CFD FOR HORIZONTAL AXIS WIND TURBINES

Ph.D. student: Sugoi Gómez-Iradi (sugoi@liv.ac.uk)
Supervisor: Dr. G. Barakos (G.Barakos@liverpool.ac.uk)
CFD LABORATORY - The University of Liverpool

PROJECT OBJECTIVES

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 their relative rotation. Numerical issues with the sliding planes must be addressed, creating opportunities for exciting research in numerical methods and interpolation techniques. The resulting schemes should not violate the conservation laws and should allow for efficient computations. In addition, the effect of the ground on the HAWT aerodynamics should also be accounted for.

NOVEL CFD METHOD

The current CFD method builds on the compressible Navier-Stokes solver of Liverpool University [1] and is extended here for flows around wind turbines.

As can be seen in Figures N1 to N3, radial and axial sliding planes are introduced, which allow for non-matching grid densities on each side.

Since the underlying solver uses the multi-block concept along with layers of halo-cells (Figure N4) it is relatively easy to device a numerical scheme to populate the halo-cells using values of the conservative variables (φ) in close proximity to the sliding plane.

This leads to an identification problem as well as an interpolation one. The key concept is the computation of halo-cell values using a weighted summation of the type:

$$\varphi_{halo} = \sum_{i}^{neighbours} w_i \varphi_i$$

The weight coefficients (w_i) are calculated using inverse distance or cell overlap.

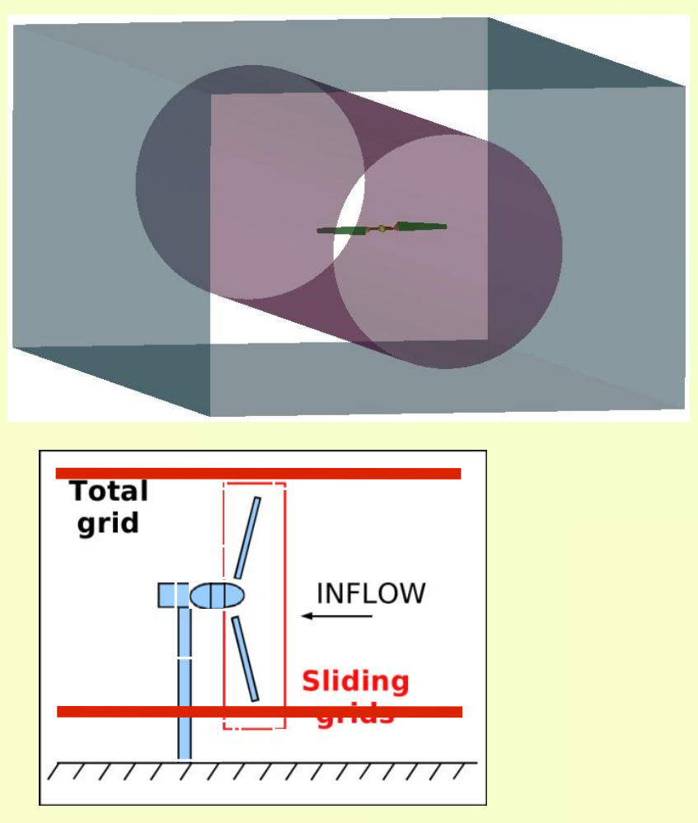


Figure N1: Sliding grid for assessing the influence of the wind tunnel walls in the WT aerodynamics.

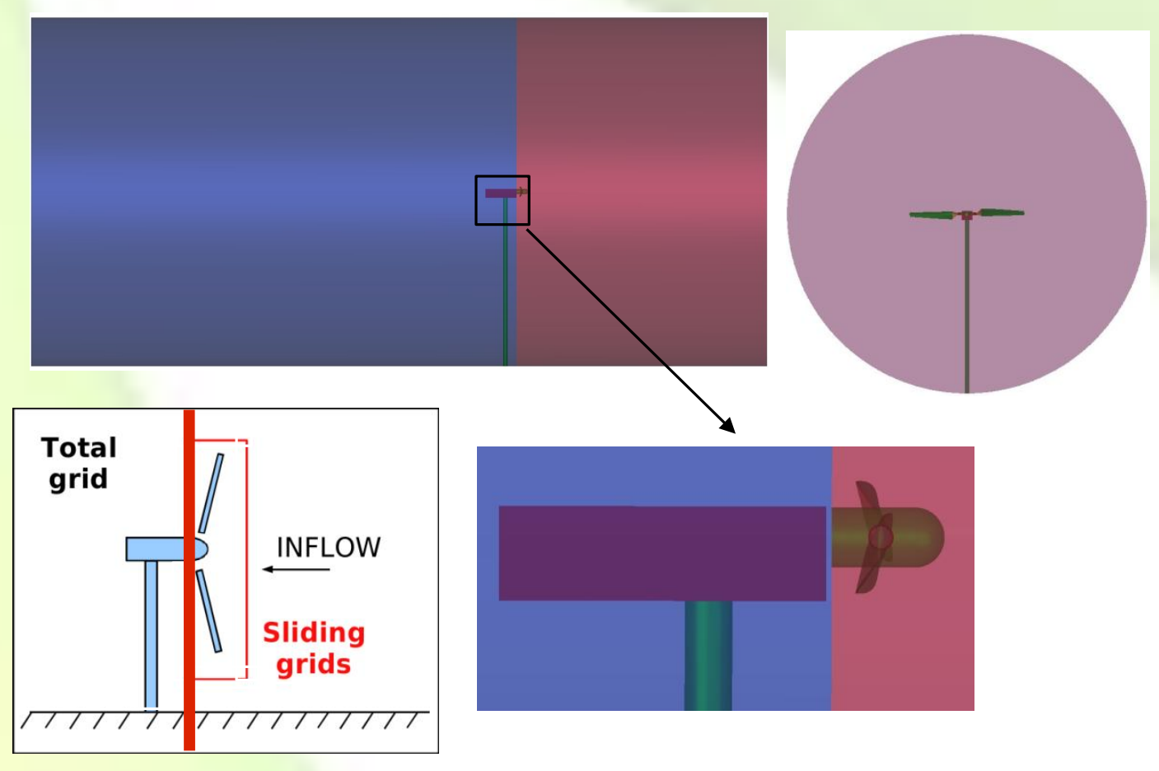


Figure N2: Sliding grid arrangement for assessing the influence of the nacelle and tower in the WT aerodynamics.

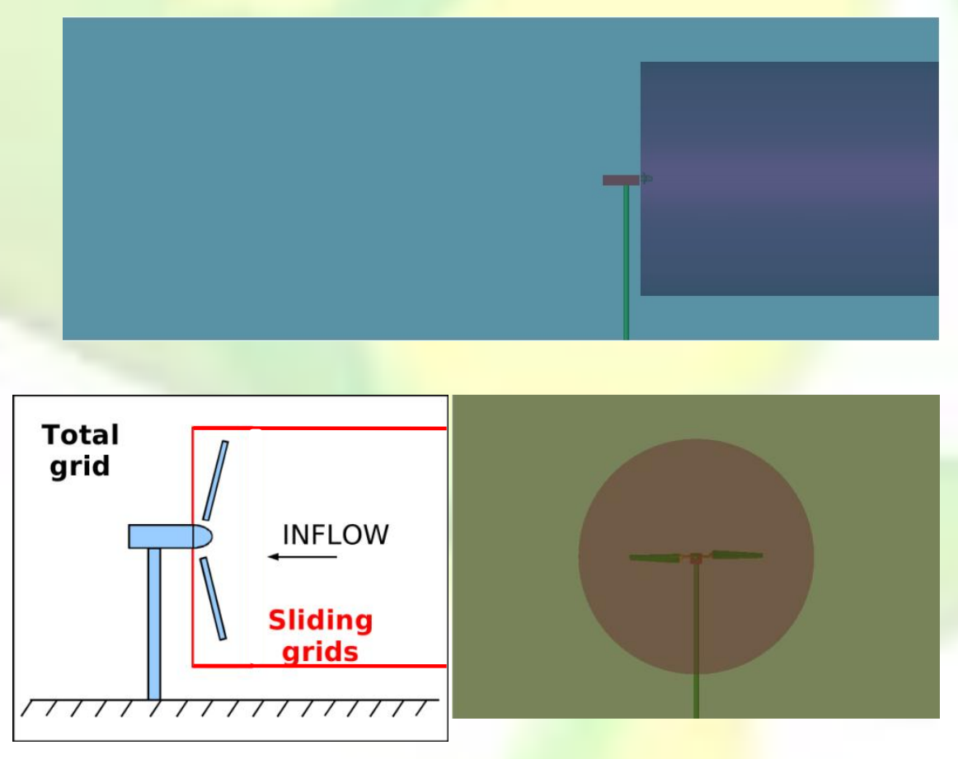


Figure N3: Combination of sliding grids for assessing the influence of the nacelle, tower and wind tunnel walls.

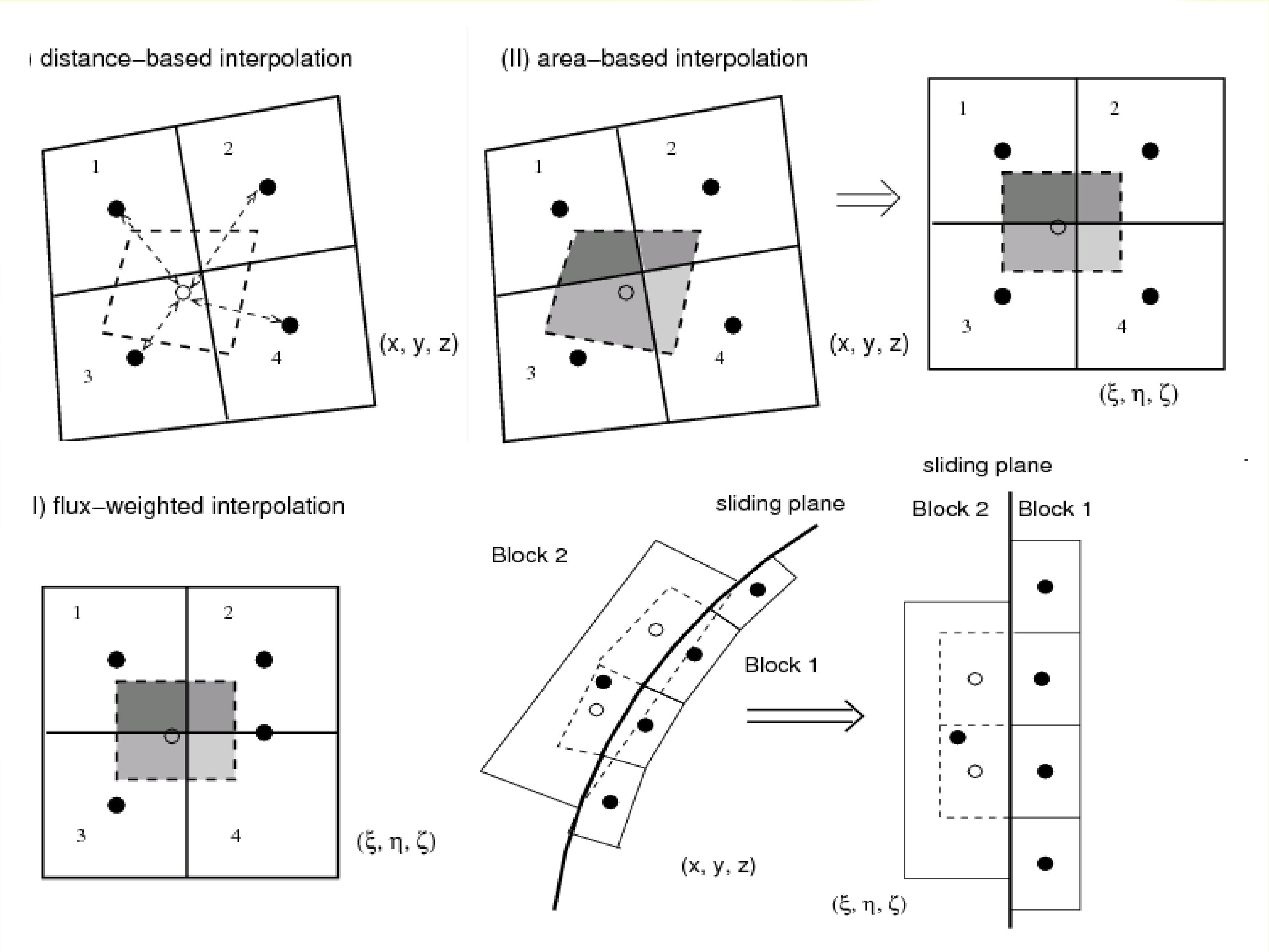


Figure N4: Interpolation across sliding planes

SUMMARY

A compressible CFD method combined with a sliding plane technique was demonstrated for the HAWT. Isolated rotors were first investigated to stablish confidence in the method and identify appropriate grid sizes for computations. Having established confidence in the method, the effect of the wind tunnel walls was investigated. The sliding planes allowed for pressure waves originating from the blades to reach the tunnel walls. The low values of the unsteady wall pressure suggest mild wall influence, though further calculations are under way to quantify this effect, as well as, the influence of the tower in the blade loads.

CFD VALIDATION

Experimental data necessary for CFD validation were obtained from the NREL database [2]. Since a full size wind turbine was tested, the influence of the tunnel walls on the obtain measurements must be investigated.

Figure C1: NASA-Ames wind Tunnel

At first, isolated wind turbine rotors were computed and encouraging results have been obtained at working and stalled conditions using a low cost Beowulf cluster of workstations. Indicative results are shown in Figures C2 to C4, where the sensitivity of the solution to grid size, time step and far-field location is presented.

It was concluded that time steps of up to 2 degrees in azimuth can be used along with CFD grids of approximately 3 million points. The standard k-ω turbulence model was selected from the library of turbulence models available in the solver, due to its robustness and fair performance.

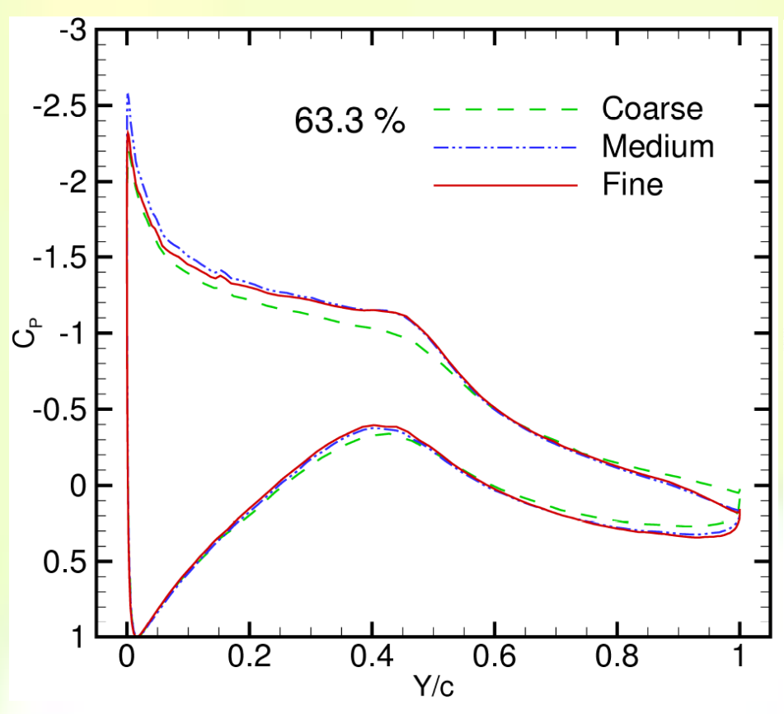


Figure C2: Effect of grid size
Coarse: 1.3 mill.
Medium: 3.4 mill.
Fine: 4.6 mill.

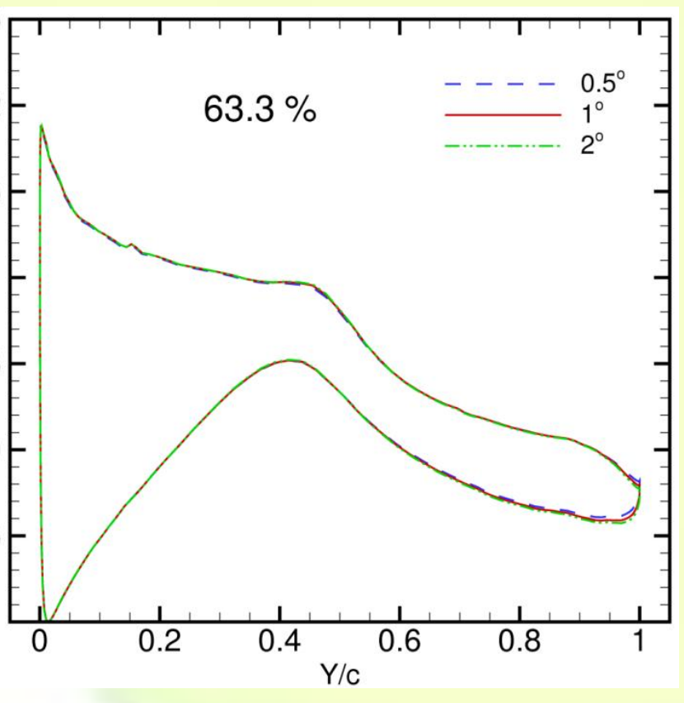


Figure C3: Effect of azimuthal step

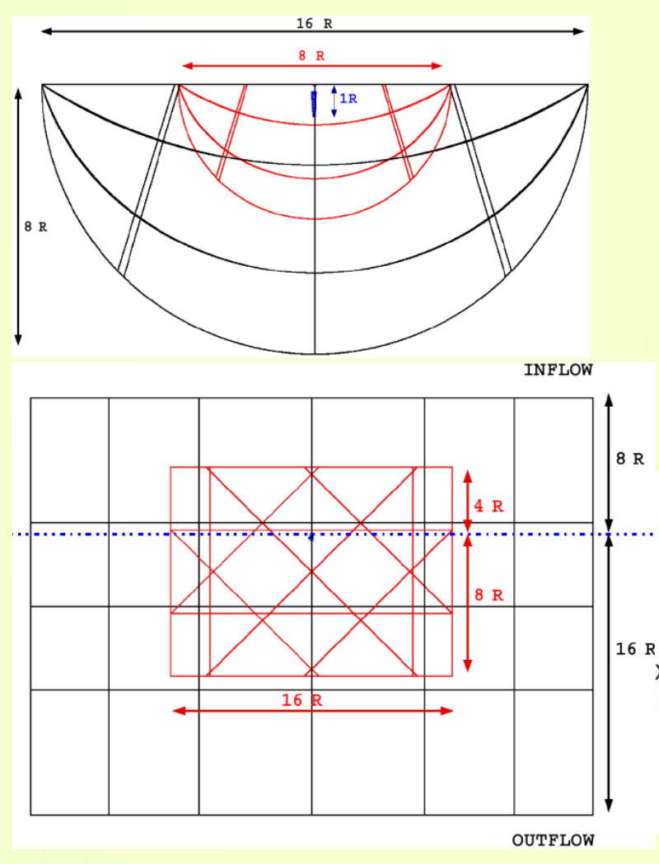
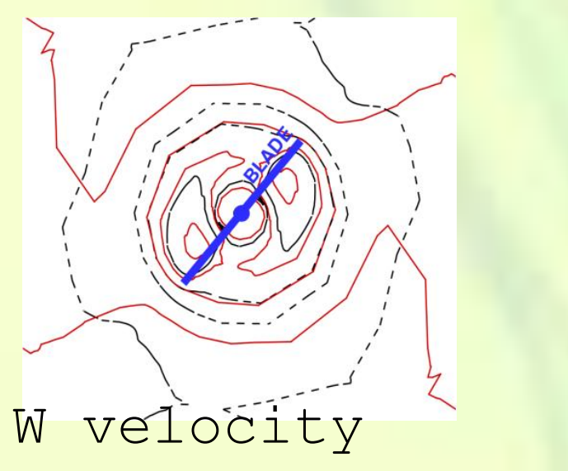
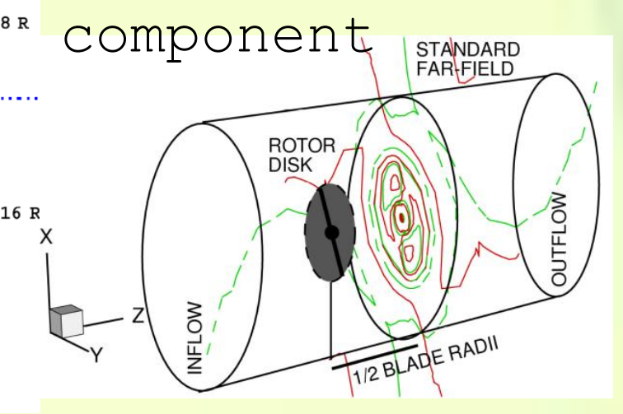


Figure C4: Effect of far-field location



W velocity component



The contours are taken half radii downwind wind turbine

To investigate the effect of the wind tunnel walls a radial sliding plane was placed outside the rotor disk and half way between the blade tip and the wind tunnel walls (see Figure N1). The unsteady flow was the computed using 0.25 degree step in azimuth. This resolution allowed for the unsteady pressure field to be resolved. Figure N3 presents the relative orientation between the blade and the tunnel walls and the corresponding pressure field at four azimuth angles (0, 90, 180 and 270 degrees). As can be seen, pressure waves originating from the blade reach the tunnel walls generating a periodic compression as the blade approaches. This is followed by the pressure reduction, as the tip of the blade approaches 30 degrees of azimuth. The pressure increases again as the distance between wall and blade decreases. This is repeated four times per revolution. The magnitude of the unsteady pressure peaks are 2.5% above the mean. The employed CFD method was capable of resolving this unsteadiness on the relatively coarse grid employed (2.7 mill points). This is a very encouraging result, suggesting that the sliding plane method allowed for proper communication between the inner rotating domain and the fix grid around the tunnel walls.

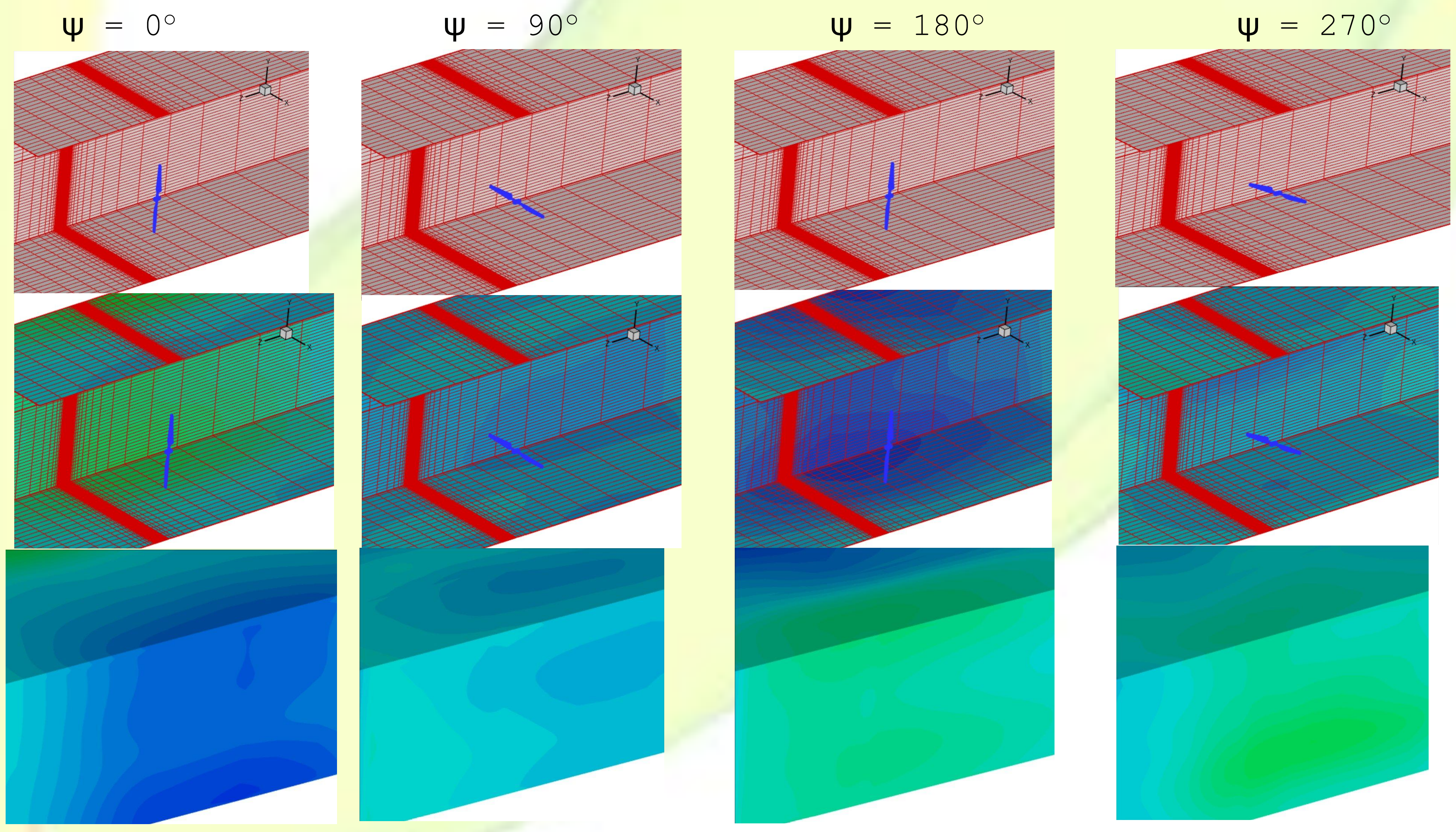


Figure N5: Blade orientation and unsteady pressure field on the tunnel walls for the 7m/s case

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

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