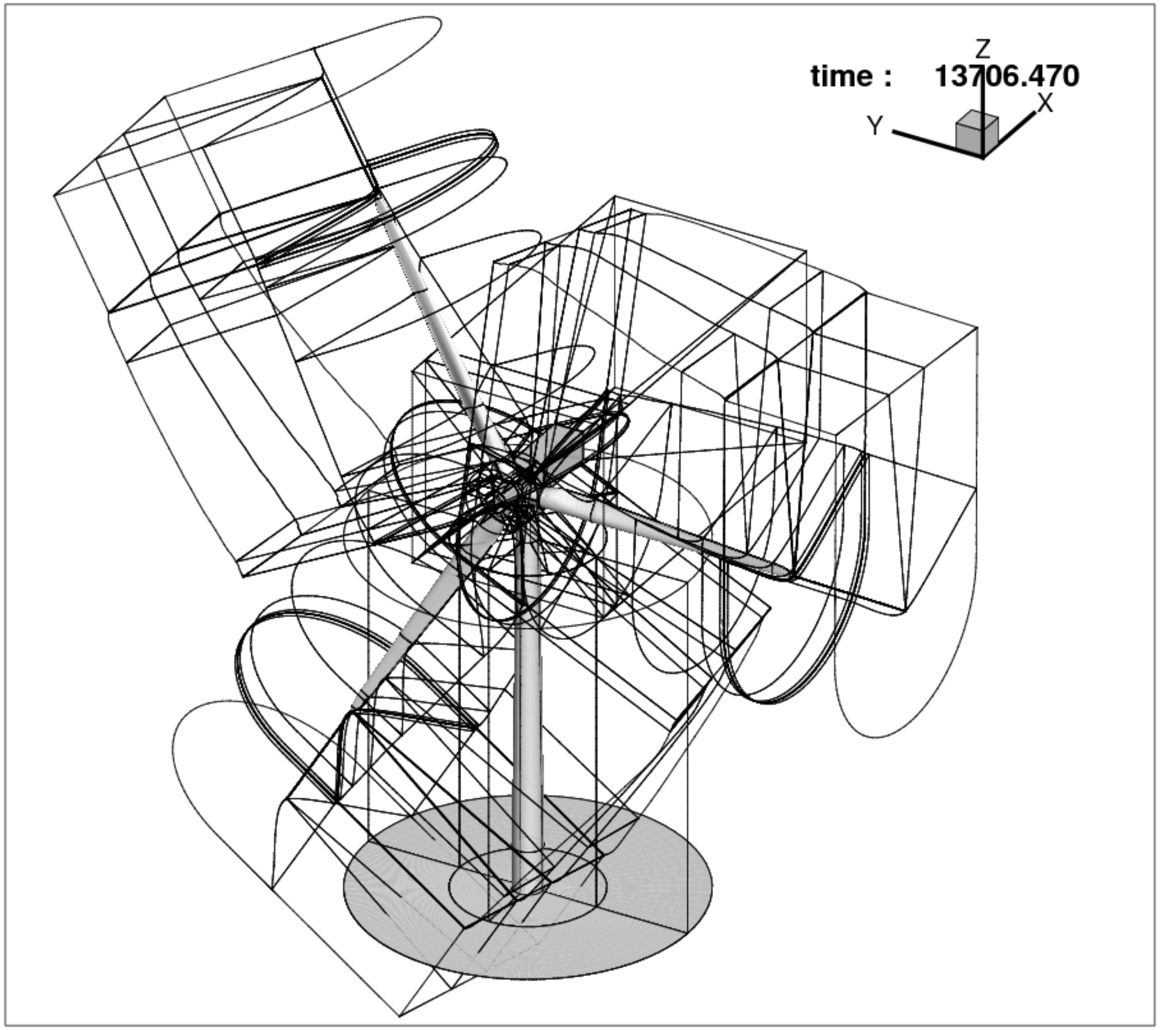
- The Navier-Stokes Equations have to be solved numerically
  - Different discretisation techniques are used
    - Finite difference
    - Finite element
    - **Finite volume (most common)**
    - Lattice Boltzmann Methods
  - Meshes are build around the geometry and fill the flow domain
    - There should be a significant distance between the outer boundaries of the flow and the geometry to reduce errors from reflection at the outer boundaries

Introduction to CFD Grid Generation - Structured Grids 2D



NACA 64418 Airfoil

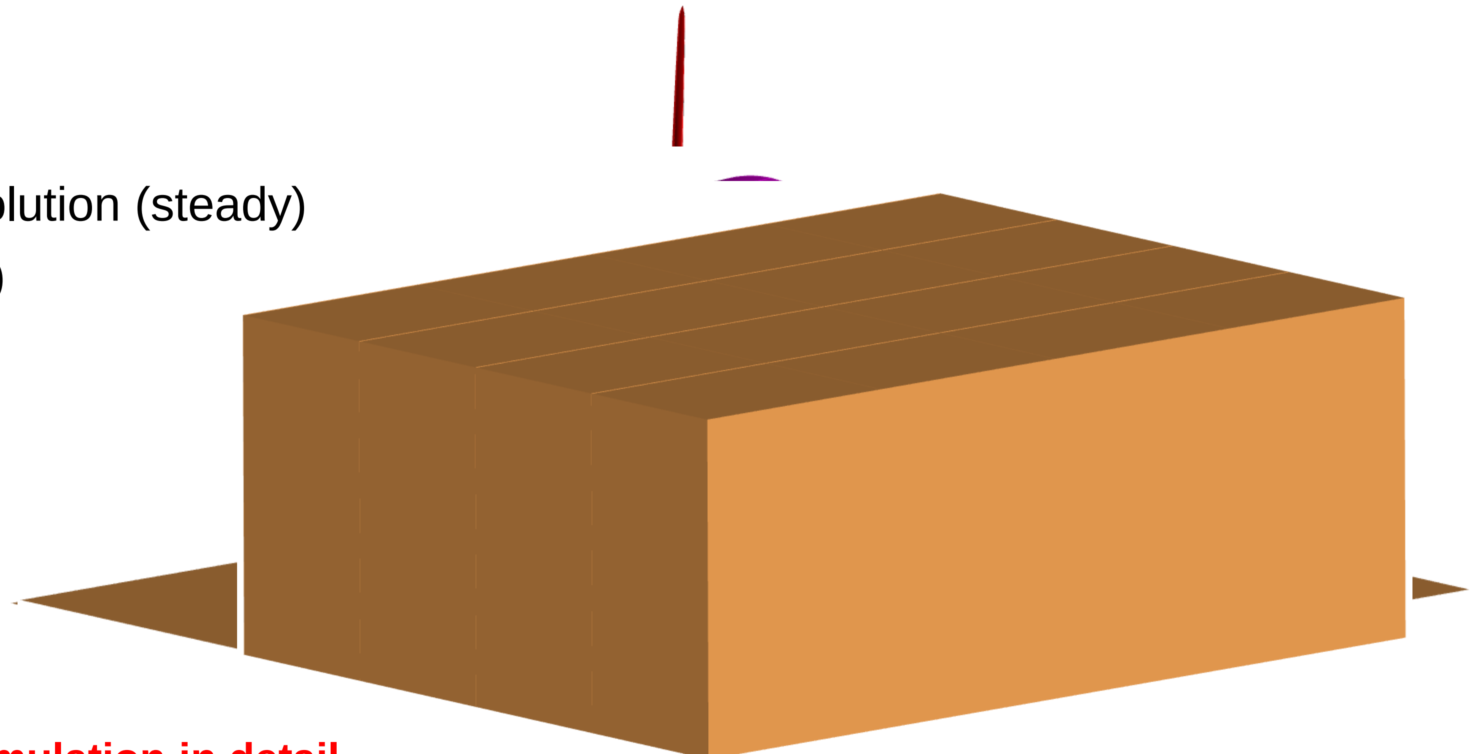
Introduction to CFD Grid Generation- Structured Grids 3D



Introduction to CFD Grid Generation- Structured Grids 3D

A wind turbine simulation consists of several meshes for:

- Spinner (rotating)
- Nacelle (steady)
- Tower (steady)
- Blades (rotating)
- Better tip vortex resolution (steady)
- Background (steady)



⇨ **Wind turbine simulation in detail affords meshing of several components**

Introduction to CFD Computational Efforts for WT Simulation

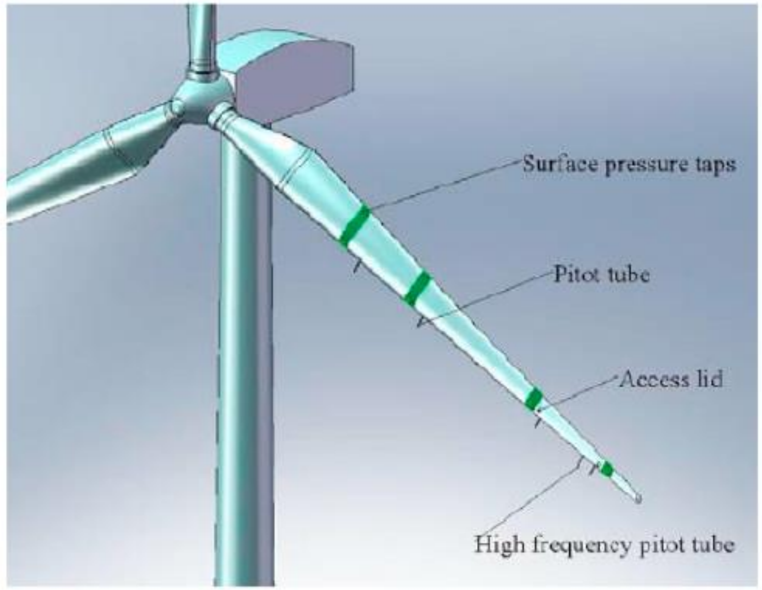
	1/3 Model (normal)	Full Model (simple)	Full Model (detailed)
Background + VortexGrid	16.7 Mio	66 Mio	97 Mio
Tower	-	1.0 Mio	1.0 Mio
Nacelle	0.8 Mio	2.5 Mio	2.5 Mio
Spinner	0.6 Mio	1.7 Mio	1.7 Mio
Blade	8.8 Mio	8.8 Mio x 3	8.8 Mio x 3
Sum	26.9 Mio	97.6 Mio	128.6 Mio
Time step	5°	5°	1°

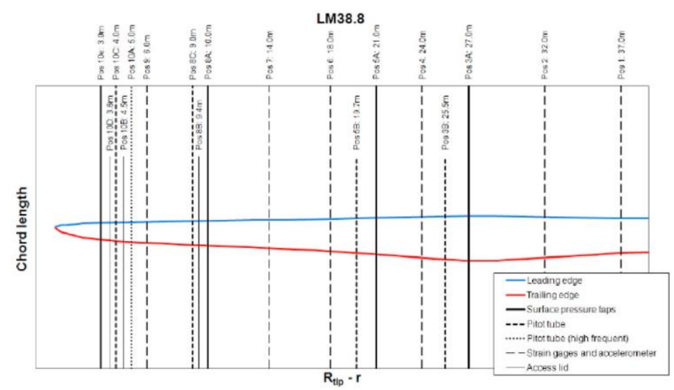
Depending on the flow situation a minimum of 10 revolutions is recommended

BEM vs. CFD DanAero Turbine

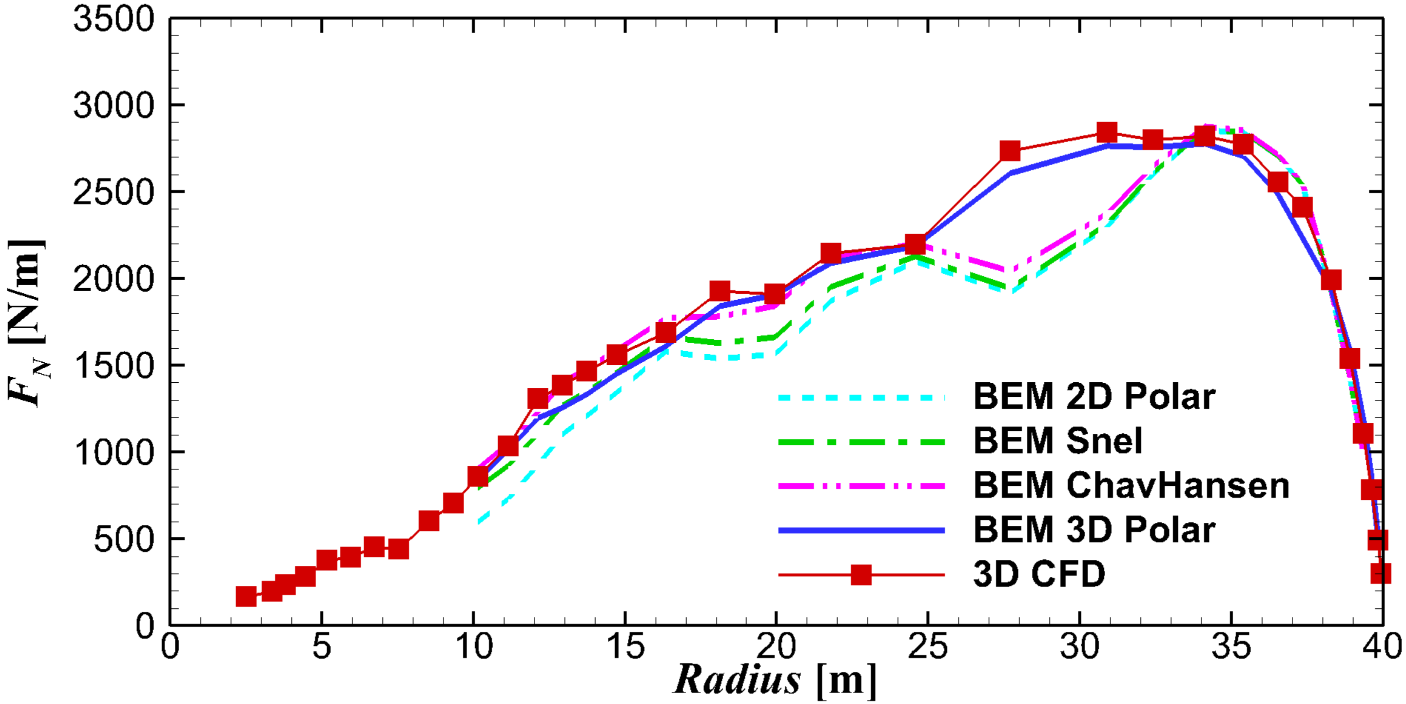
Madsen et al. The DAN-AERO MW experiments: final report. Danmarks tekniske universitet, risø nationallaboratoriet
for bæredygtig energi. 2010. available at: orbit. dtu. dk, . [Accessed 13 November 2017].

- Turbine has a diameter of 80 m, tilt 5° and 1.4 m prebend at the tip
- The field measurement was conducted from 2007-2010 at Tjaereborg, 1 km from the north sea
- The nominal rated power is 2.3 MW
- The turbine is investigated within IEA Task 29 Phase IV project





BEM vs. CFD Results – DanAero Turbine

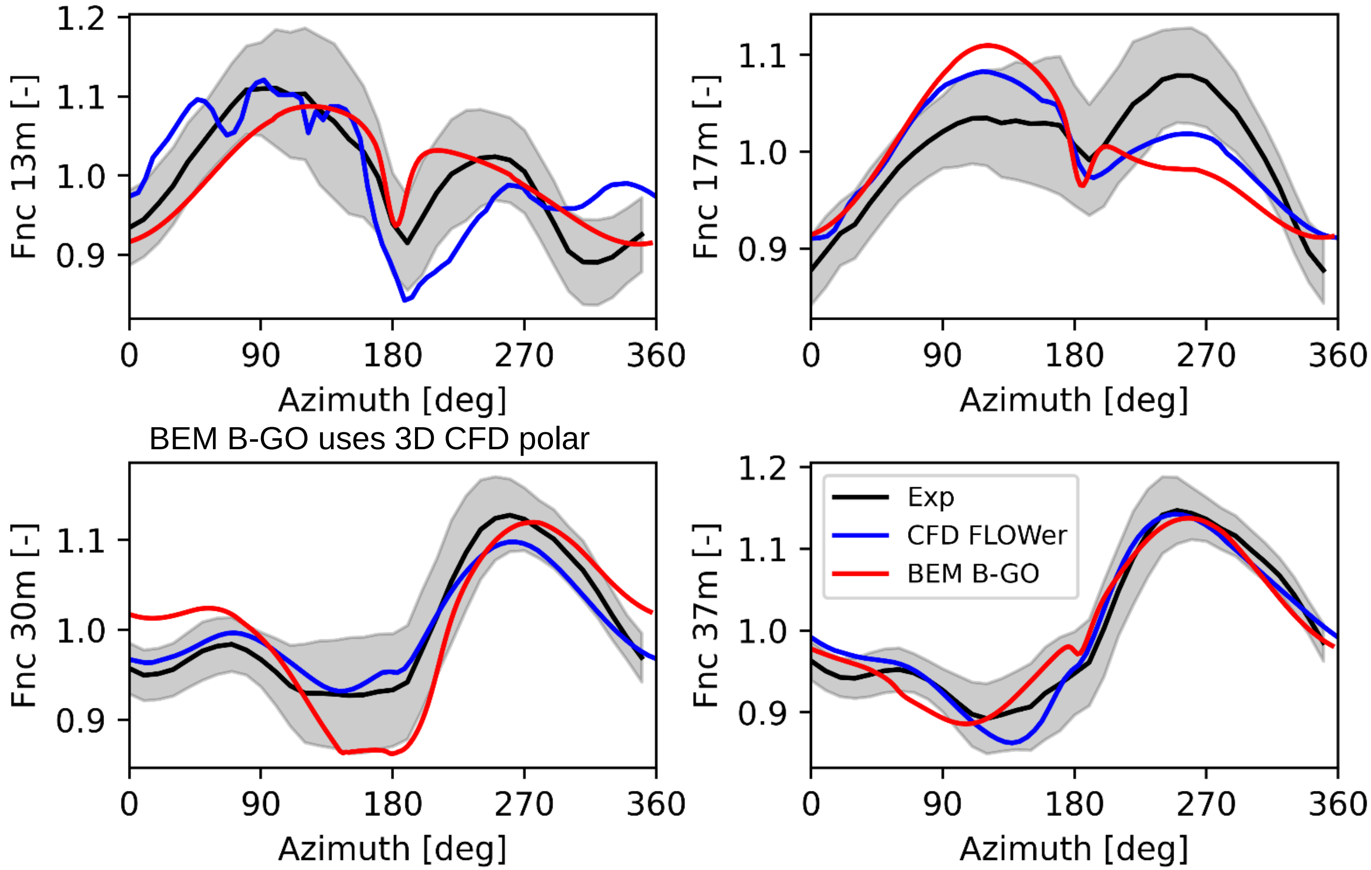


CFD extracted 3D polar data improves BEM prediction for a 2.3 MW DanAero turbine!

G. Bangga

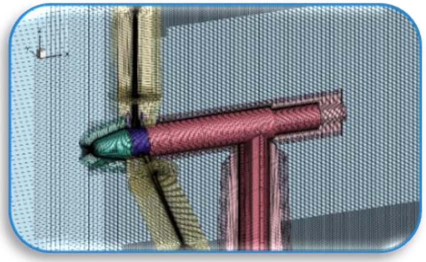
BEM vs. CFD Results – DanAero Turbine

Also in complex flow condition under 38° yaw misalignment!

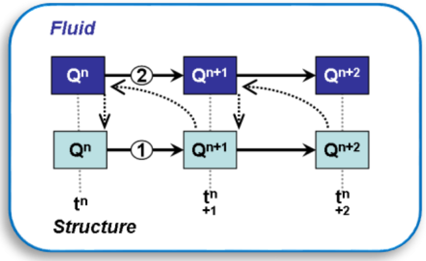


G. Bangga

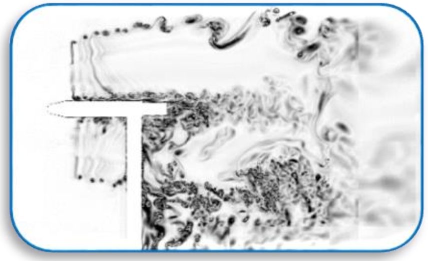
Advanced CFD Chain USTUTT-IAG Chain Based on CFD Solver FLOWer



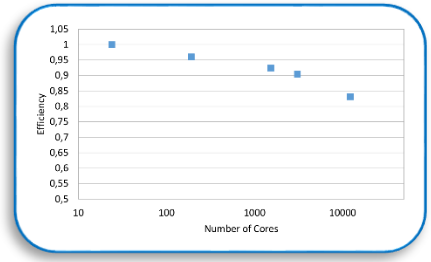
Moving, deformable grids



Fluid-structure interaction

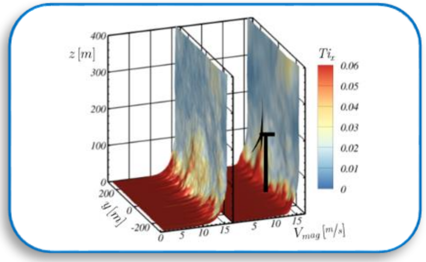


Higher-order numerics

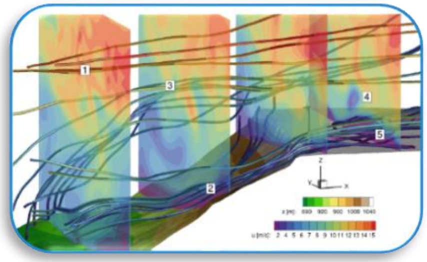


Efficient parallelization

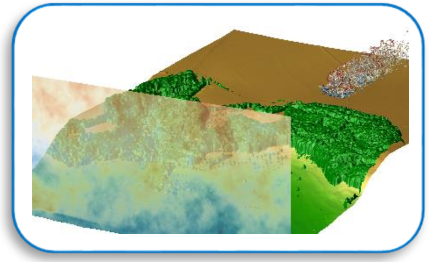
Aerodynamics, loads and noise emission of the controlled, flexible turbine in atmospheric inflow



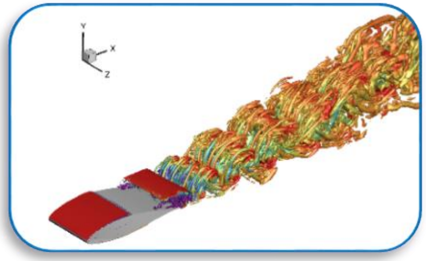
Inflow turbulence & shear



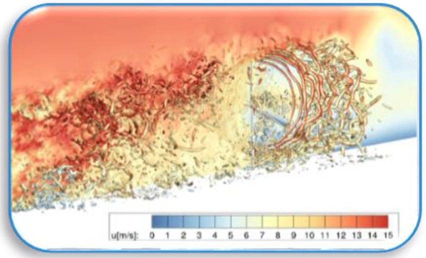
Complex terrain, vegetation



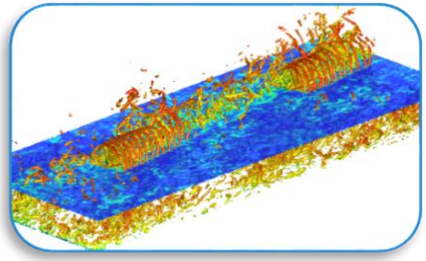
Stratification



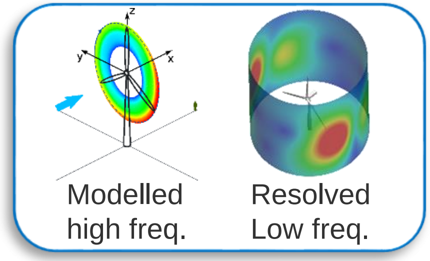
Airfoil aerodynamics



Rotor aerodynamics

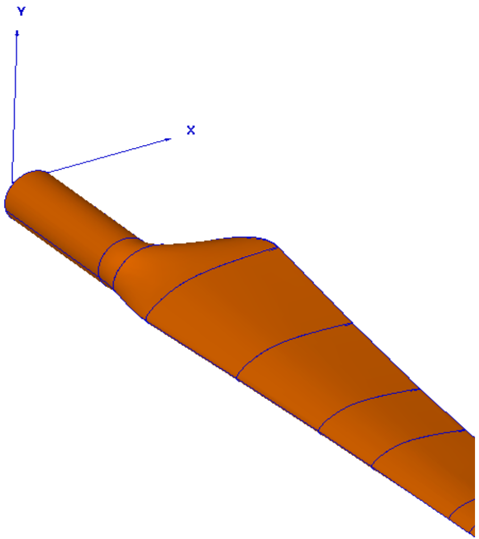


Wake interactions

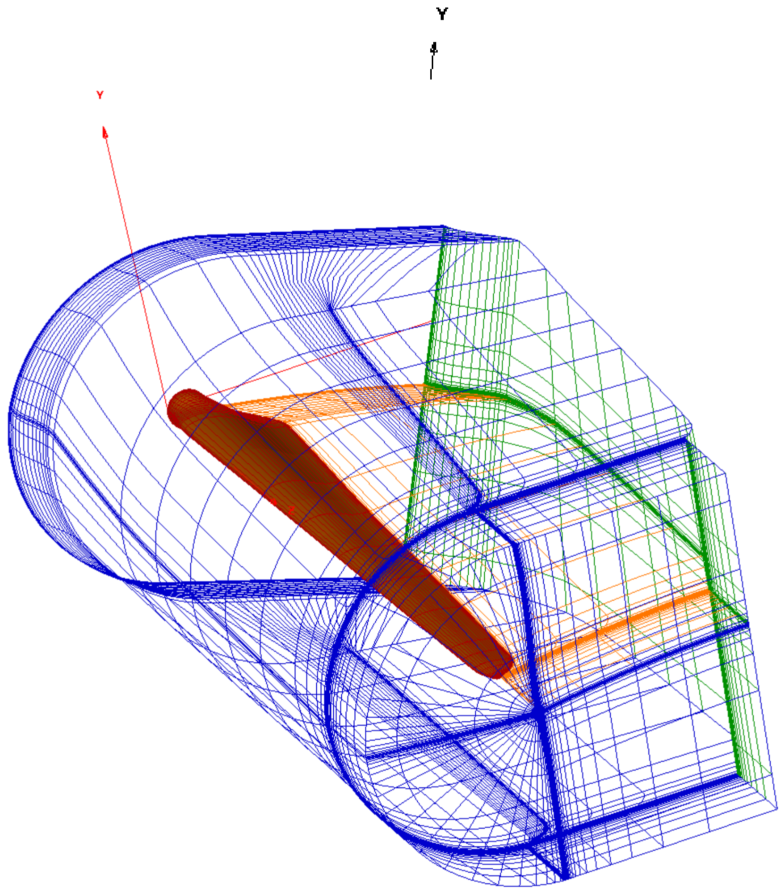


Aeroacoustics

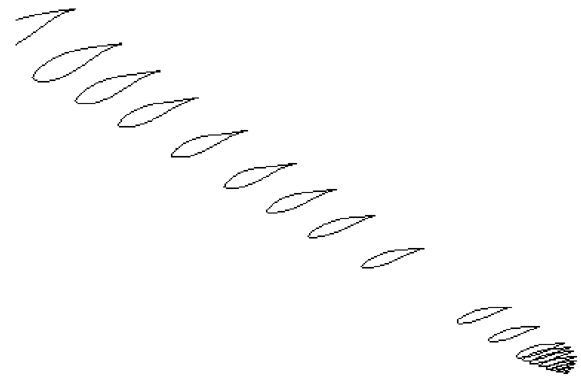
Advanced CFD Chain Automated Blade Meshing



CAD data in iges format



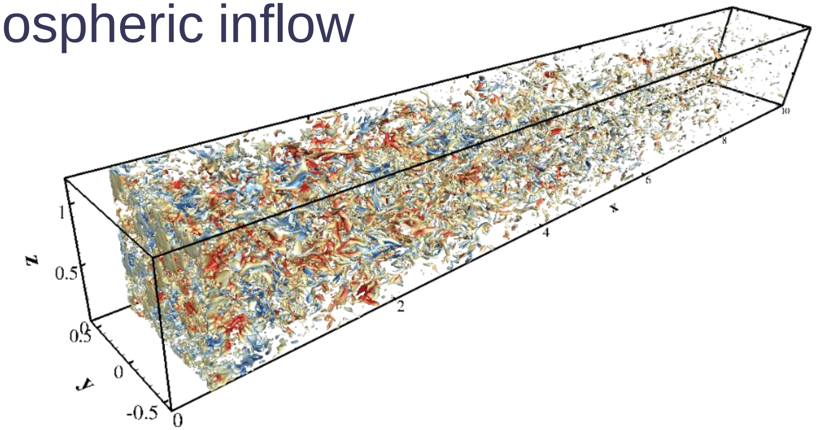
Meshed blade



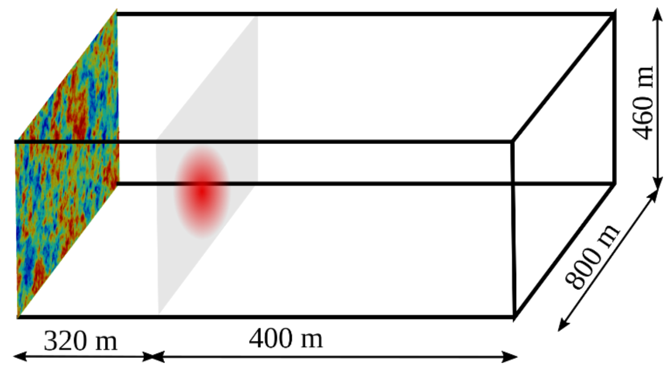
Cuts of the blade
(actual basis of mesh generation)

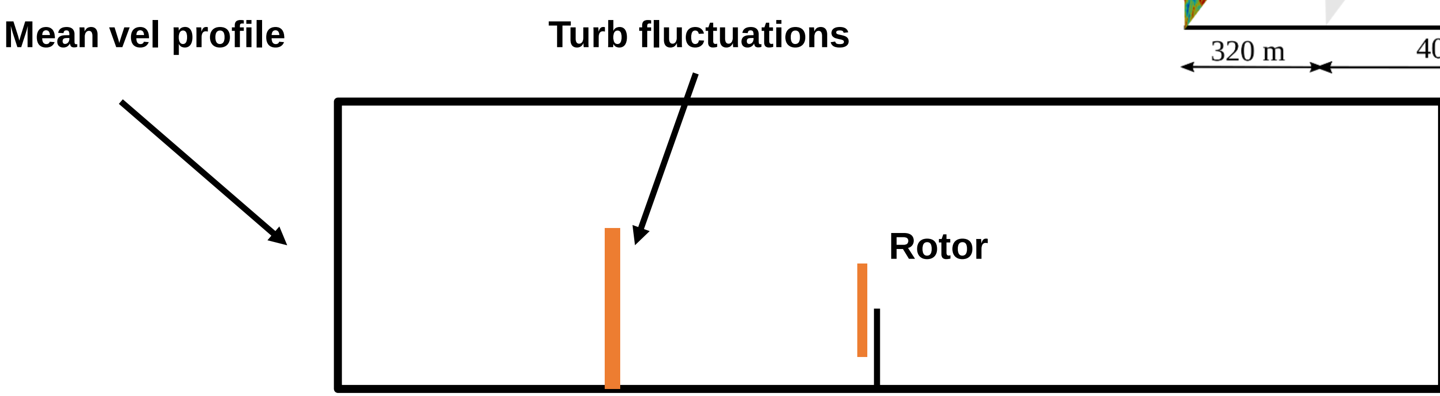
Advanced CFD Chain Unsteady atmospheric inflow

As a forcing term



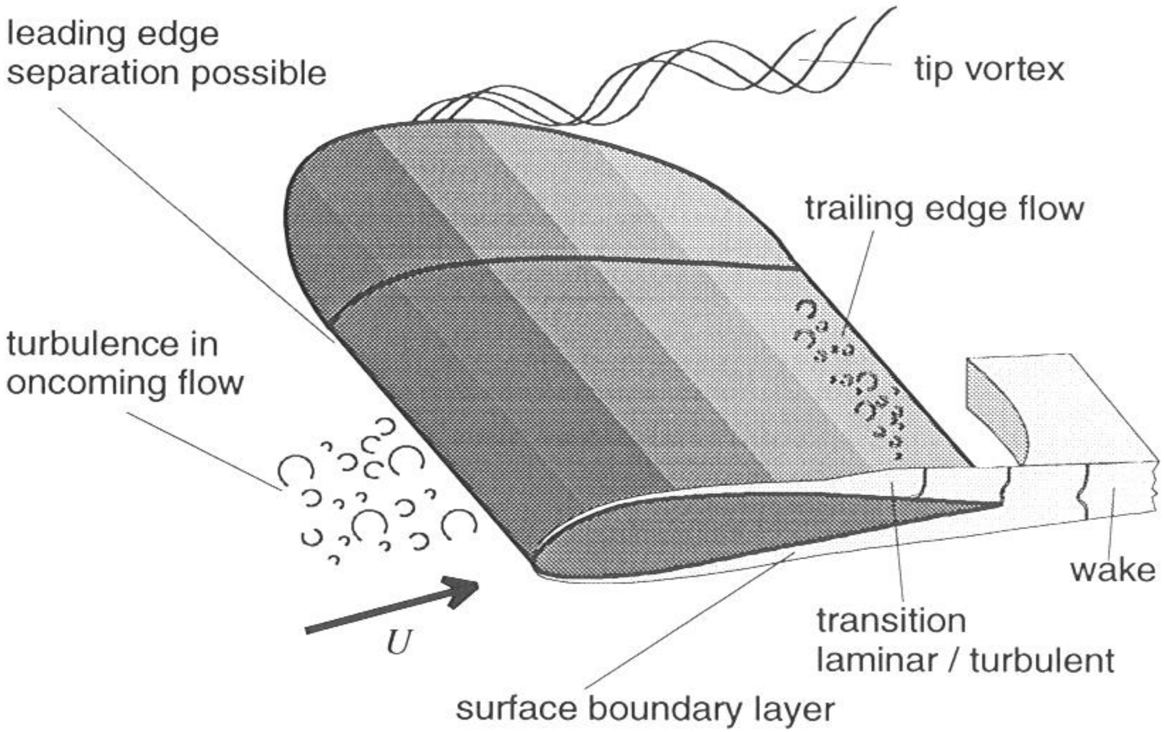
- Turbulent flow is fed into the simulation domain a forcing term in three directions, using only its fluctuations
- Mean velocity profile is specified at the inflow domain
- Both are combined by means of superposition
- Minimizing numerical dissipation



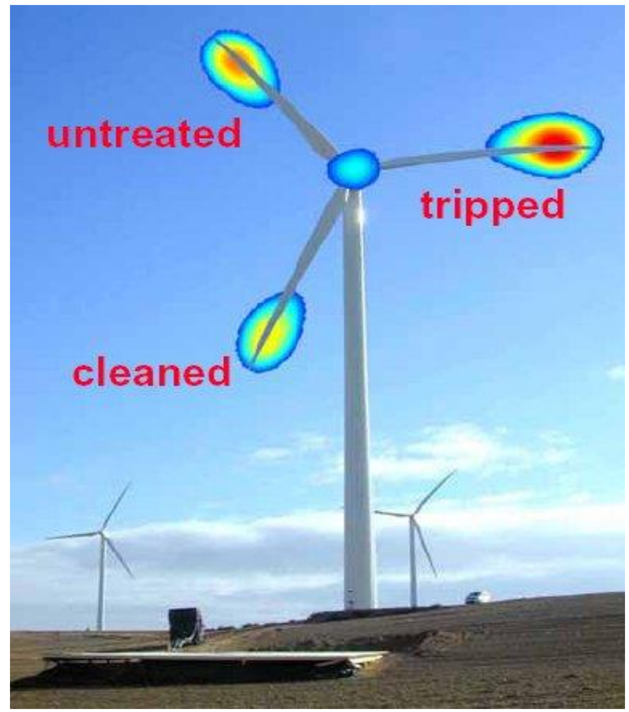


Exemplary CFD Applications

Noise Prediction & Reduction Relevant Aeroacoustic Noise Sources

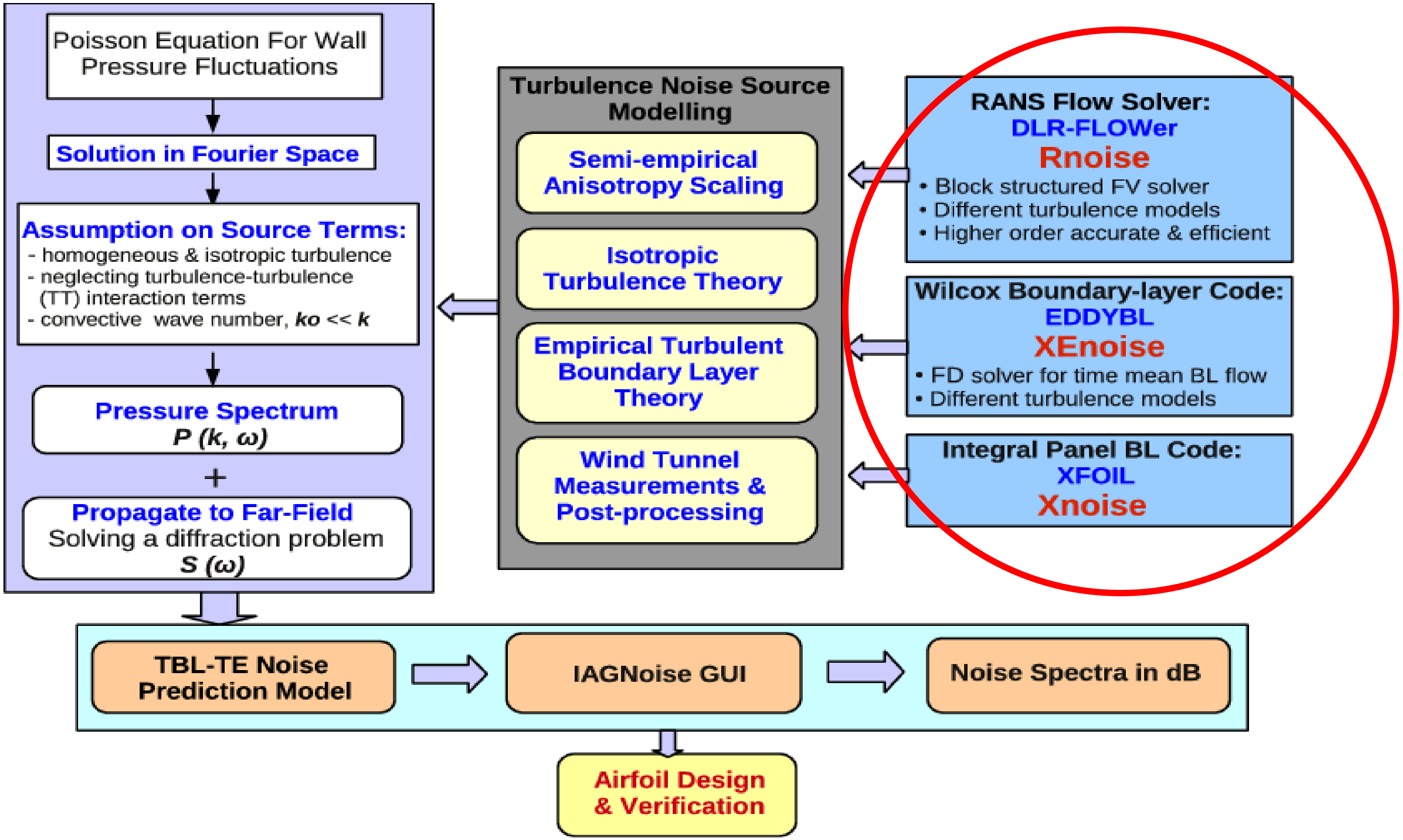


Wagner et al.: Wind Turbine Noise



S. Oerlemans (NLR)

Noise Prediction & Reduction **Airfoil** Self Noise Prediction Schemes



Th. Lutz, USTUTT

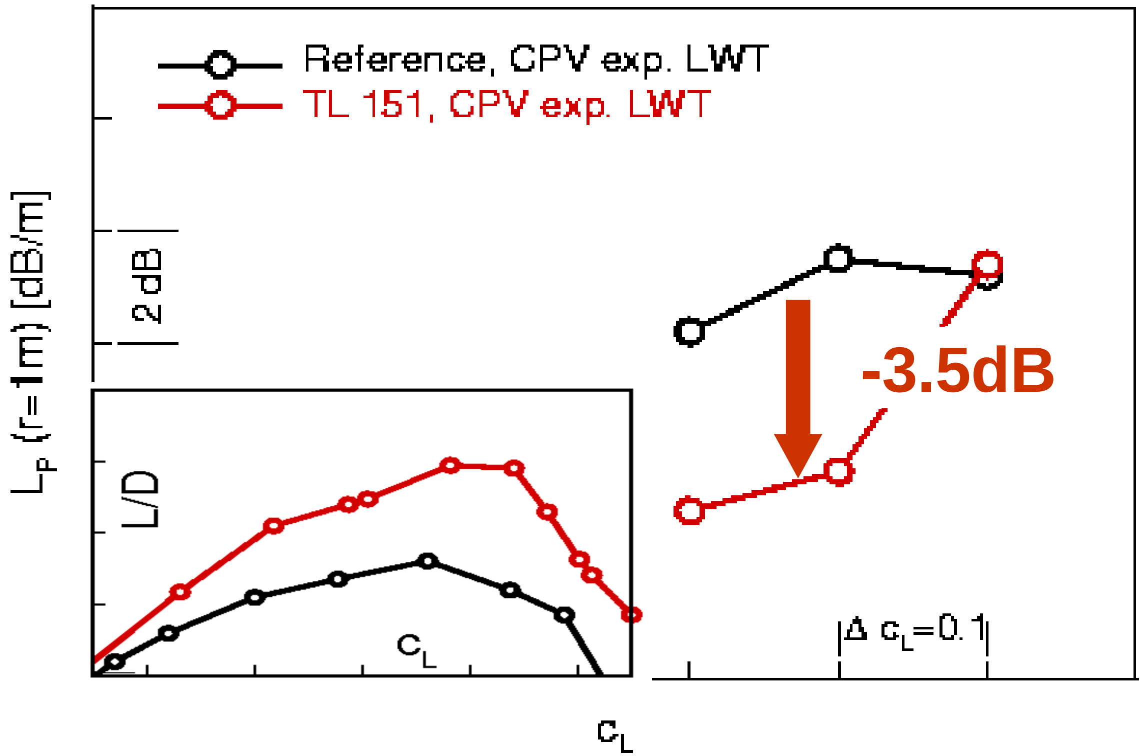
Passive Noise Reduction Aeroacoustic Airfoil Design

GE Reference Turbine

(2.3 MW, D=94m, Wieringermeer, NL)

Achieved Gain

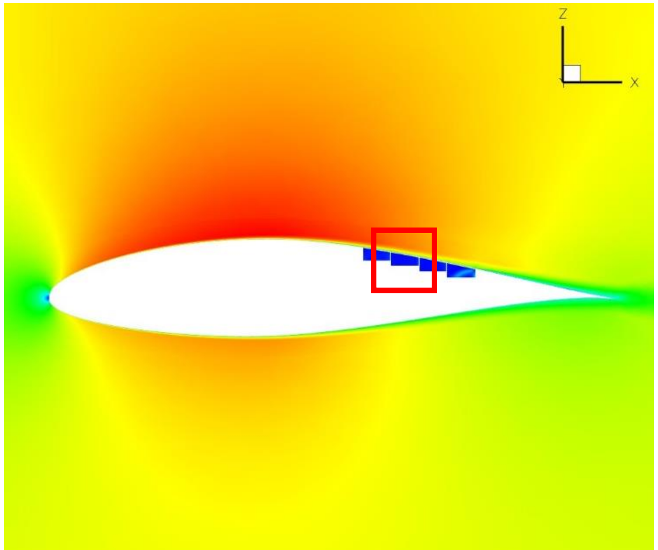
(Wind Tunnel Results)



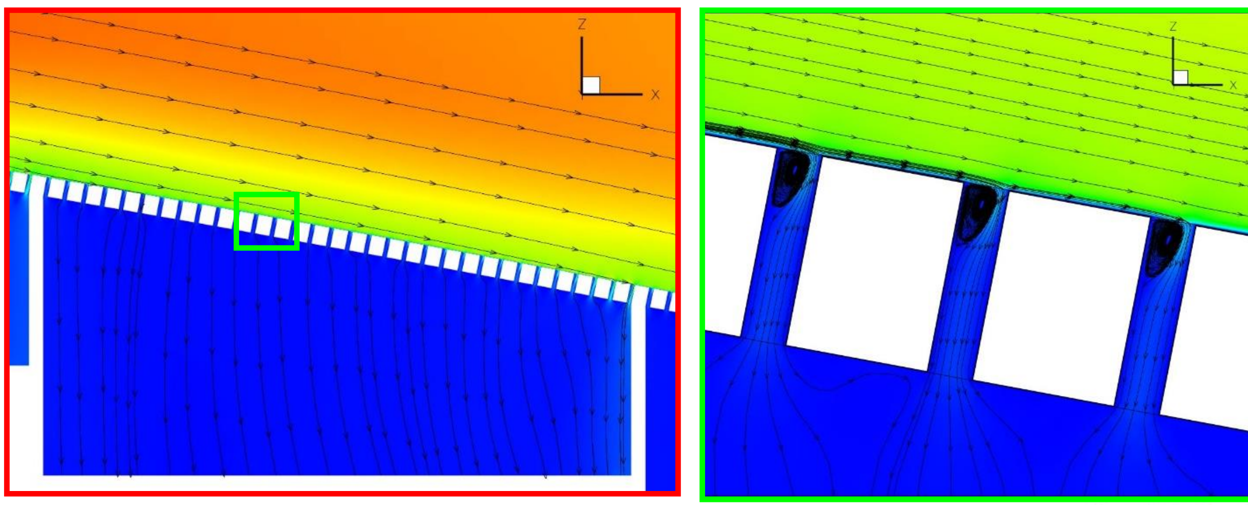
- *Reduction of trailing-edge noise*
- *Increase of performance L/D*

Th. Lutz: AIAA Journal, 2007

Noise Reduction by AFC



- Hybrid Mesh for NACA64-418 airfoil with 4 suction chambers on upper side
- Each 0.25mm hole is resolved with 10 points
- 320.000 points, $y^+=1$, BL resolved with 50 points
- Resolving the porous plate delivers a more reliable suction velocity distribution
- Pressure losses at porous plate are considered



TAU simulation performed:

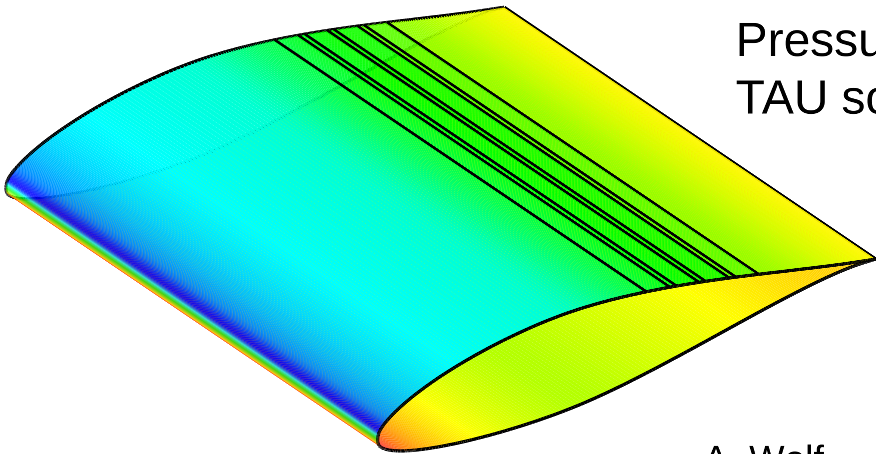
Ma=0.206

Re=0.6

TM: Wilcox k-ω

A. Wolf

Noise Reduction by AFC CFD vs. Wind Tunnel Experiment

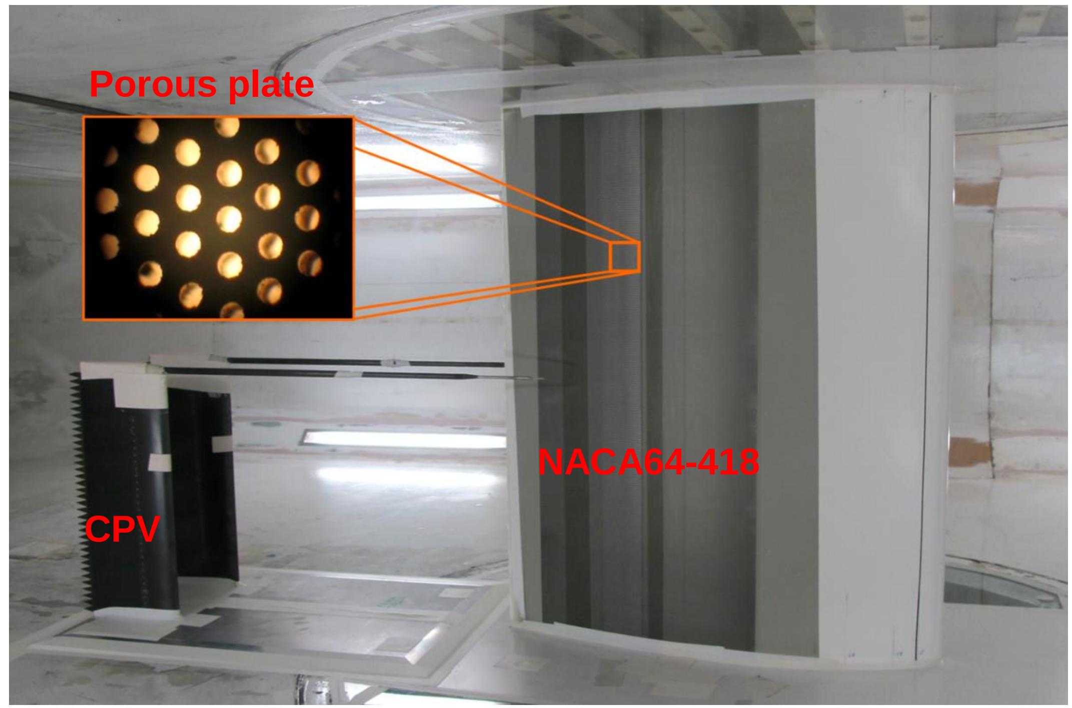


Pressures surface plot of NACA64-418
TAU solution

A. Wolf

NACA64-418 Wind Tunnel Model with suction device + CPV System for acoustic measurements

Magnified picture: Porous plate of suction device, porosity: 25%, hole diameter: 0.25mm



Noise Reduction by AFC CFD vs. Wind Tunnel Experiment

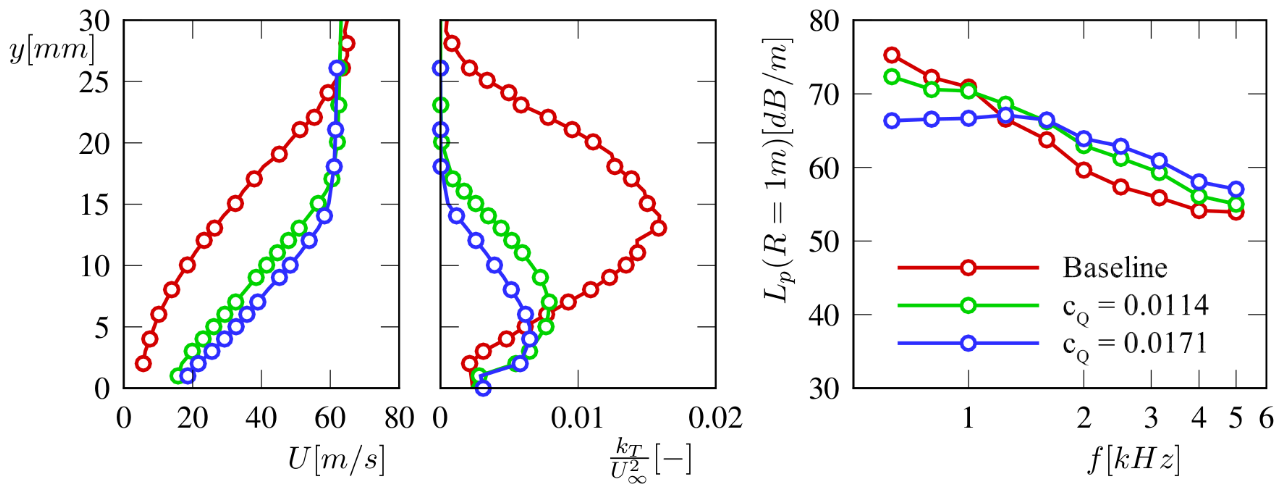
Distributed Suction: Impact of mass flow coefficient

Experiment:

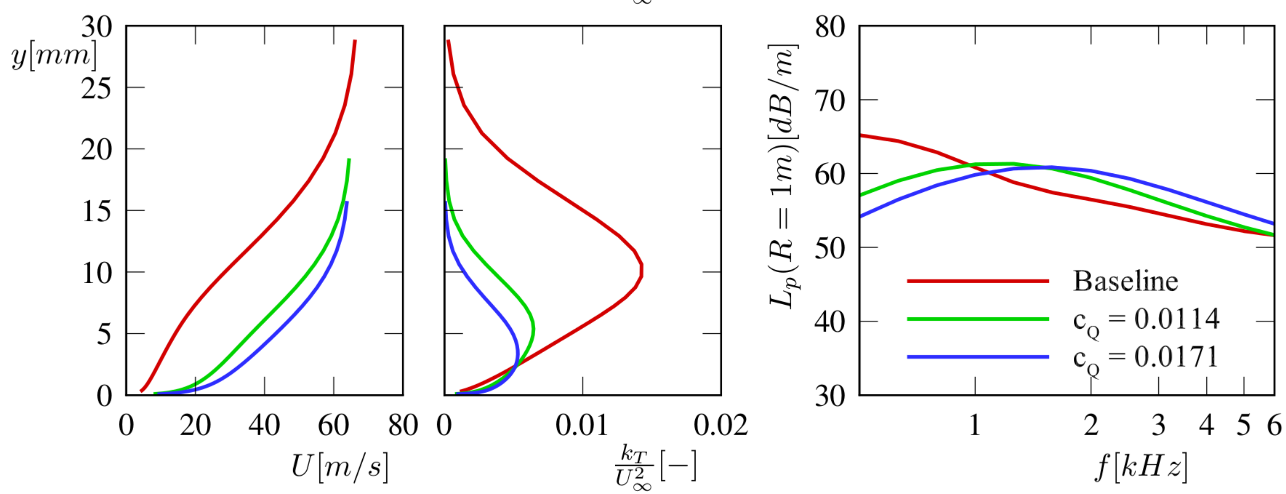
NACA 64-418

Re=2.5e6, α=6°

$x_{tr}/c=0.05$



Prediction (Rnoise):

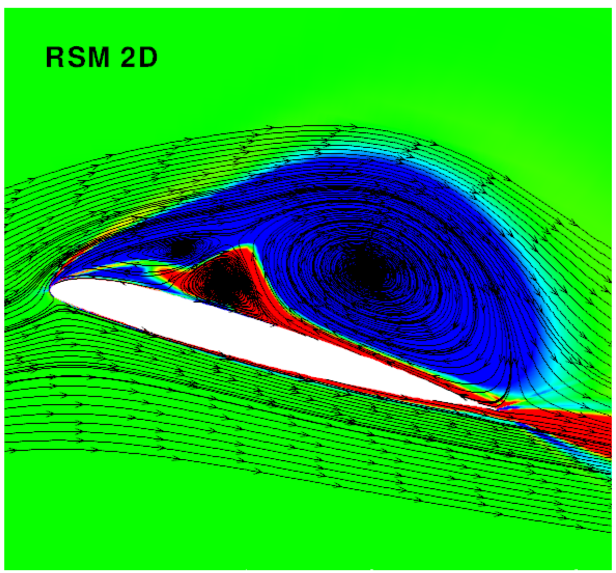


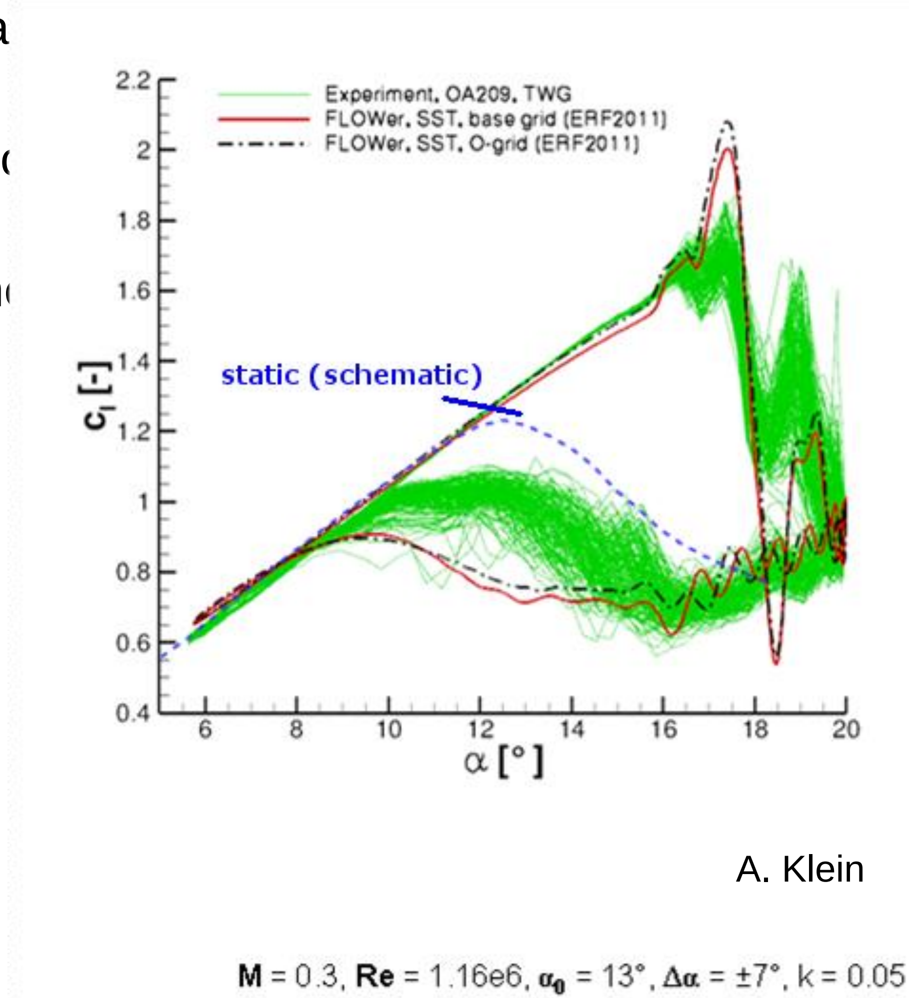
$$c_Q = \frac{\dot{m}_s}{\rho_\infty \cdot U_\infty \cdot A_s}$$

A. Wolf

Dynamic Stall Observations

- Enhanced maximum lift by delay of the sta angle with respect to stationary stall
- Formation of a characteristic large dynami stall vortex on the suction side
- Strong breakdown of lift and pitching mom due to the downstream convection of the vortex leads to high dynamic loads

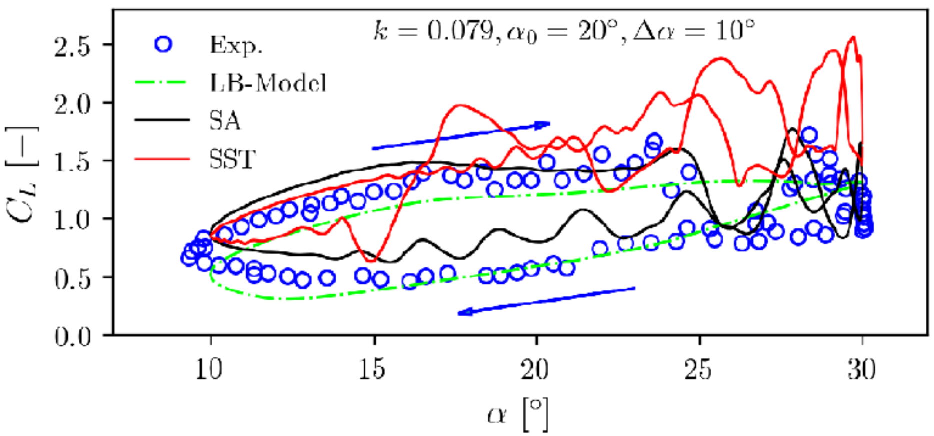


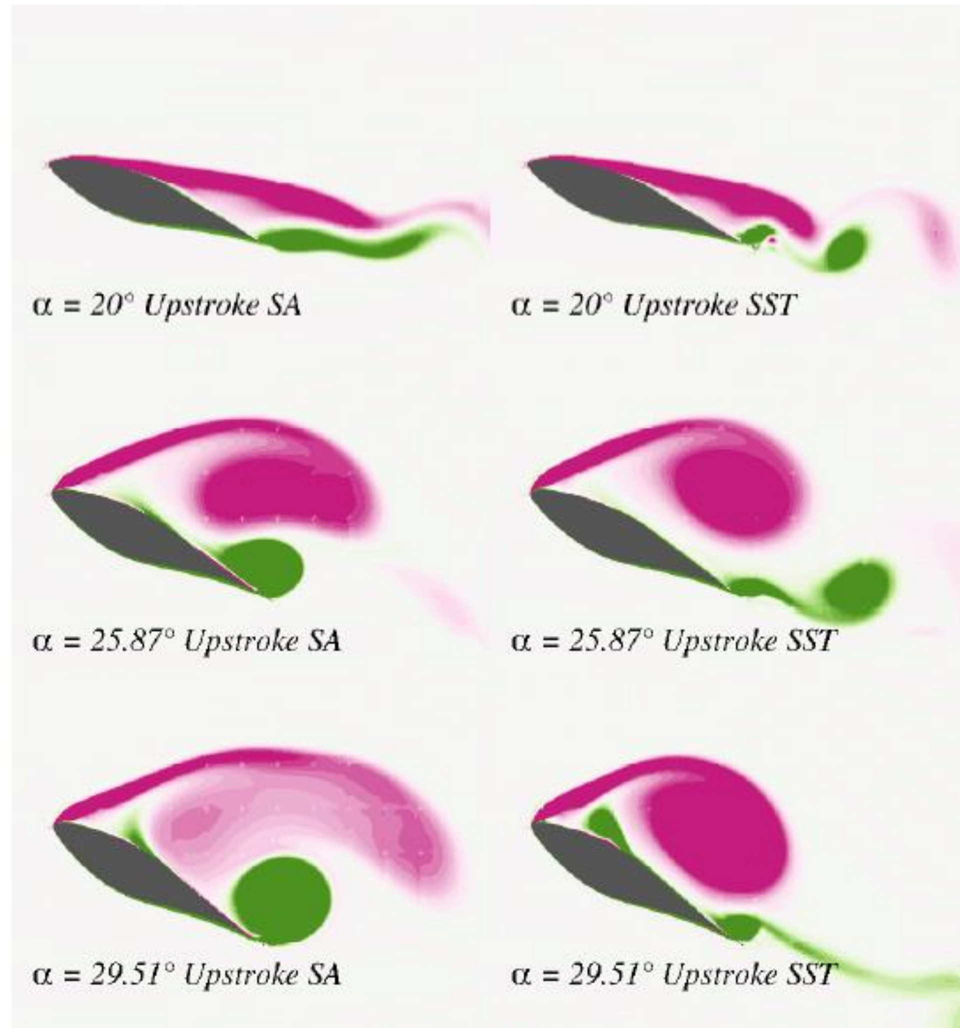


Th. Lutz, USTUTT

Dynamic Stall Prediction

Dynamic stall

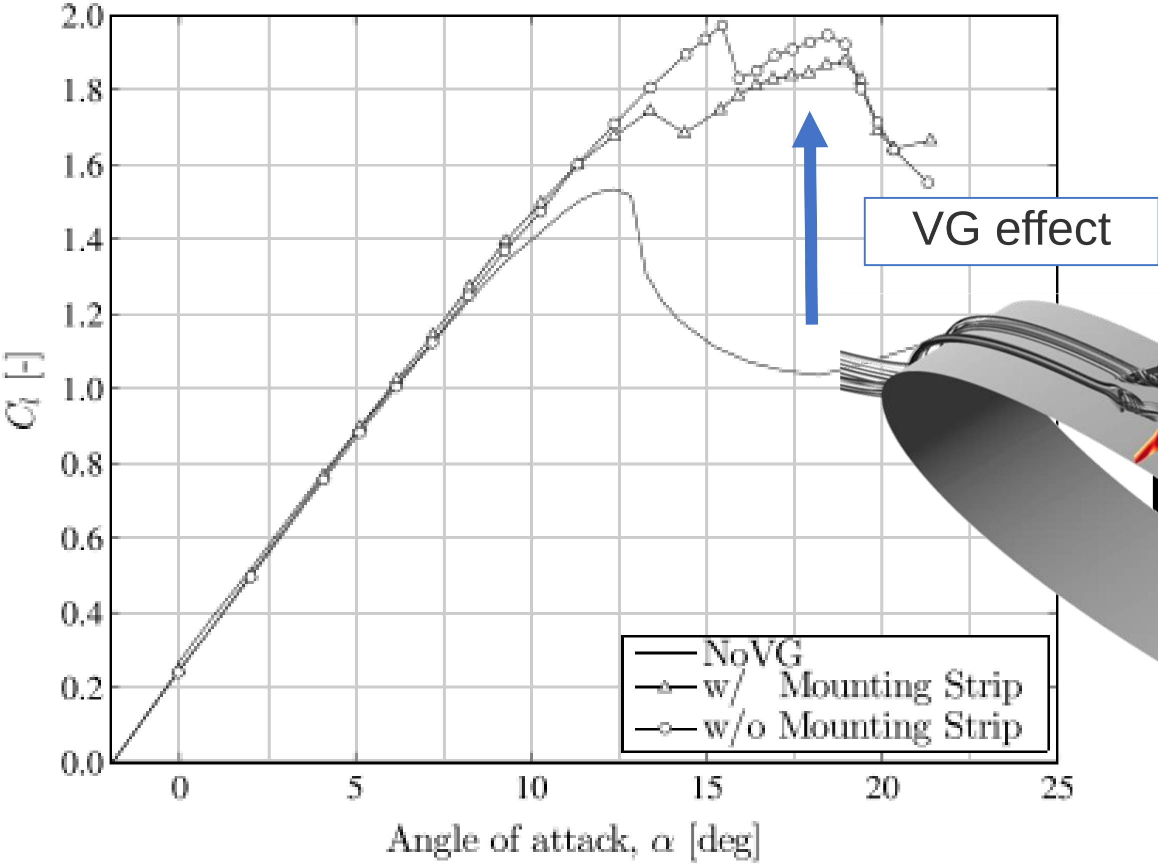




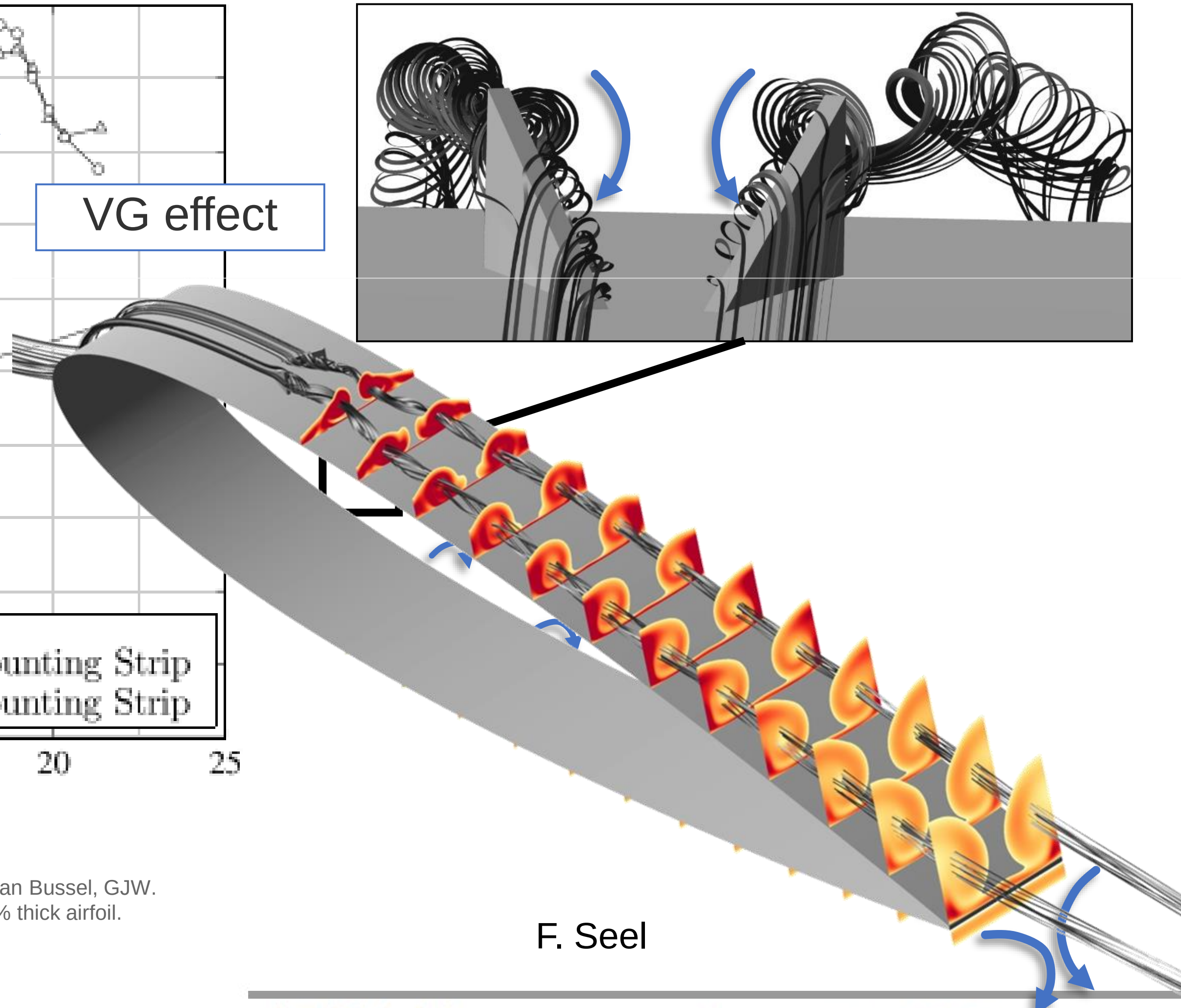
Force and flow field characteristic predictions depend on the employed CFD approaches especially on turbulence model

G. Bangga

Vortex Generators Separation Delay & Lift Enhancement



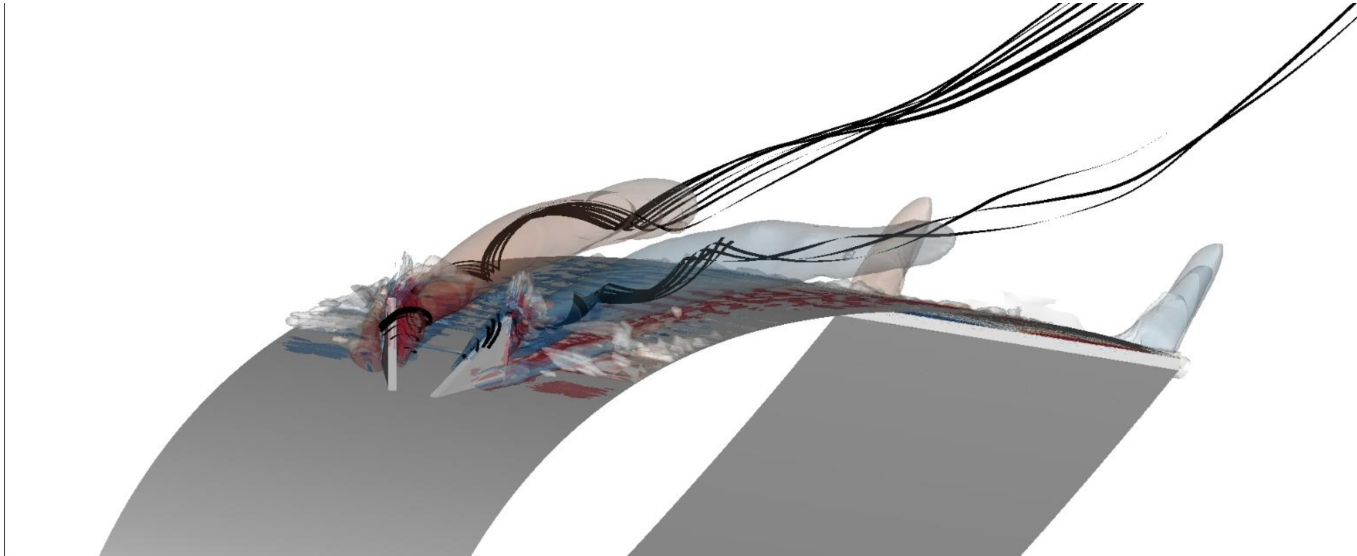
Front view:



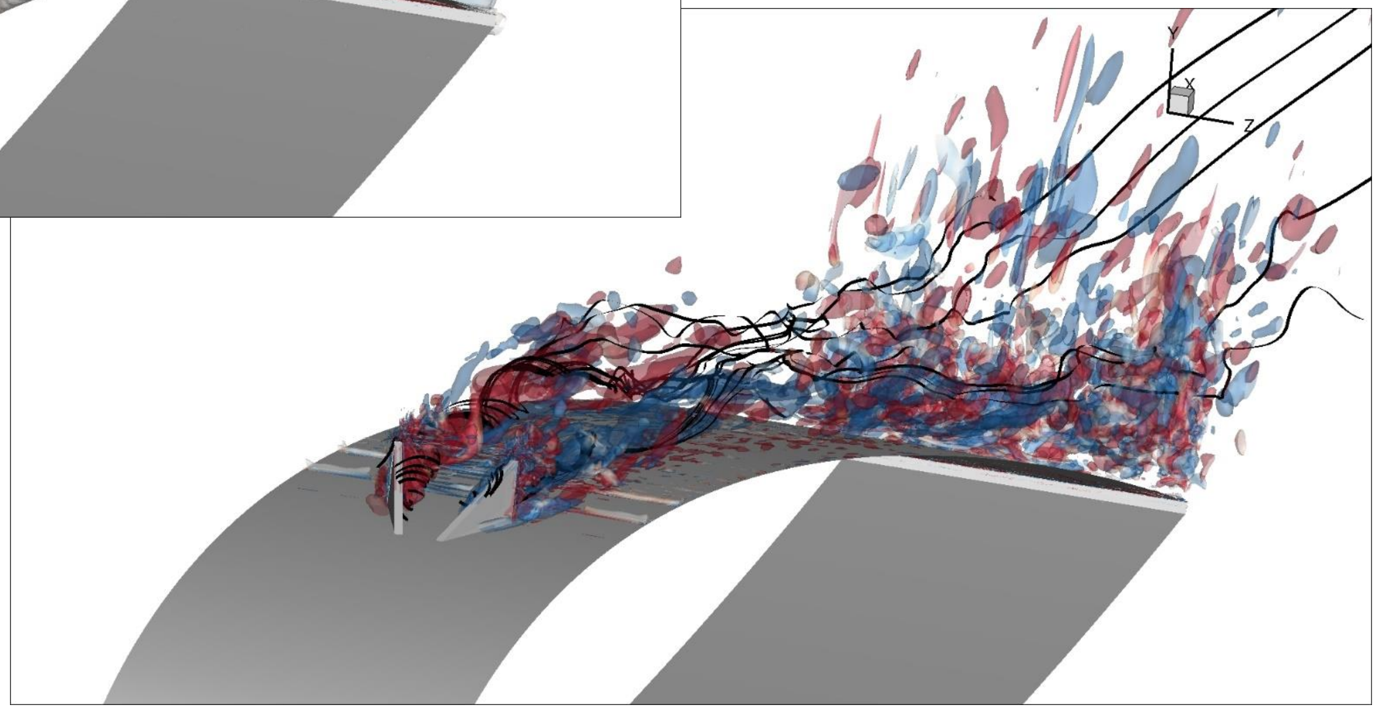
Source: Baldacchino, D, Ferreira, C, De Tavernier, D, Timmer, WA, van Bussel, GJW. Experimental parameter study for passive vortex generators on a 30% thick airfoil. Wind Energy. 2018; 21: 745– 765. https://doi.org/10.1002/we.2191

F. Seel

Vortex Generators Impact of Turbulence Model on Predicted Mixing



Scale resolving iDDES



URANS result

F. Seel

Rotor & Wake Flow Physics The Mexico Rotor

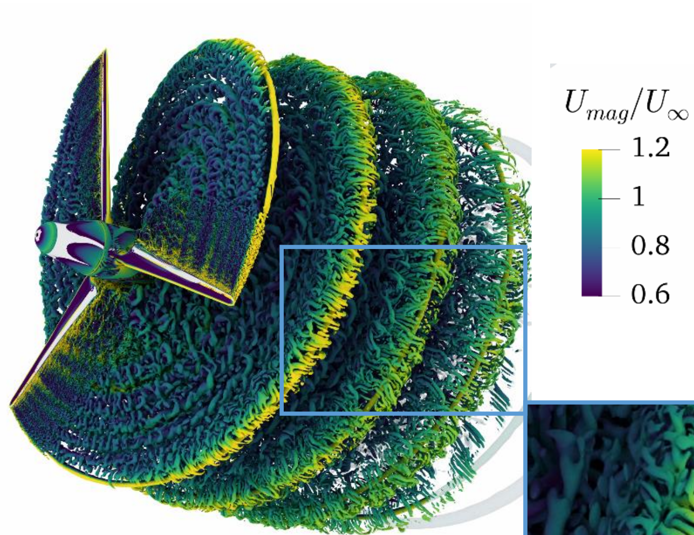
DNW LLF

Mexico Rotor

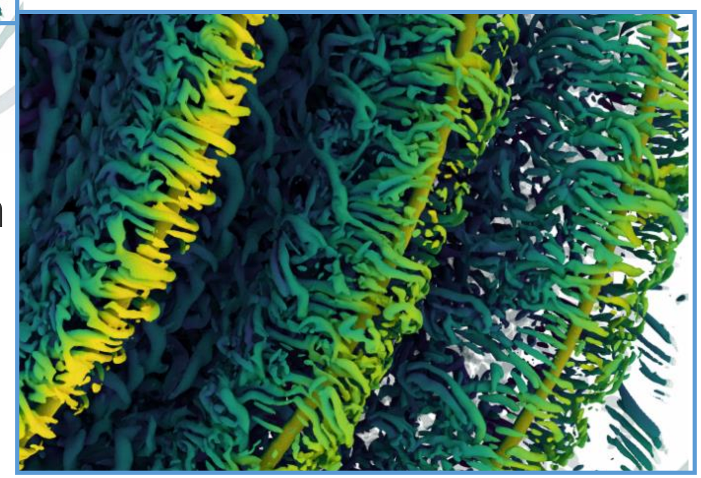
Rotor & Wake Flow Physics Tip Vortex Development

Simulation of the MEXICO model wind turbine at low TSR

Smoke visualization in the experiment



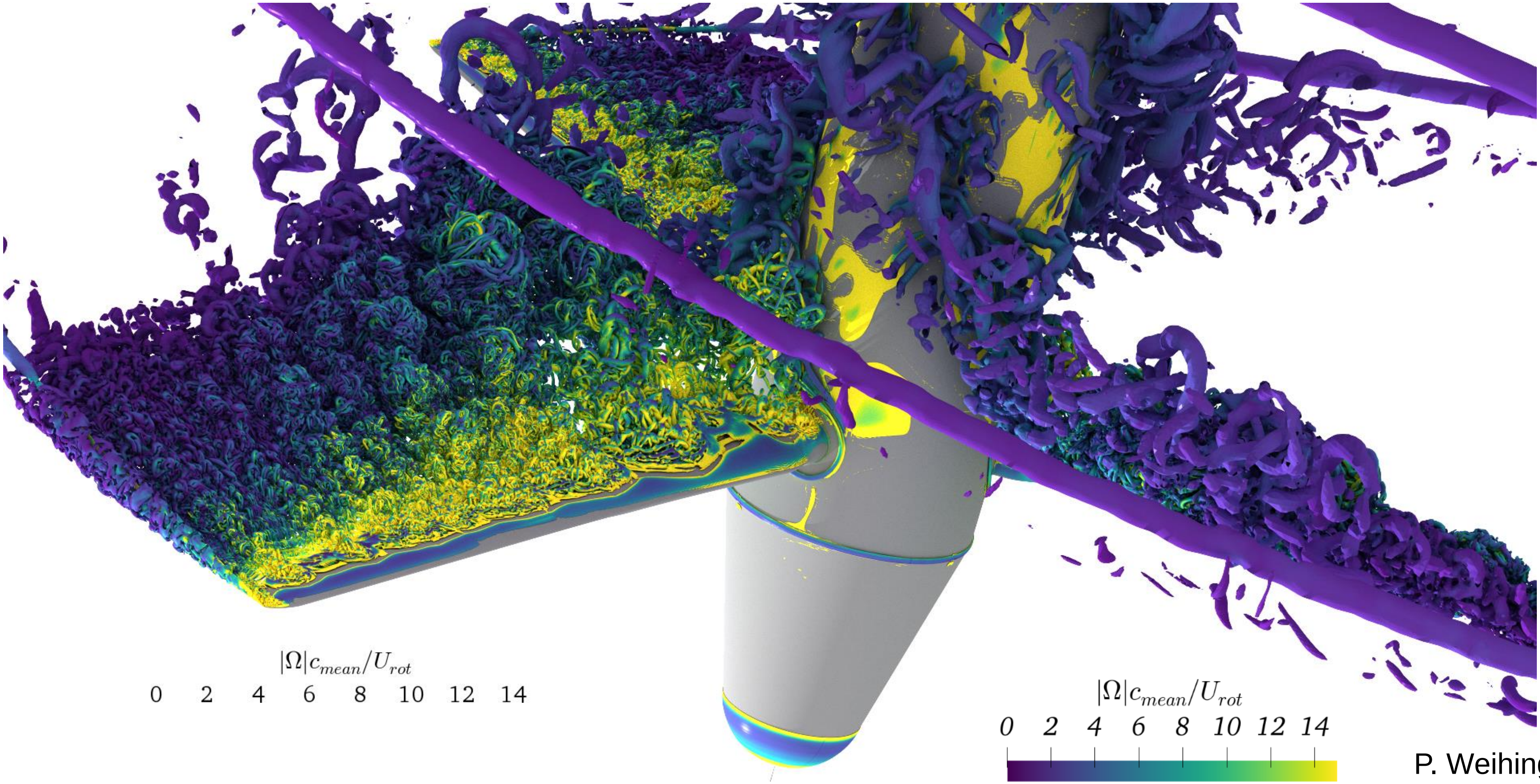
High-fidelity simulation



P. Weihing

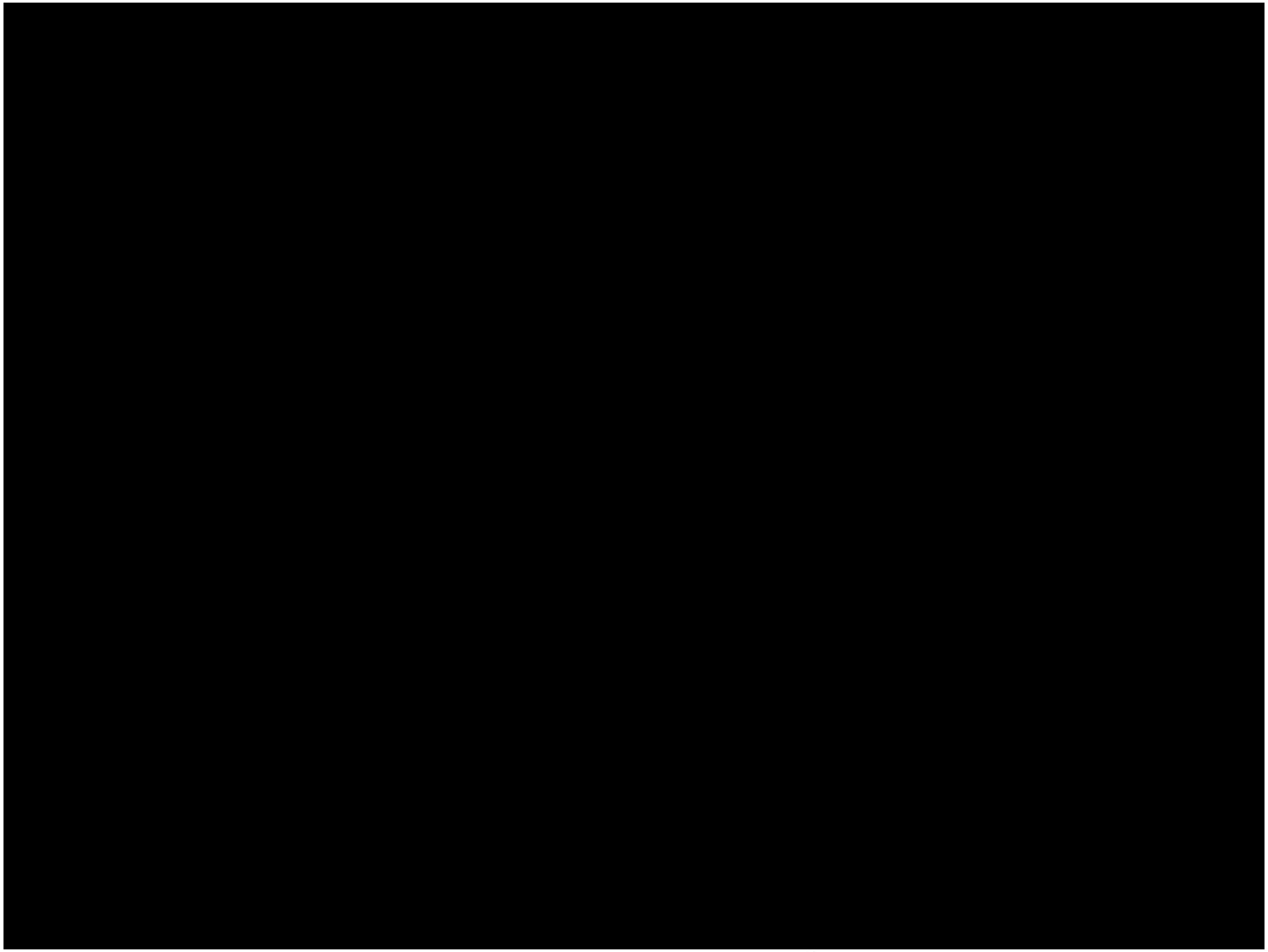
Formation of “turbulent worms”

Rotor & Wake Flow Physics BDES Result

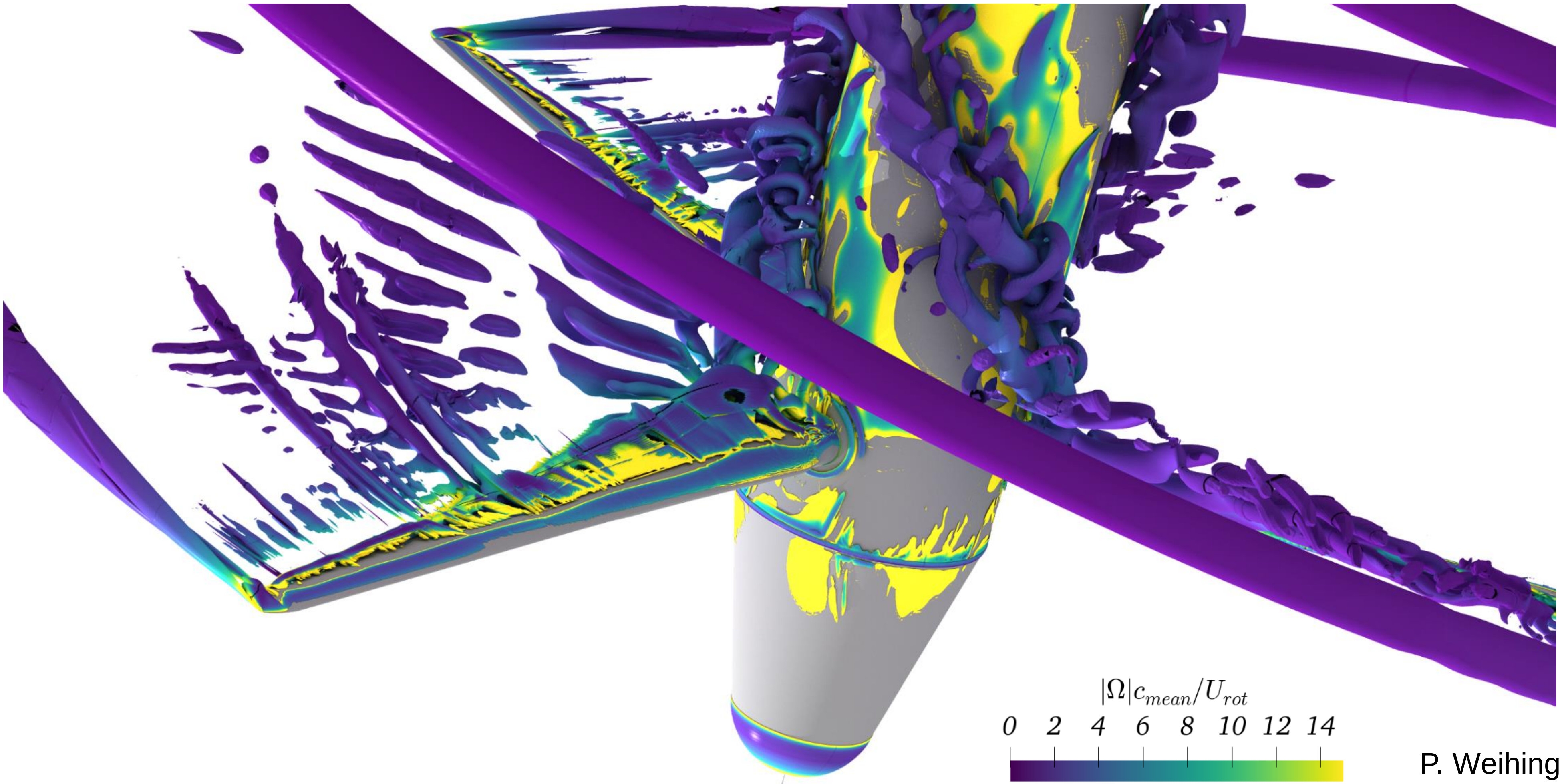


P. Weihing

Rotor & Wake Flow Physics BDES Result - *Video*

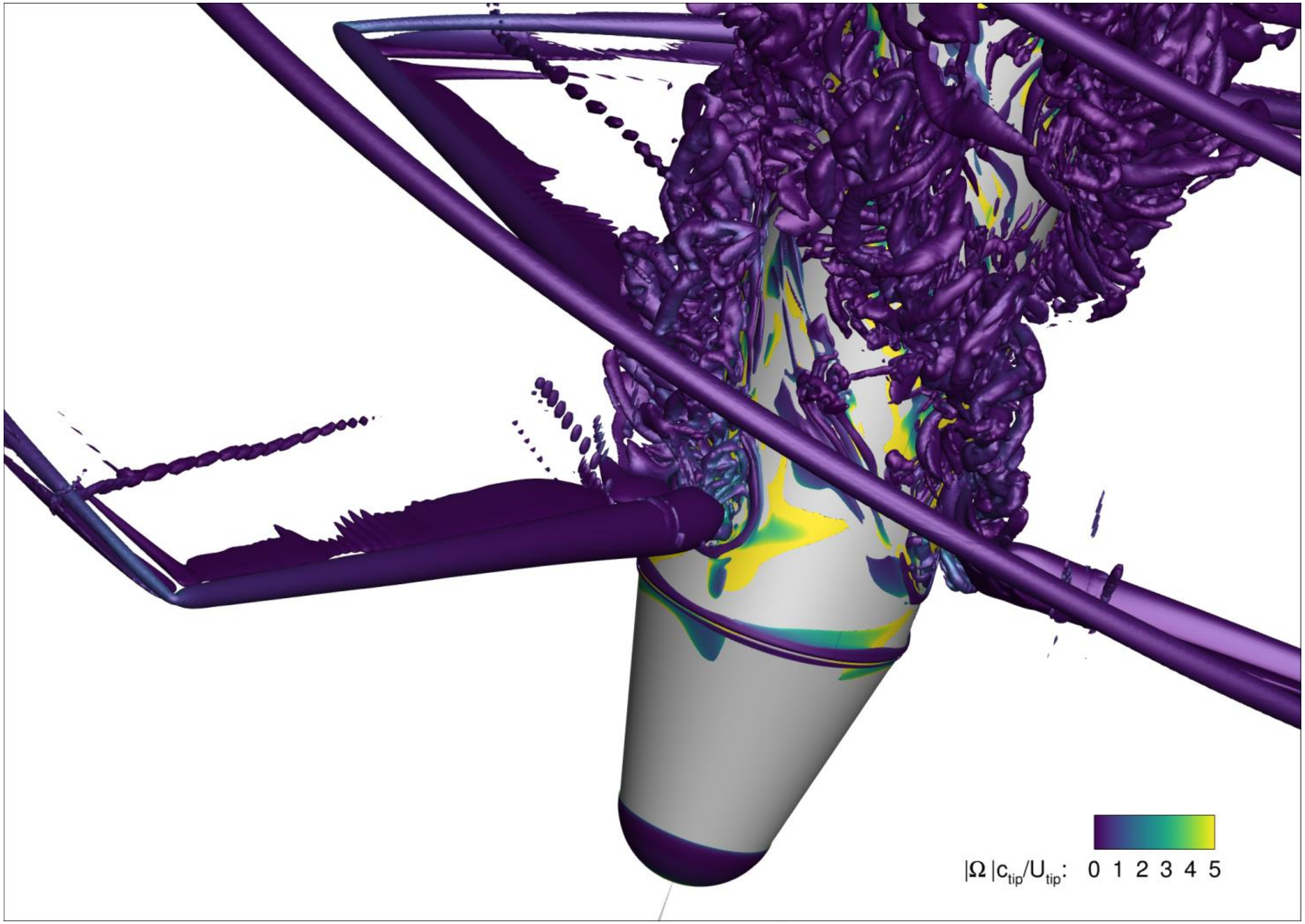


Rotor & Wake Flow Physics URANS Result



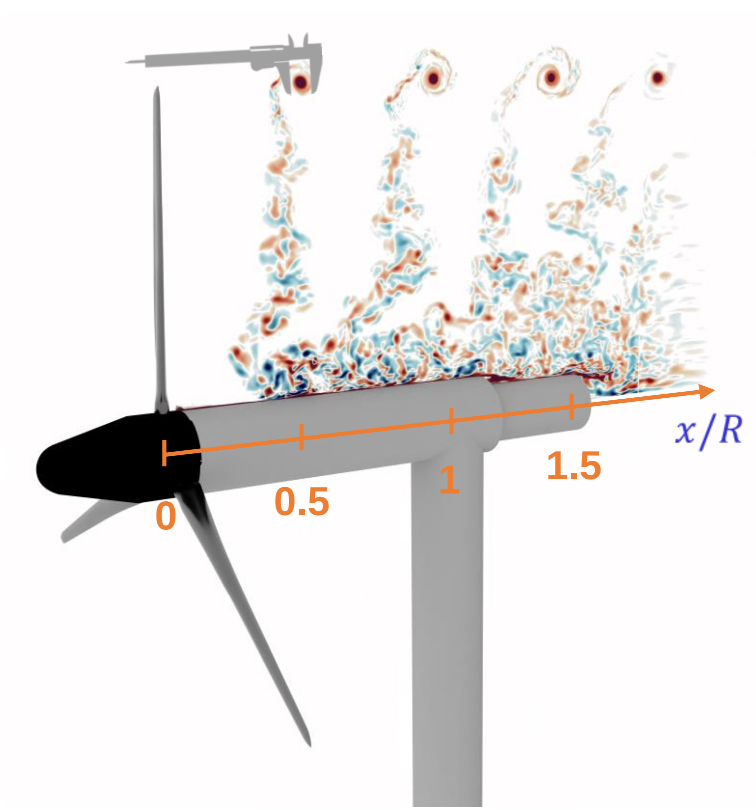
P. Weihing

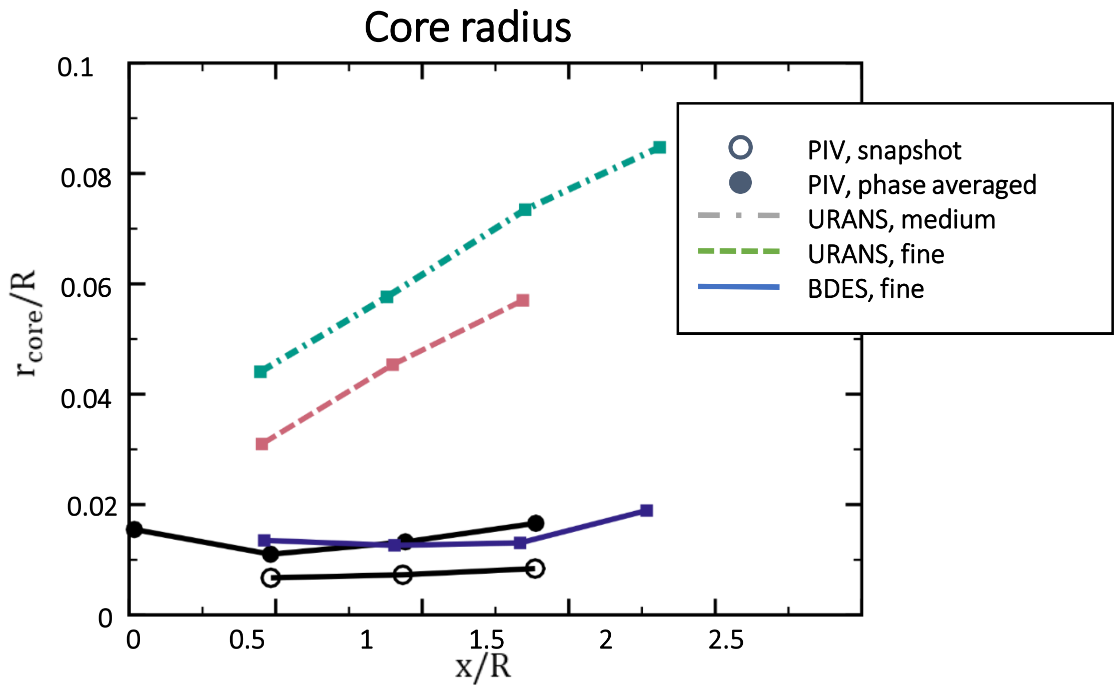
Rotor & Wake Flow Physics Actuator Line Result



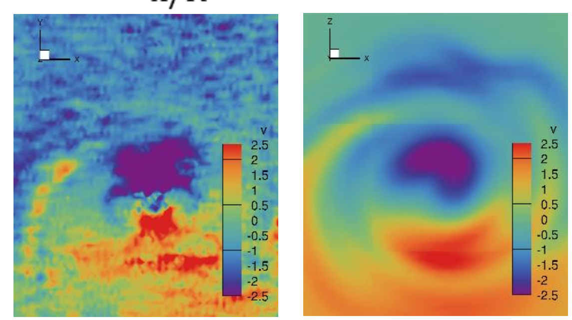
M. Cormier

Rotor & Wake Flow Physics Development of Tip Vortex Radius





- Excessive vortex diffusion for URANS
- Highly improved vortex preservation with DES



P. Weihing

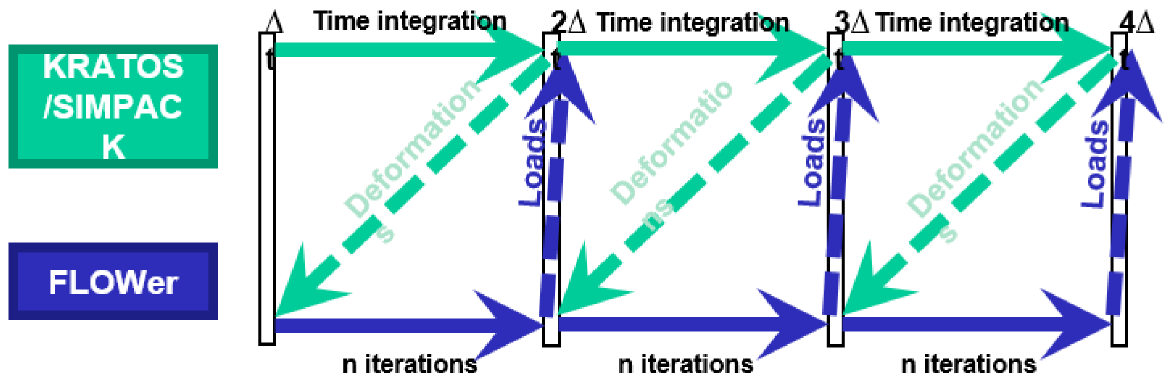
Th. Lutz, USTUTT

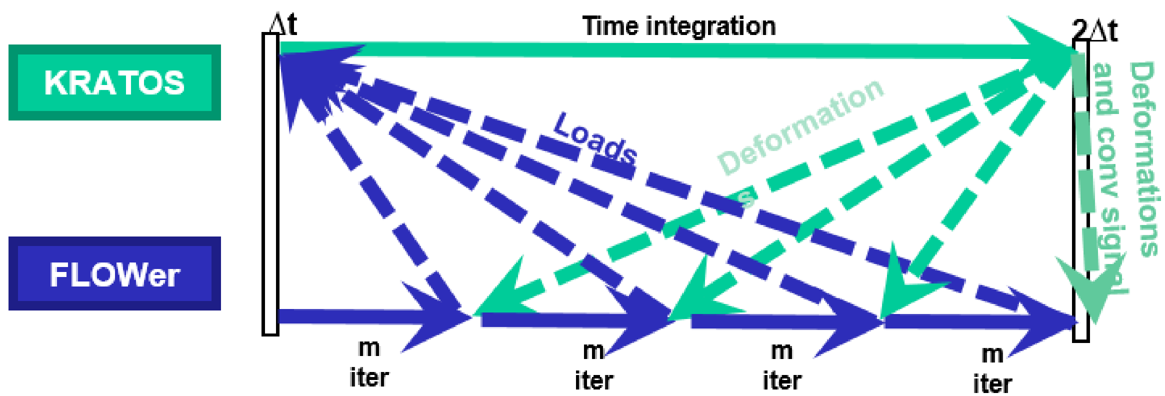
Aeroelasticity Fluid Structure Coupling Schemes

FLOWer is coupled to two different structural solvers:

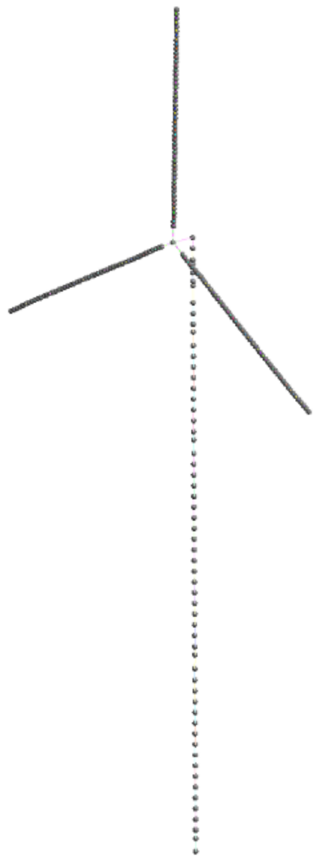
- MBD with beam elements in *SIMPACK*
- FEM with beam or full FE shell elements in *Kratos*

Both, implicit and explicit coupling schemes are possible

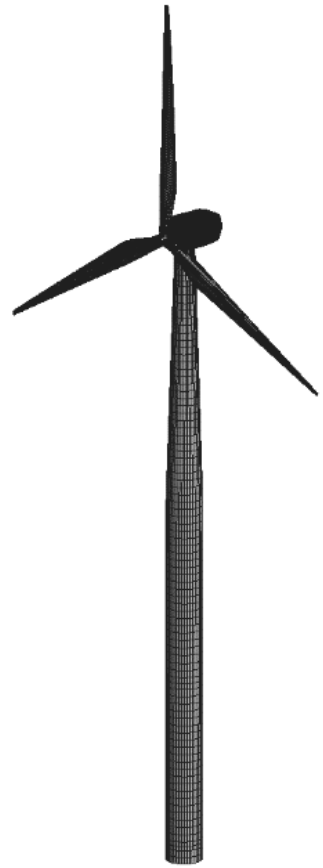




Aeroelasticity Structural Models in Kratos



- Easy to model
- Cross-Sectional properties
- Fast simulation
- Euler-Bernoulli & Timoshenko
- Non-linear
- 133 nodes for the blade



- Accurate description of local effects
- “Real“ material properties
- Non-linear
- Geometry of wet surface available for FSI
- 19215 nodes for the blade

Beam - Model

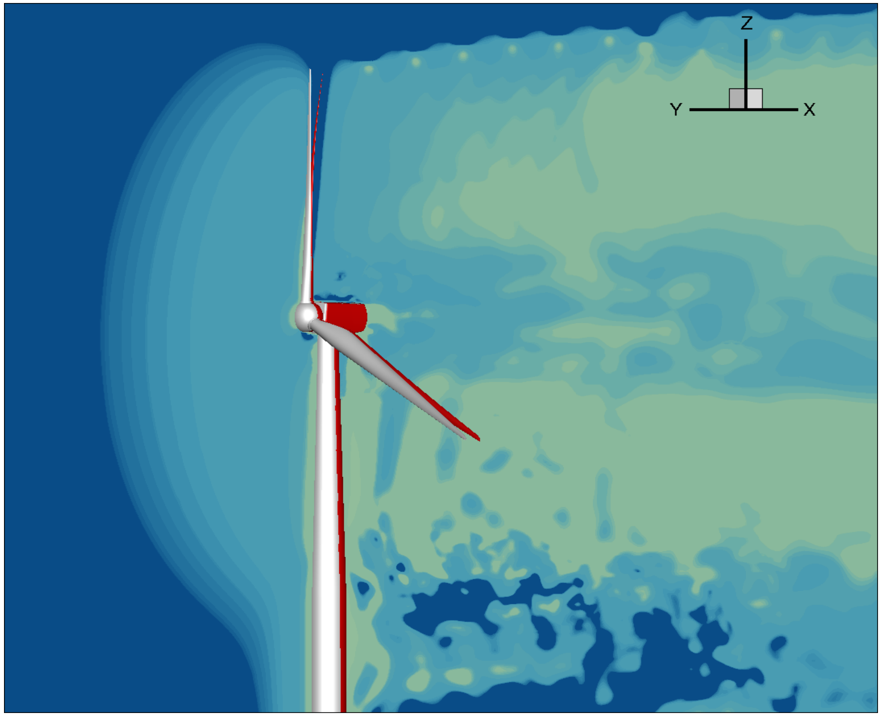
~ 10^3 DOFs

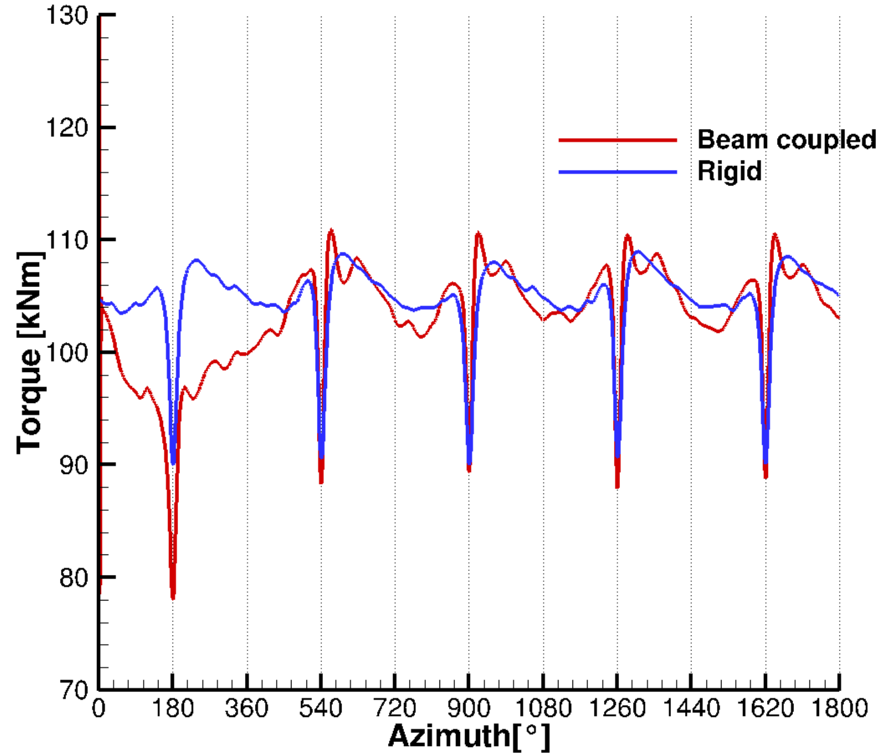
Shell - Model

~ 10^6 DOFs

G. Guma

Aeroelasticity Flexible Turbine in Uniform Inflow



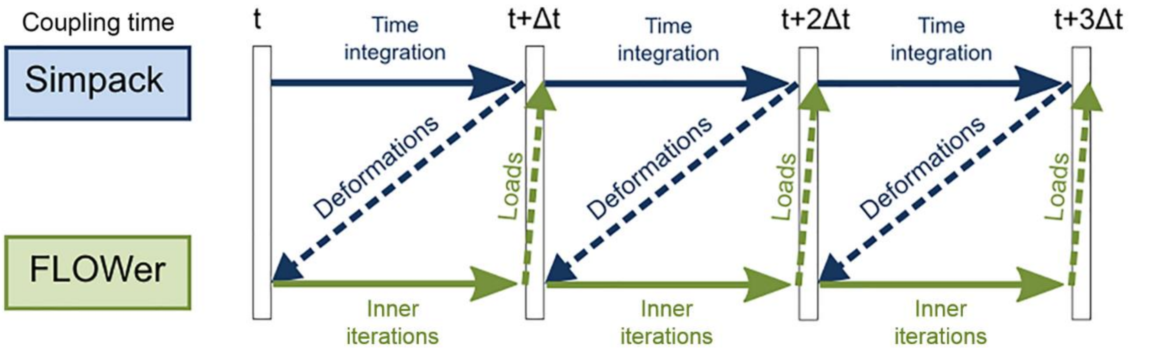


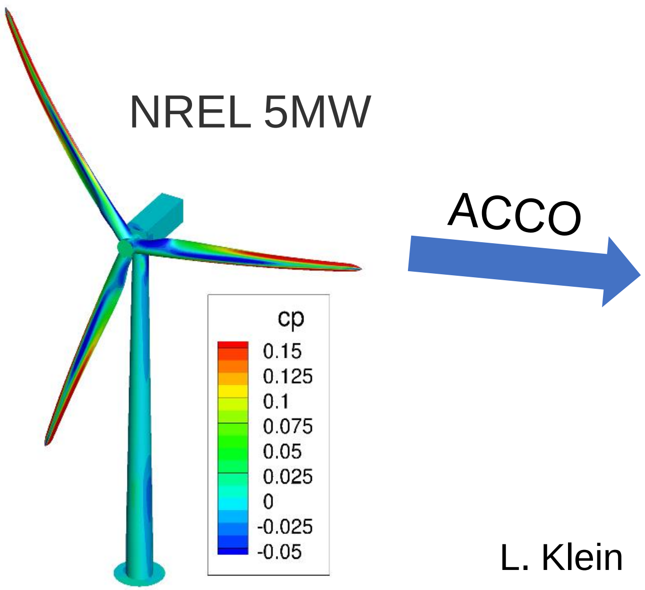
- Loads relaxation at the beginning to support convergence
- Power: approx. -0.5% with flexibility
- Max tip deformation: 6.6% radius (vs 5.8% if only a blade model)
- Tower top contributes with ~0.6% radius

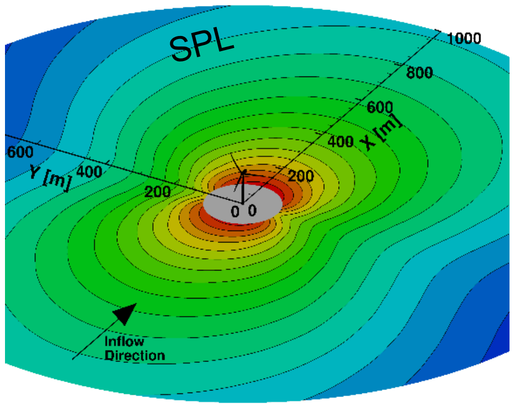
G. Guma

Low Frequency Noise CFD – MBS – FW-H Chain

- Explicit fluid-structure coupling
  - Non-linear beam deflection
  - Rigid body motion
  - Variable pitch / rotation speed
- Ffowcs-Williams Hawkings acoustic propagation
  - Use pressure fluctuations on turbine surface
  - Get acoustic pressure time series at discrete observer positions



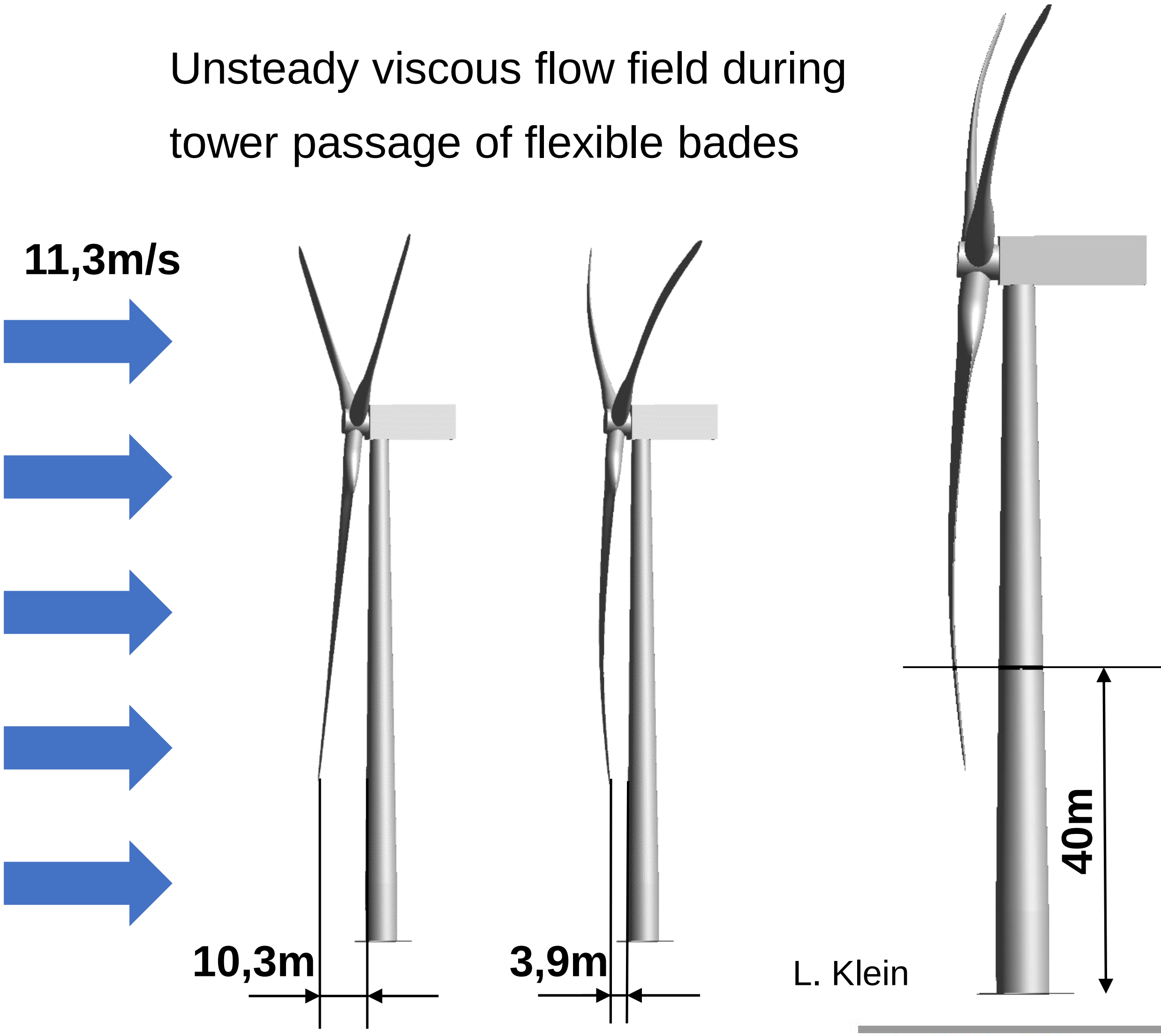




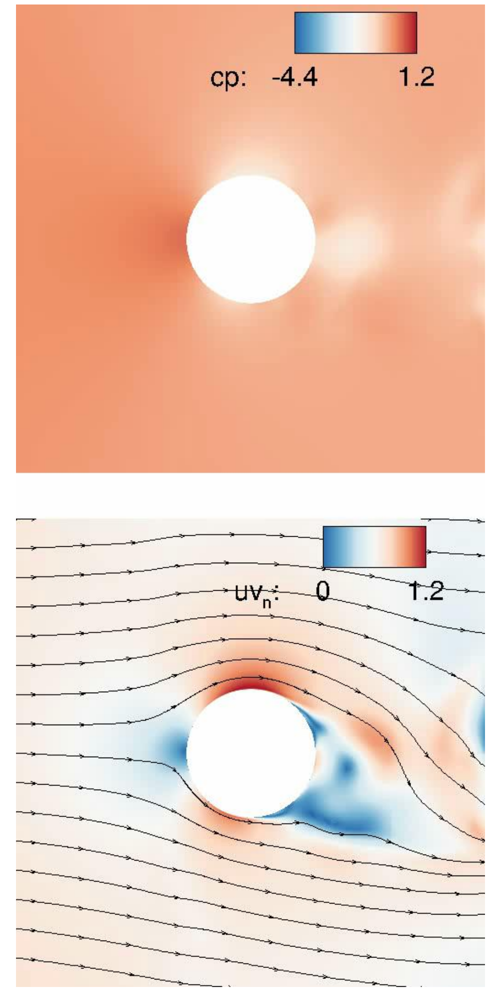
L. Klein

Low Frequency Noise Flexible Turbine in Uniform Inflow

Unsteady viscous flow field during tower passage of flexible bades



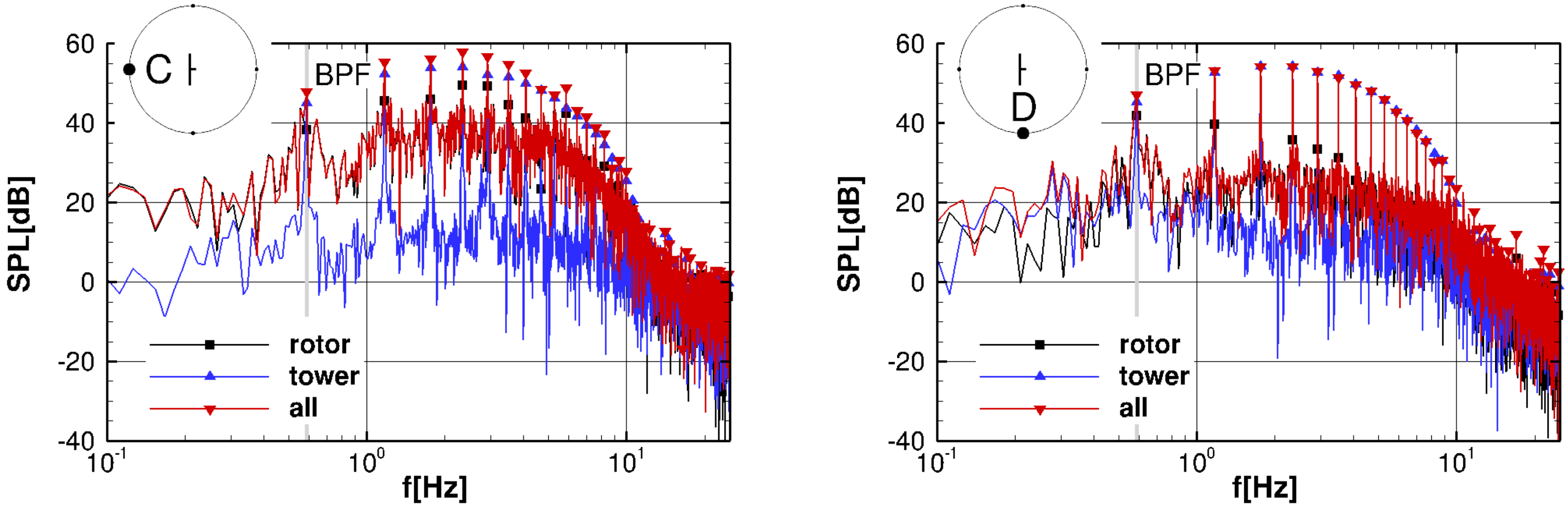
L. Klein



Th. Lutz, USTUTT

Low Frequency Noise Flexible Turbine in Uniform Inflow

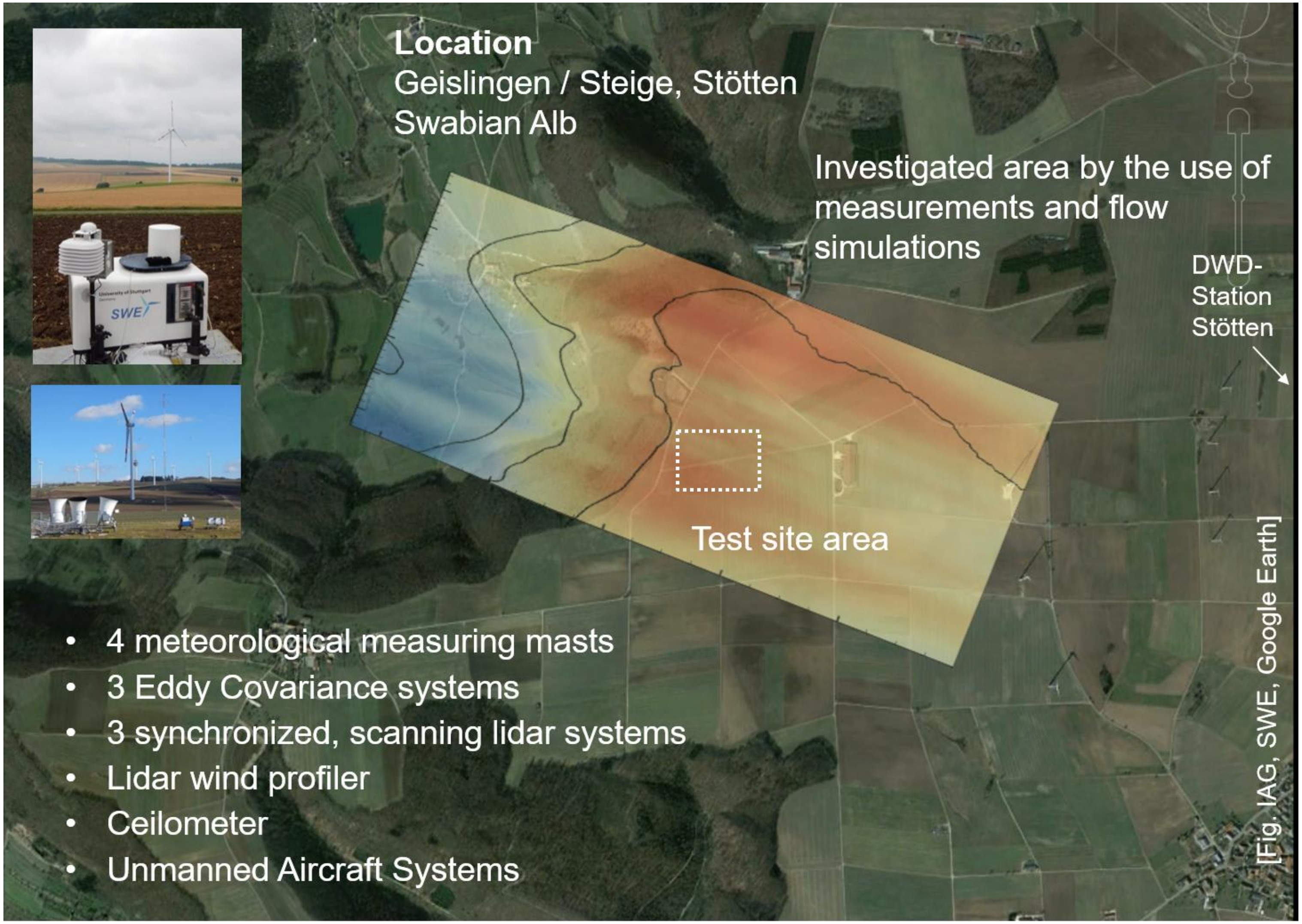
Rigid but statically deformed NREL 5MW turbine at uniform inflow



- **Rotor**: Emits less broadband noise to lateral direction than to for/aft and peaks at BPF harmonics are not very prominent
- **Tower**: Peaks at BPF harmonics have much higher amplitudes and are more prominent
- **Turbine**: Tower emits majority of low frequency sound

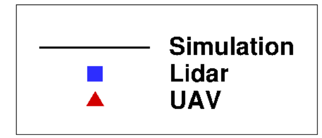
Source: Advanced CFD-MBS coupling to assess low-frequency emissions from wind turbines; L. Klein et al.; Wind Energy Science

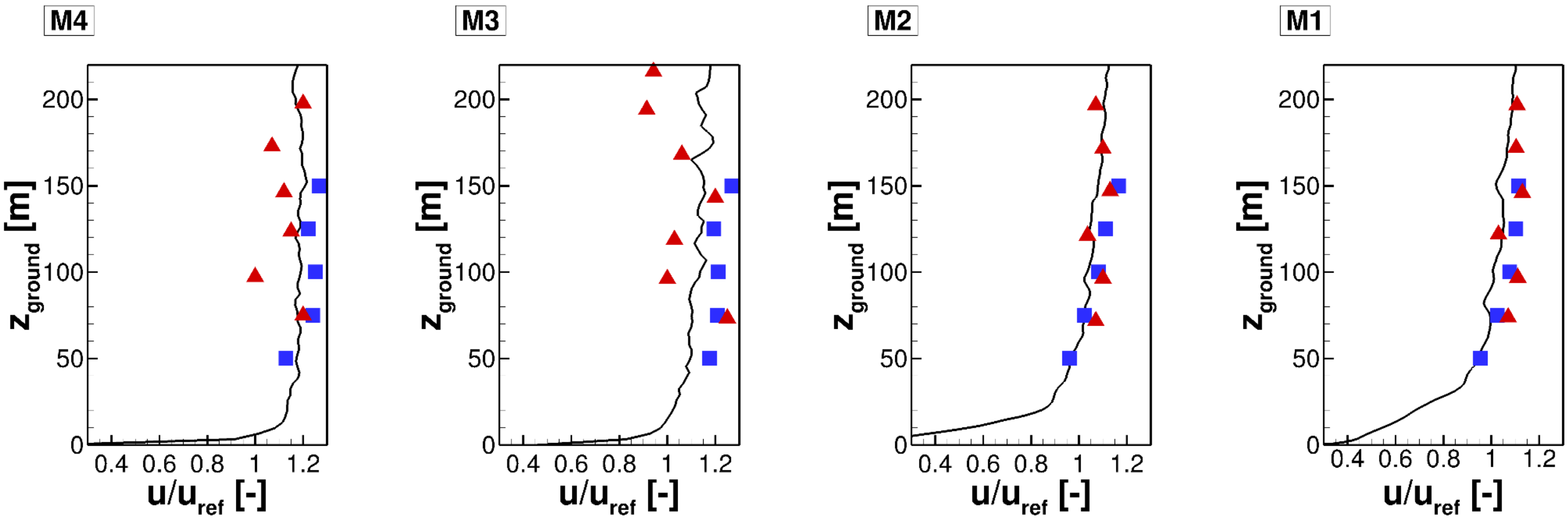
Complex Terrain WindForS Test Site (Under Construction)



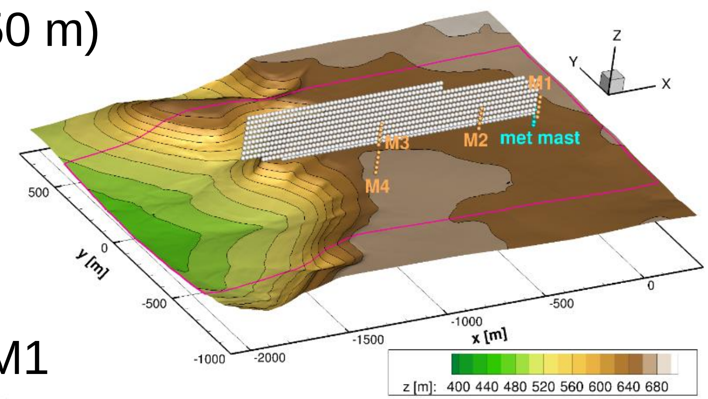
Complex Terrain Terrain Simulation Without Turbine

Predicted vs. Measured Wind Speed Profiles





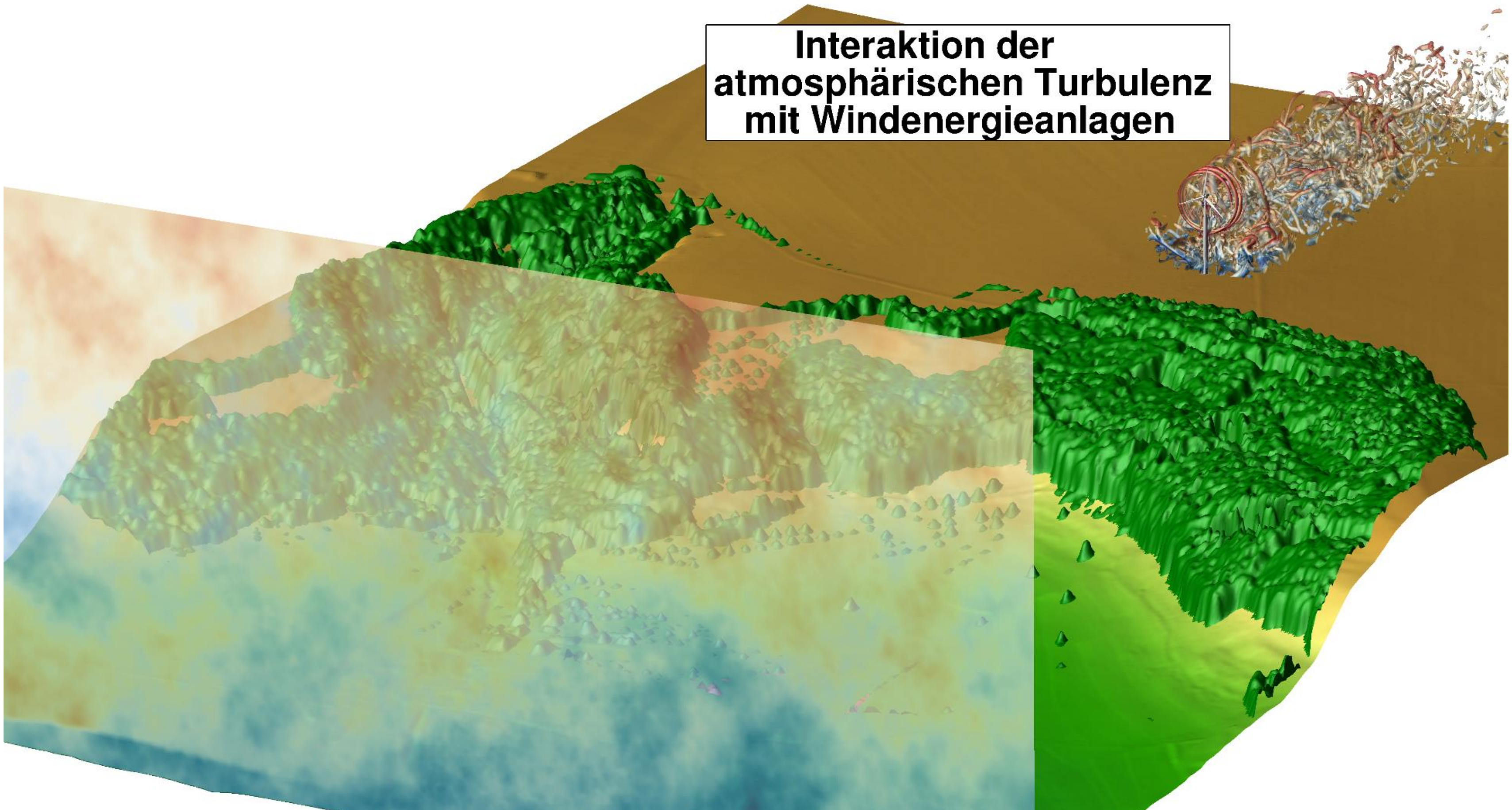
- Increase of velocity at the edge (factor 1.2 in 50 m)
- Recovery of profile downstream
- Lidar: good agreement with simulation
- UAV: deviations at M4 and M3
- UAV: zigzag shape at M4 and M3
- Good agreement between all data at M2 and M1



C. Schulz

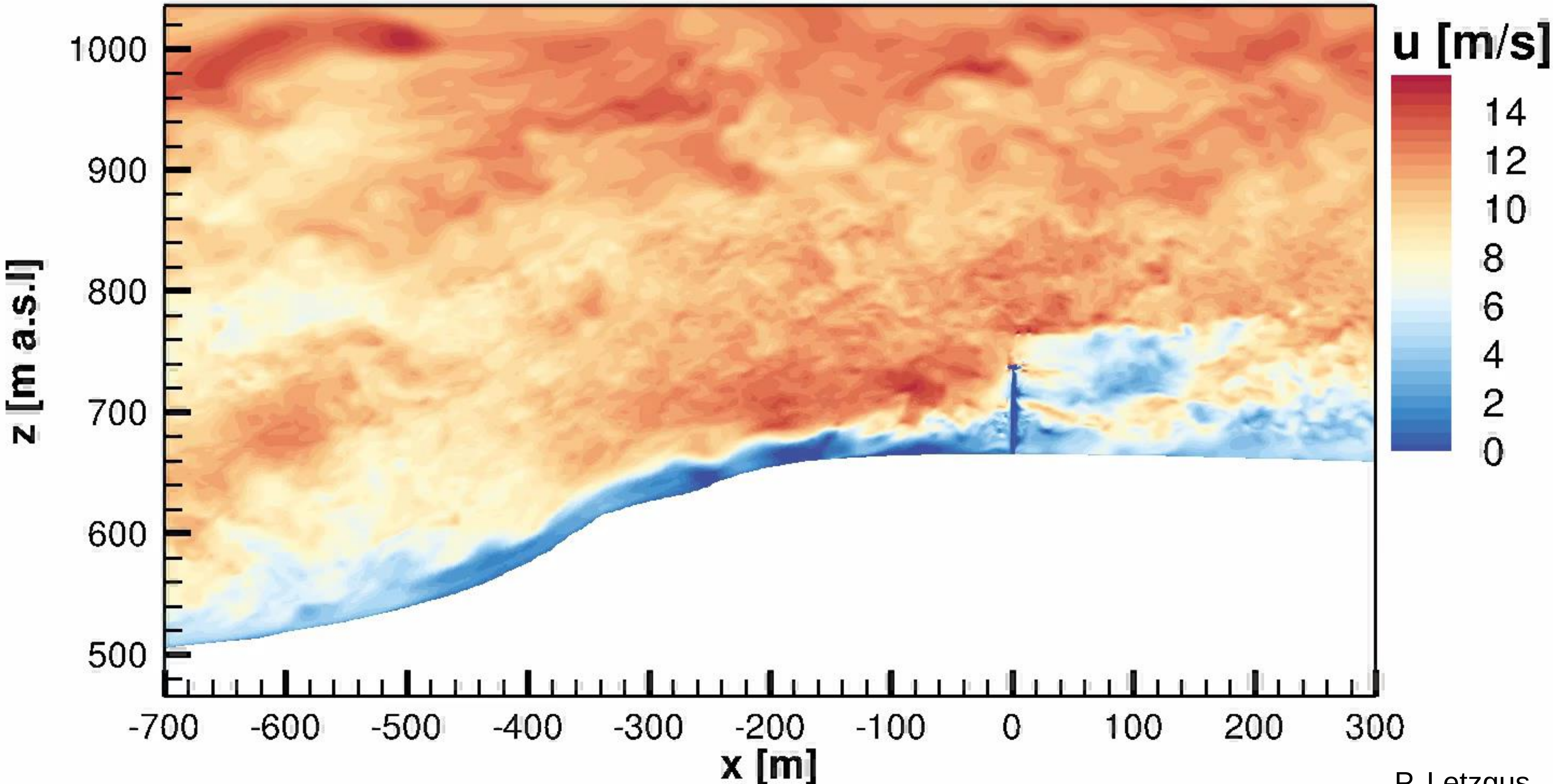
Th. Lutz, USTUTT

Complex Terrain Rigid Turbine in Complex Terrain With Forrest



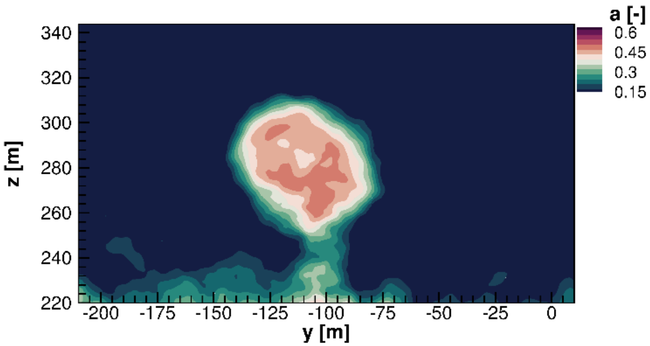
P. Letzgus

Complex Terrain Rigid Turbine in Complex Terrain With Forrest

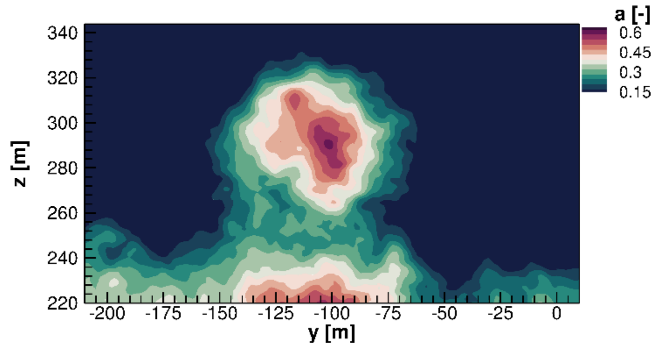


P. Letzgus

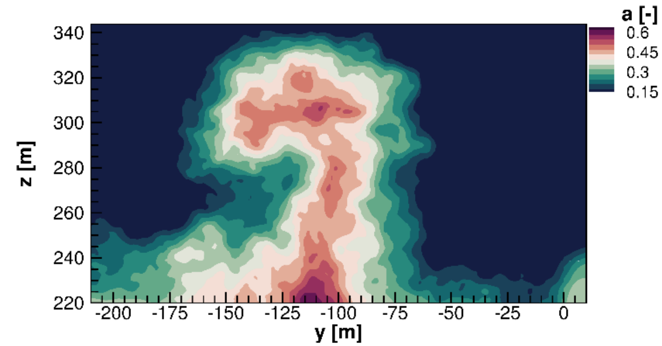
Complex Terrain Impact of Stratification on Turbine Wake



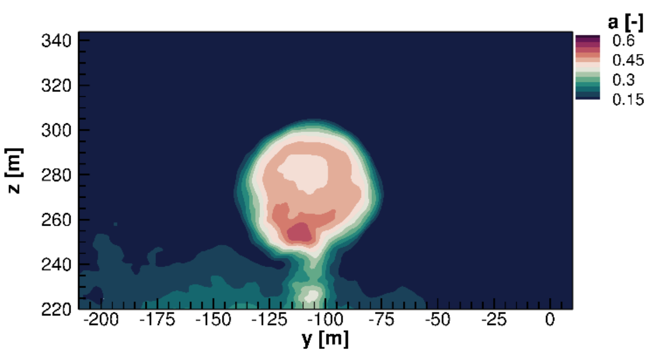
Unstable 2D



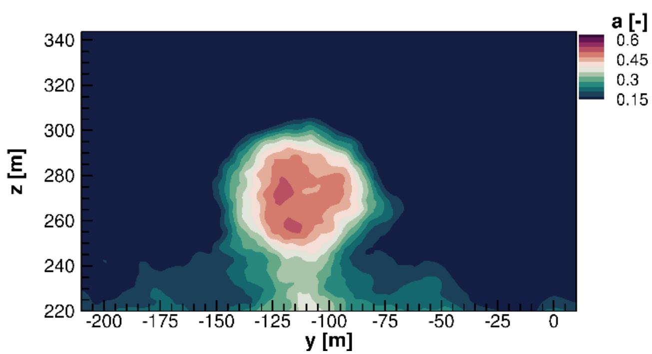
Unstable 4D



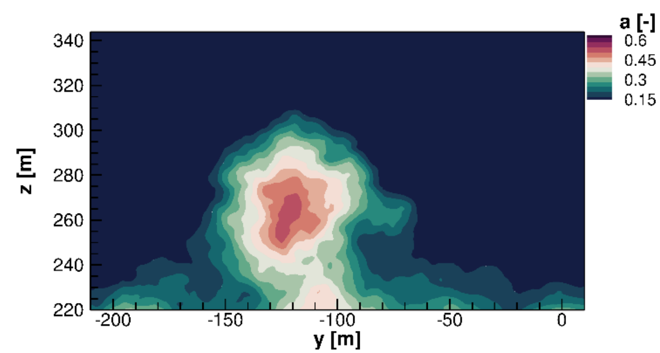
Unstable 6D



Stable 2D



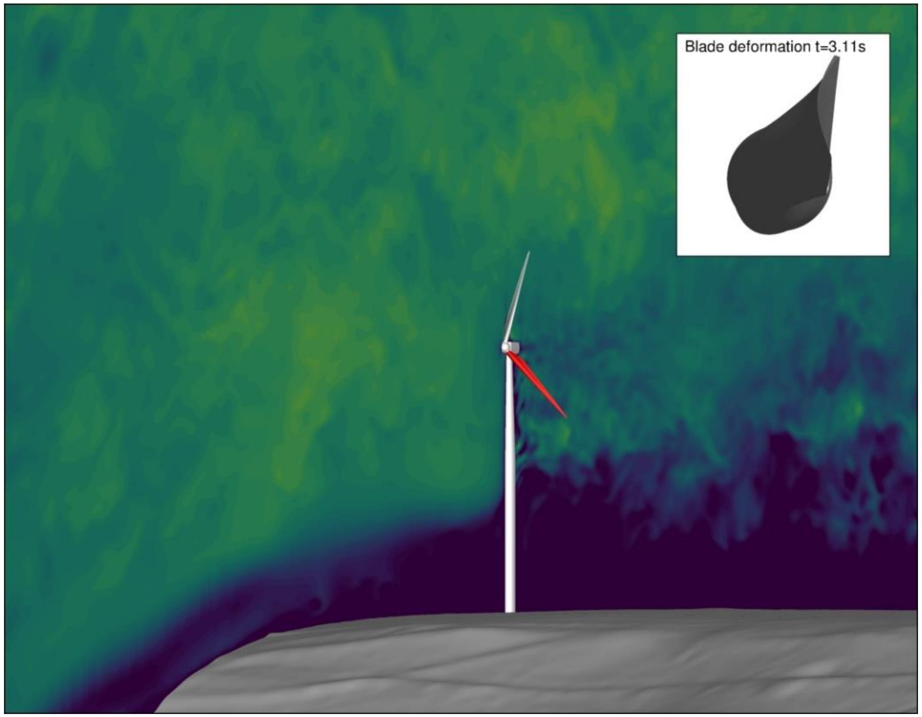
Stable 4D

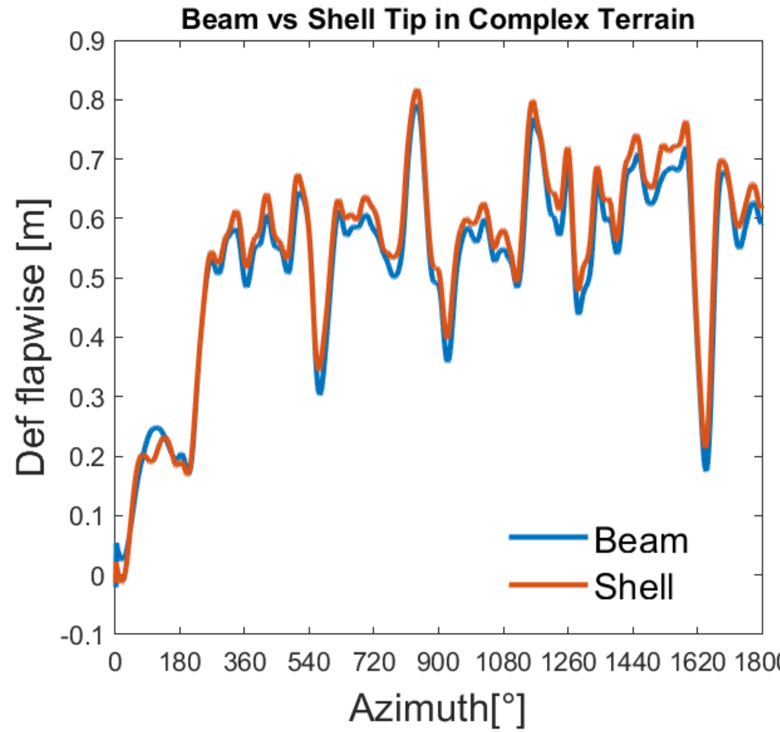


Stable 6D

P. Letzgus

Aeroelasticity Flexible Turbine in Atmospheric Inflow and Complex Terrain

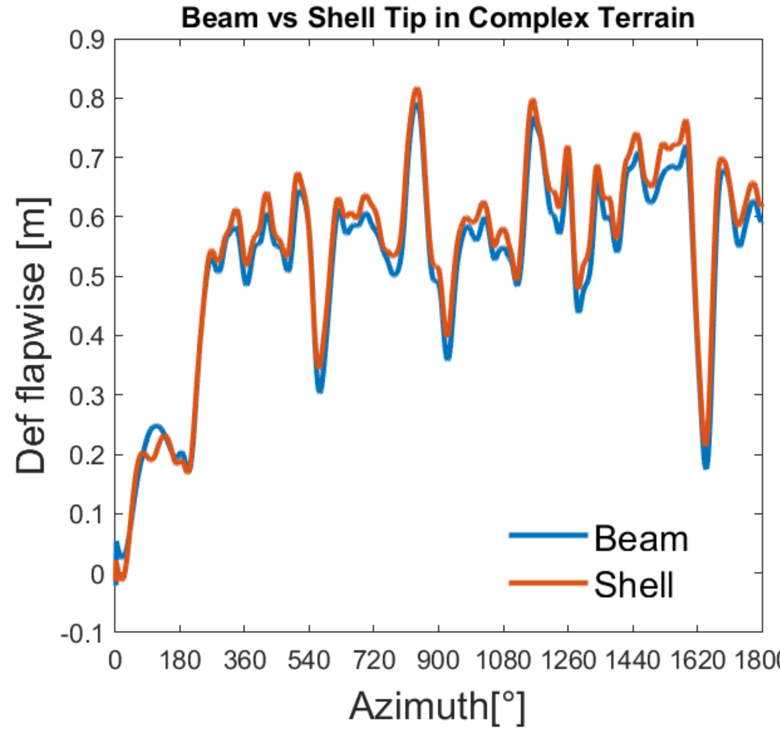




Tip deformation shows similar shape, but shell model predicts up to 5% more deformation. This is directly affecting Thrust and Torque

G. Guma / P. Bucher

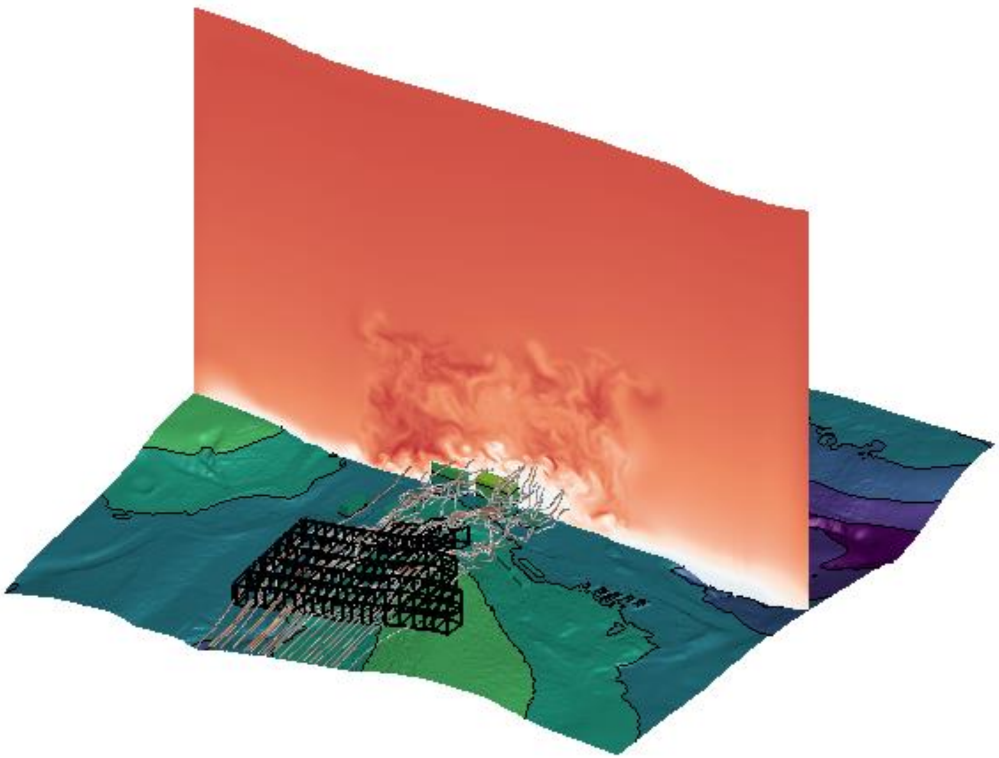
Aeroelasticity Flexible Turbine in Turbulent Inflow and Complex Terrain - *Video*

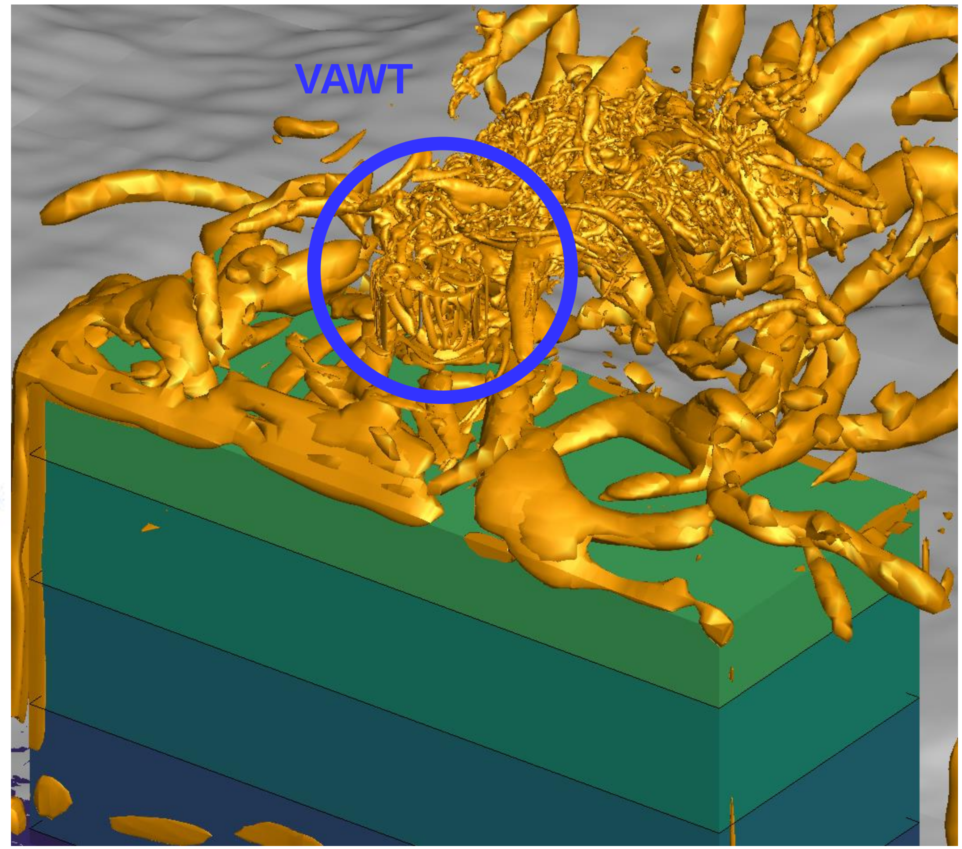


Tip deformation shows similar shape, but shell model predicts up to 5% more deformation. This is directly affecting Thrust and Torque

G. Guma / P. Bucher

Urban Terrain Forrested and Urban Terrain Simulation With VAWT





P. Zamre

Th. Lutz, USTUTT

University of Stuttgart
Germany
Germany

Thank you!

Thorsten Lutz

e-mail lutz@iag.uni-stuttgart.de
phone +49 (0) 711 685- 63406

University of Stuttgart
Institute of Aerodynamics and Gas Dynamics
Pfaffenwaldring 21, 70569 Stuttgart

Acknowledgement

Results shown in this presentation stem from the following national research projects funded by the German Federal Ministry for Economic Affairs and Energy (BMWi):

WINSENT, TREMAC, KonTest, ActiQuieter, IndianaWind, Schall_KoGe

WindForS
Windenergie Forschungscluster
Wind Energy Research Cluster