CFD Analysis to Evaluate a Passive Ventilation System for a Building in Hot and Humid Climate – Auroville Language Laboratory, India.

Santi Qirko - Transsolar Academy 6 - Transsolar Energietechnik GmbH - 2019

Abstract

This paper explains the process of using a steady state CFD analysis in OpenFOAM to simulate pressure coefficient (Cp) data for evaluating a passive ventilation system in an existing building in Auroville, India. Three external points on the building are analyzed in this study: The inlet tower, the exhaust shaft, and a room window. The major finding of the study is that the proximity of dense vegetation close to the building has a dominating influence on the pattern of the wind flow around the building. In addition, the position of the evaluation points in relation to the wind direction i.e. windward or leeward, plays an important role in affecting the Cp values. Four wind directions are simulated based on the weather data. In all the cases, a pressure drop between the inlet tower and the exhaust shaft is observed. These results serve to indicate that there is a potential for air flow in the ventilation system in the desired direction. It is also found that the impact of the exact geometry & location of the inlet in the inlet tower is negligible compared to the impact of the turbulence caused by the surrounding vegetation. These findings help to identify and understand the primary factors that affect the Cp values in the building. These data are further used as input in a separate network flow simulation software (TRNFLOW) to further evaluate the dynamic annual performance of the passive ventilation system, which is discussed in another paper.

Introduction

The language laboratory in Auroville, India is a building that has a passive ventilation system composed of three air inlet towers, coupled with an underground tunnel and exhaust shafts. When fully complete, there will be a constant flow of pre-cooled and dehumidified air throughout the building. The pre-cooled air will come from 3 wide air inlet towers located outside the main building. It will be distributed throughout the building to every room through a network of wide underground air channels and vertical air shafts (Figure. 1). The exhaust shafts will be located on the external façade. The central courtyard dome roof is designed to create a vacuum effect, which draws out the hot air while creating an under-pressure that will suck the cool and dehumidified air in from the towers.
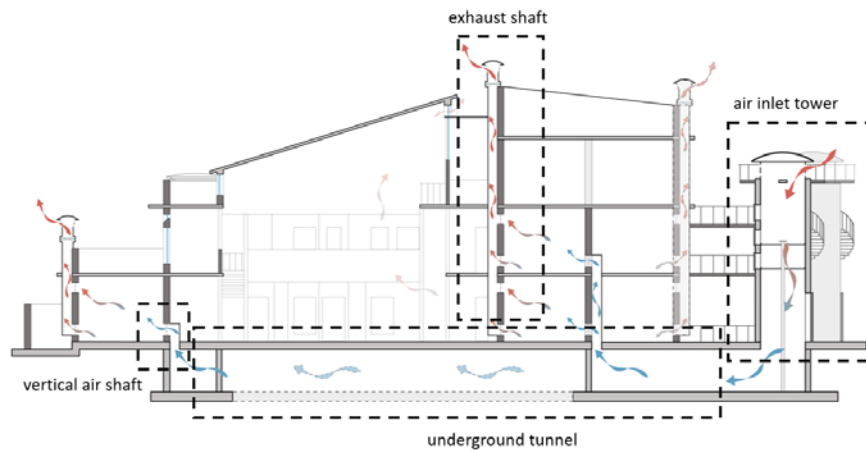
Figure 1.Passive ventilation system of the building

In order to test the effectiveness of the system, there are different parameters that need to be evaluated. First, the existing system is tested simulating an airflow network in TRNFLOW (a module of airflow network for coupled simulation with the multi-zone building of TRNSYS). The aim is to measure all the air parameters (temperature, humidity, velocity, pressure) in certain nodes of the network to define later strategies that will optimize the system: Sizing the elements that compose this system, usage of active system to enhance the airflow, designing the inlet towers to increase their efficiency, usage of active or passive systems to control the air humidity.

According to the TRNFLOW manual, the air flows in a building can be calculated with a multizone air flow model. Such a model idealizes the building as a network of nodes and air flow links between them. The nodes represent the rooms and the building surrounding. The links depict openings, doors, cracks, window joints and shafts as well as ventilation components like air inlets, outlets, ducts and fans. The wind pressures on the façade, and the indoor and outdoor air temperatures are the important input parameters.

For each external node, a set of Cp-values for as many wind directions as desired can be defined. At least one external node per façade with area-averaged Cp-values is necessary. Cp-values are always referenced to wind velocity at a certain height. This reference height should be given together with the Cp data set. Usually, it is the height of the eaves of the building. (Transsolar Energietechnik GmbH, 2009)

Weather data for Auroville are gathered and analyzed to set up the air temperature and velocity as boundary conditions for the simulation (Figure. 2).
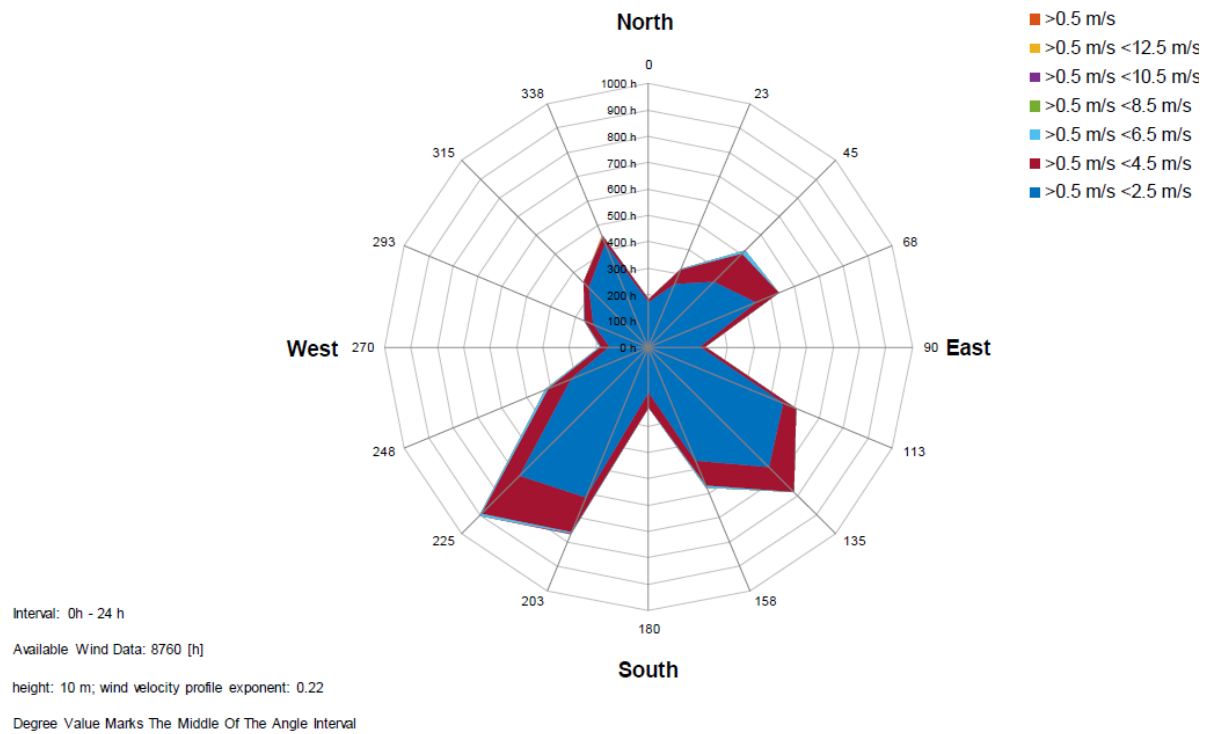
Figure 2. Yearly wind rose

According to the climate analysis, the most wind comes from the southwest and southeast directions, however, there are high levels of wind in all directions yearly, with an average wind speed of 2.5 m/s. Wind speeds at night are more common from Northwest and Southwest. The average wind speed is also 2.5 m/s.

To calculate the Cp values for the boundary conditions in TRNFLOW, a CFD simulation is made using OpenFOAM.

Creating the 3D Model and the Surrounding Vegetation

The 3d model of the building is created using 2d draftings and pictures that are provided by the client. The model is created using the 3d software Rhinoceros (Figure. 2).
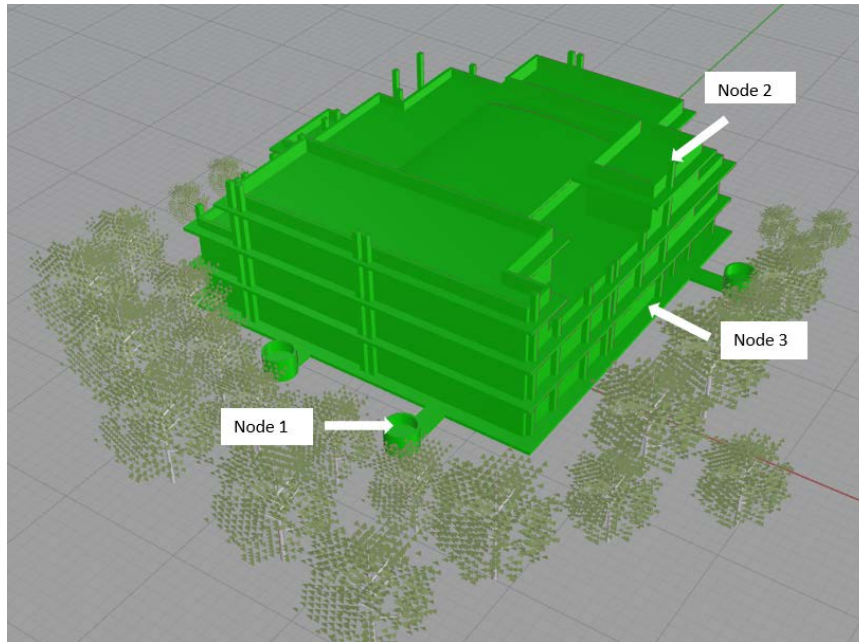
Figure 3. 3D model of the building and the surrounding vegetation. Location of the nodes.

Data regarding the type, dimensions, and proximity of the vegetation surrounding the building are collected on-site by the client and used to create the 3d environment. A tree model with a porosity of 91% is simplified and used to create the surroundings (Figure. 4).
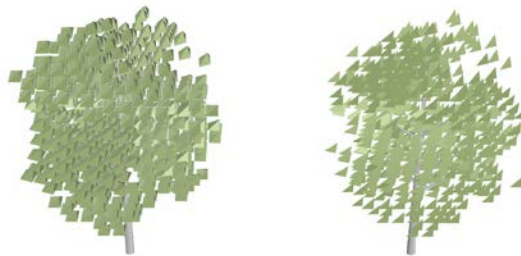

Figure 4. On the left side the 3d model of the tree with 91% porosity. On the right side the simplified model

The vegetation can be categorized into 3 groups: Small, medium and large trees (Table. 1).

Tree category	Height (m)	Tree radius (m)	Trunk size (m)	Porosity (%)
Small	6.8	1.7	0.15	91
Medium	8	3.2	0.30	91
Large	10.7	4.2	0.36	91

CFD Case 1

The first case is created in order to evaluate and understand the best meshing parameters and processing time for those parameters.

Three external locations are chosen in the building to be analyzed and evaluated in the CFD simulations. The first element is the inlet tower in the south-west façade, the second element is the outlet tower in the south-east façade, and the third element is the window in the south- east façade. The wind directions chosen to be analysed are: south-west, south-east, north-west, north-east (Figure. 2).

First the 3d model of the building and the vegetation is tested to understand if the meshing is acceptable for the expected results. Then the model is simulated using OpenfFOAM to understand if the case would produce acceptable results in terms of values and processing time.

Setting up the Environment

To set up the environment, the blockmesh is created in relation to the size of the buildings geometry and the rules shown in Figure. 5.
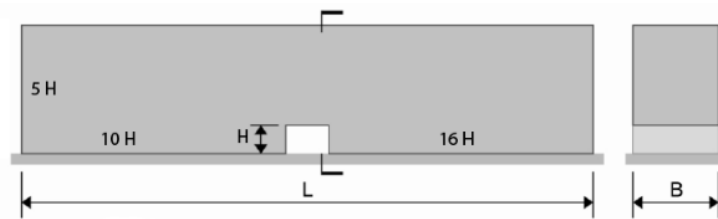

Figure 5. Suggested dimensions for the blockmesh.

Table. 2 shows the dimensions of the 3d model

Description	Dimensions width x length x height (m)
Buildings outside dimensions (including the inlet towers)	40 x 44 x 13
Scene dimensions (building and vegetation)	57 x 59 x 13
Simulation domain dimensions	180 x 360 x 60
Ground dimensions	207 x 383

The number of cells defined for the block mesh for both cases is 5 x 5 x 5m.

SnappyHexMesh

The geometry is divided into two files: 'building.stl' (building and vegetation) and 'ground.stl'. The smallest dimension of the surface where Cp value need to be collected is 0.45m (the outlet shaft). There are needed at least three cells on this side of the surface to evaluate an average Cp value (Figure. 6).
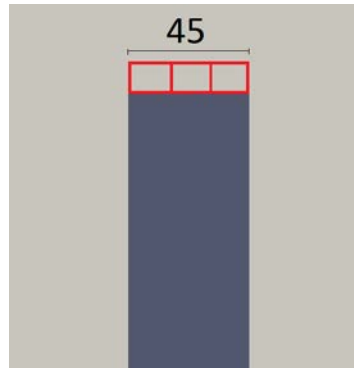
Figure 6. Section of the exhaust shaft showing the minimum number of cells to evaluate an average Cp value

The smallest cell size should be 0.45/3=0.15m. The formula defining the cell size is: BlockMesh/2^n, where 'n' is the level number. In this case we have: $5/2^5$= 0.156m. For this reason, meshing level 5 is used for the building to get an accurate amount of data. The maximum cells for the model is set to be 13000000. The number of processors used to run the meshing is 22, so the maximum local cells is: 13000000/22=590000 . Table. 3 shows the snappy Hexmesh boundaried that are used for the case.

castellatedMeshControls	
building.eMesh	Level 4
building	Level 5 5
Ground	Level 2 2
refinementRegions	
mode inside	Level 5 5
maxLocalCells	590000
maxGlobalCells	13000000
snapControls	
nSolverIter	500
Settings for the layer addition	
Building – nSurfaceLayers	3
Ground – nSurfaceLayers	3
expansionRatio	1.05
FinalLayerThickness	0.2
nLayerIter	100
errorReduction	0.75

Table. 4 shows information regarding the mesh results.

Meshing time	41 minutes
Number of cells	3796395
Max skewness	14.3831
Number of skewed faces	1318

Case 1 is meshed successfully. The number of cells in the horizontal surface of the exhaust shaft is enough to gather data for the Cp values (Figure. 5).
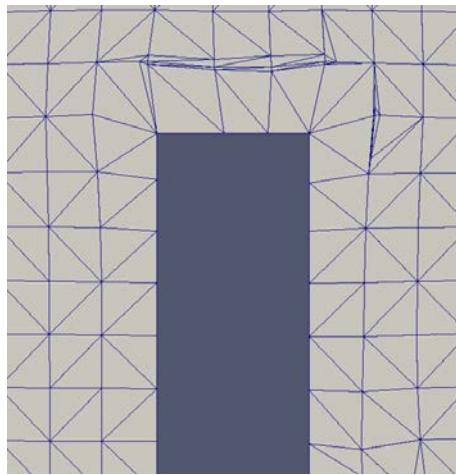

Figure 7. Section of the exhaust shaft showing the number of the cells on the horizontal surface.

Calculation of the Turbulence Properties

The yearly average wind speed and wind directions are collected from the Auroville weather data. In this case, the wind direction is defined to South-West. The atmospheric boundary layer is chosen as 'Agricultural landscape'.

Table. 5 shows the Turbulence properties used for the case.

Freestream velocity	2.5 m/s
Turbulence kinetic energy	0.0234375
Turbulence dissipation	0.3295898
Specific turbulence dissipation	156.25
Turbulence intensity/level	5
Kinematic viscosity	1.5e -5
Eddy viscosity ratio	10
Turbulence length scale	0.5%

Table. 6 shows the parameters set for the simulation.

Number of iterations	2000
fvSolution	
P tolerance	1e -9
U tolerance	1e -8
K tolerance	1e -8
Epsilon tolerance	1e -8
Omega tolerance	1e -8

Postprocessing and results

From each element are collected different Cp values that are measured in different positions. The reason is to get an average value for each external node and to understand how the values change in different positions of the same node. The first location is the inlet tower. From the horizontal surface of the inlet tower are collected 5 Cp values in different positions, at the height of 2m (Shown in magenta colour) (Figure. 6).
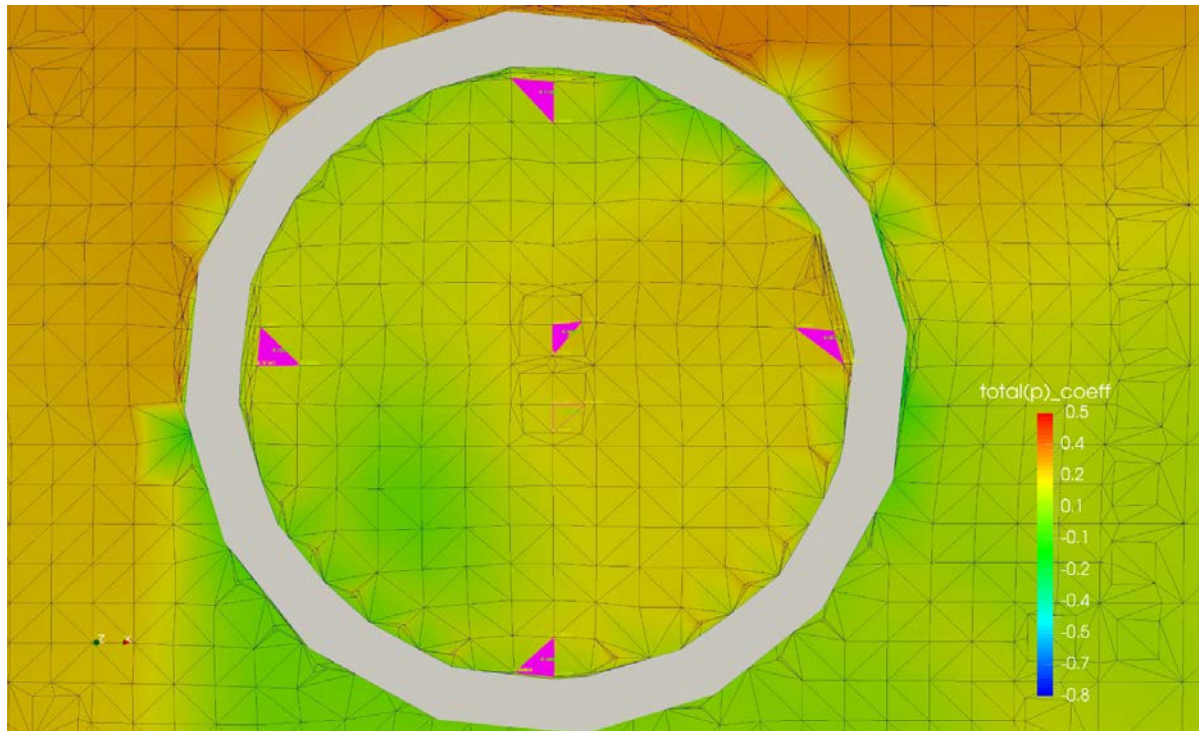

Figure 8. Inlet tower Cp values.

This inlet tower is located on the south-west façade which is also the wind direction for this case. From the results we can say that all the Cp values on this node are positive. This is because the wind flow is affected by the vegetation in front of the tower. After passing through the vegetation, the wind creates a swirl effect that pushes air inside the inlet tower (Figure. 7).
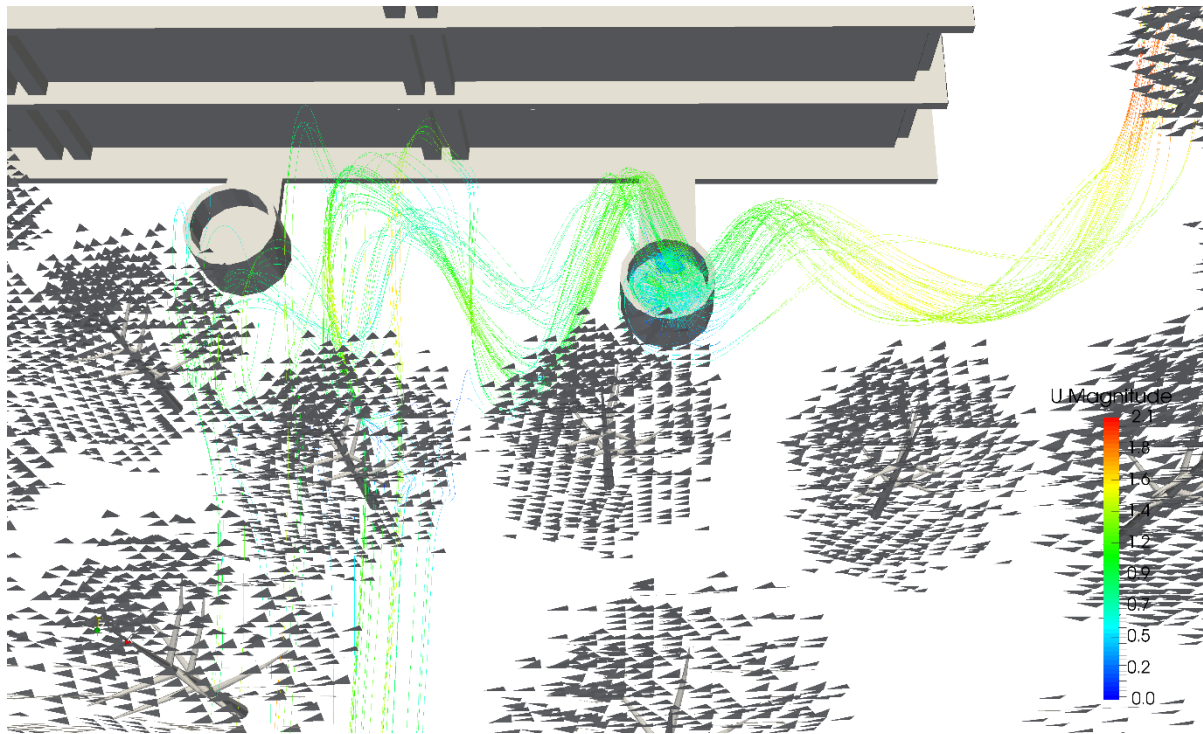

Figure 9. Wind flow affected by the vegetation in front of the tower.

The second location is the exhaust shaft. In this node are collected 5 Cp values on the horizontal surface at the height of 13.1m (Shown in magenta colour) (Figure. 8).
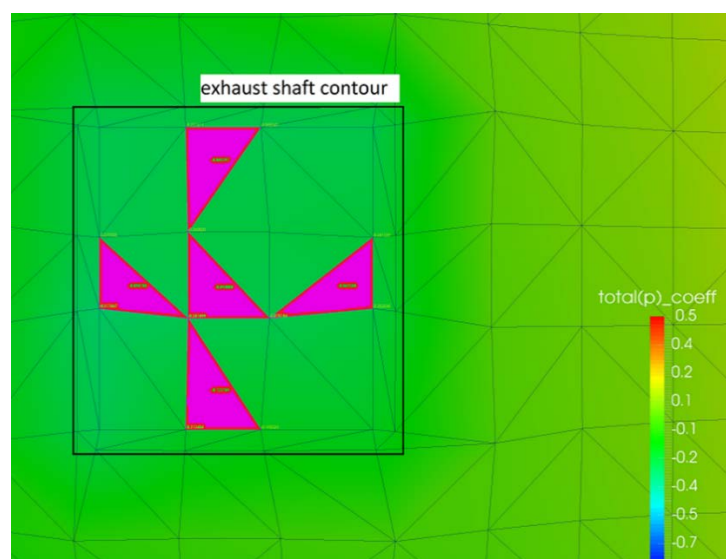

Figure 10. Exhaust shaft Cp values

All the Cp values on the exhaust shaft are negative. The wind flowing on the surface of the shaft is not affected by the vegetation or other obstacles. The surface of the shaft is at the height of 13.1m, which is the highest point of the building. The wind flows on the horizontal surface creating negative Cp values. (Figure. 11).
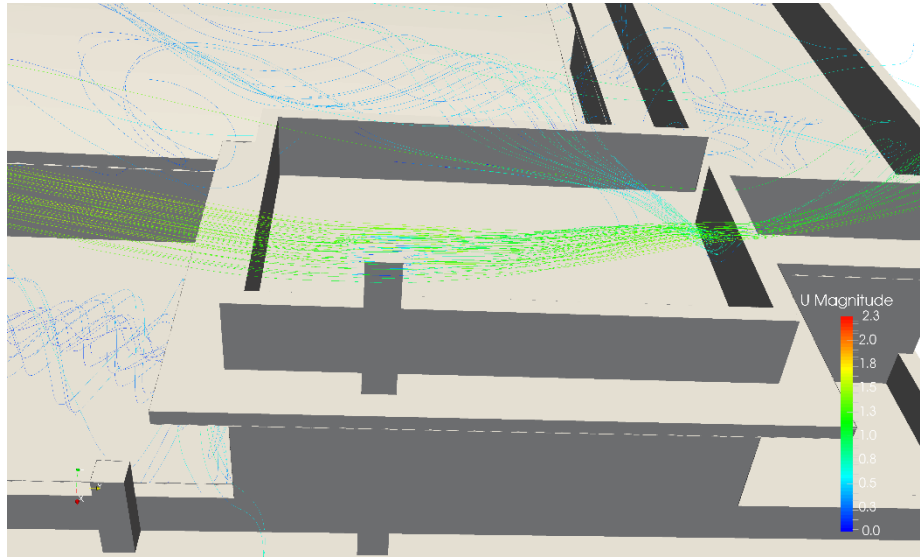

Figure 11. Wind flow on the horizontal surface of the exhaust shaft

The third location is the window located on the South-East façade. From the window are collected 9 Cp values (Shown in magenta colour) (Figure. 10).
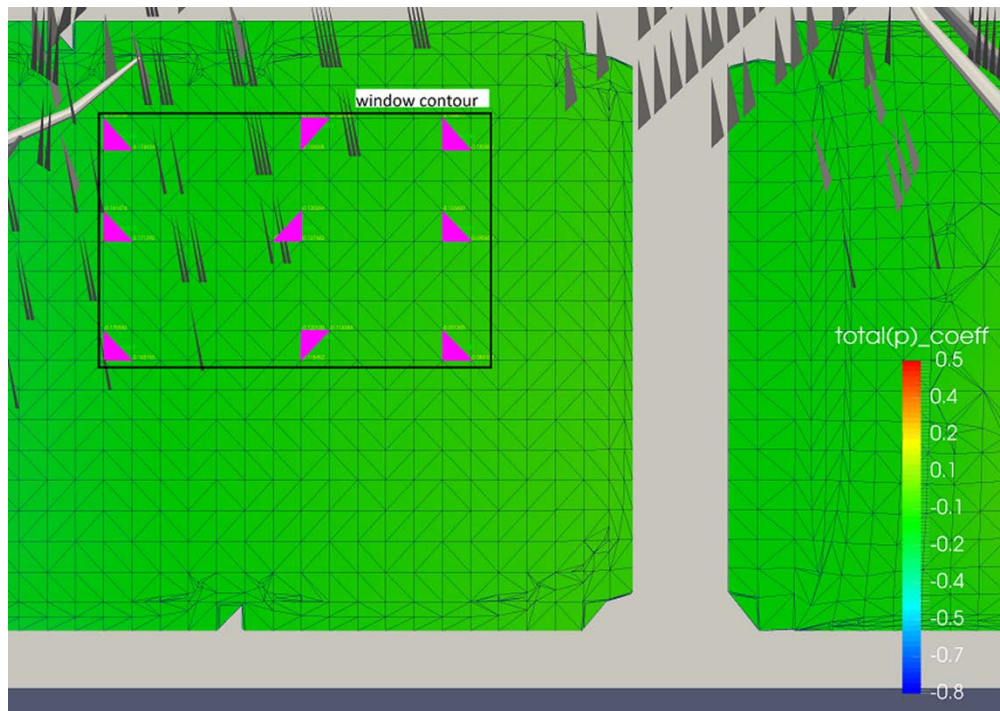

Figure 12. Window Cp values

The wind that comes from the south-west direction creates negative Cp values on the part where the window is located (Figure. 11).
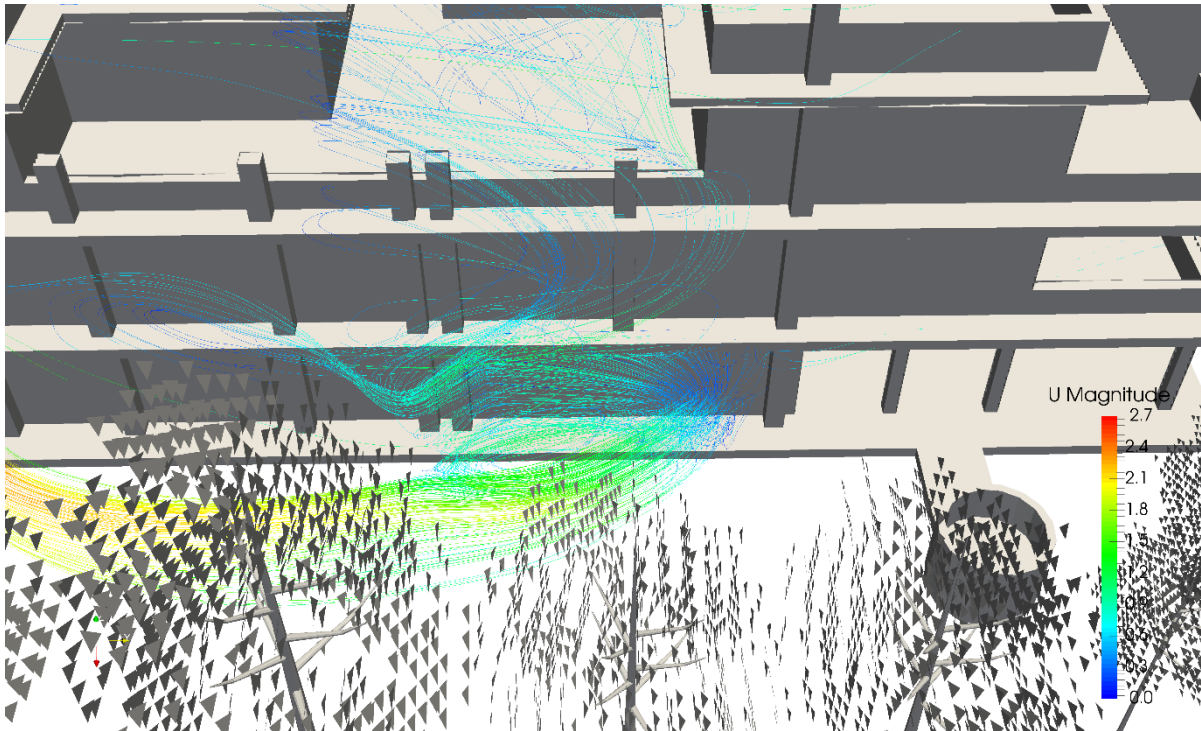

Figure 13. Wind flow affected from the building's geometry.

CFD case 2

This case is created to test if the effect of the wind on the inlet tower would be different if the values are collected from inside of the tower instead of the surface. For this reason, the 3d model of the inlet tower is changed so that the values can be measured inside its volume (Figure. 12).
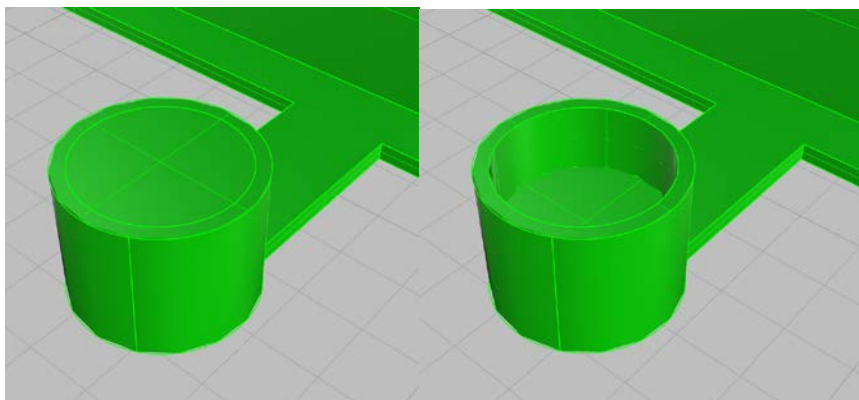

Figure 14. The old inlet tower on the left side and the new one on the right side

The meshing and the simulation process is done using the same parameters as in the first case.

The Cp values for the inlet tower are collected at the height of 1.8m (or 0.2m inside the tower).

Table. 7 shows a comparison between the Cp values of case 1 and case 2.

Case	Position	Cp 1	Cp2	Cp3	Cp4	Cp5	Average Cp
1	Inlet Tower	0.142	0.112	0.206	0.164	0.206	0.166
2	Inlet Tower	0.073	0.114	0.046	0.115	0.161	0.102

The results show that even though there is a small difference between the Cp values (due to the height difference of the collected valus), the effect of the wind flow in case 2 is the same as in case 1 because it is affected by the vegetation and not by the shape of the tower. After this conclusion, the other scenarios were the wind changes direction, are simulated using the 3D model of case 2.

CFD Final Results

The final Cp values for the three locations are assembled in Table. 8.

Case	Position	Cp 1	Cp2	Cp3	Cp4	Cp5	Cp6	Cp7	Cp8	Cp9	Average Cp
1. Wind direction SW	Inlet Tower	0.142	0.112	0.206	0.164	0.206					0.166
	Exhaust shaft	-0.099	-0.086	-0.064	-0.124	-0.095					-0.093
	Window	-0.154	-0.117	-0.094	-0.085	-0.116	-0.151	-0.151	-0.108	-0.082	-0.117
2. Wind direction SE	Inlet Tower	-0.159	-0.325	-0.153	-0.154	-0.187					-0.196
	Exhaust shaft	-0.057	-0.042	0.039	-0.124	-0.127					-0.062
	Window	0.319	0.316	0.316	0.323	0.324	0.335	0.363	0.347	0.349	0.332
3. Wind direction NE	Inlet Tower	-0.111	-0.111	-0.112	-0.111	-0.111					-0.111
	Exhaust shaft	-0.214	-0.212	-0.213	-0.222	-0.210					-0.214
	Window	-0.216	-0.217	-0.218	-0.218	-0.217	-0.215	-0.216	-0.218	-0.219	-0.217
4. Wind direction NW	Inlet Tower	-0.320	-0.385	-0.241	-0.215	-0.162					-0.265
	Exhaust shaft	0.079	0.092	0.115	0.085	0.073					0.089
	Window	-0.119	-0.120	-0.121	-0.119	-0.118	-0.118	-0.119	-0.119	-0.119	-0.119

Conclusions

After the simulated data for the three external points in the model, we can say that the direction of the wind, the vegetation surrounding the building and the position of the surface where the data is measured, are factors that affect the Cp values.

In all the four cases, there is a difference between the Cp values measured in the inlet tower and the exhaust shaft. This difference suggests that there might be airflow in the ventilation system. The highest difference is seen in the case when wind comes from the north-west and the lowest difference is seen in the case when the wind comes from north-east.

The inlet tower has positive Cp values when is facing the wind direction in the south-west. This is because the wind creates a swirl effect from the vegetation in front of the tower. This effect pushes the wind in the tower, creating positive Cp values (Figure. 9). The lowest Cp values for this point are collected when the wind comes on the opposite direction.

The exhaust shaft has positive Cp values when the wind comes from north-west. This is because the wind is affected from the roof parapet in a way that creates positive Cp in the horizontal surface of the exhaust shaft (Figure. 13).
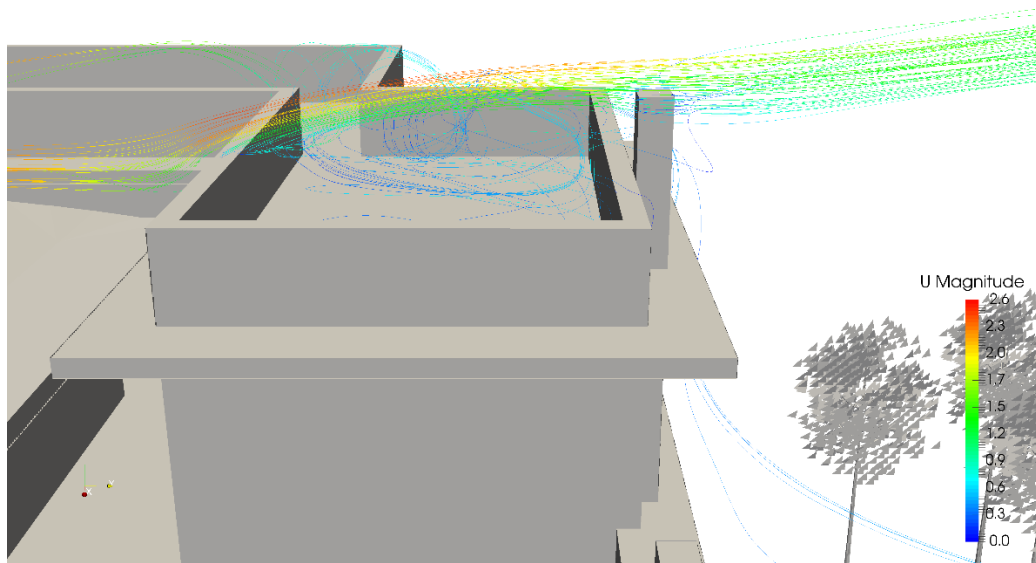

Figure 15.Roof parapet affecting the wind flow on the exhaust shaft.

The lower Cp values are recorded in the case when the wind comes from the north-east.

The third node has the highest Cp value when it is facing the wind direction, which in this case is south-east. The lowest Cp values are collected when the wind comes from the north-east. The wind is not facing any obstacles when flowing in the surface and creates negative Cp values.

These findings help to identify the factors that affect the Cp values in the building.

Bibliography

Transsolar Energietechnik GmbH. (2009). *TRNFLOW Manual.* Manual, Stuttgart.

Transsolar Energietechnik GmbH. (April 2017). OpenFOAM Documentation. Stuttgart, Germany.