Influence of jet flow on the aerodynamics of a floating model wind turbine

Levin Klein, Christoph Schulz, Thorsten Lutz, Ewald Krämer
Institute of Aerodynamics and Gas Dynamics
University of Stuttgart
Germany
klein@iag.uni-stuttgart.de

ISOPE-2016 Conference, Rhodes, Greece
The 26th International Ocean and Polar Engineering Conference
Rhodes (Rodos), Greece, June 26–July 1, 2016: www.isope.org;

The research leading to these results has received funding from the European Community’s Seventh Framework Programme under grant agreement No. 308974 (INNWIND.EU).
Overview

• Introduction
• Approach
• Results
• Summary and conclusion
Introduction – research project

INNWIND.EU Project (www.Innwind.eu)

“high performance innovative design of a beyond-state-of-the-art 10-20MW offshore wind turbine”

Including:

• design of platforms for offshore installation
• validation of different simulation tools by model tests in wave tank

The research leading to these results has received funding from the European Community’s Seventh Framework Programme under grant agreement No. 308974 (INNWIND.EU).
Introduction – experimental setup

The research leading to these results has received funding from the European Community’s Seventh Framework Programme under grant agreement No. 308974 (INNWIND.EU).

ISOPE-2016
Rhodes

scale: 1/60

wind generator outlet
model turbine
wave basin
floating platform
Introduction – main task

Wind generator outlet size ~ rotor diameter

Simulations:
- Most aerodynamic models (like BEM) can’t consider jet flow and use uniform inflow conditions
- Approach needed that can consider jet flow as well as uniform inflow

→ CFD
Main function:

- provide realistic aerodynamic forces → thrust force

How?

- Froude scaled (1/60) + new design
- low Re airfoil (Re~45,000)
- matches thrust coefficient of full size turbine
- lower power coefficient
Approach – CFD solver

FLOWer by DLR (German Aerospace Center)
• block structured finite volume solver
• 2\textsuperscript{nd} order in time and space
• chimera technique (overlapping grids)
• extended at IAG for wind turbine simulation
  – atmospheric turbulence and complex terrain
  – load control (e.g. flaps)
  – fluid structure coupling
• experimental validation (e.g. MEXICO project) and code to code validation (e.g. AVATAR project)
Approach – CFD setup

Turbine mesh – overlapping meshes
Approach – CFD setup

uniform inflow (29M cells)  jet flow (35M cells)

only sections of grids shown
Approach – CFD setup

- rated wind speed
  1.48 m/s corr. to 11.4 m/s
- 70.9 rpm
- fixed turbine
  (no floating motion)
- fully turbulent
- time step ~ 0.75° azimuth
- prescribed velocity at wind generator outlet
Approach – CFD setup

- rated wind speed
  1.48 m/s corr. to 11.4 m/s
- 70.9 rpm
- fixed turbine
  (no floating motion)
- fully turbulent
- time step ~ 0.75° azimuth
- prescribed velocity at wind generator outlet
Results – flow field

uniform inflow

jet flow

induction factor: 27.2%

induction factor: 27.8%
Results – flow field

**uniform inflow**

induction factor: 27.2%

**jet flow**

induction factor: 27.8%
Results – velocity in rotor plane

uniform inflow

jet flow

induction factor: 27.2%

induction factor: 27.8%
Results

velocity difference

+ = higher in jet case

spanwise thrust at r/R=90%

The research leading to these results has received funding from the European Community’s Seventh Framework Programme under grant agreement No. 308974 (INNWIND.EU).
Results – Thrust

uniform inflow  
jet flow

Thrust [N/m]

Azimuth [°]

Radius [m]
Results – Thrust

**blade thrust**

**Rotor thrust**

Average Thrust 0.7% lower for jet case
Results – Thrust FFT

Blade thrust

Rotor thrust

The research leading to these results has received funding from the European Community’s Seventh Framework Programme under grant agreement No. 308974 (INNWIND.EU).
Summary and conclusion

- CFD simulation of model wind turbine in uniform inflow and jet flow
- rotor thrust comparable, blade thrust different

Is uniform inflow approach suitable?
Yes but, …

- only if turbine is placed in the center of the jet (and at this position)
- influence might be higher when turbine is moving (floating motion)
- real jet might be less perfect than simulated one (inhomogeneous, turbulence)
Thank you!

Questions?