CFD FOR HORIZONTAL AXIS WIND TURBINES

Sugoi Gómez-Iradi (sugoi@liv.ac.uk)
Supervisor: Dr. G. Barakos
CFD LABORATORY

AIM OF THE THESIS

The objective of this research is to develop a CFD [1] method for the analysis of horizontal axis wind turbines (HAWT). Sliding planes will be introduced between CFD meshes fixed on the turbine blades and the support tower to account for the rotation of the blades. Several numerical issues need to be addressed, creating opportunities for exciting research in numerical methods and interpolation techniques, while not violating the conservation laws and maintaining the efficiency of the calculations. The sliding grids will also allow for modelling the relative motion of the blades and the ground.

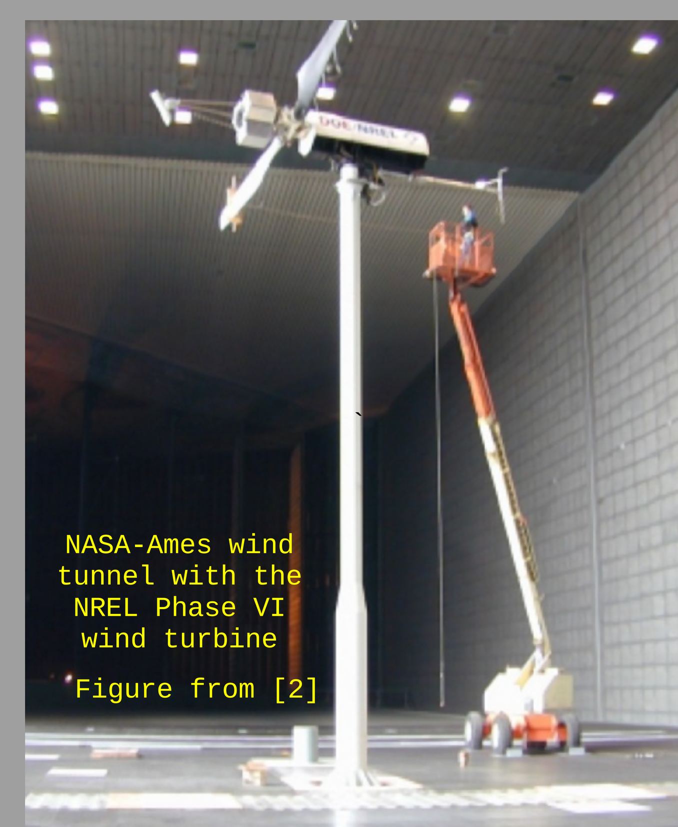
Moving to the structural analysis of the HAWT, an finite element model for the turbine blades based on beams which allow deflection and torsion and which are suitable for high diameter blades will be implemented.

To account for the low Reynolds number flow on parts of the blades and the transition from laminar to turbulent flow conditions, transition modelling is required. Transition models already available in the literature will be implemented and this will allow better predictions for the aerodynamic forces on the blades.

All these will be validated with experimental data obtained in a wind turbine and field data from an actual size wind turbine.

CFD VALIDATION

Experimental data are needed for the validation of the CFD code. The field data are full of uncertainties and the wind tunnel data generally are scaled, so the measurements need to be extrapolated. There is one set of experiments [2] carried on in the NASA Ames wind tunnel (24.4m x 36.6m) where a real size wind turbine was experimented in controlled environments. These data are very complete comparing with any other.



The measurements in a two bladed wind turbine contains pressure along the span of the blade at different stations, the strain of the blades at root, accelerations (edge and flap of the blades and the nacelle), wind tunnel's dynamic, static and total pressures, density, temperature, velocities,...

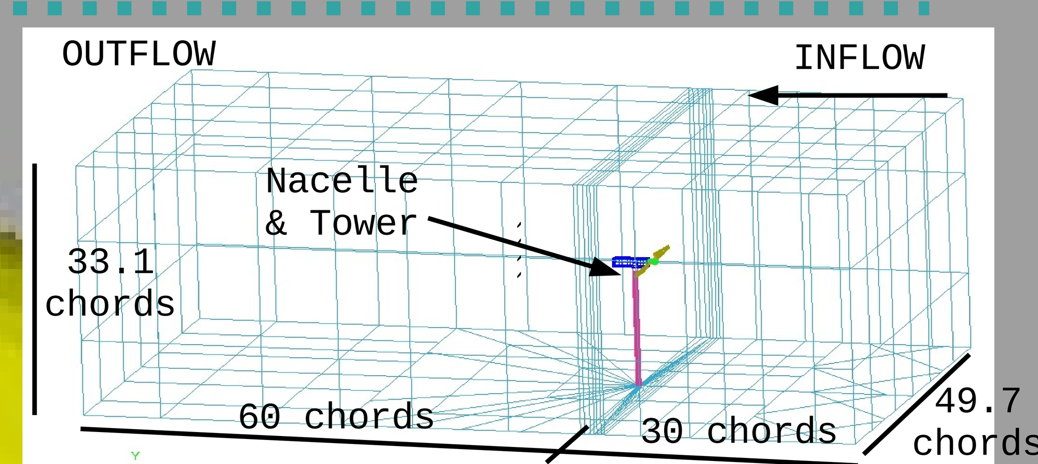
These experiments are known as the best ones in the wind turbine area. Due to this reason they were selected for validation for many researches.

NOVELTY

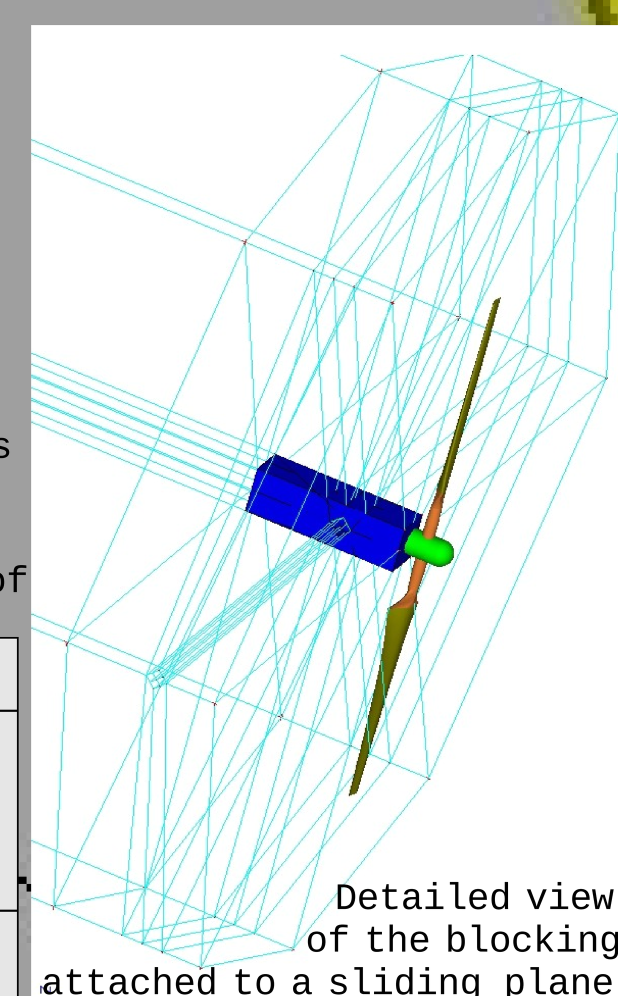
CFD give us the chance to study fluids that others they can not. The high aspect ratio blades design trend will bring new uncertainties as blade tip Mach number (compressibility) effect which is going to be addressed using CFD.

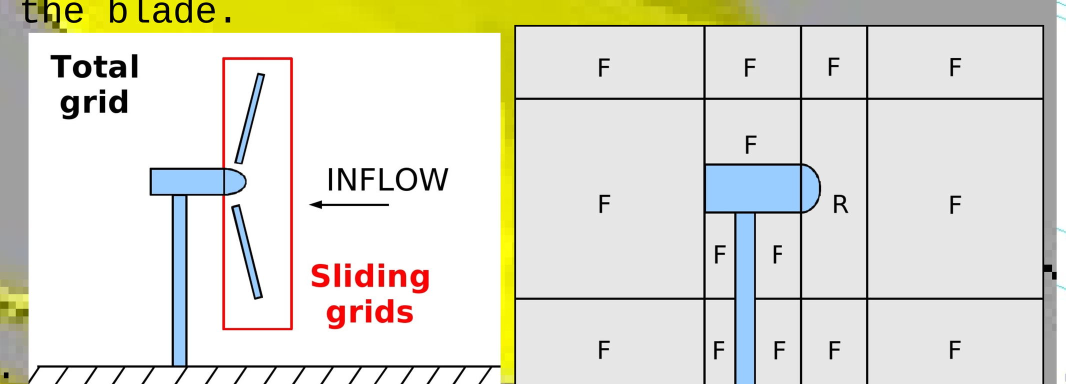
The stalled flows have to be solved using CFD with the implementation of a transition/turbulence model. This is a novelty because before the post stalled flow regimes were avoided in a nominal operation of an aerodynamic body or section but wind turbines needs to be designed to operate also in this regime.

Sliding grids are necessary to take into account the tower, nacelle, the ground and even the wind tunnel wall effects. The sliding grid will allow to make a detailed study of the relatively fix bodies around the rotor. A novelty will be a new sliding grid technique based on the grid regularity.



Wind tunnel full configuration with nacelle, tower, spinner and blades. All dimensions are normalised for the maximum aerodynamic chord of the blade.





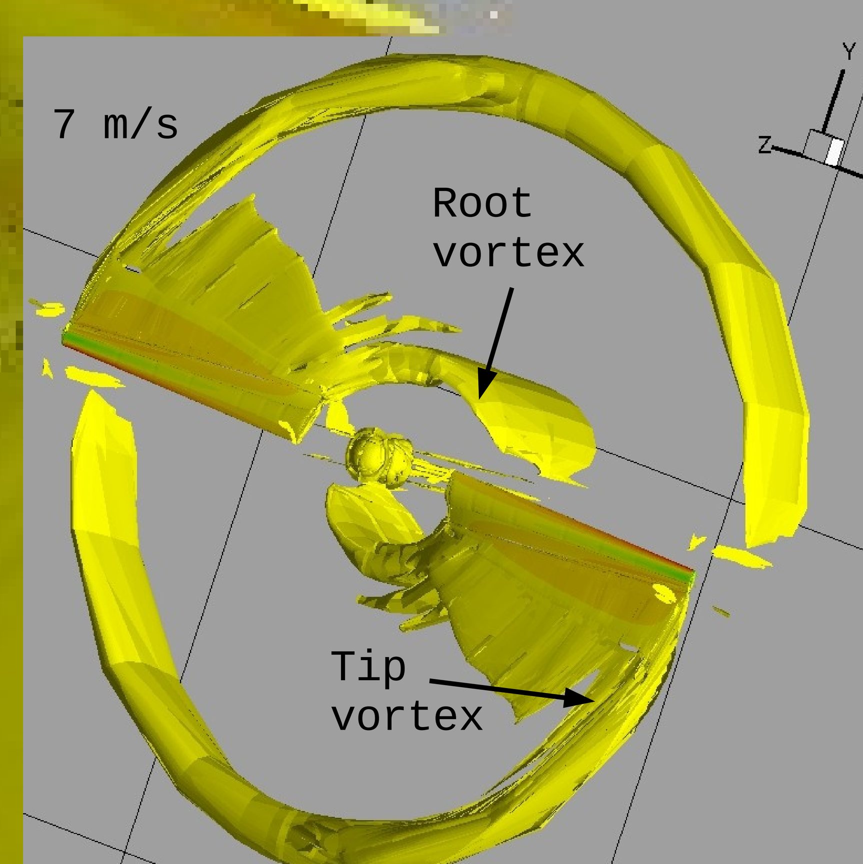
Schematics of sliding grids where the fix and rotating grid parts are shown.

RESULTS

The CFD results are not easy to obtain. One of the reasons is that the efficient parallel computing is a difficult task. For the four cases shown, grids of 2.4 million (parked), 4.6 million (rotating 7m/s) and 6.6 million (rotating 20m/s) points were used. These were solved with 8 to 55 computers, from 7 hours to 42 days.

For the CFD calculations isolated rotors were first considered. Starting from the parked case with attached flow, the parked and pitching case with separated flow was computed and analysed. Then, the rotational effects were studied at the 7m/s rotating and attached flow case. Finally, a more challenging case was computed and analysed at the post stall regime.

Turbulence has been modelled using either κ-ω and SST models. The selected experimental cases and computational results are:



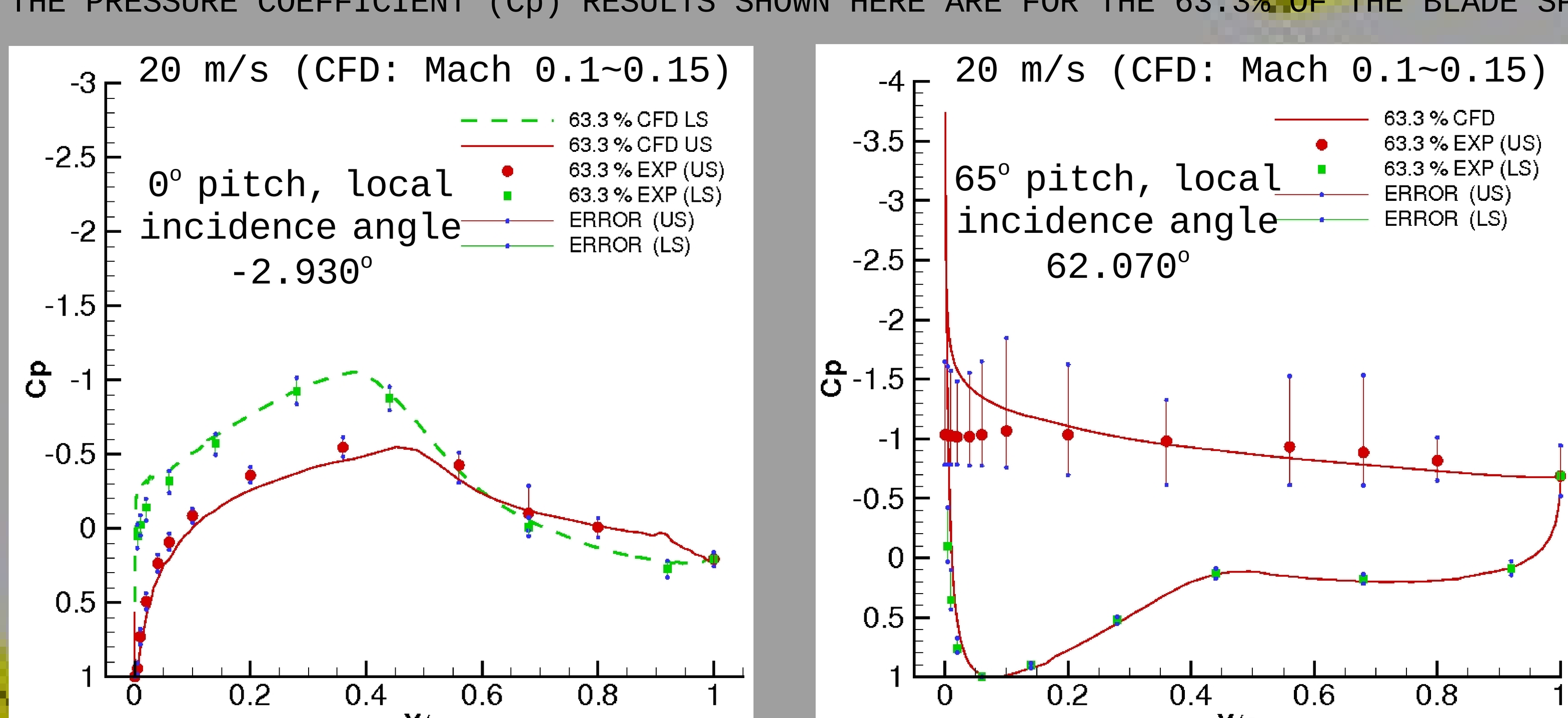
Wind turbine blades and spinner wake with pressure plotted as contour.

For separate flow cases the SST model was found to be better than the κ-ω one although deep stall conditions were harder to predict than design ones.

For unsteady rotating cases three revolutions were computed to allow wake development. This is very important due to the blade wake interaction and due to the effect that this wake will have in the rest of the wind turbines of the wind farm.

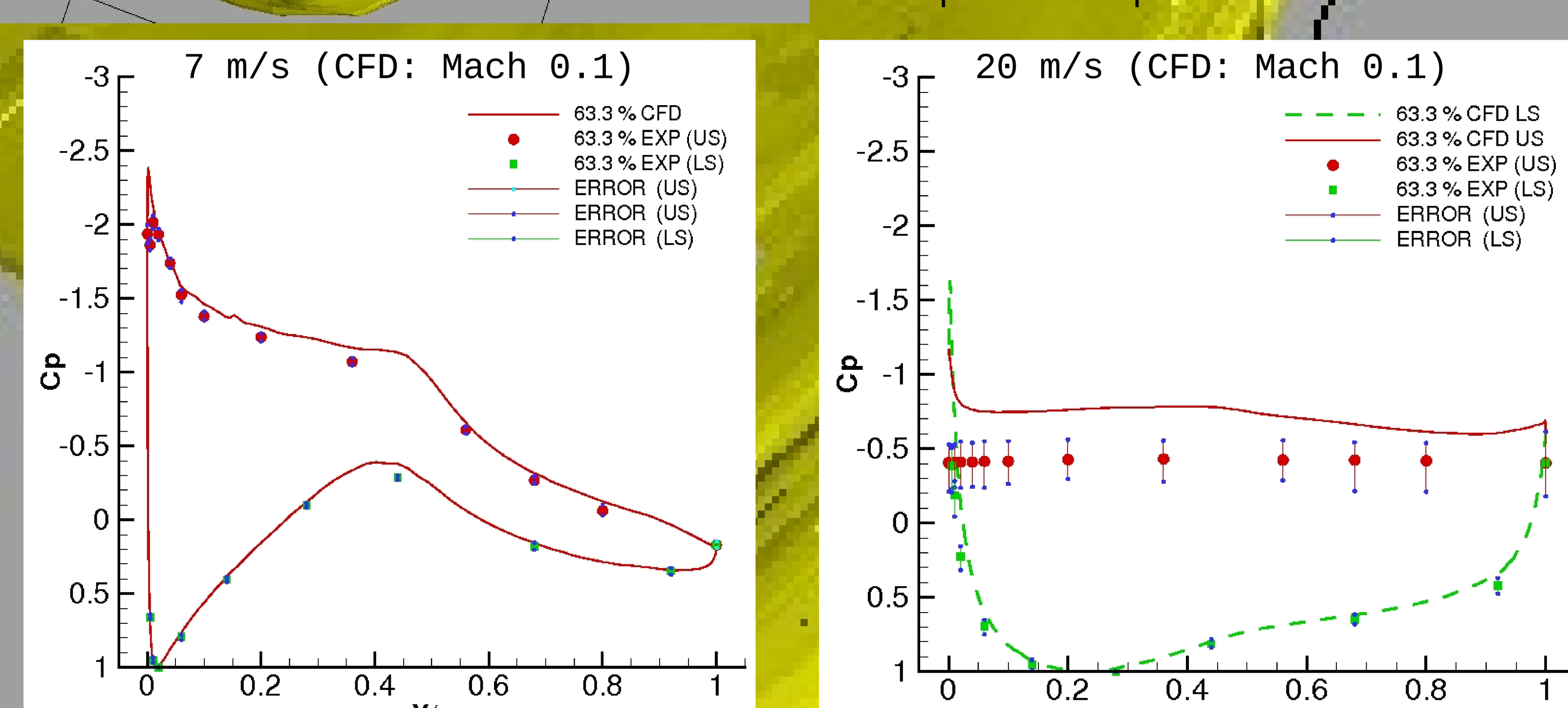
ALL THE PRESSURE COEFFICIENT (Cp) RESULTS SHOWN HERE ARE FOR THE 63.3% OF THE BLADE SPAN.

PARKED CASES



Parked and parked and pitching cases from 90° to -15° at a ratio of 0.682 °/s.

ROTATING CASES



The 72 rpm rotational cases with 3° of pitch at the tip.

FUTURE STEPS

Previous cases should be computed and analysed with blade root and full configuration, instead the isolated rotor, to study their influence in the generated wake and the blades loadings. Side wind case also will be addressed due to its importance in the real wind turbine operation regime.

An implementation of sliding planes in CFD is also needed in order to take into account the tower, nacelle and ground effects. Once this is done, a model for blade dynamic simulation will be added to the CFD solver to take into account the influence of aerodynamic loading of the blades due to their large deflections (larger than 10% at the tip).

Finally, an appropriate turbulence/transition model will be merged with the CFD solver to asses the behaviour of the fluid at the post stall regime.

REFERENCES

[1] Barakos, G. et al. CFD Capability for Full Helicopter Engineering Analysis, 31st European Rotorcraft Forum, Florence, Italy, September 2005.

[2] Hand,M.M. et al. Unsteady Aerodynamics Experiment Phase VI: Wind Tunnel Test Configurations and Available Data Campaigns, Technical Report NREL/TP-500-29955, NREL, December 2001.

ACKNOWLEDGEMENTS

THE UNIVERSITY of LIVERPOOL
CFD LABORATORY
cener centro nacional de energías renovables
FUNDACIÓN CENER CIEMAT