Development of a System to Optimize Wind Farm Layout

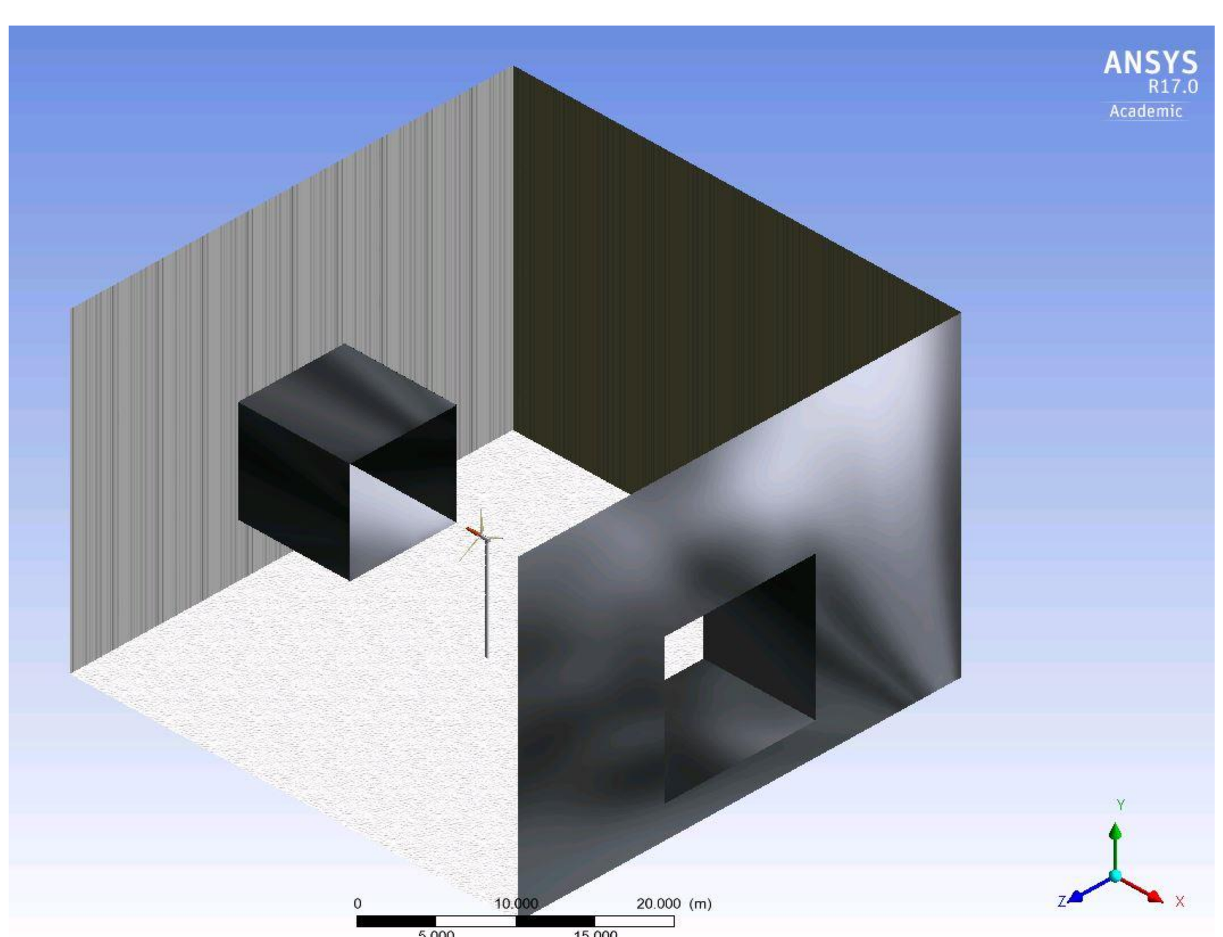
Rafael Valotta Rodrigues/ Corinne Lengsfeld
University of Denver / Mechanical and Materials Engineering Department

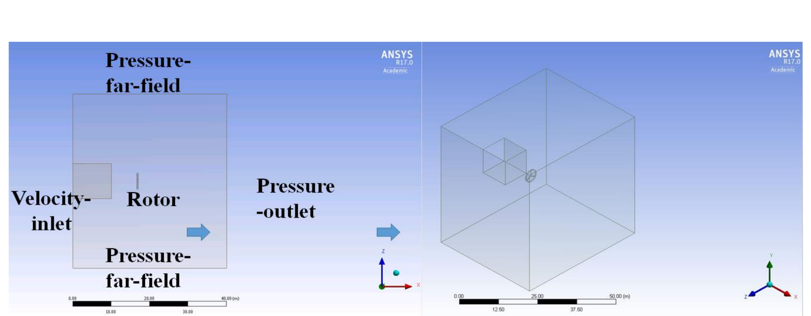
Introduction

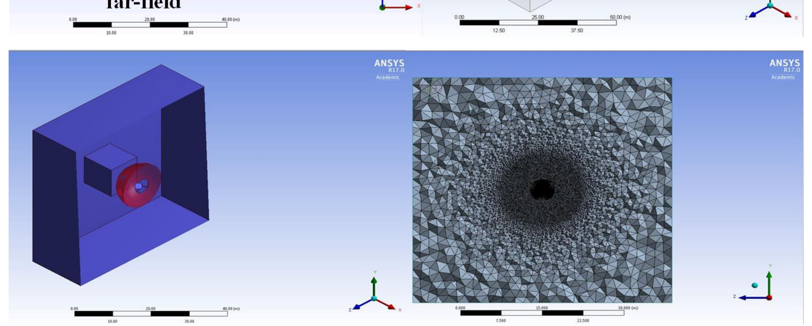
This work attempts to describe the validation of a wind turbine farm CFD (Computational Fluid Dynamics) simulation using velocity wake data from previous experiment from literature. Ultimately, optimization routines are going to be implemented using the mathematical solver MATLAB and the CFD solver ANSYS Fluent. This initial work is to validate the simulation and identify parameters influencing the wake or eliminate variables that have little effect in order to set up a robust optimization. The primary goal is to develop a robust computational model capable of simulating specific operational conditions of commercial wind turbines. Additionally, the optimization routine attempts to optimize the layout of commercial wind farms with regards to output power/components fatigue lifetime and footprint extension.

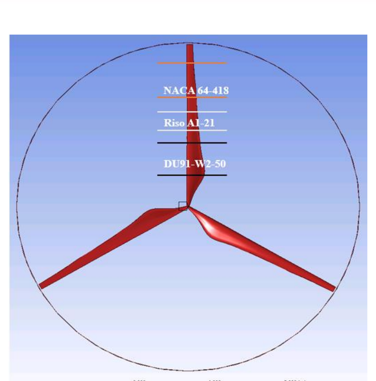
Computational Fluid Dynamics Model

- CFD solver: ANSYS Fluent 17, two server-computers with 64GB RAM/ 8 processers. Moving Reference Frame approach. Turbulence model: k-ω SST (suitable for swirl flow).
- Computational model validation: performed by plotting the velocity in the wake region of the blade and directly comparing it with experimental data from literature (MEXICO Experiment).



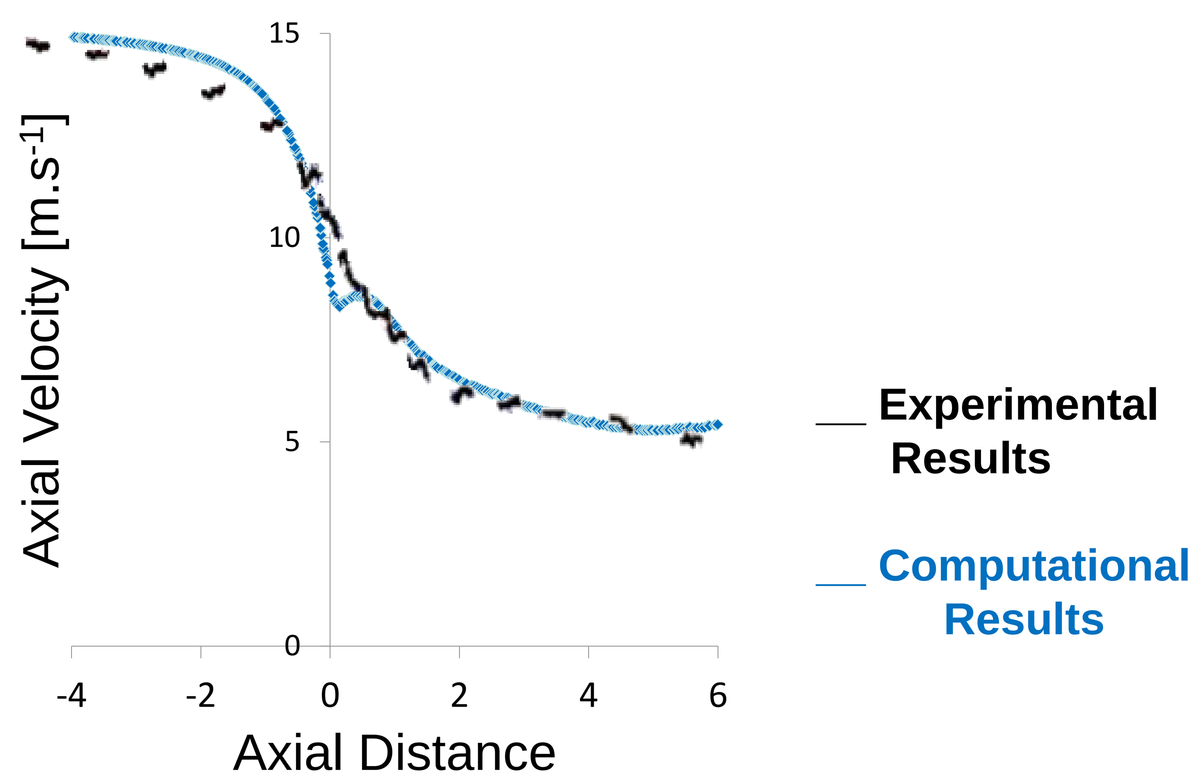


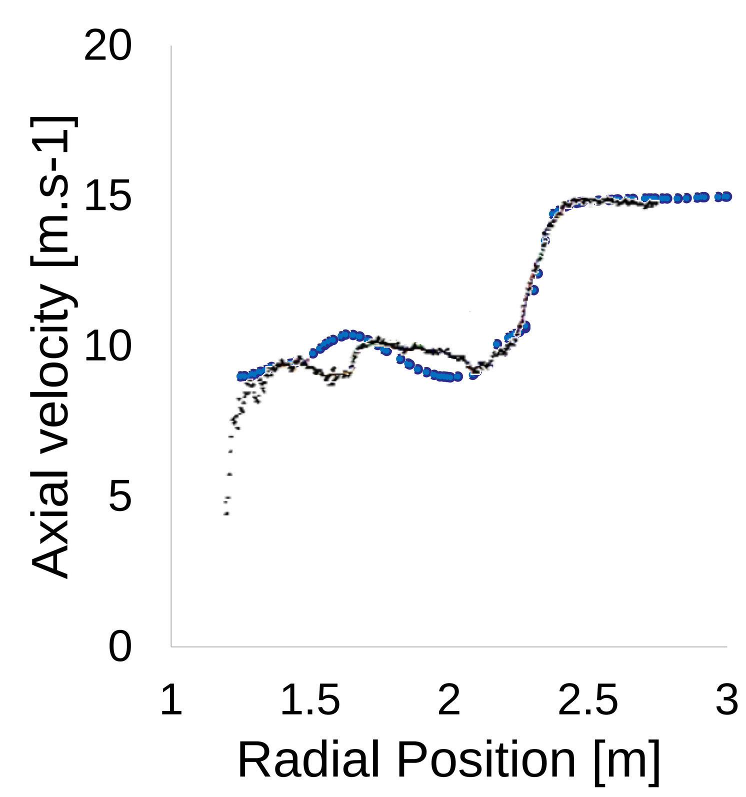




MEXICO Rotor: 3-bladed (twisted) 4.5m small wind turbine)

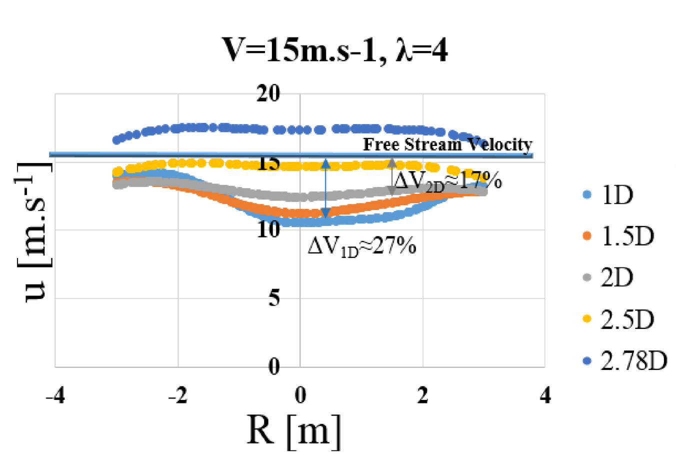
Validation

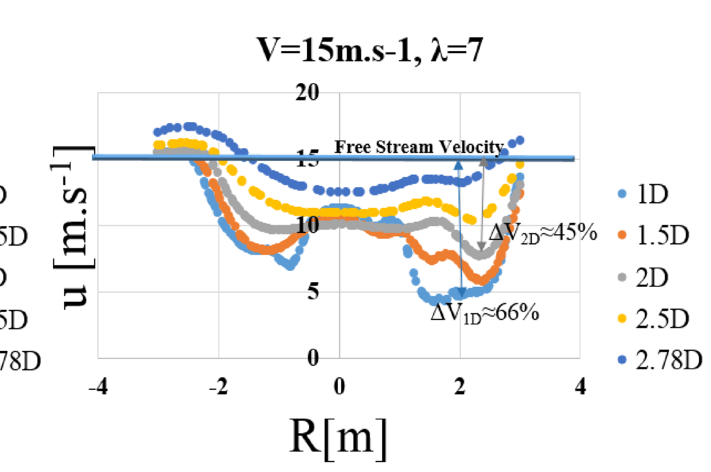


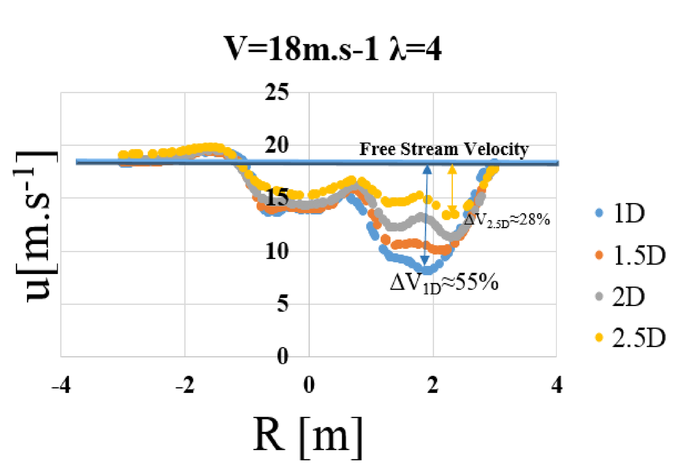


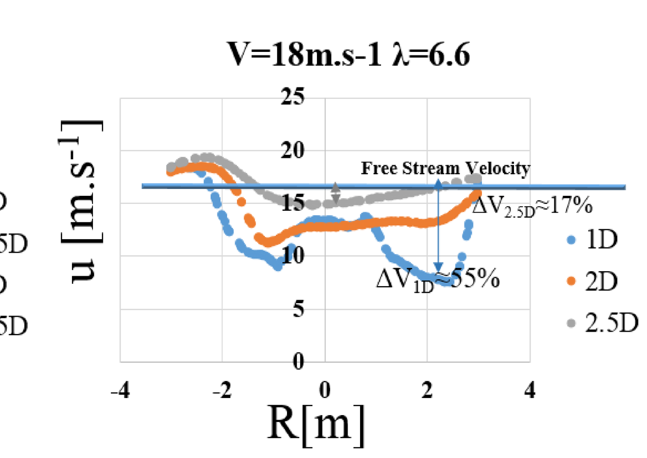
Results

Tip Speed Ratio (λ) effect on the Near Wake

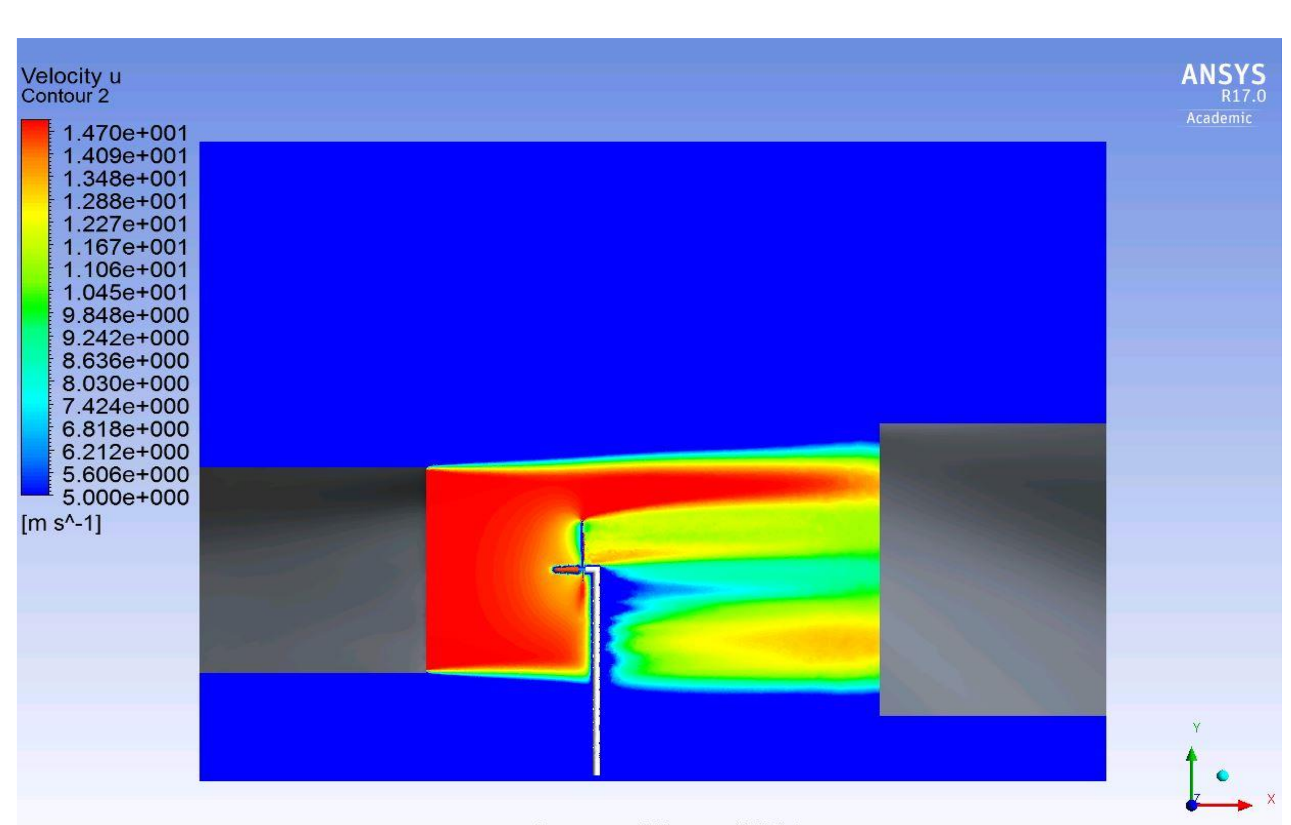






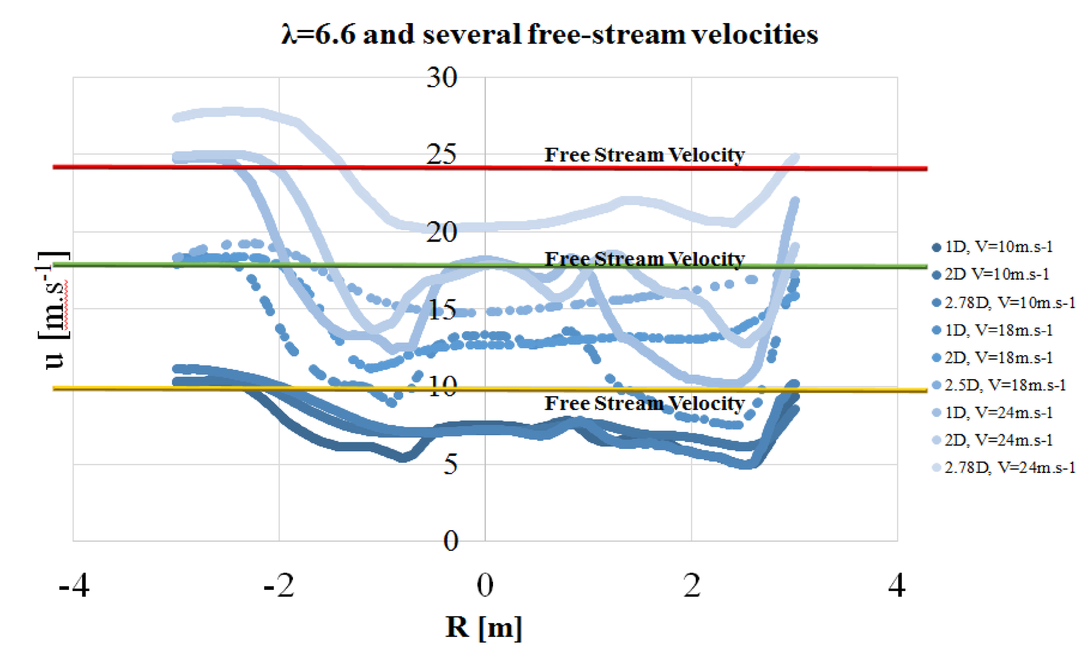


$$\lambda = \frac{\omega \cdot R}{U_{FreeStream}}$$

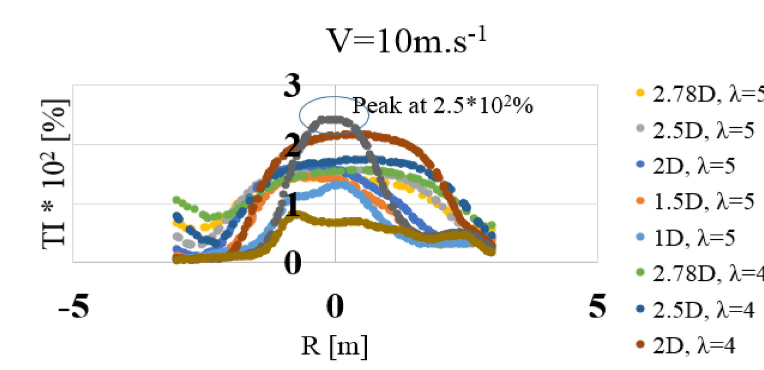


Results

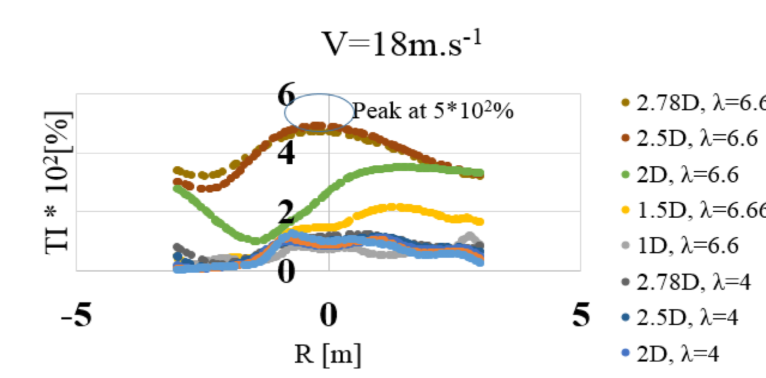
Free Stream Velocity effect on the Near Wake



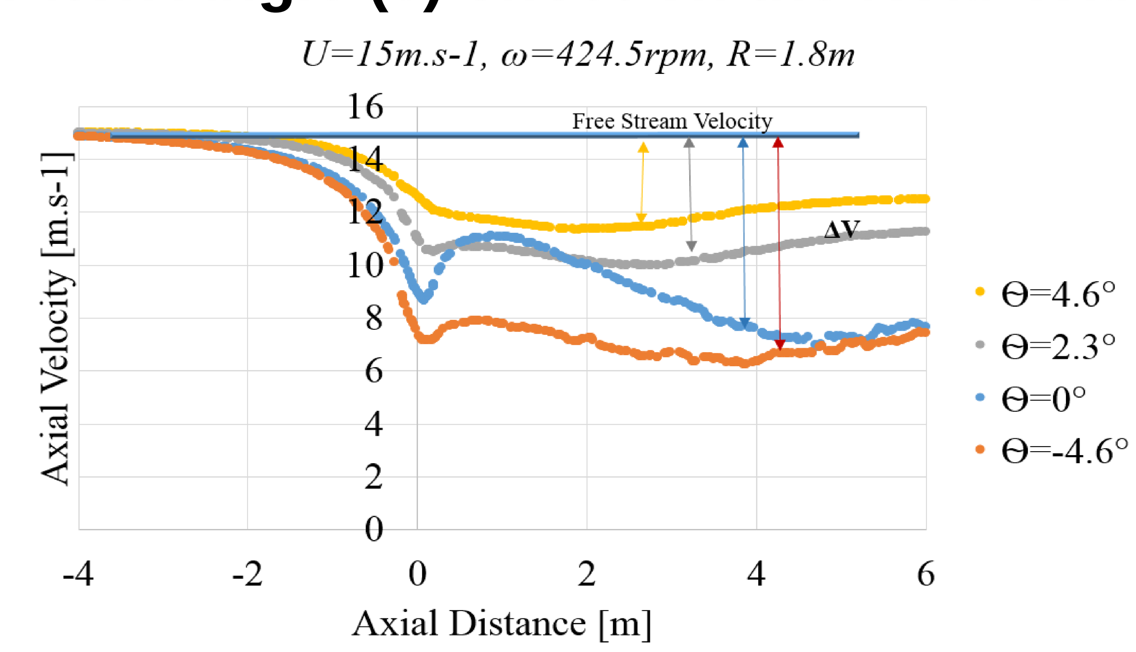
Tip Speed Ratio effect on the Turbulence Intensity (Near Wake)



$$TI = \frac{\sigma_U}{U_{freestream}}$$



Pitch Angle (θ) effect on the Near Wake



Conclusions

- CFD simulation demonstrates that the tip speed ratio and the pitch angle greatly influence the wake behavior and the velocity deficit in this region.
- The turbulence intensity is critically influenced by the free-stream velocity intensity, where higher velocities lead to more turbulent wake behavior.
- **Future directions:** The computational domain needs to be increased to capture the entire downstream wake (near and far wake). Additionally, other design parameters are going to be considered for the optimization task: yaw angle and blade geometry.

References

[1] J.G. Schepers, K. Boorsma, T. Cho, S. Gomez-Iradi, P. Schaffarczyk, A. Jeromin, W.Z. Shen, T. Lutz, K.Meister, B. Stoevesandt, S. Schreck, D. Micallef, R. Pereira, T. Sant, H.A. Madsen, N. Sorensen. **Final Report of IEA Task 29, Mexnet (Phase 1): Analysis of Mexico Wind Tunnel Measurements**, Technical Report, 312p., Netherlands, 2012.

Small Wind Conference 2017