T2 D8 ITC5: Wind mill specification

This document describes the wind mill test case. This test case is interesting since it contains a fixed piece (mast) and a moving part (rotor and blades). It means that the mesh should be able to track the blades while rotating without degrading the mast which modifies the flow past the wind mill.

A picture of the rotor and blades part is given in Figure 1. A stl file is available upon request to test this case. The blades represented in Figure 1 are 17.8 m long and 0.2 m large. To be realistic a mast has to be added. Generally a wind mill is about 80 m high; a cylindrical mast of 80 m will be added from the ground to the rotor.

These dimensions represent the second difficulty of this test case. The mesh has to be stretched to capture all the dimensions from a tenth of meter (and even more for the vortex) to one hundred of meters.

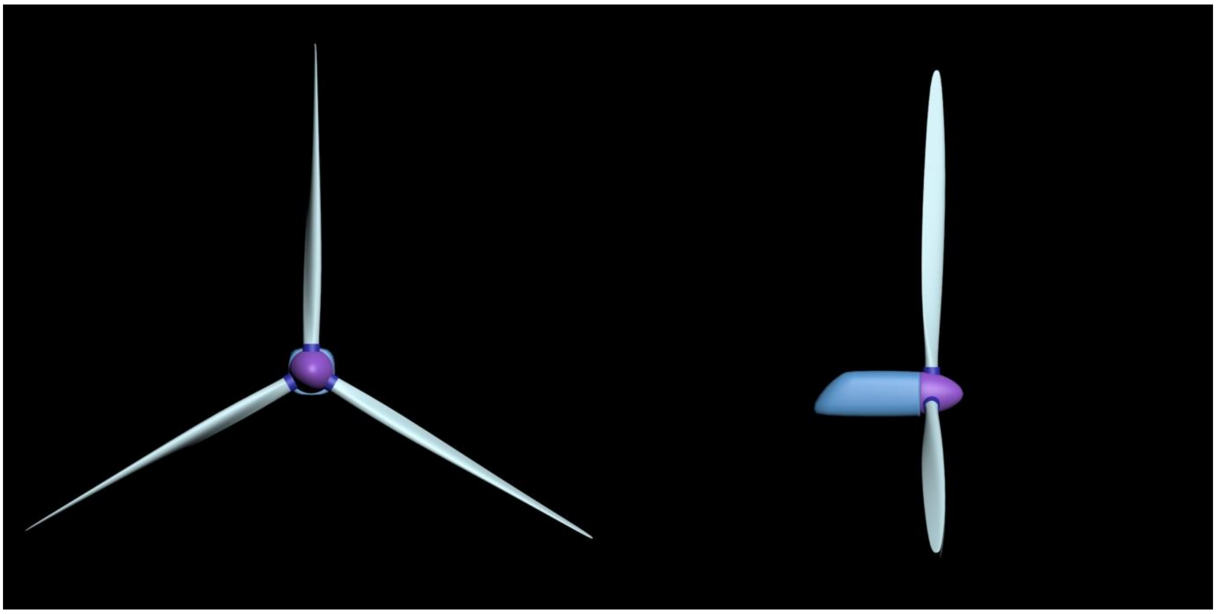

Figure 1 - Illustration of the rotor and blades part of the wind mill

Due to overall dimensions, the domain has to be at least 120 m high, 200 m long and 80 m large.

In reality, in a stable stratified atmosphere, the vertical wind speed has a logarithmic shape depending mainly on the surface roughness. A typical roughness of 0.1 m will be taken into account. This represents a flat surface with hedges and some small and distant buildings. The wind speed is chosen to be 7 $m.s^{-1}$ at the rotor level. This means the wind speed increases from approximately 4.8 $m.s^{-1}$ at a height of 10 m to 7.4 $m.s^{-1}$ at the top of the domain.

Given these values the Reynolds number is given by:

$$Re = \frac{\rho L U}{\mu} = \frac{1 * 17 * 7}{\mu} = \frac{119}{\mu}$$

By changing the viscosity (μ) value a wide range of Reynolds number can be obtained. With the true air viscosity value, a very high Reynolds number is obtained (10^7). It seems difficult to solve the problem with complete realistic values.

A high number of mesh nodes will be mandatory. It seems that approximately a million nodes will be necessary.

Step 1 - Stationary simulation

Given the objects described in introduction, the first step is to realize a stationary simulation. The wind mill is not moving. In this step, the robustness of the meshing process is evaluated. The pressure on the blades can be determined and the vortex generated in the lee of the blades is represented.

Step 2 - Constant rotation

This simulation is the same than in step 1 except that the blades are rotating with a given constant rotation speed. Depending on the model used a special attention has to be paid to the extreme part of the blade which can rapidly reach high speed. The air may not be incompressible anymore but we do not intend to include this effect in our simulation for the time being.

In that case, if we achieve to simulate a rotation of 360° or more, it will be interesting to observe the effect of a blade upon the others.

Step 3 - Free rotation

This is the final complete simulation. The blades are set free to rotate around the hub axis. The wind speed is imposed on the side of the domain facing the wind mill and the efforts generated on the blades start and maintain the rotation of the blades. A counter torque will be applied to simulate the effect of the power generation. The hub will be simulated as a hydraulic bearing with a high viscosity to create the resistant torque.