Numerical Methods: Final Project Report

Limitations of a Computational Fluid Dynamics (CFD) simulation: Why one should not build a wind turbine over the KTH campus …

Guillaume Bournique
Master in Sustainable Energy Engineering
Royal Institute of Technology (KTH)
Stockholm, Sweden
guibou@kth.se

Florian Remy
Master in Sustainable Energy Engineering
Royal Institute of Technology (KTH)
Stockholm, Sweden
fremy@kth.se

***Abstract* — The purpose of this project is to investigate the feasibility of the installation of a Vertical Axis Wind Turbine (VAWT) at the KTH main campus. In spite of its lower efficiency, a VAWT is preferred to a Horizontal Axis Wind Turbine (HAWT) because it is more adequate in areas close to buildings and housings. In the first time, the wind speed profile over the campus is modelled using a Computational Fluid Dynamics (CFD) software. In the second time, the most appropriate locations for the wind turbine are reviewed. Two possible locations to install the wind turbine are put into light based on the simulation results. However, none of these locations is suitable in real life because of the relief and forests surrounding the campus.**

Keywords* — *computational fluid dynamics, CFD, star cmm+, wind speed profile, vertical axis wind turbine, VAWT, simulation, feasibility study, KTH campus, limitations, disillusion

I. Introduction

Today, renewable energies account for about 60% in the electricity production in Sweden [1]. In order to increase this share, small capacity wind turbines located close to the electricity demand can be used.

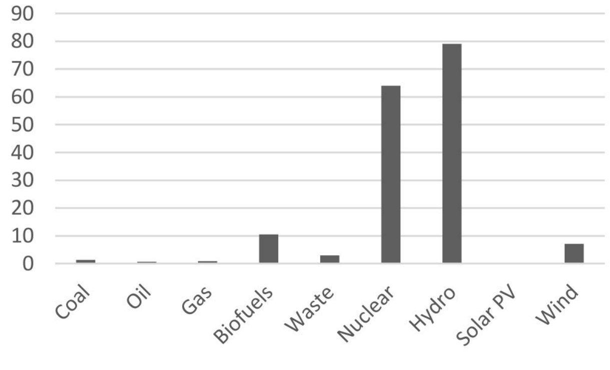

Figure 1. Electricity Production in Sweden (MWh) [1]

Wind turbines can be split into two categories: Horizontal Axial Wind Turbines (HAWT) and Vertical Axial Wind Turbines (VAWT). HAWTs are the most frequently used because they provide higher power outputs and higher efficiencies [2]. However, they need to be installed away from cities in order to reduce noise and vibrations. VAWTs, on the other hand, can be installed in living quarters. As a consequence, the need for electrical cables and losses are lower.

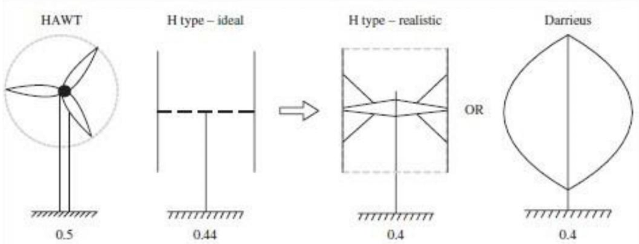

Figure 2. HAWT, VAWT: Savonius and Darieus designs [2]

There are primarily two types of Vertical Axial Wind Turbines (VAWT): the Savonius (or H type) design and the Darrieus design. Both design work under similar wind velocities (about $11\ m/s$) [3]. Their principle is the same, the blades harness the kinetic energy of the wind, particularly in up-flows and vortex which form in the shade of buildings [2]. The rotor power coefficient reaches 15% for the Savonius design, and more than 25% for the Darrieus design.

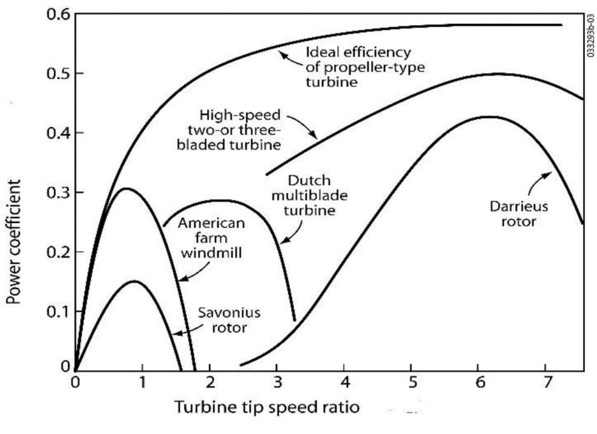

Figure 3. Savonius vs. Darrieus designs regarding efficiency and speed [4]

As both designs have the same rated speed, but the efficiency is higher for the Darrieus design, the latter is preferred.

II. GENERAL DESCRIPTION OF THE PROBLEM AND PURPOSE OF CFD SIMULATION

The power production of a wind turbine is directly related to its location. In every wind farm project, the wind speed is monitored on-site during one year, in each direction. Such precautions are legitimate by the cost of a wind turbine. Simulating the wind speed profile in a location can be seen as a previous step in a wind farm project. It is easy to put into practice, does not cost much, but the results are less accurate than on-site measurements.

III. CODE CHOSEN FOR SOLUTION OF PROBLEM

The software used to run the simulation was STAR CCM+.

IV. COMPUTING PLATFORM USED FOR RUN

The final simulation was run on a Windows 7 based computer, with an Intel i7 @ 2.8 GHz processor.

V. SCHEMATIC DIAGRAM OF THE REGION OF INTEREST WITH ALL KEY DIMENSIONS, FLOW INLETS AND OUTLETS

The 3d model of the campus, as well as the North East face through which the wind enters, is illustrated in Figure 4:

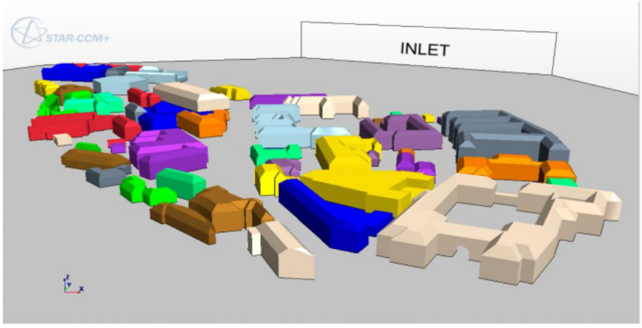

Figure 4. Schematic diagram of the region and wind inlet

The model was made at a reduced scale. A scaling factor of 8.68 needs to be applied to get the real lengths from the lengths of the 3d model.

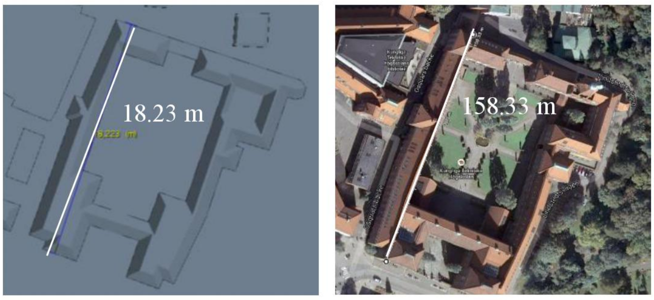

Figure 5. Scale difference on a horizontal plane between model and reality

The height of KTH main building was measured at 2.90 m on the 3d model. By assuming that the scaling factor according to the z axis is the same as the one on a horizontal plane, this building would measure 25 m. This estimation may be a bit overestimated, but the order of magnitude is correct.

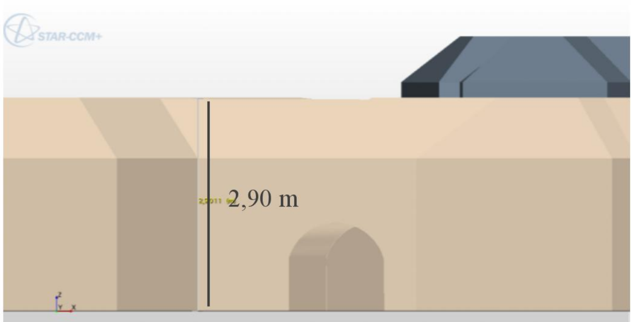

Figure 6. Measure of the height of KTH main building in the model

VI. GRID DESIGN

The geometry was meshed using a tetrahedral pattern and a surface remesher. The base size of the mesh was chosen to be 0.7 m. The data entered into the software are summarized in Table 1:

Continua / Mesh		
Models		
Surface Remesher		
Tetraheadral Mesher		
Reference Values		
Base size	0.7	m

Table 1. Input parameters – mesh model

The tetrahedral mesh was preferred to the triangular mesh because it is more accurate. As a consequence, there are more nodes in the mesh, and more calculations to do at each iteration. The convergence is also often slower than when using a triangular mesh.

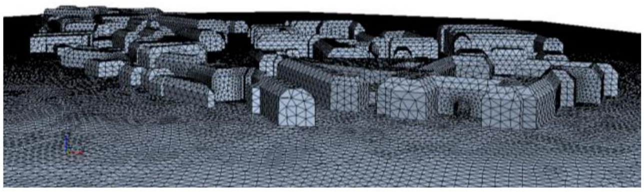

Figure 7. Mesh scene of the 3d model

The mesh appeared to be fine enough with a parameter of 0.7 m. This value was kept for the final simulation.

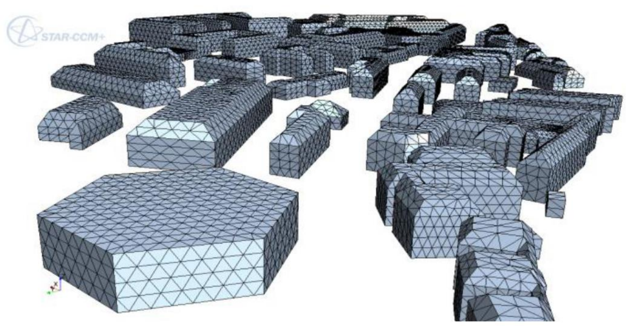

Figure 8. Mesh scene of the 3d model (2)

VII. BOUNDARY CONDITIONS

All the input parameters regarding the boundary conditions are summarized in Table 2:

Regions / Region / Boundaries		
Default 1 -> 57		Wall
Physics Conditions		
Shear Stress Specific.		No-Slip
Wall Surface Specific.		Rough
Physics Values		
Roughness Height	0.001	m
EE, NE and NN		Velocity Inlet
Physics Conditions		
Flow Direction Specific		Component
Turbulence Specificat.		Int. + Visc. Ratio
Velocity Specification		Magn. + Direct.
Physics Values		
Flow Direction	Cartesian	[-1,0,0]
Turbulence Intensity	0.013342	
Turbulent Viscosity Rat	100	
Velocity Magnitude	45	m/s
Ground		Wall
Physics Conditions		
Shear Stress Specificat.		No-Slip
NW and SE		Wall
Physics Conditions		
Shear Stress Specificat.		Slip
SKY		Symmetry Plane
SS, SW and WW		Pressure Outlet
Physics Conditions		
Backflow Direction Specification		Normal
Turbulence Specific.		Int. + Visc. Ratio
Physics Values		
Relative Pressure	0	bar
Turbulence Intensity	0.013342	
Turbulent Viscosity Rat	100	

Table 2. Input parameters – boundaries types

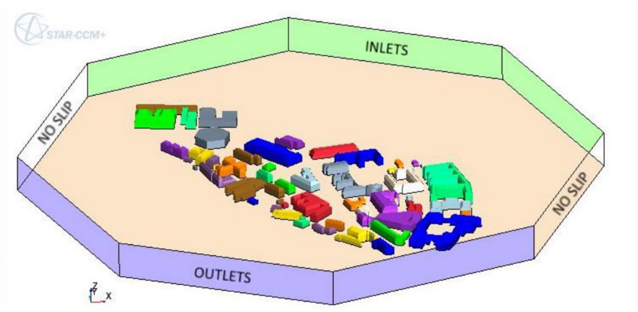

Figure 9. Types of boundaries in each direction

The walls are assumed to be rough, with a roughness height of 1 mm. This is an average value for concrete [5].

The wind enters through the East East (EE), North East (NE) and North North (NN) faces, and goes in the direction of the South West (SW). To set up the wind direction, a coordinate system was created. To do so, the initial coordinate system was duplicated and rotated by an angle of 67.5 deg. The x-axis is thus pointed toward the inlet face. The way the physical values were found is detailed in section X.

The South South (SS), South West (SW) and West West (WW) faces are set as pressure outlets. This type was preferred to split-flow outlet, because the share of the wind exiting by each of the faces is not known a priori.

The default wall values were kept for the ground, and the sky was put as a symmetry plane.

VIII. INITIAL CONDITIONS

The physic model, as well as the initial conditions, allow to converge toward the solution. They also decrease the number of steps required to reach the solution. They are summarized in Table 3:

Continua / Physics		
Models		
Constant Density		
Gas		
Density	1.18415	kg/m3
Dynamic Viscosity	1.86E-05	Pa.s
K-Epsilon Turbulence		
Segregated Flow		
Steady		
Turbulent		
Reference Values		
Min Wall Distance	1.00E-06	m
Reference Pressure	1	bar
Initial Conditions		
Pressure	1	bar
Turbulence Intensity	0.013342	
Turbulent Velocity Scale	4.5	m/s
Turbulent Viscosity Ratio	100	
Velocity	[45,0,0]	m/s

Table 3. Input parameters – physics model

The choice of the model for the flow is explained in section X. For the reference values, the default values were kept. The initial conditions are the same as the input velocity initial conditions. There values are calculated in section X.

IX. FLUID PROPERTIES

The default values of the fluid properties, that is to say the density and dynamic viscosity of the air, were kept. They have already been given in Table 3, but they are repeated below:

Fluid properties		
Density	1.18415	kg/m3
Dynamic Viscosity	1.86e-05	Pa.s

Table 4. Fluid properties – gas: air at standard conditions

X. MODELING OPTION SELECTIONS:

The choice of the physics model determines which equations will be used by the software, and which assumptions will be made (eg. constant density). Choosing an appropriate model is essential in order to converge toward the solution. It can also significantly decrease the calculation time (eg. segregated flow vs. coupled flow). The modeling options are detailed below:

- **gas**: the fluid going through the campus is air, more or less humid. In the software, it just needs to be put as gas, without précising its nature. The density and viscosity properties allow to model air at standard conditions.
- **segregated flow**: this assumption is required to use the k-epsilon model: it could diverge otherwise. It makes the simulation quicker than when using the coupled flow option. Not to go too much into the details, this modelling option allows the software to solve the flow equations independently, which as a result saves calculation time.
- **constant density**: the air going through the campus is supposed to have the same density everywhere. That implicates that the temperature field is also constant. This assumption considerably decrease the calculation time in comparison to a gradient of temperatures. Another justification of this modelling option is the lack of data regarding the temperature in KTH campus.
- **turbulent**: the air flow is obviously turbulent. The Reynolds number is calculated below, and its value support this assumption.
- **k-epsilon turbulence**: k-epsilon is the standard model for separated flow under full turbulence. It is particularly appropriated for this kind of external simulation with a lot of obstacles. It is also relatively fast to converge.
- **steady**: the flow is supposed to be steady, as the wind never stops blowing.

It is necessary to calculate the turbulence parameters in order to get the right wind profile. The first thing to calculate is the characteristic length L. Here, it is equal to the hydraulic diameter, whose value can be assessed by analogy with a square section pipe:

$$L = D_H = \frac{4 * Section\_Area}{Perimeter} \quad (1)$$

The corresponding square section pipe would have the same height as the octagon defining the domain. Its length would be the distance between the two parallel sides which are neither the velocity inlets nor the pressure outlets. They were measured in the 3d-model, and are respectively $h = 128\ m$ and $l = 158\ m$. As a consequence, the characteristic length is about 141 m.

The Reynolds number is defined as the ratio of the inertia forces by the viscous forces:

$$Re = \frac{\rho * u * L}{\mu} \quad (2)$$

The Reynolds number is a dimensionless number characteristics of the flow regime. In order to get coherent results, it is crucial to give it the same value as the one in the reality. The scaling ration has been assessed in section III. As the characteristic length is 8.68 times smaller in the model than in the reality, the velocity of the wind must be 8.68 times larger in the model in order to keep Re unaffected.

The real wind velocity can be assessed using wind reports for Stockholm that can be found on the internet [6]:

Wind Data	
Average velocity - reality	$5\ m/s$
Average velocity - model	$45\ m/s$

Table 5. Input parameter - Wind inlet velocity

The wind speed is highly susceptible to be higher than the average value of 5 m/s, as the wind mainly comes from the South West in Stockholm. This is illustrated in Figure 10:

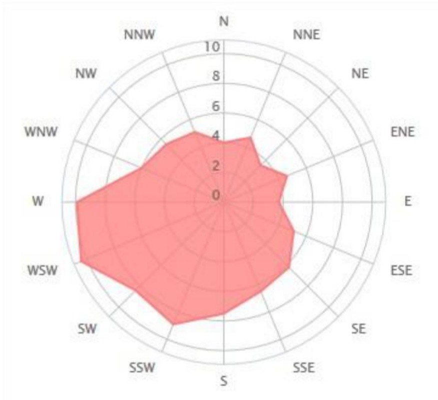

Figure 10. Wind distribution (%) over the year at Stockholm

As a consequence, the Reynolds number is about $427.6\ e8$. The turbulence intensity is defined for an intern flow as:

$$I = 0.16 * Re^{-\frac{1}{8}} \quad (3)$$

By analogy with a square pipe, the turbulence intensity is about $I = 0.013342$. The viscosity ratio, β, can be assumed to be 100. The length scale, l, is equal to:

$$l = 0.07 * L \quad (4)$$

This length gives an order of magnitude of the vortex forming behind the walls. Here it is about 10 m. Finally, the k and epsilon parameters can be assessed from the other parameters:

$$k = \frac{3}{2} * (U * I)^2 \quad (5)$$

$$\varepsilon = C_\mu^{\frac{3}{4}} * k^{\frac{3}{2}} * l^{-1} \quad (6)$$

By assuming that the constant is about 0.09, k and epsilon are respectively 0.5407 and 0.006. The three different set of data characterising the turbulence are summarised in the table below:

Turbulence specifications	
K + Epsilon	$(k = 0.5407\ ;\ \varepsilon = 0.006)$
Intensity + Length Scale	$(I = 0.013342\ ;\ l = 9.87)$
Intensity + Viscosity Ratio	$(I = 0.013342\ ;\ \beta = 100)$

Table 6. Turbulence specifications

XI. Solution algorithm choices

The flow was assumed to be segregated for the reasons given in section X. The default values were kept for the other algorithm parameters. They are summarized below in Table 5:

Continua / Physics / Models	
Constant Density	
Gas / Air / Material Properties	
Density	Constant
Dynamic Viscosity	Constant
Gradients	
Gradient Method	Hybrid Gauss-LSQ
Limiter Method	Venkatakrishnan
K-Epsilon Turbulence	
Realizable K-Epsilon Two-Layers	
Two_Layer Type	Shear Driven (Wolfstein)
Curvature Correction Option	Off
Convection	2nd-order
Reynolds-Averaged Navier-Stokes	
Segregated Flow	
Convection	2nd-order
Steady	
Three Dimensional	
Turbulent	
Two-Layer All y + Wall Treatment	

Table 7. Input parameters – physics model and algorithms

XII. Iterative convergence criteria choices:

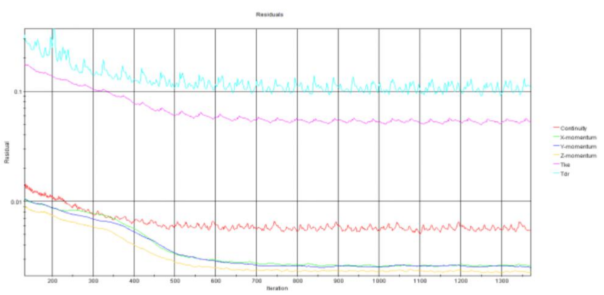

Figure 11. Evolution of the resilduals over 2000 iterations

It is necessary to specify stopping criteria for the simulation. The maximum number of steps was put at 2000 iterations, One can see in Figure 11 that the simulation was quite slow to reach such an accuracy (it took about 1000 steps).

XIII. Results

A. *Wind profile over the KTH campus*

At the end of the simulation, one can see the wind profile over the KTH campus by creating a section plane. That requires to choose the heights at which the wind velocity profile will be visualize. Depending on the type of turbine chosen, the height of interest will change. VAWTs are generally smaller than 20 m. The wind velocity profile at heights ranging from 10 m to 30 m are given in Figures 12 to 14 respectively.

Each of the building is taller than 10 m. As a consequence, at this height the wind velocity profile is greatly dependant on the location of the buildings:

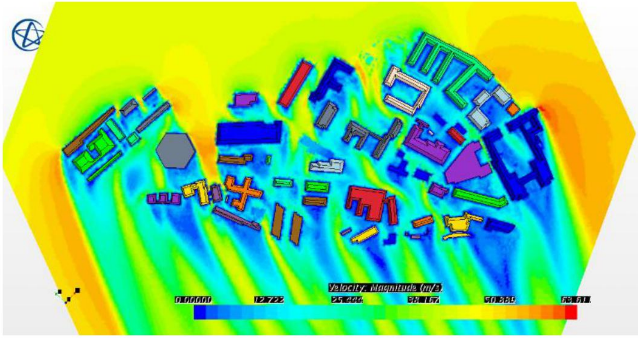

Figure 12. Scalar scene - velocity field at 10 m

At 20 m high, most of the wind flows over the buildings, but the depression triggered by the buildings can be seen on the velocity of the wind:

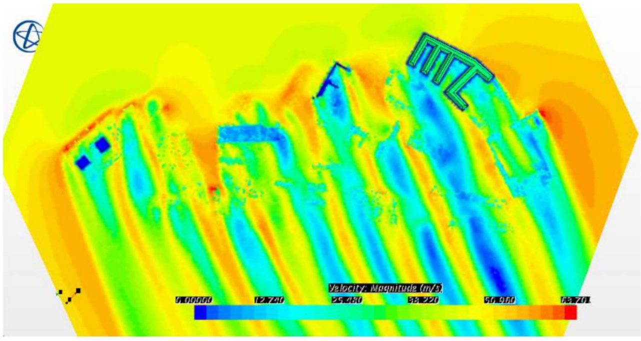

Figure 13. Scalar scene - velocity field at 20 m

At 30 m high, the winds totally passes over the buildings. The wind maximum velocity is a bit lower than at 20 m, mainly because there is no bottleneck effect anymore:

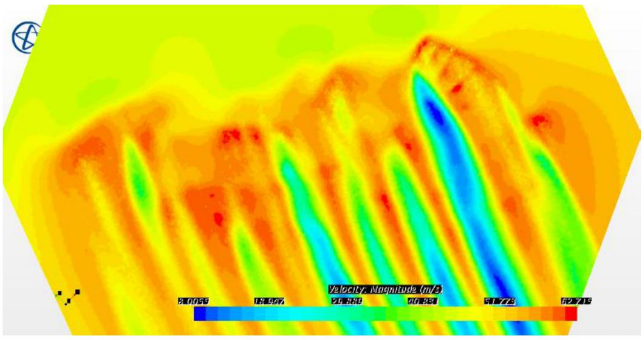

Figure 14. Scalar scene - velocity field at 30 m

A vector scene is useful to look into details at the wind direction close to the areas of interest. One can for instance see on Figure 15 that the buildings act as a bottleneck close to the B-building. As a result, the wind speed increases, as well as the mass flow taking this way.

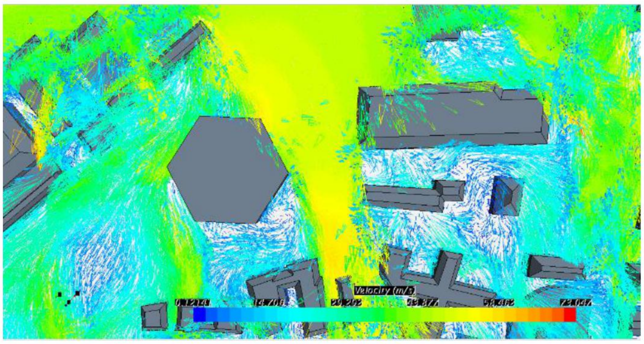

Figure 15. Vector scene – wind velocity magnitude next to the B-building

A similar phenomenon appears behind the V-building: the wind is concentrated in one point by one of the walls. As a result, the wind speed is relatively high at this particular location.

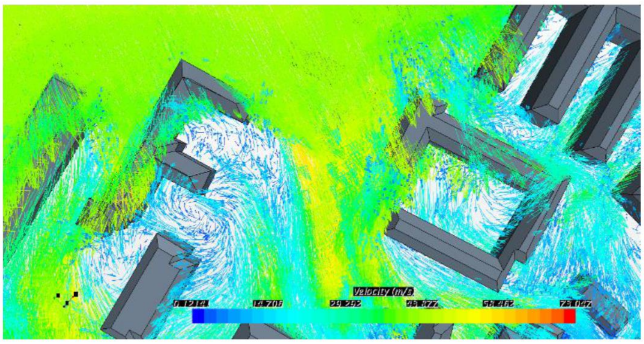

Figure 16. Vector scene – wind velocity magnitude next to the V-building

The wind reaches the speed of about 70 m/s in the simulation, which corresponds to 9 m/s in the reality. These winds are quite slow to produce electricity out of it.

B. Where to place the wind turbine?

There are several suitable places to put a wind turbine on the KTH campus. Among them, the location close to the B-building is particularly interested. A satellite view of the location is presented in Figure 17:

Figure 17. Possible location for a wind turbine, B-building

The octagonal building creating the bottleneck effect does not exist in reality. Instead, there are trees which do not guide the wind in the same way as a rough wall.

C. If a second wind turbine was to be constructed …

If a second wind turbine was to be constructed, it would surely be behind the V-building. The wind is lower in this area, but it is sufficient to legitimate the construction of a wind turbine.

Figure 18. Possible location for a wind turbine, V-building

XIV. Discussion

The two proposed locations would not be suitable in reality, as they are surrounded by trees which affect the wind. Globally, they slow it down. The relief over the KTH campus are neither taken into account in the 3d-model. For instance, the location next to the B-building is in a drop. This is the reason why there is no wind at this particular location. What seems to be the best location for the wind turbine appears to be the worst in the real case.

Others locations surrounding the KTH campus are also subject to high velocity winds. However, the simulation does not take into accounts the forest in this area, which significantly slows down the wind and even change its direction. Thus, these locations cannot be retained for the emplacement of the wind turbine.

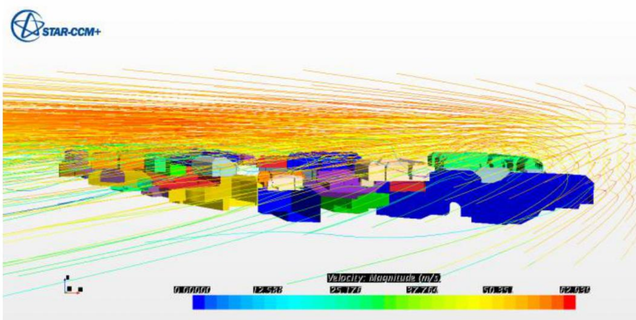

Figure 19. Streamline scene – wind flowing over the buildings

One could conclude that there is no appropriate location over the KTH campus to build a wind turbine. In a larger extent, this project highlights the limits of a simulation software: it totally relies on a model. Here, the neglect of the relief and trees leads to inaccurate results. That is also the reason why companies spend so much money on getting accurate on-site wind measurements.

XV. SUMMARY

As there is a high density of people during work days at the KTH campus, it is not possible to install a Horizontal Axial Wind Turbine (HAWT). The only alternative left is to build a Vertical Axis Wind Turbine (VAWT), despite its lower power output and efficiency. VAWTs require strong up-flow winds to operate. Such conditions can be found in the vortex created by buildings. The simulation has highlighted two high potential locations for installing the wind turbine. However, when confronted with reality, both locations turned out not to be suitable. This is the consequence of approximations taken into the model.

XVI. REFERENCES

[1] International Energy Agency (IEA), "Sweden: Electricity and Heat for 2012," IEA, 2012.

[2] P. Jamieson, "Innovation in wind turbine design," John Wiley & Sons, 2011.

[3] M. Raghed, "Vertical Axis Wind Turbines," 2015.

[4] Today's energy solutions, "VAWT vs. HAWT technology," 2015. [Online]. Available: http://www.onlinetes.com/wind-sail-type-vertical-axis-wind-turbines-111011.aspx.

[5] The Engineering Toolbox, "Roughness & Surface Coefficients of Ventilation Ducts," 2015. [Online]. Available: http://www.engineeringtoolbox.com/surface-roughness-ventilation-ducts-d_209.html.

[6] Wind Finder, "Wind and Climate in Stockholm - Bromma," 27 04 2015. [Online]. Available: http://www.windfinder.com/windstatistics/bromma.