

ENE 505 – Applied Computational Fluid Dynamics in Renewable Energy Technologies

WEEK 10: CFD PROBLEM SET UP

PROBLEM SETUP:

- **Geometry Model**

- In the present study [1], the geometry is created using a design tool called CATIA V5R20. Symmetrical straight bladed NACA0022 is selected to be the airfoil with 22 mm of thickness, resulting in a 100 mm blade chord. The selection is made in accordance with the vertical axis wind turbine created by Howell et al. [2] for validation purposes with the existing experimental results. [1].
- The diameter and the overall domain length in the flow direction are set to 0.6 m and 15 m (25D), respectively. This is due to the fact that the effect of the boundary conditions and the wall functions should be kept to minimum to reduce the error factor and also to examine the flow in the wake of the blades.
- The sides of the domain are considered to be walls to represent the similar physical wall boundary constraints to those in the wind tunnel.
- The same square tube section of the computational wind tunnel test area of 1.2 m x 1.2 m is employed as in the wind tunnel experiment. The inlet and outlet on the other hand are positioned further away to allow a full development of the wake of the turbine as shown in Fig. 1.
- The computational domain is constructed using a pre-processor module (Gambit V.5) after building the geometry in CATIA
- A cylinder with diameter of 0.7 m is generated in Gambit to set up a rotating zone and finer mesh could be created in the center of the circle and less mesh cells

would be used in the outer volume as illustrated in 2-D mesh layouts of the present flow domain together with the 3-D local mesh representing the rotating inner zone and stationary outer zone as shown in Fig. 2 and Fig 3. , respectively.

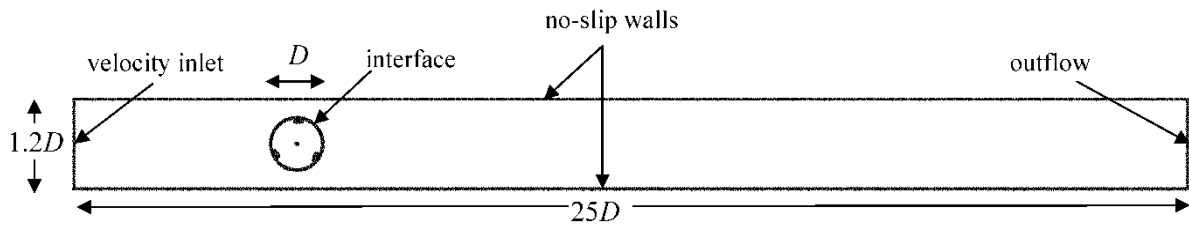


Figure 1. The model geometry set-up and the computational boundary conditions set-up [1].

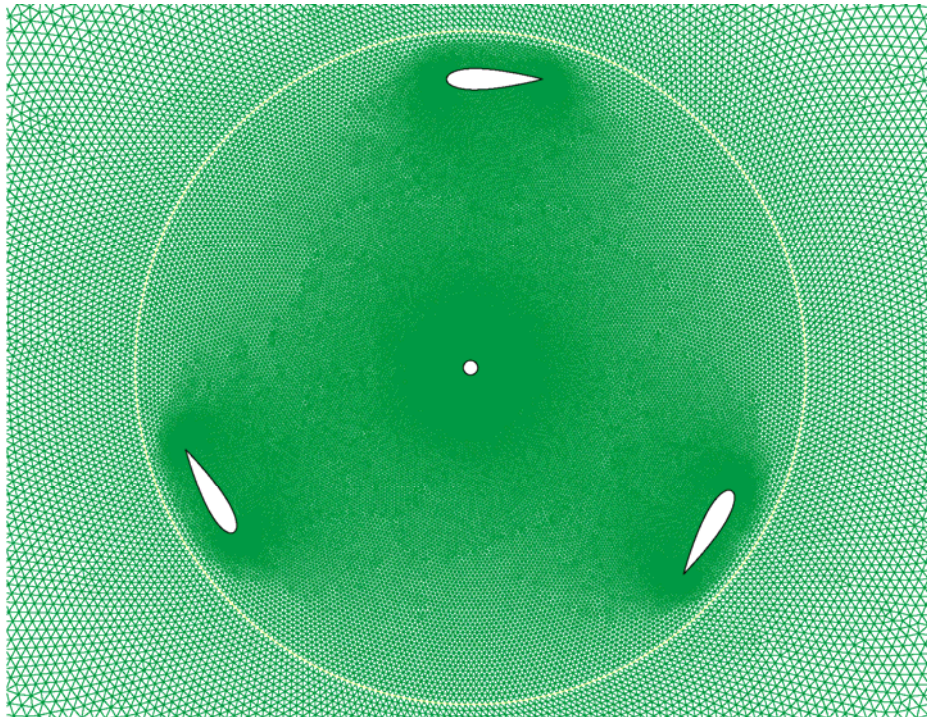


Figure 2. Mesh of rotating zone which includes blades [1].

- The mesh should be fine close to the blades to accurately simulate the large velocity and pressure gradients within the boundary layer.
- Within the stationary outer region, the flow would settle down after the wake of the turbine and relatively less number of cells would be enough to simulate the flow characteristics.

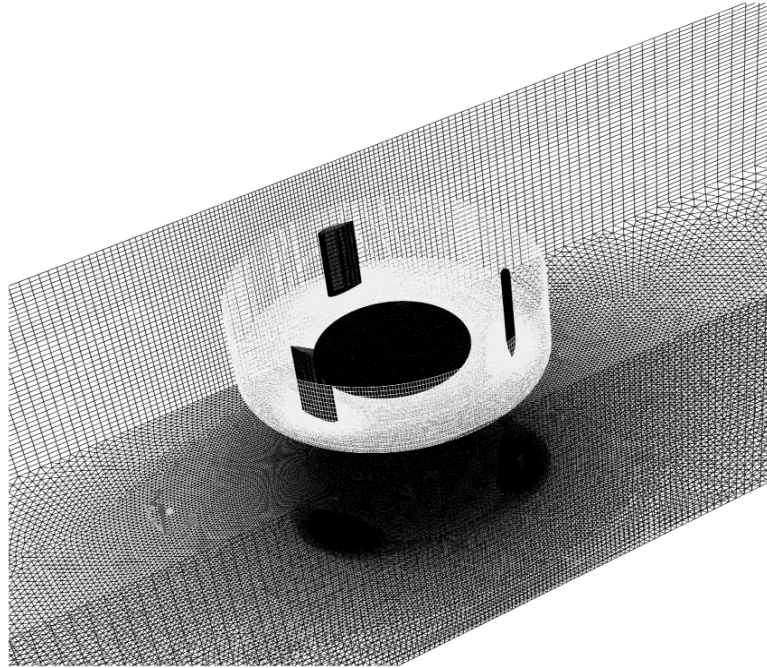


Figure 3. 3-dimensional mesh [1]

References:

1. Aresti, L., Tutar, M., Chen, Y., and Calay, R. K., "Computational study of a small scale vertical axis wind turbine (VAWT): comparative performance of various turbulence models" *Winds and Structures*, Vol. 17(6), pp. 647-670

2. Howell, R., Qin, N., Edwards, J. and Durrani, N. (2010), "Wind tunnel and numerical study of a small.