CFD Simulations of 2.5 MW turbine using ANSYS CFX and OpenFOAM

Bastian Dose[12], Wided Medjroubi[3] and Bernhard Stoevesandt[2]
[1] University of Applied Science Kiel
[2] Fraunhofer IWES, Oldenburg
[3] ForWIND, Oldenburg

First Symposium on OpenFOAM in Wind Energy 2013, March 21th, Oldenburg

FACHHOCHSCHULE KIEL
University of Applied Sciences

ForWind
Center for Wind Energy Research

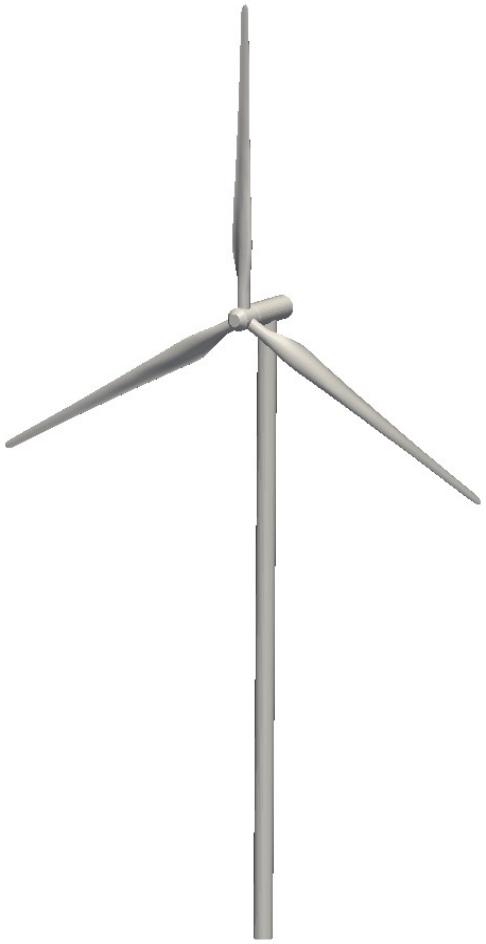
Fraunhofer
IWES

Outlook

- Complete wind turbine (incl. tower) was simulated
- Focus on comparison of ANSYS CFX and OpenFOAM
- Structured mesh generated in ANSYS ICEM CFD
- Steady-state and transient simulations

Turbine Data

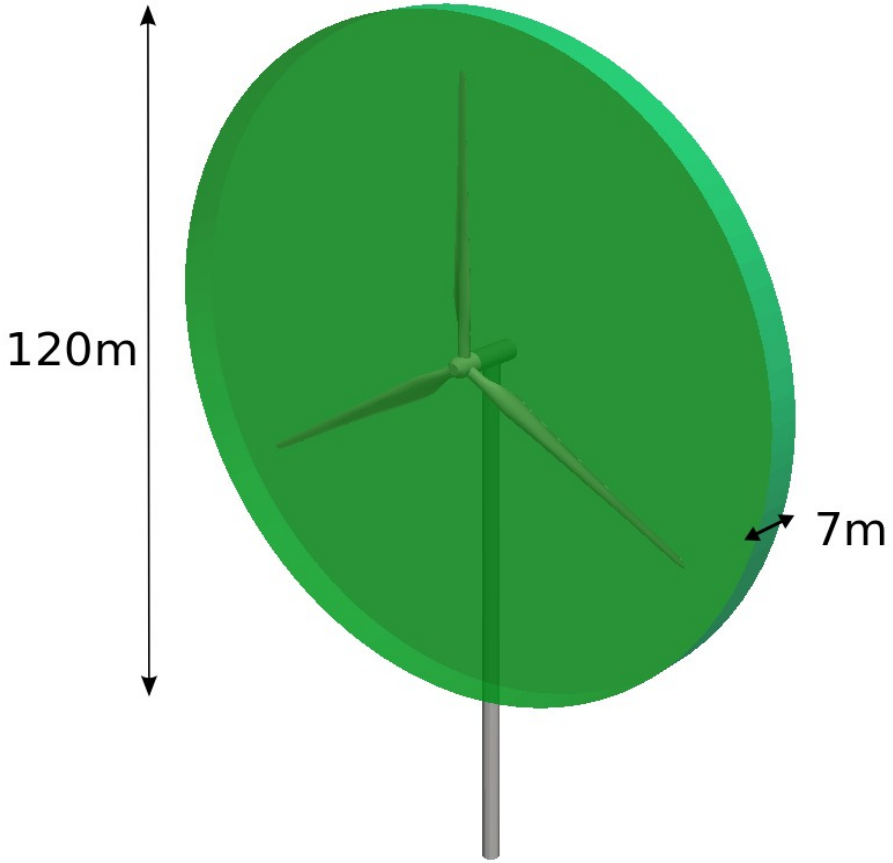
- Reference 2.5 MW wind turbine
- Designed by IWES
- Upwind configuration
- Rotor diameter: 100 m
- Hub height: 100 m
- Rated inflow velocity: 10.8 m/s
- Rated rotational speed: 13 rpm



Meshing strategy

- Two separate structured meshes (rotor and far field)
- Both simulations use the same mesh
- Total cell count: 52 million (36 + 16)
- Mesh quality verified by checkMesh
- Y+ 200

Meshing strategy

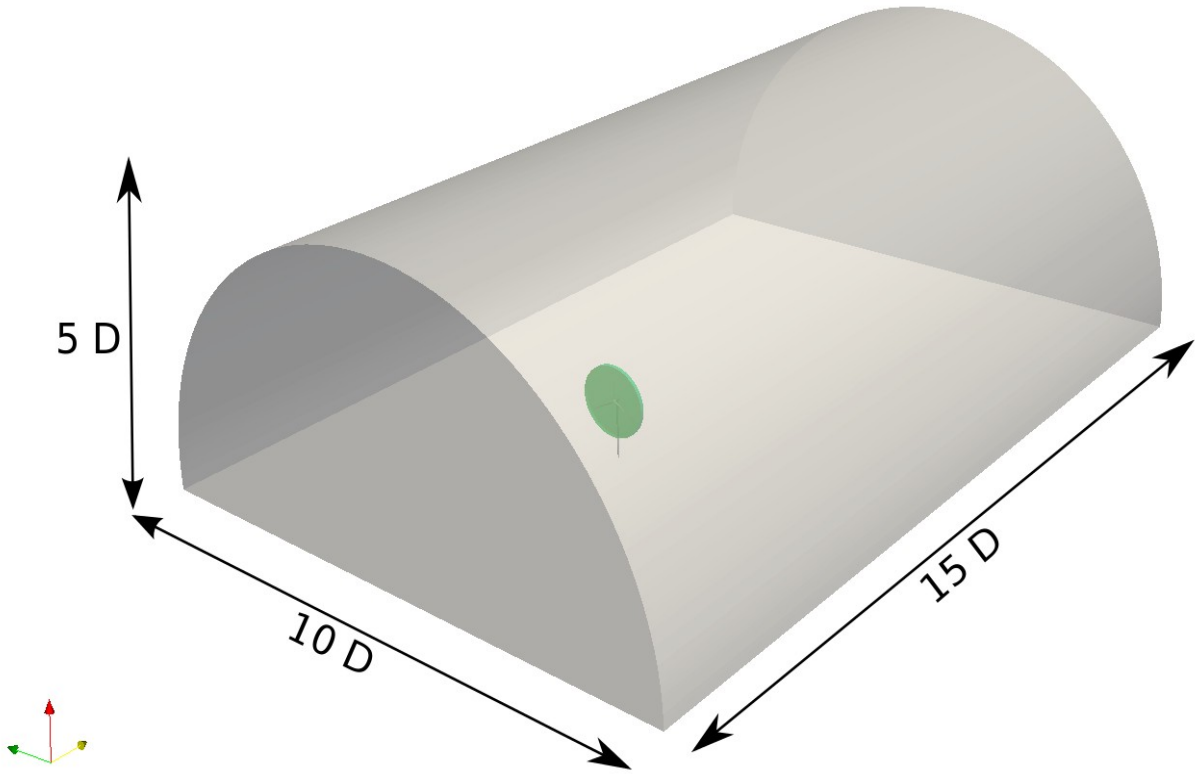


FACHHOCHSCHULE KIEL
University of Applied Sciences

ForWind
Center for Wind Energy Research

Fraunhofer
IWES

Meshing strategy

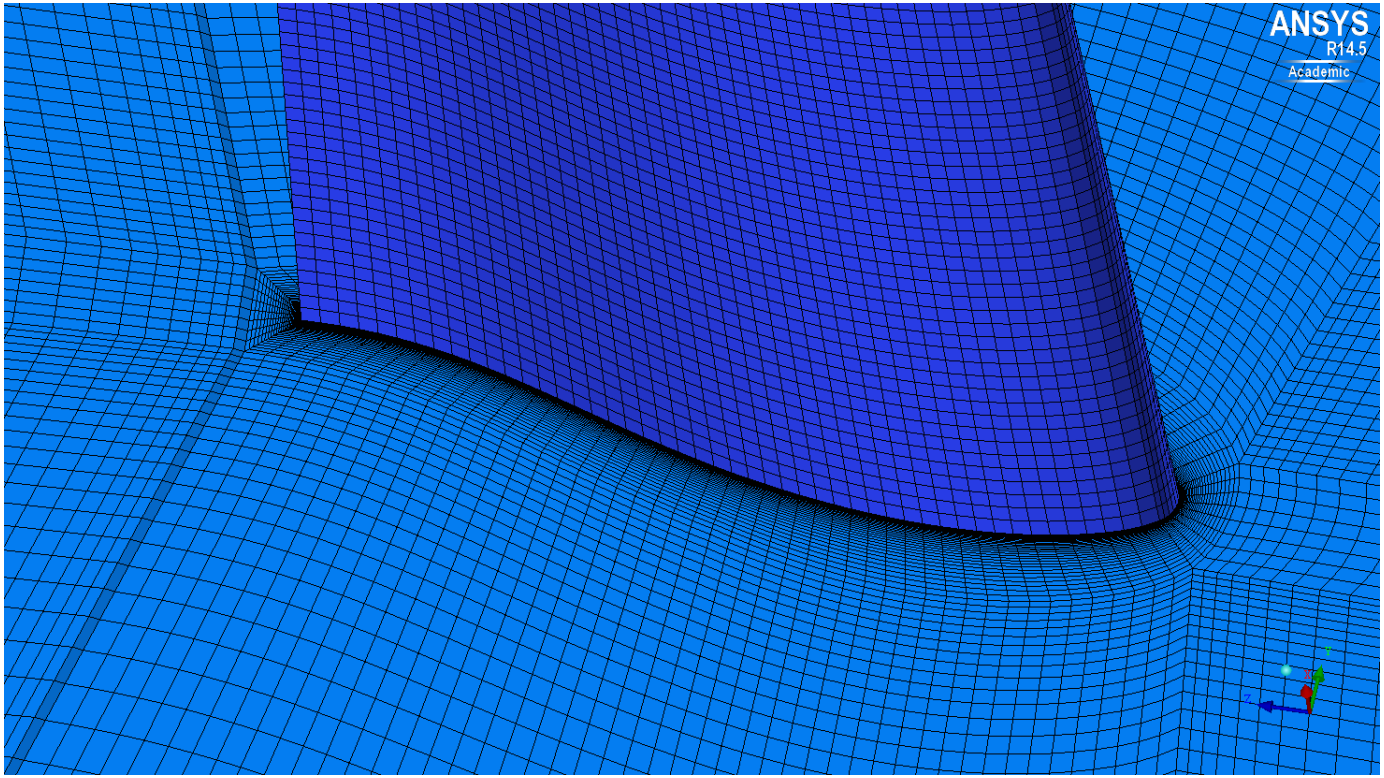


FACHHOCHSCHULE KIEL
University of Applied Sciences

ForWind
Center for Wind Energy Research

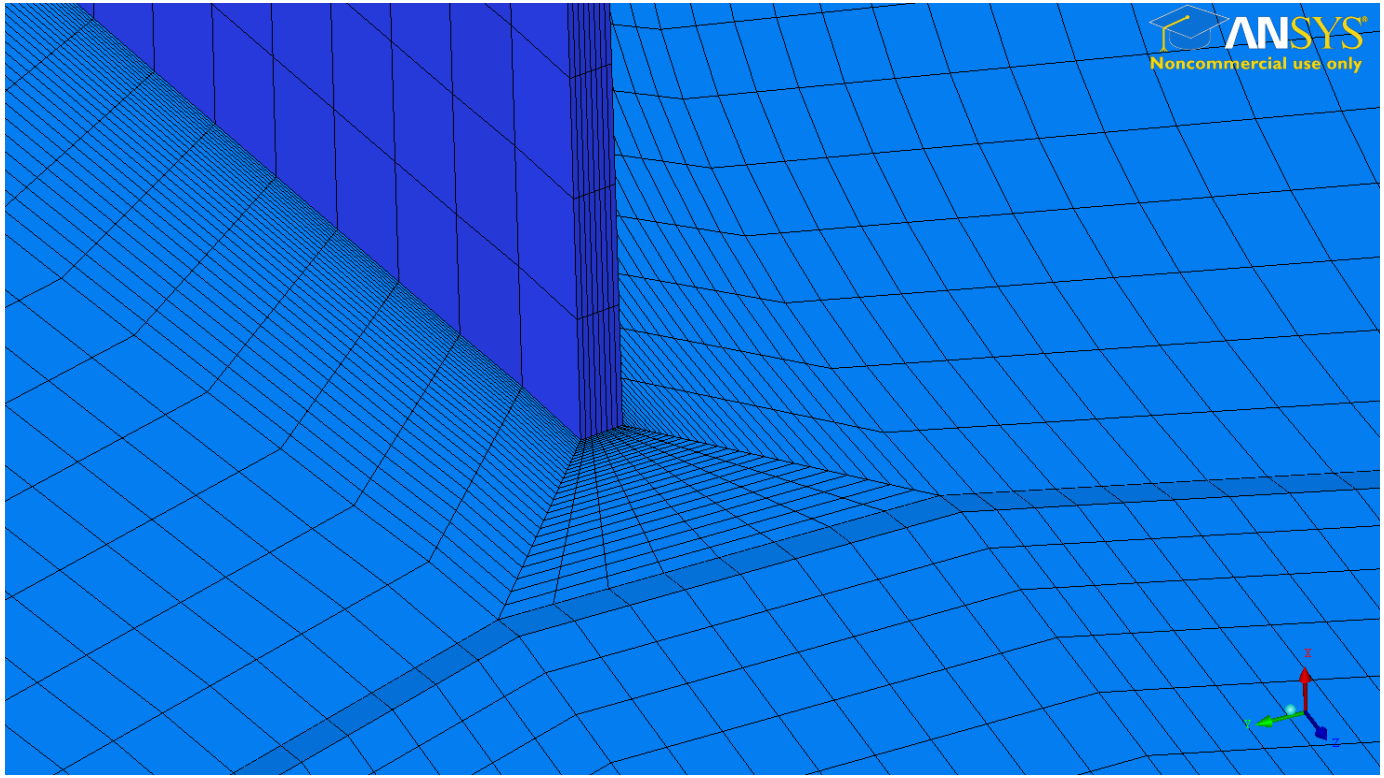
Fraunhofer
IWES

Meshing strategy



Chord length: 2m, r/R=0.5

Meshing strategy



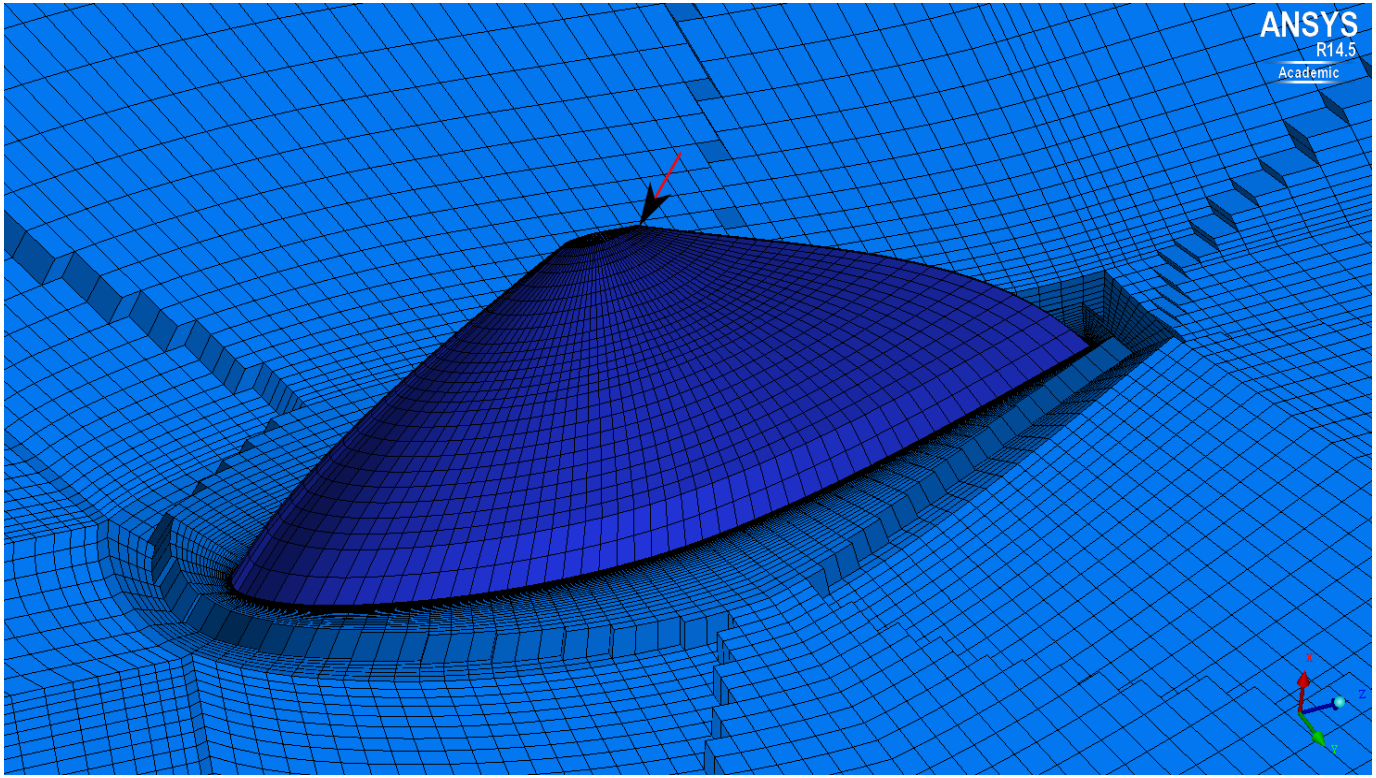
Chord length: 2m, r/R=0.5

FACHHOCHSCHULE KIEL
University of Applied Sciences

ForWind
Center for Wind Energy Research

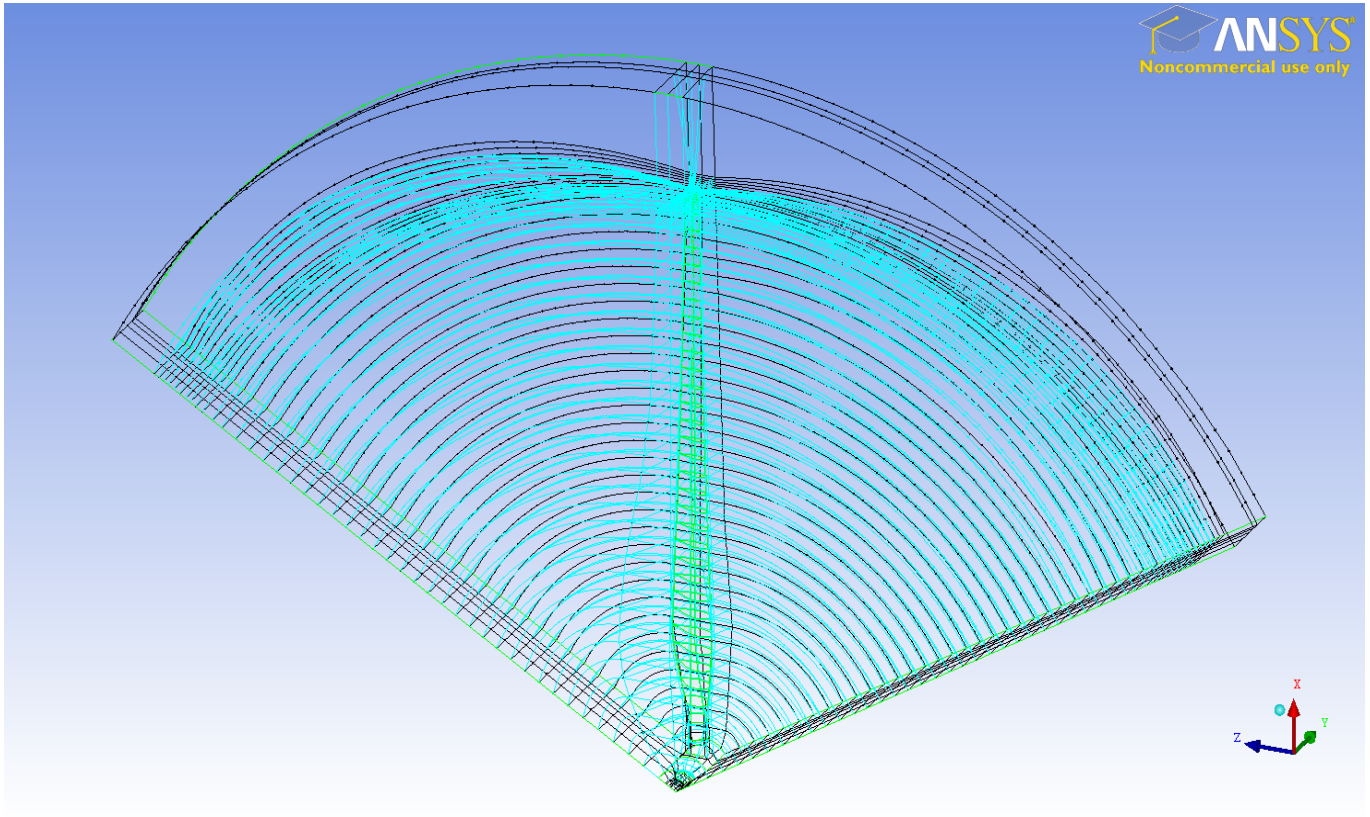
Fraunhofer
IWES

Meshing strategy



Tip chord length: 6cm

Meshing strategy



FACHHOCHSCHULE KIEL
University of Applied Sciences

ForWind
Center for Wind Energy Research

Fraunhofer
IWES

Set-up ANSYS CFX 14.5

- Steady-state (RANS)
- Frozen-Rotor concept
- GGI interfaces
- k-Omega SST turbulence model
- Standard solver settings

Set-up OpenFOAM 2.1.1

- Steady-state (RANS)
- Mesh import using fluent3DMeshToFoam
- MRF concept
- AMI interfaces
- k-Omega SST turbulence model
- MRFSimpleFoam [I. Herraez]

Preliminary Results

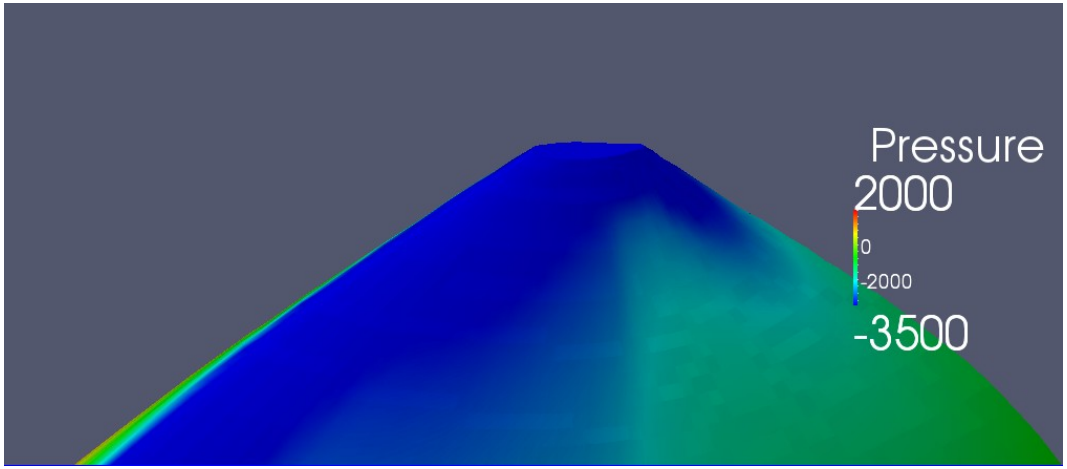
- Power output calculated based on pressure & viscous forces
- CFX-Post: torque function, OpenFOAM: turboperformance lib
- Calculated aerodynamic wind turbine power output:

  ANSYS CFX 14.5: 2.045 MW

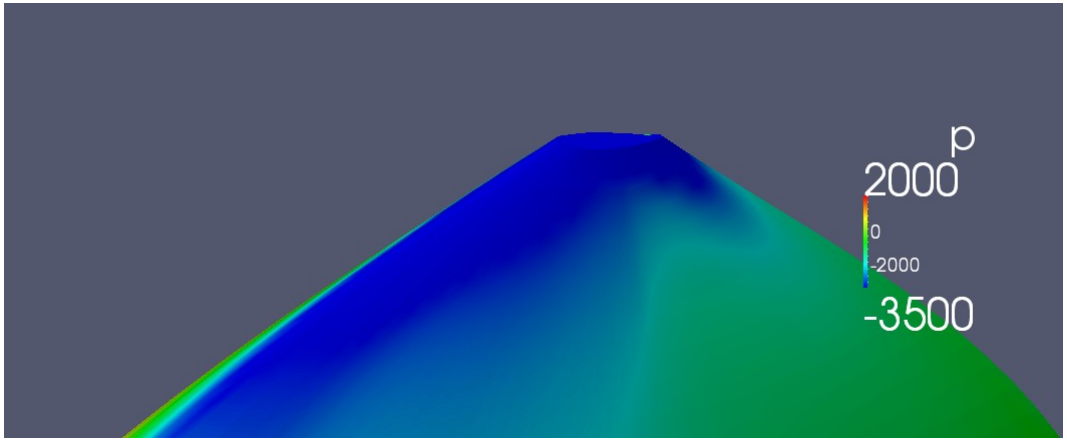
  OpenFOAM 2.1.1: 1.98 MW

Preliminary Results

ANSYS CFX



OpenFOAM



FACHHOCHSCHULE KIEL
University of Applied Sciences

ForWind
Center for Wind Energy Research

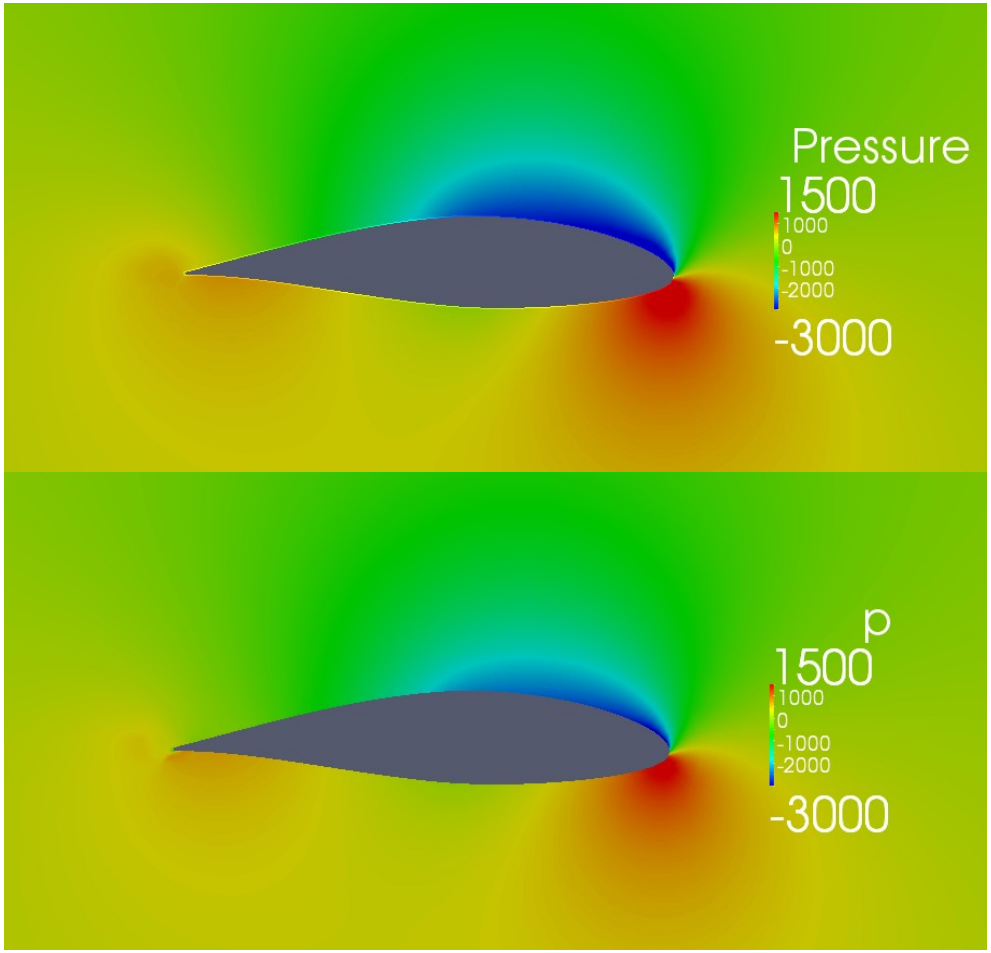
Fraunhofer
IWES

Preliminary Results

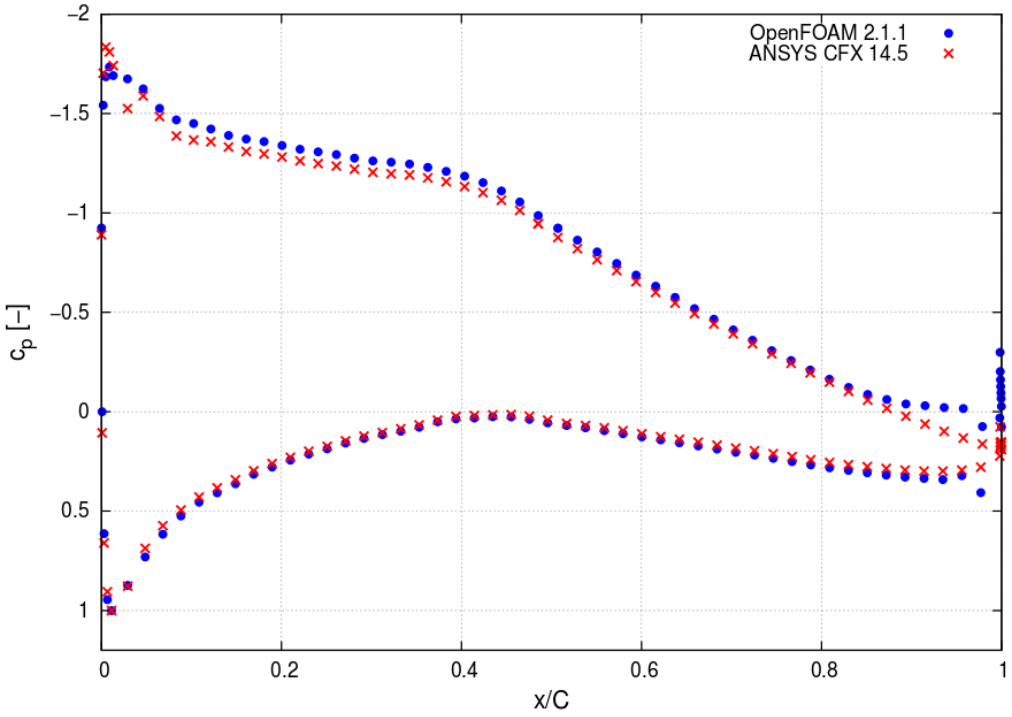
ANSYS CFX

OpenFOAM

r/R=0.9 (45m)



Preliminary Results



r/R=0.9 (45m)

Conclusion

- Structured meshes generated in ICEM can be used within OF
- Meshing approach used worked
- Similar results with both ANSYS CFX and OpenFOAM
- CFX faster (3-4x) and more stable

Thank you for your attention!

bastian.dose@iwes.fraunhofer.de

FACHHOCHSCHULE KIEL
University of Applied Sciences

ForWind
Center for Wind Energy Research

Fraunhofer
IWES