Chapter Five

CFD Applications in Wind Studies

5.1. Introduction

Chapter Four discussed different techniques used for the analysis of wind pattern and air movement in urban areas. The techniques include wind tunnels and CFD program. The chapter concluded that CFD program is the best because it is available, easy to use and accurate with results.

This chapter is devoted to study CFD as one of the important tools for wind and air analysis in urban areas. It has underpinned different examples (towns and areas) from different parts of the whole world where CDF is employed to analyze wind pattern and air movement. The chapter also tends explore and understand CFD techniques and methods of analysis.

The CFD simulates or predicts of fluid using a computer. It is also a computer modeling of fluid behavior, for instance the flow of fuel and air. CFD refers to computational solutions of differential equations, such as Navier stokes set. Derived from mathematical equations, CFD has become an important and essential analysis tool in different disciplines especially engineering.

5.2. Use of CFD to study ventilation in urban area

The CDF program is used for different purposes to analyze wind pattern and air movement in urban areas. It can be used to study natural ventilation, role of air movement in reducing high temperatures in towns and urban environment. Different samples from different urban areas where CFD is used for wind or air analysis are discussed in this chapter in the following pages.

5.3. Campinas City, Brazil

A research on the use of CFD to studies of ventilation in urban areas in Campinas, Brazil was carried out by Oliveira, M.C.A, Labaki, L.C, and Vatavuk Paimod. The research aimed to study wind flow and natural ventilation in urban plots in the city of Campinas, Brazil, with input parameters data values of wind velocity and direction. The adopted model used the standard size of a typical block in the settlement Sao Jose, a typical self-construction settlement in the city. [Oliviera, et al, 2009]

The study has discussed the effect of urban block on natural ventilation of self-built houses in the city of Campina.. The research adopted CFD simulation, using the software PHOENICS (CFD) to calculate air velocities in the external region of the residences. [Oliviera, et al, 2009]

The problem of research is the negative impact of spontaneous housing on urban environment. Due to specific local urban growth patterns as well as to economic and social structures of the country, self-built houses, that is, houses built by owner families, make up a

Substantial percentage of Brazilian housing production, around 60% (Oliveira et al, 2005). Due mainly to low quality design solutions, self-built houses present on the whole a low environmental comfort standard. [Oliviera, et al, 2009]

Buildings of one up to five stories, the implantation usually follows the pattern of the urban design based on the orthogonal mesh and in a simple repetition of identical units. The plot dimensions are small, with geometry that limits design possibilities, and they are mainly rectangular narrow strips of land, to reduce street front dimensions and thus subdivision infrastructure costs. [Oliviera, et al, 2009]

Subdivision layout is not ideal for the sitting of desirable house designs. The orientation of streets does not take into account sun exposure and ventilation conditions. [Oliviera, et al, 2009]

There are three types of houses: [Oliviera, et al, 2009]

a. Houses built in the back of the plot.

b. Houses built along most of the sidewall.

c. Houses built in the center of the plot.

Due to small size of the houses and poor quality, the houses are extended horizontally and vertically to accommodate more family members.

According to Oliviera, et al the simulation using CFD is powerful tool for studying natural ventilation. The programme is useful for the numerical calculus or for the visualization of the wind flow indoors and outdoors [Oliviera, et al, 2009]. Much software defines fixed values for pressure coefficients, as well as turbulence. The values come from empirical models, based in wind tunnel studies. The presented analysis in such cases defines internal and external flow lines. [Oliviera, et al, 2009]

The study used the CFD simulation to quantify the positive or negative effect of the most common modifications in houses facades on natural ventilation and wind flow. [Oliviera, et al, 2009]

5.3.1. CFD Simulation

Phoenics 3.6 CFD program was employed in this research. The turbulence model was modified k-ε model by Chen and Kim (1987). The model includes extra time scale in the ε equation and several coefficients are adjusted to maintain the good agreement with classical experimental data. The simulation consist of many objects, which are defined precisely, these objects are: [Oliviera, et al, 2009]

1. Studied objects

Domain size in urban climate, the studied objects and the boundaries of the domain maintain a certain proportion to the height of the object. From the flow inlet surfaces to the objects, they propose a minimum distance of 8 to 10 times the object height, to the flow outlet surfaces a minimum of 8 times the height and that the height of the calculation domain is at least 6 times the object height. [Oliviera, et al, 2009]

According to Oliviera, et al (2009), the simulation was employed in two different stages:

In the first stage, the simulation model contained many blocks, including the characteristic block, and the immediate surroundings. The model was defined by a calculation domain with the size of 800m, 400m and 70 meters in directions x, y and z, respectively. [Oliviera, et al, 2009]

The mesh contained 125 subdivisions in the x direction, 56 in y direction and 100 in z direction, resulting in 700.000 cells in a Cartesian coordinate system. An upper view of the mesh can be seen in figure (5-1). It can be noticed that the mesh spacing is gradually varied using a geometric progression. The larger mesh spacing is about 20 m in the flow inlet region and the smaller spacing is about 1 m in the regions near the houses. [Oliviera, et al, 2009]

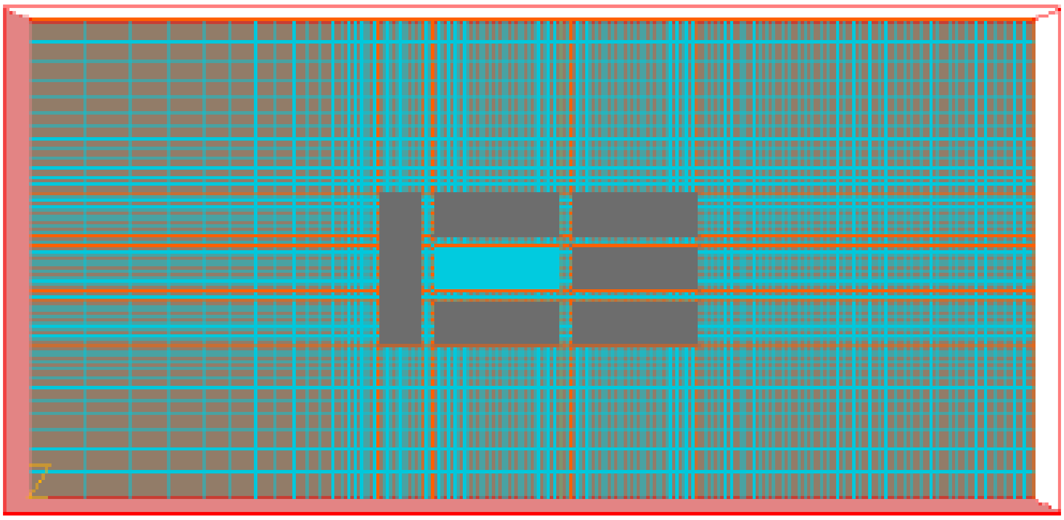

Figure 5-1 Created mesh with the studied characteristic block appearing in the center [Oliviera, et al, 2009]

Boundary conditions were applied in the six faces of the domain. Two flow inlet faces, two flow outlet faces and the lower and upper faces. [Oliviera, et al, 2009]

Wind speed and direction measured by INFRAERO, was inputted in the WRPLOT view program, which calculated the hourly medium velocities and directions. It can be seen the strong predominance of south east direction.

Material properties are selected in the (object) dialog box. Air was considered to flow without exchanging heat around concrete buildings, whose properties are shown in table 5-1. [Oliviera, et al, 2009]

Table (5-1), Wind speed (m/s) calculated in the frontal cells. [Oliviera, et al, 2009]

position	year	X							
Y		2	3	4	5	6	7	8	9
11	2005	0.34	0.39	0.44	0.47	0.48	0.49	0.48	0.47
	2000	0.45	0.47	0.48	0.53	0.62	0.70	0.78	0.86
12	2005	0.28	0.31	0.34	0.33	0.35	0.75	0.42	0.50
	2000	0.47	0.54	0.58	0.63	0.69	0.75	0.82	0.89
13	2005	0.20	0.21	0.23	0.23	0.24	0.30	0.41	0.50

	2000	0.50	0.57	0.63	0.68	0.72	0.78	0.83	0.91
14	2005	0.24	0.23	0.22	0.20	0.28	0.39	0.44	
	2000	0.47	0.57	0.64	0.69	0.73	0.78	0.83	0.91

Convergence of the calculations was carefully assessed. The convergence criteria suggested by Ferziger and Peric (2002) was used. According to researchers, the value of the residues obtained after a number of iterations can be compared to the residues of the first iteration to assess convergence. A reduction of the residues of about three to four orders of magnitude is recommended. [Oliviera, et al, 2009]

The second stage is the region containing the characteristics block. The follow inlet velocities used in this stage are the areas obtained in the first stage of the study. [Oliviera, et al, 2009]

The objects used for establishing the boundary conditions are modified only in the two flow inlet surfaces and depend on the values obtained in the first stage. To obtain the velocities in the entry cells that were located in the vertical planes directed to the south and east directions, [Oliviera, et al, 2009], an interpolation was made using the data of the first part of the simulation. With this intent, a program was written in C language that reads the data of the first stage of the simulation, interpolates the data, and writes a file that contains a series PHOENICS commands that establishes the boundary condition in each entry cell. [Oliviera, et al, 2009]. The executable program was generated using the DEV- C++ compiler. In part, two of the simulation, the file with the commands is read by PHOENICS, which executes them. The file has 33179 lines containing PHOENICS commands. [Oliviera, et al, 2009]. There is no need to use the WIND-PROFILE object that was used to establish the boundary conditions in the first stage of the study, because the flow inlet velocities are obtained from the first one in the second stage. The insertion of the characteristic block is done by the STLOUT command of PHOENICS, that imports the three dimensional drawing of the block created in CAD. [Oliviera, et al, 2009]

The characteristic block is formed by residences that were modified between 2000 and 2005, for comparison. The block contains models of the residences according to their design of the years 2000 and 2005. To obtain these data a survey was made with all residences in the neighborhood in those years. Figure 5-2 shows the same residence depicted in 2000 and 2005 respectively.

Figure 5-2- Residence depicted in São José neighborhood at different times, on the left in 2000 and on the right in 2005, [Oliviera, et al, 2009]

Plans have been modify within years 2002 to 2005. Figure 5-3 shows a residence plan in 2002, and figure 5.3 shows it in 2005 after modifications.

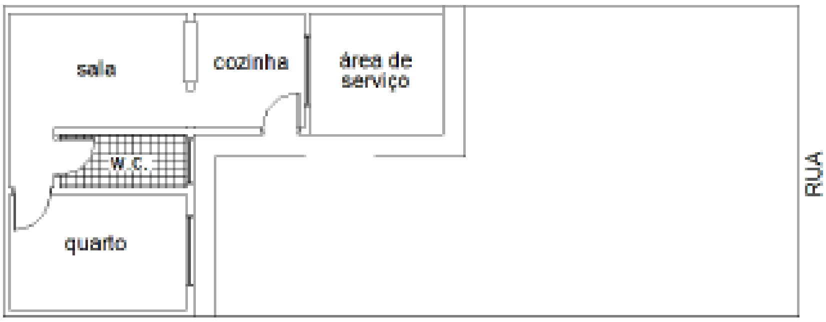

Figure 5.3 Plan of the residence of figure 5-2 as observed in 2000, [Oliviera, et al, 2009]

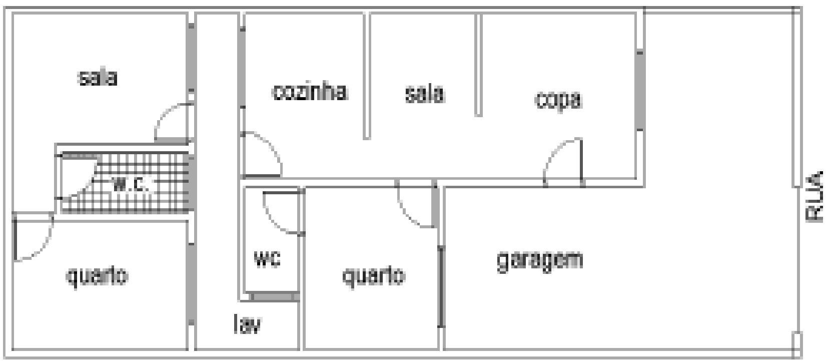

Figure 5.4 Plan of the residence as observed in 2005, [Oliviera, et al, 2009]

Survey was done randomly with 13 houses chosen in the Sao Jose neighborhood resulting in the standard block shown in figure 5.5. [Oliviera, et al, 2009].

Figure 5.5 Characteristic block used in the simulation. [Oliviera, et al, 2009]

5.3.2. Results:

The authors have arrived at the following results:

Figure 5.6 shows a top view with the calculated velocity values in the horizontal plane at 1 m height above the ground. [Oliviera, et al, 2009]

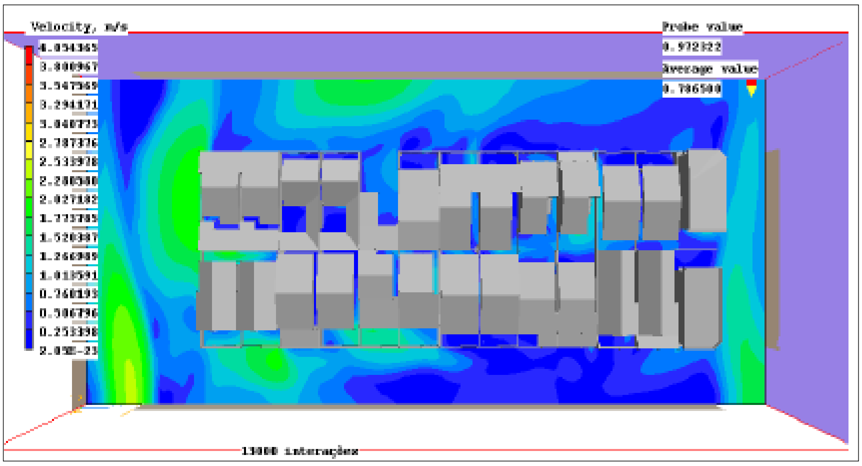

Figure 5-6 Upper view of the model implantation, [Oliviera, et al, 2009]

A number of iteration totaling 26000 is done. With this number of iterations, the reduction of the residues was five to six orders of magnitude for all variables except for wind velocity in direction z. The residues for this variable are very small at the start of the iterations and then they increase, since the final residues are four orders of magnitude smaller than the maximum residues it was considered that convergence was attained because fully the criteria of Perice Ferziger, 1990 was accomplished with a substantial margin of safety. [Oliviera, et al, 2009]

Part 2 of the modeling could be analyzed in a first stage under the aspect of the insertion of the block in the urban area. Part 1 of the modeling stipulates that wind speed in urban area is less than 2.5 m/s, for an average speed of about 4 m/s in the city of Campinas. The highest velocities, between 0.9 and 1.4 m/s occurred in houses 12 and 13, in the corner of the predominant flux. [Oliviera, et al, 2009]

Houses with façades oriented to SE (the direction of the prevailing winds) could be more benefited by ventilation if the width of streets and sidewalks was otherwise planned. In the existing reality (6m street width), the recirculation occurs mainly in the corners, which reduces wind speed, so that in the houses lots, it is not higher than 1 m/s. Figure 5-7 represents the streamlines that originate in the first cell of each lot, at the height z=1m, for the houses facing prevailing SE winds. [Oliviera, et al, 2009]

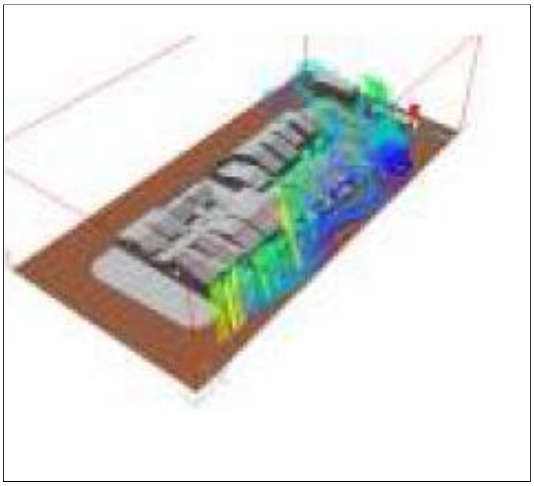

Figure 5-7 streamlines for houses facing prevailing SE winds. [Oliviera, et al, 2009]

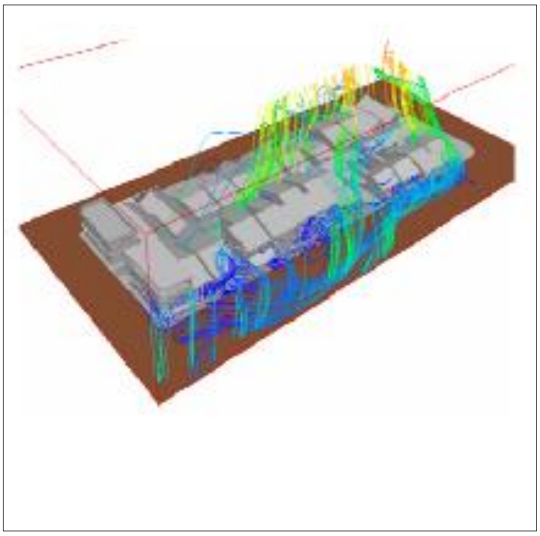

Figure 5-8 Recirculating wind flow in the street, consequence of the houses facing west [Oliviera, et al, 2009]

Houses facing east receive greater influence of the recirculation (vorticity) effect produced by the 6m street width in this neighborhood. In the west side, as shown in figure 5.9, the recirculation in the streets results from the flow passing through the houses facing west. [Oliviera, et al, 2009]

Resulting streamlines originated in the first cell in each lot, for z = 1m, both for east and west façades are shown in Figure 5.9. The urban design of the settlement São José, for low-income housing, does not allow blocks with four facades, only two, a common feature in Brazilian urban settlements. The most important factor considered in settlement design is the topographic leveling. Wind profile is usually not taken into account. [Oliviera, et al, 2009]

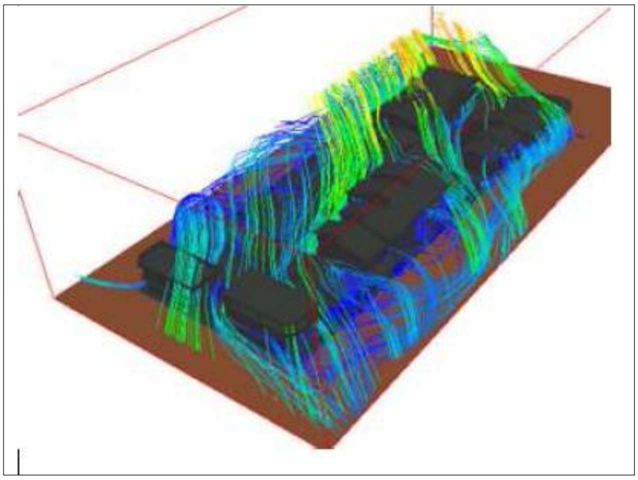

Figure 5.9 Upper view of the model implantation. [Oliviera, et al, 2009]

The author had come out with the following results:

It was observed that in the four residences, the wind flow reduced, in two the flow was about the same and in five; there was an increase of wind flow inside the terrain. [Oliviera, et al, 2009]

It was also observed that the position of the houses in the block affects the wind velocities. Some houses which were not modified had their wind flow changed due to the other houses and the pattern of the surroundings. [Oliviera, et al, 2009]

The authors presented the results, which they obtained as following:

Figure 5-10 presents the results for the two plans of the house shown in figure 5-2. The two side-by-side houses correspond to the plant of 2005 on the left and the plan of 2000 on the right. The wind speed is also presented in the figure and is measured in a height of 1 m. The values vary from 0.2 to 0.9 m/s.

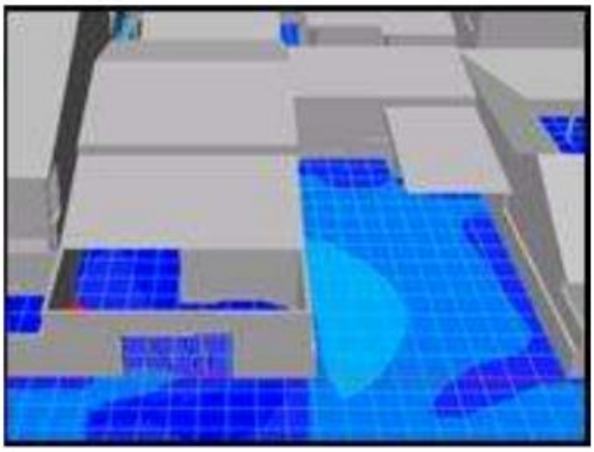

Figure 5.10 shows results for the two plans of the house depicted in figure 5-2. [Oliviera, et al, 2009]

Table 5-1, presents wind velocities calculated at 1m height at 32 cells of each plan of the house depicted in figure 5-2. These 32 cells correspond to the eight by four first cells in the front of the terrain.

5.3.3. Discussion

Finally, the researchers have come up with the following recommendations: [Oliviera, et al, 2009]

1- It is important to study simulations largely to reach a result with best possible exactitude. [Oliviera, et al, 2009]

2- Phoenics is very complex software, so the insertion of data should fulfill with the best clearness as possible. [Oliviera, et al, 2009]

3- Simulations had largely studied so that it could be reached a result with the best possible exactitude. It is important to state that PHOENICS is a very complex software, but the insertion of data was fulfill with the best clearness possible. [Oliviera, et al, 2009]

4- Low wind velocities and the tendency of wind to follow its natural flow contribute to the creation of vortices in almost all free areas in the interior of individual lots. This contribution comes even from side corridors, which act as natural wind catchers, from which the wind reaches larger areas. [Oliviera, et al, 2009]

5- The slope orientations of the roof also interferes with orientation and velocities of air flows; two-slope roofing, with slopes oriented to the front façades also can contribute to air recirculation in the lots. This was not the main objective of authors work, but they recommended that to be considered in future studies. [Oliviera, et al, 2009]

6-it is important to take the direction of prevailing winds, orientation to the sun during the early stages of the new urbanization. [Oliviera, et al, 2009]

7-The external walls (fences) which describe the plot borders, side corridors, roofing slope should be considered because of their negative effects n wind follow between plots. [Oliviera, et al, 2009]

8- CFD simulation program was used to study wind and air pattern around 26 houses in the settlement, the results show values between 0.1 and 1.2 m/s for the wind velocity in the external region surrounding the houses, and the greatest values were found in the corner houses, where there is a stronger incidence of winds. [Oliviera, et al, 2009]

5.4. Use of CFD for Wind Analysis in Singapore City

A study conducted by Danei Hii Jun of the Center for Sustainable Asian Cities, has discussed the alteration of the wind by buildings, The study has used CFD to predict how much of the wind follow is altered by buildings. The study area is situated at the south-eastern part of Singapore around Kallang River. [Chung, 2010]. The study area is one of four areas identified in the Singapore master plan, 2008. The area is with a projected population of 5.5 million in the future. [Chung, 2010]

The area of the site studied is 64 hectares. The proposed plot ratio (floor space ratio) for the site ranges from 1.5 to 5.6 with buildings heights ranging from three to 36 stories. [Chung, 2010]

CFD simulation program has been used to determine the wind pattern around the buildings and to specify the area with bad circulation, This area is highlighted by the red box (Figure 5-11) for both predominant wind directions (north-east, south-east). [Chung, 2010]

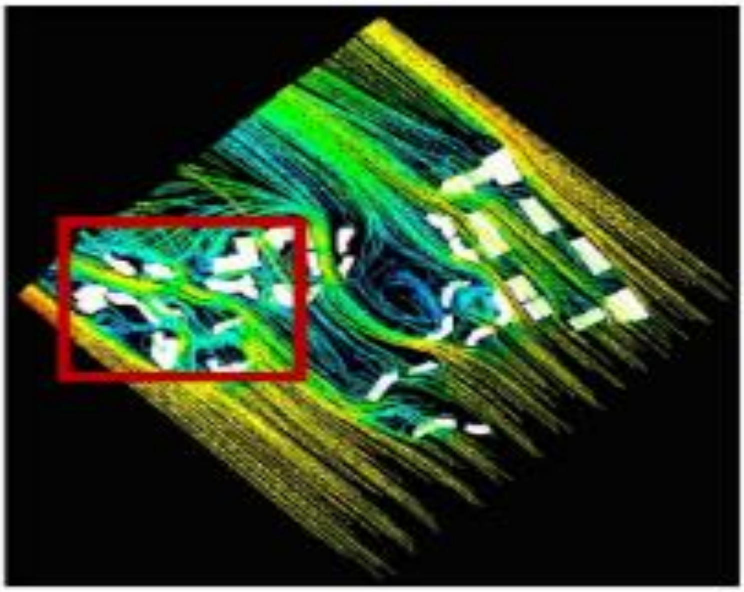

Figure 5-11- Area of bad wind circulation with the direction (south east). [Chung, 2010]

From the (figure 5-12), it can be noted that the wind movement pattern around the buildings and the path lines colored by velocity magnitude (m/s) for the study area north- east wind follow analysis in ANSYS fluent. The color of the line ranged from red (higher velocity) to blue (lower velocity). [Chung, 2010]

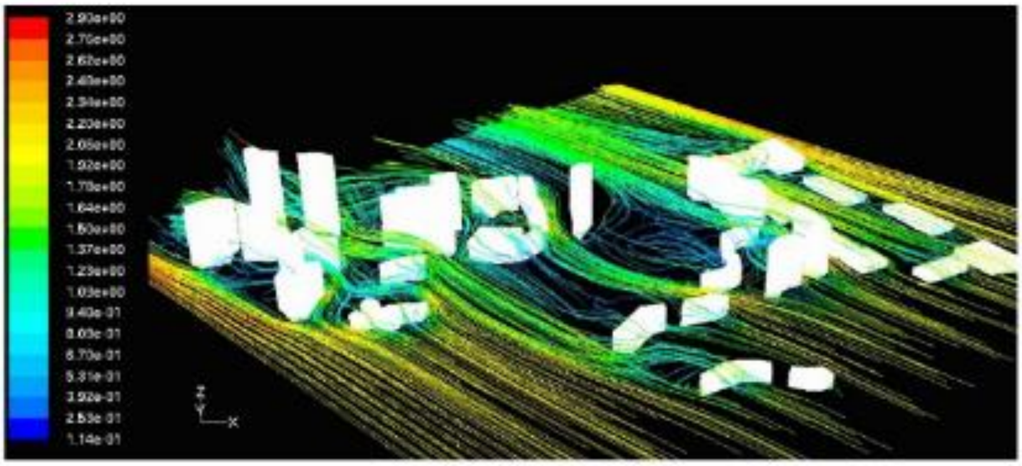

Figure 5-12 shows air movement pattern around buildings (south- east). [Chung, 2010]

Wind pattern in the area of study shown in figure 5-13 and figure 5-14, the lines colored by velocity magnitude (m/s), south – east, the wind velocity was analyzed by ANSYS fluent. The red line means higher velocity and the blue one means lower velocity. [Chung, 2010]

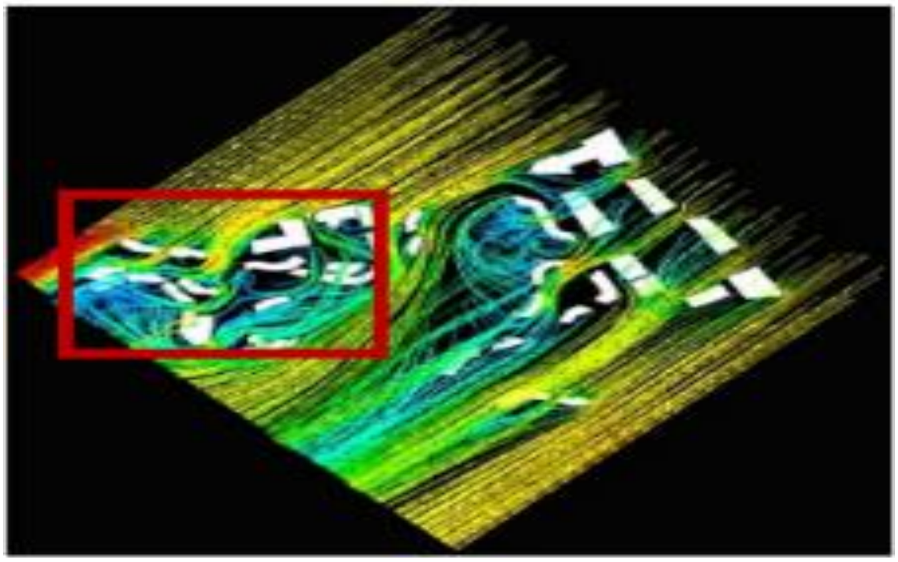

Figure 5-13- shows area of bad wind circulation with the direction (north- east). [Chung, 2010]

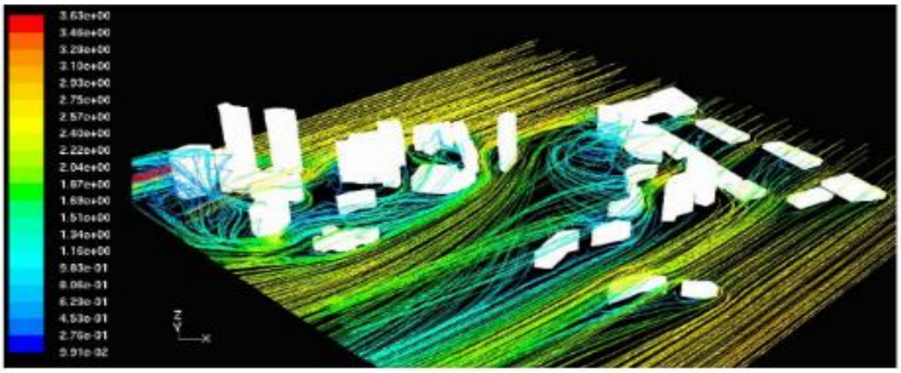

Figure 5-14 shows air movement pattern around buildings (north- east). [Chung, 2010]

5.5. Urban Form Effect on Wind Movement Using (CFD) as Analysis Tool

Local climatic parameters and thermal comfort conditions are changed by the disorderly urban growth of cities. The haphazard cities, however, is responsibility of urban planners. [Renan, 2011]

Elements of the climate and ventilation conditions suffer major changes during the process of urbanization. Wind is the main factor of change in the climate elements in urban areas. [Renan, 2011]. Furthermore, there are no serious integrated studies on urban organization and exploitation of natural wind potentials during urban planning process.

In Brazil, there was no emphasis on wind or wind application in urban planning. The concern with wind has recently emerged. The direct effect on cities and the population is consequent of decisions by public administrators on urban planning. [Renan, 2011]

To sum up, it is important to control solar radiation and natural ventilation in urban areas and indoor spaces of buildings, especially in regions of hot and humid climate, where there is no difference in temperature during the whole hours of the day. [Renan, 2011]

The urban form and its components, which include building heights, the distance between buildings, is the main factor that affect negatively permeability of the wind within the urban space, natural ventilation, exploitation of the wind in positive conditioning of the buildings, and energy conservation. [Renan, 2011]

However, understanding physical phenomena and factors that affect natural ventilation process is very important to use the wind for passive cooling of buildings objectively. [Renan, 2011]

Therefore, the creation of microclimates within urban form by compatibility between heights of buildings, high-density and maintenance of ventilation in urban areas is necessary to provide natural ventilation in urban spaces. [Renan, 2011]

However, it is very important to carry out studies on wind speed and direction and provide natural ventilation in the urban areas to pave the way before architectural design and urban planning process. [Renan, 2011]

There are important tools for the analysis of wind movement, natural ventilation, and climatic elements in the urban areas. The tools include wind tunnels tests, software and mathematical models. [Renan, 2011]

According to the author, it is important to integrate natural ventilation strategy and thermal comfort in the city of Fortaleza distribute densities process, as happened in the south-eastern sector of the city. This will develop natural ventilation strategy and achieve thermal comfort in Fortaleza. [Renan, 2011]

Although the master plan for urban development is reviewed, there are no studies about amending legislation of urban growth of this part of the city, and the treatment of this sector of the city is associated mainly to market to buy and sell houses. [Renan, 2011]

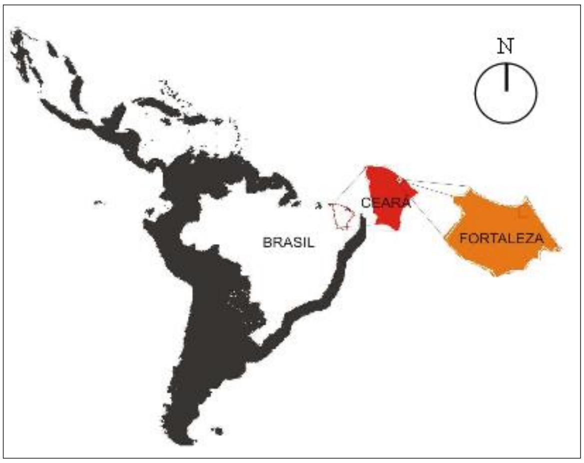

Figure 5-15- shows localization of Fortaleza, Ceará, Brazil. [Renan, 2011]

This study aims at investigating the impacts of the height of buildings, on natural ventilation in the southeast part of Fortaleza. [Renan, 2011]. The author has used a CFD to analyze the airflow around buildings at various study points within three virtual tridimensional scenarios that simulate urban built form possibilities for this part of the city. [Renan, 2011]

5.5.1. Simulation Experiments

a. Data Analysis

The data was collected from city airport to identify the frequency of wind speeds and directions. The data analyzed within the period between February 12, 2002 and January 30, 2009, (61,032 hours), figure 5-16.

Wind rose was plotted to determine the two predominant wind directions: east and southeast directions with 40% and 37% of total hours. The average speed of the wind blowing from the

East is about 3, 85 m/s. in the southeast, the wind has 4, 51 m/s with average velocity during this period, as shown in figure 5-16. [Renan, 2011]

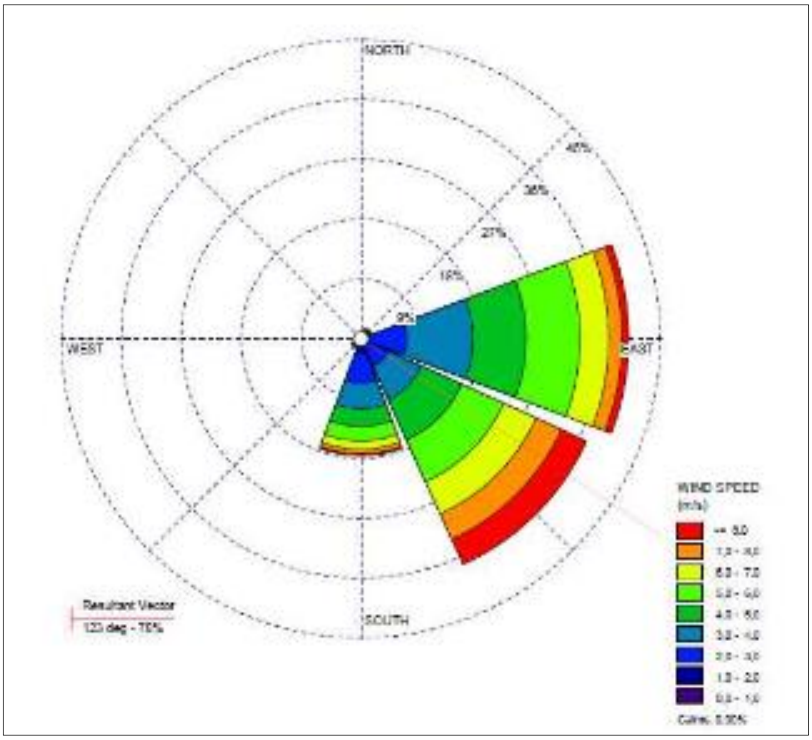

Figure 5.16. Wind rose plotted for 2002-2009 period (Renan, 2011)

5.5.2. The Use of CFD for Airflow Analysis

The CFD is advanced software. It is used to simulate fluid in motion. Currently the programme is available in a wide range. [Renan, 2011]

Navier-stoke equations are considered as a base of CFD models. The equations are used for solve all points of a two or three-dimensional mesh. In these models, the buildings and surrounding areas are represented in the mesh, and the pressure and velocity are given for determining the initial and boundary conditions. [Renan, 2011]

In this study, the CFX 5.6 CFD software has been used for analysis. The software was produced by the ANSYS Company. It has been used to simulate any situation involving fluid mechanics in any scale and boundary conditions, as long as the computational capacity has provided. [Renan, 2011]

ANSYS CFX 5.6 CFD license is available at the Laboratory of Environmental, Comfort and Energy Efficiency in the Faculty of Architecture and Urbanism, University of Sao Paulo.The accumulated experience has contributed to the choice of this software. This software can also be used in other researches. [Renan, 2011]

Software consists of four modules, in processing level; the geometry has prepared by CAD software, the area of the model should be determined to be suitable in ANSYS ICEM for the mesh, and to define the points where the equations should be calculated. [Renan, 2011]

For the model construction, the buildings should be simple to reduce considerably the processing time and computational capacity for calculating the simulation. [Renan, 2011]

After preparing the model, and before starting the simulation, you must determine the definition of the system simulation, the initial and boundary conditions and the turbulence model and equations prior to model analysis by CFX software. [Renan, 2011]. Then Insert points, lines and planes in various locations to allow visualization of the airflow in different parts of the model through velocity contours and vectors or streamlines. [Renan, 2011]

5.5.3. Modeling for Computational Simulation and the Boundary Conditions

The study used three virtual three- dimensional models. The first model represents the present situation of this part of the city, the second one expresses the maximum density permitted by the current urban legislation in terms of buildings height (72 m) and a lower density rate and a third one indicates an intermediate proposal designed as a criticism of the permissiveness of the law, with building's height about 48 m but with a higher density rate. . [Renan, 2011].

The wind pattern around building was simulated by ANSYS CFX CFD 5.6 software and compared with the existing models and the scenarios of maximum and intermediate densities. [Renan, 2011]

There are two main directions of the wind had been determined to analyze the data. Two domains were created to settle each of the three models: the first has been directed to the eastern direction, and the other has been directed to the southeast direction. Moreover, the form of this domain allows air to be perpendicular to the model in 90°. [Renan, 2011]

In this case study, the definition of boundary conditions is performed by determining characteristics to the domain faces (parts) that will influence the flow. [Renan, 2011]

At the inlet (E and SE wind orientation), the average of wind speed has been assumed for the both domains as 3.85 m / s for the eastern direction, and 4.51 m / s for the south-east direction. [Renan, 2011]

At the outlet (W and NW wind orientation), zero static pressure has been assumed for the both domain. [Renan, 2011]

The top and sidewalls of both domains are modelled as free slip walls (zero normal velocity and zero normal gradients of all variables). [Renan, 2011]

At the ground and building surfaces the standard wall functions used is smooth wall, no slip. This condition (smooth wall) is applied to the sides of buildings and the model floor to represent the typical surfaces of urban environment materials. [Renan, 2011]. The RNG k-ε turbulence model was chosen for this study because of its good performance for predicting the surface pressures on the windward building facades. The flow regime is subsonic and the domain motion option is stationary and non-buoyant. [Renan, 2011]

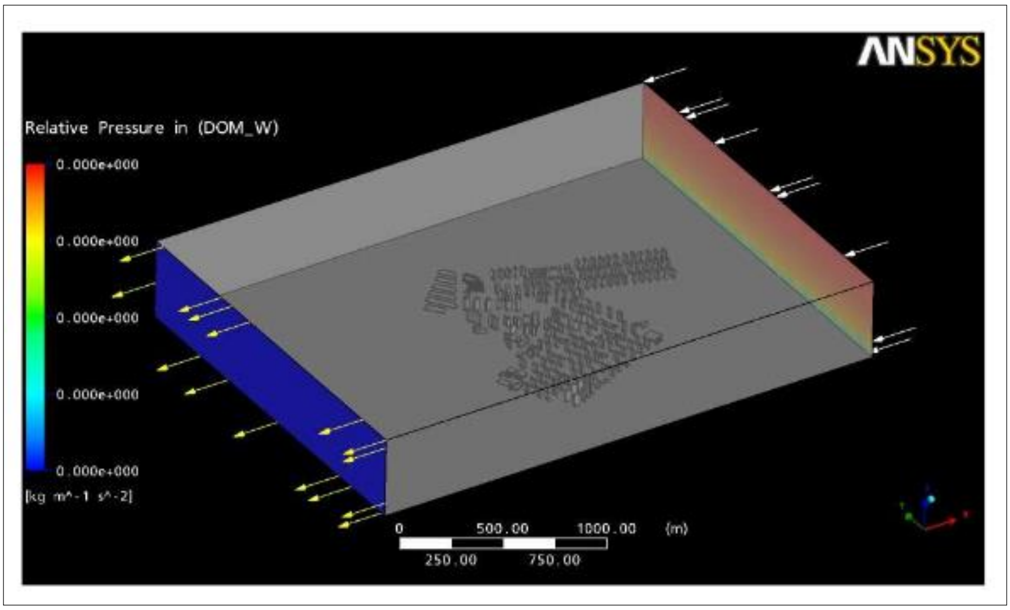

Figure 5-17- General aspect of the boundary conditions defined for a simulation of the maximum density model with east wind direction. [Renan, 2011]

In order to analyze the wind pattern within the three models simulated, horizontal planes at different heights were generated. Over these planes velocity vectors and contours were plotted. [Renan, 2011]

The horizontal plane 1 (PH1) is about 1.5 m high, corresponding to the pedestrian level. The second plane (PH2) is 10 m high, at which the wind data was collected by the weather station located in the city's airport. The third plane (PH3) is located 36 m above the ground, half the maximum height of buildings permitted by law in the study area. [Renan, 2011]

The simulation also used points of comparison distributed along the models for testing and validating the results and velocity vectors in different plans to evaluate the wind field around buildings. [Renan, 2011]

The six simulations reached reliable results according to the convergence criteria recommended by CFX CFD (2003). The results may be evaluated through the Residual Mean Square (RMS) graphs generated during the simulation calculation. They indicated a RMS about 1 x 10 -4, which is considered regular and enforceable. [Renan, 2011]

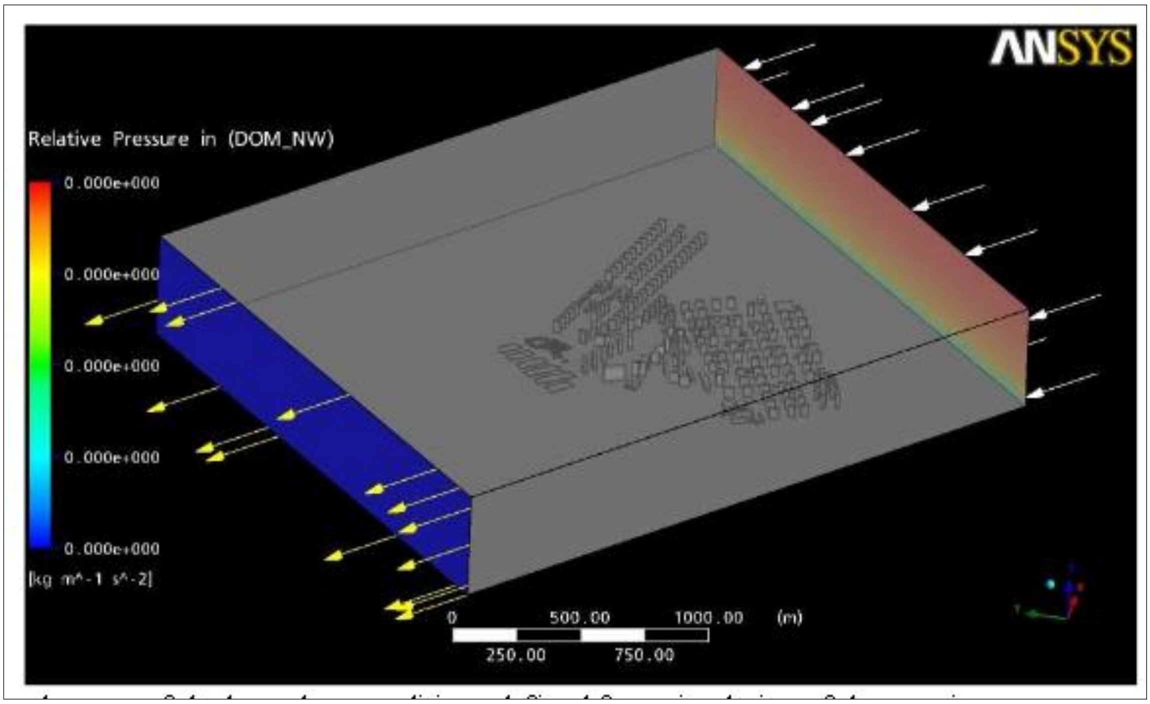

Figure 5-18- show general aspect of the boundary conditions defined for a simulation of the maximum density model with southeast wind direction. [Renan, 2011]

5.5.4. Results Analysis

1- East Wind Analysis

The east wind has been analyzed and simulated over the plans PH1, PH2 and PH3. The areas with static and stagnant wind flow are highlighted in blue color, the wind speed a round these areas is around 0 and 0.2 m/s.

In the existing model, and when the velocity contours at 1.5 m above ground, as is shown in figure 5-19, the speeds are verified in the range between 1.1 and 2.2 m/s. [Renan, 2011]

Corner effects are verified and, at the beginning of the streets which the direction coincides with the east wind, an acceleration of the flow due to channeling along the axis of these stretches is indicated reaching speeds of 1.9 m /s. [Renan, 2011]

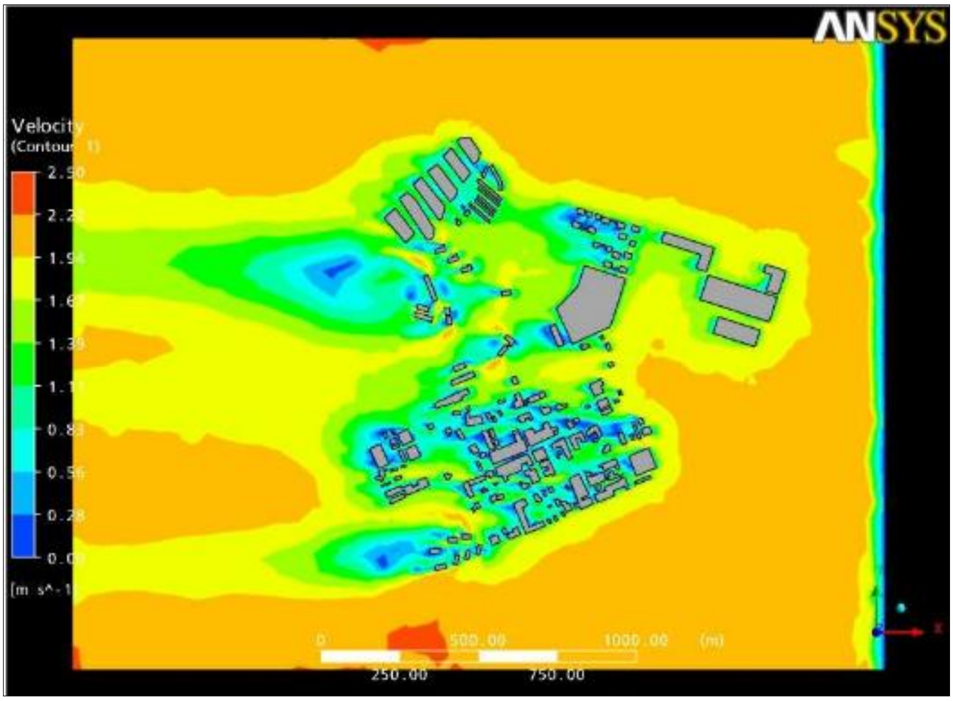

Figure 5-19- shows velocity contours over PH1 (1.5 m height) in the present situation model (E-wind) [Renan, 2011]

In the maximum occupation model, and when the wind flow at a pedestrian level about 1.5 m height, it is noted that, the areas of static and stagnant are located further away from the buildings in the model. The size of the biggest stagnant wind flow area is about 4 blocks (400m) as presented in figure 5-20.

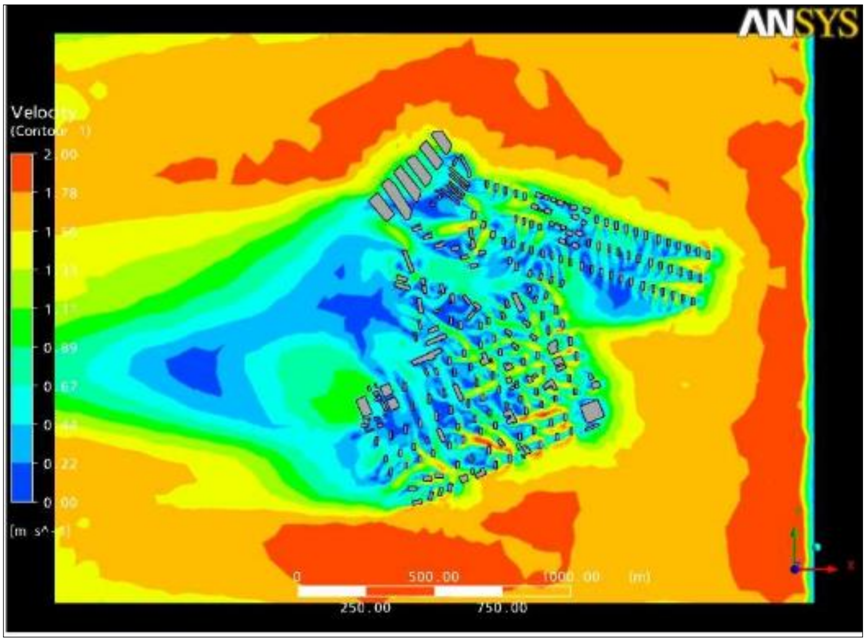

Figure 5-20 Velocity contours over PH1 (1.5 m height) in maximum density model (E-wind) [Renan, 2011]

On intermediate model, it can be found that the areas with the static wind close to the buildings are smaller compared with the maximum model. There are still areas between the buildings with stagnant wind flow as a result of increase in land density rate due to the reduction in the height of the buildings. This is presented in figure5- 21.

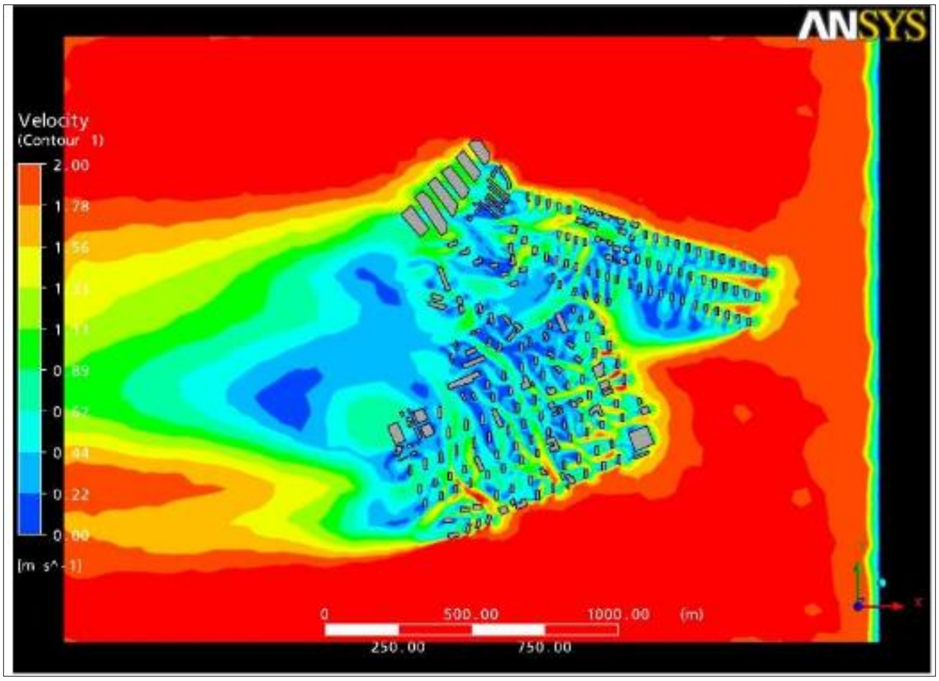

Figure 5-21: Velocity contours over PH1 (1.5 m height) in intermediate density model (E-wind). [Renan, 2011]

In the center of the existing model, we observe the effect of high-rise buildings corners on the vortex, which appear in the longer form, the order of these towers will help generate an increase in the speed of the wind as a result of the existence of a network of wind flow channels, as shown in Figure 5-22.

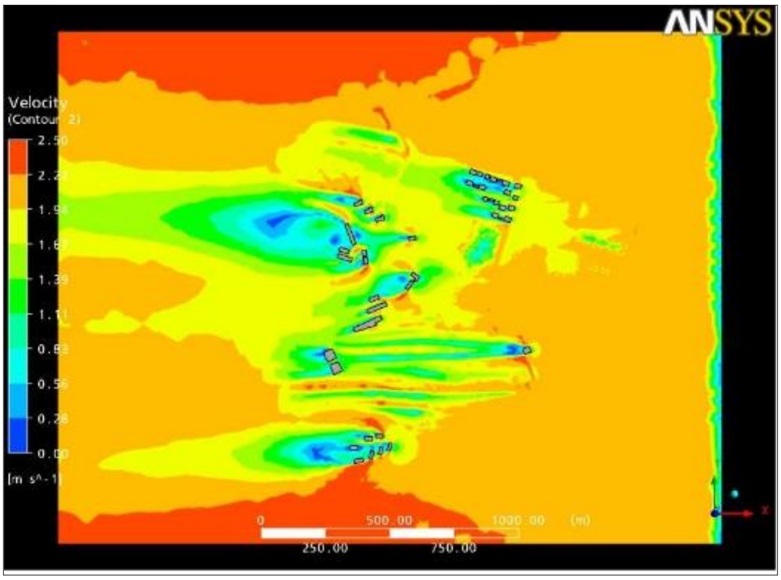

Figure 5-22: Velocity contours over PH2 (10 m height) in the existing situation model (E-wind). [Renan, 2011]

The two major stagnant areas are observed after the buildings in the model, as shown in figure 5-23, when the wind flow is analyzed in the maximum density model, about 10 m above the ground (PH2). [Renan, 2011]

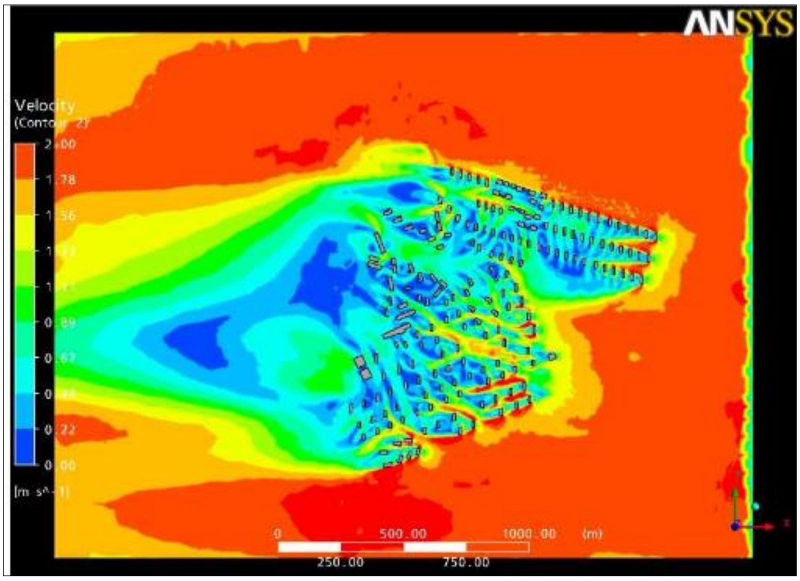

Figure 5-23: Velocity contours over PH2 (10 m height) in maximum density model (E-wind). [Renan, 2011]

There are two phenomena observed in the intermediate model compared with the maximum model, the first is decrease of the vortices after the buildings, and the other one is dilution of the flow in the areas adjacent to the buildings. Within this area, wind velocities about 0-0.2 m/s are verified as shown in (figure 5-24). [Renan, 2011]

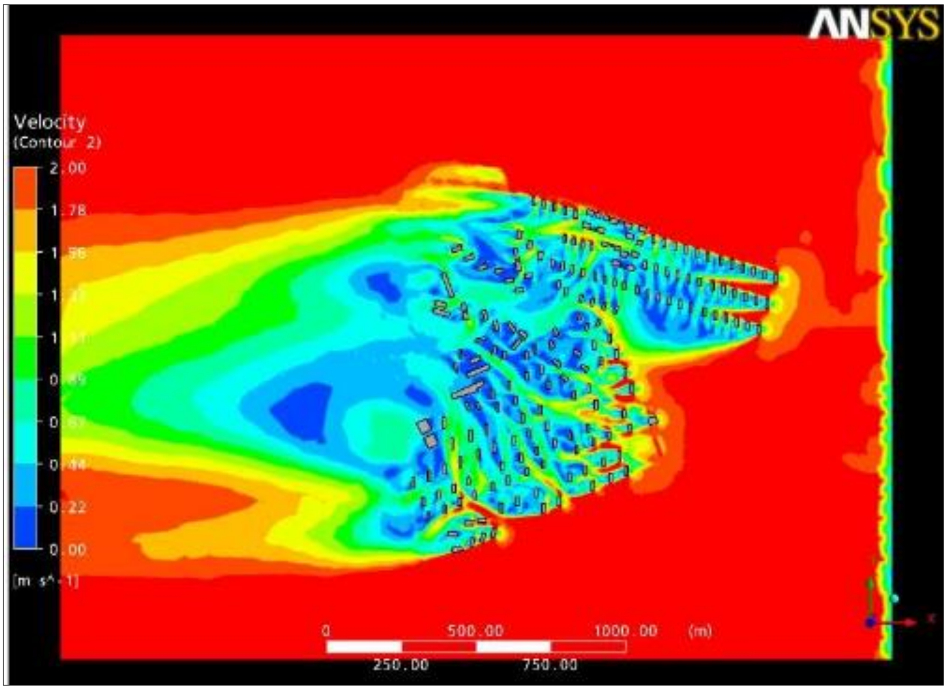

Figure 5-24: Velocity contours over PH2 (10 m height) in intermediate density model (E-wind) [Renan, 2011]

When the wind speed at a height of 36 meters above the ground level in the existing model, the area of deceleration is observed due to the main vortices that caused by the tallest buildings. In addition, tallest buildings reduce the dimensions and proportions of deceleration area significantly. However, a third vortex is generated in the area, which is located after the two buildings, which placed near to each other, in the corner, which corresponds to the eastern direction of the wind. See the figure 5-25. [Renan, 2011]

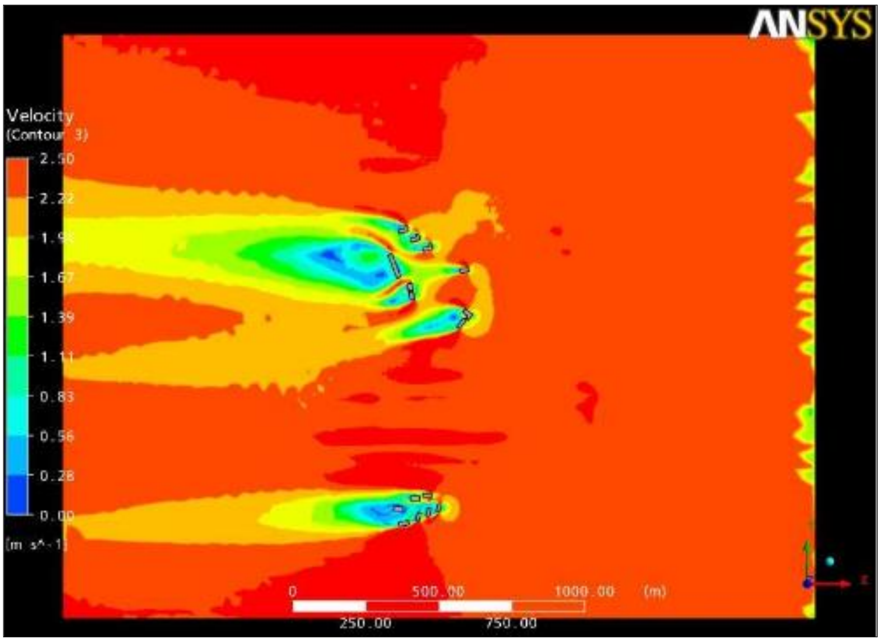

Figure 5-25: Velocity contours over PH3 (36 m height) in the existing situation model (E-wind) [Renan, 2011]

Comparing the maximum density model to the intermediate model at a horizontal plane 36 m above the ground (PH3), the effect of the buildings over natural ventilation is the same. However, it is quite evident the size reduction of the vortex created after the buildings in the intermediate model. See figure 5-26. [Renan, 2011]

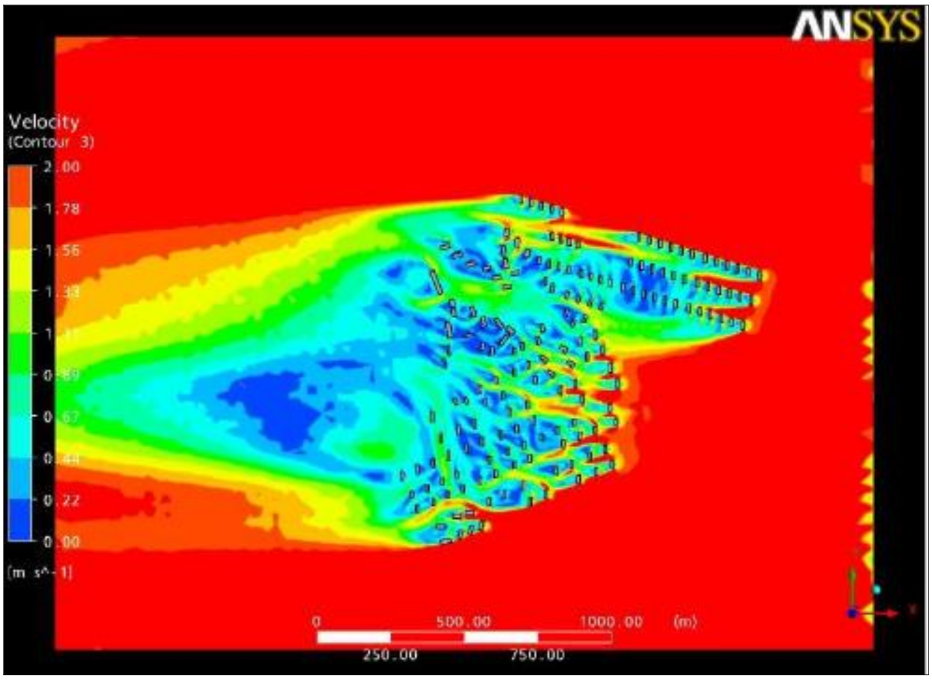

Figure 5-26: Velocity contours over PH3 (36 m height) in maximum density model (E-wind) [Renan, 2011]

In the intermediate density model, the buildings, which placed at the smaller areas of slow wind flow, create the biggest vortex. See figure 5-27.

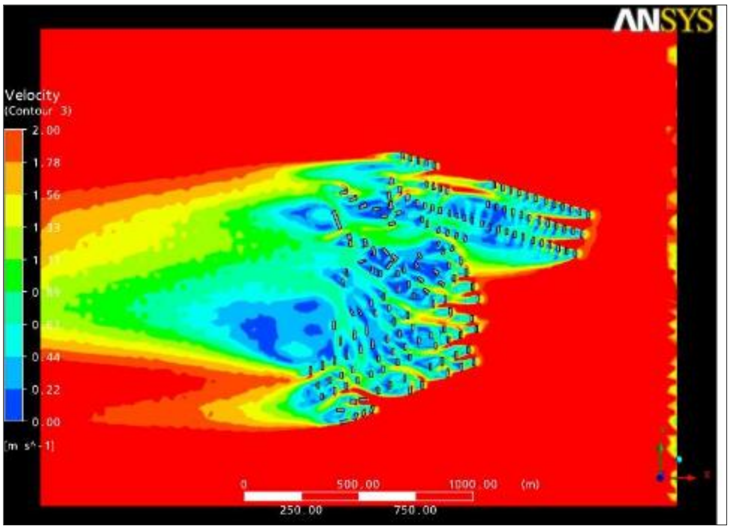

Figure 5-27: Velocity contours over PH3 (36 m height) in intermediate density model (E-wind) [Renan, 2011]

The streamlines, indicate that, the tallest buildings that located in the center of the two models (intermediate and maximum model), caused hange in the wind flow. They act as a barrier to the wind and create a vortex zone and a turbulence flow in the areas after them. See figure 5-28. [Renan, 2011]

Yet, there is structurally impact on wind circulation at the windward facade of the buildings caused by the increase in the flow velocity at higher levels of the maximum occupancy model. [Renan, 2011]

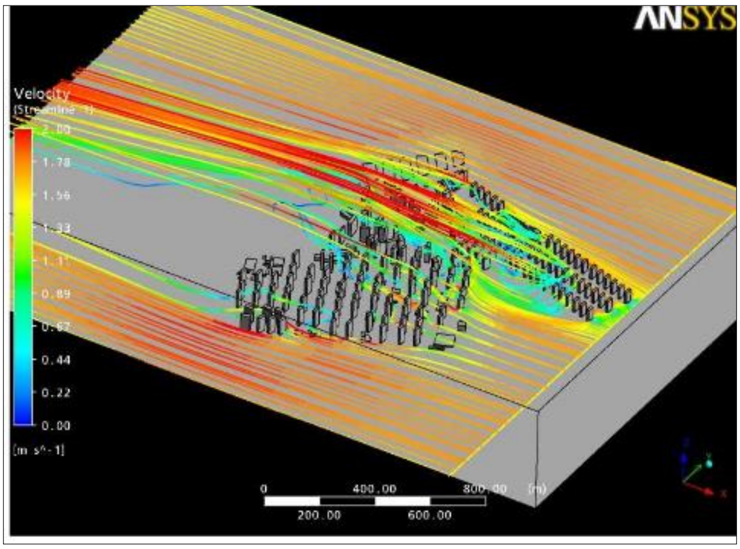

Figure 5-28: Using streamlines to visualize the airflow over the maximum density model (E-wind). [Renan, 2011]

Notably, the streamlines in the intermediate model do not rise as much as in the maximum density model, building with 48 m height has less impact on the vortex creation than the arrangement of buildings of 72m height. See figure 5-29. [Renan, 2011].

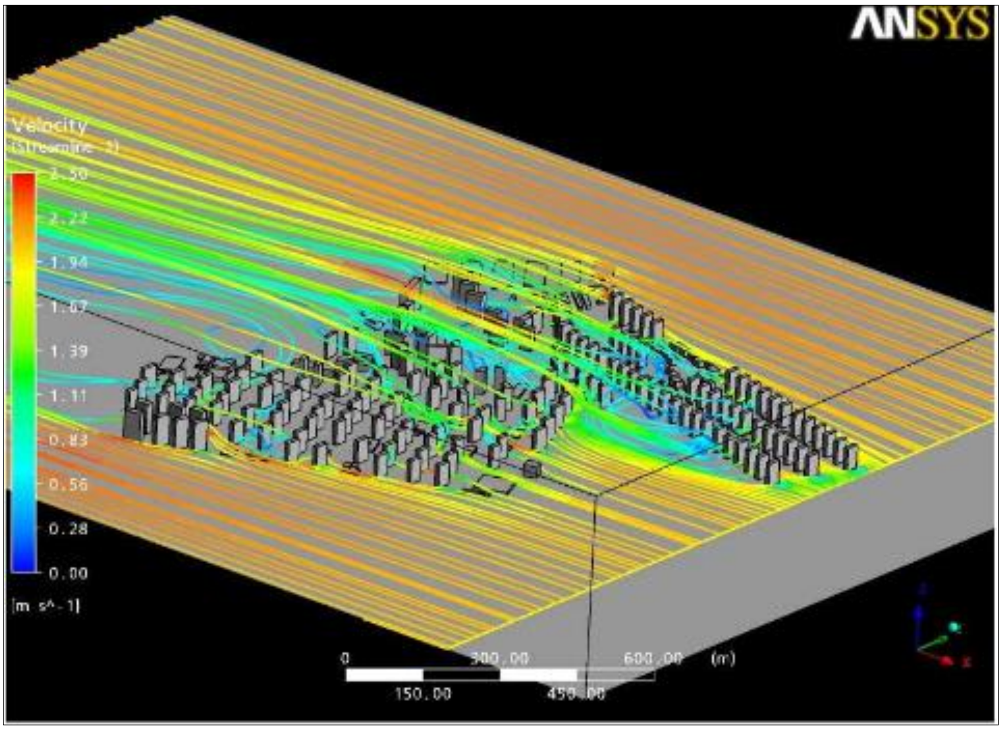

Figure 5-29: Using streamlines to visualize the airflow over the intermediate density model (E-wind) [Renan, 2011]

2- Southeast Wind Analysis

When the southeast wind simulated, we find that, the important effect of the buildings to the movement of wind is the existence of a network of streets between them. We find that the spatial distribution of the buildings in the first model helps reduce the areas of the low wind velocities. [Renan, 2011]

However, despite the street effect, a vortex of larger proportions in both models of maximum and intermediate occupation is created. Compared to the simulations with east wind, the effect over the windward facade is quite evident as shown by the area highlighted in blue (wind velocities between 0 – 0, 2 m/s). See figures 5-30 and 5-31. [Renan, 2011]

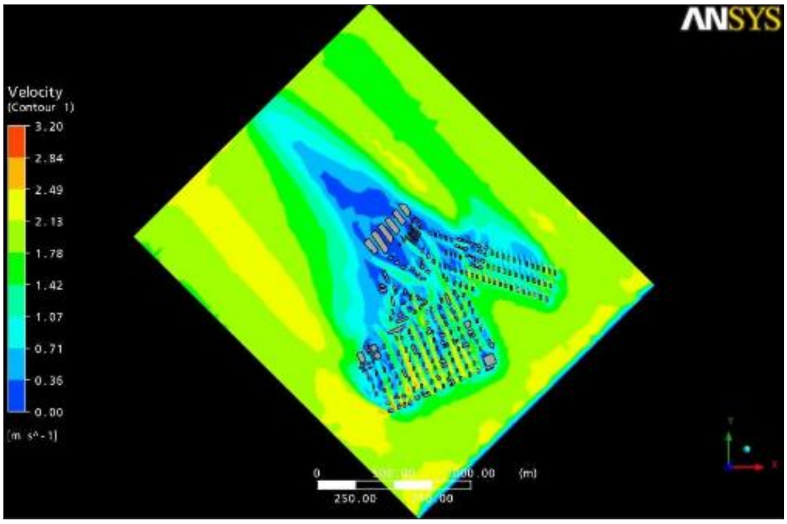

Figure 5-30: velocity contours over PH1 (1,5m height) in maximum density model (SE-wind) [Renan, 2011]

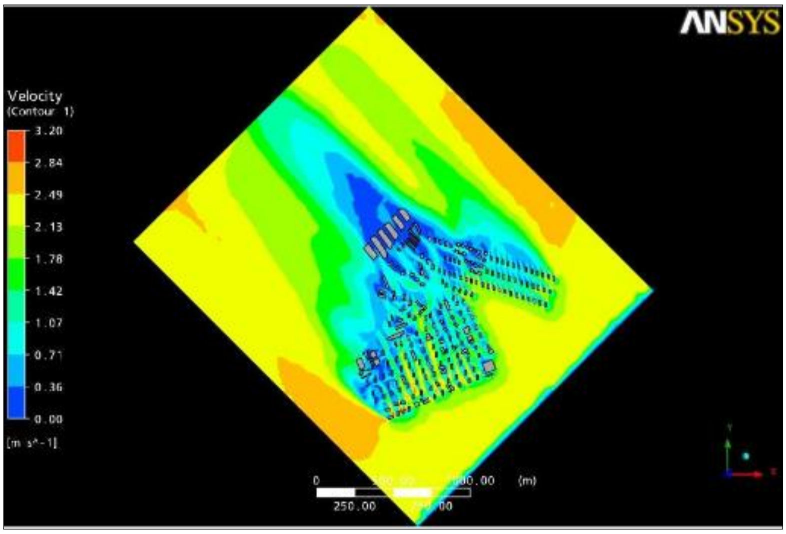

Figure 5-31: Velocity contours over P1 (1,5m height) in intermediate density model (SE-wind) (Renan, 2011)

The main vortex is verified at 10 m above the ground (PH2) in the maximum density model, which has more spaced density. The maximum land density model, as shown in figure 5-32 minimizes the negative effect on wind at the windward facades, when the wind is coming from the southeast. [Renan, 2011]

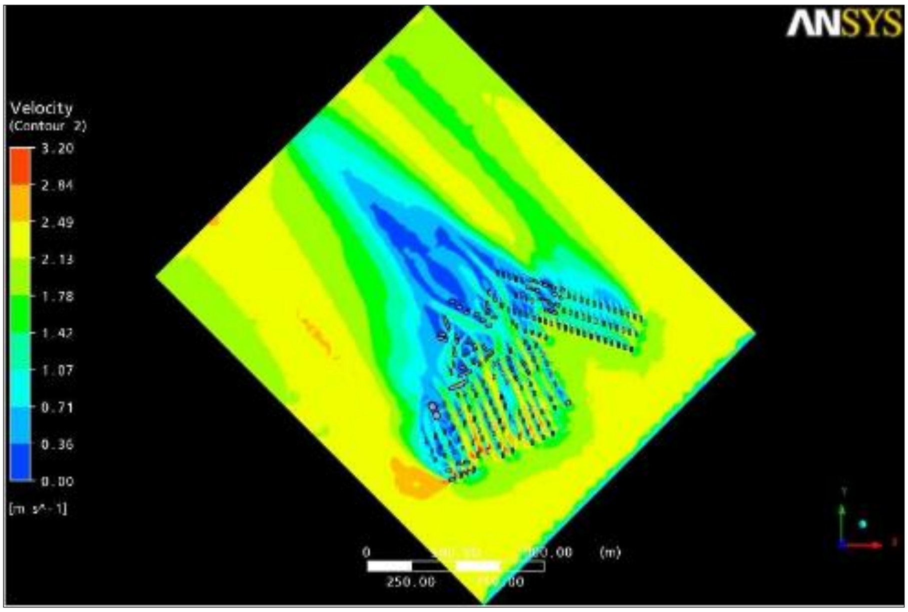

Figure 5-32: Velocity contours over PH2 (10 m height) in maximum density model (SE-wind). [Renan, 2011]

On the other hand, in the intermediate density model, which contains lower buildings, the area of the stagnant wind at 10 m height (PH2) is markedly larger and concentrated compared with the maximum density model. In addition, the wind speed at the windward facades is decreased, as shown in figure 5-33.

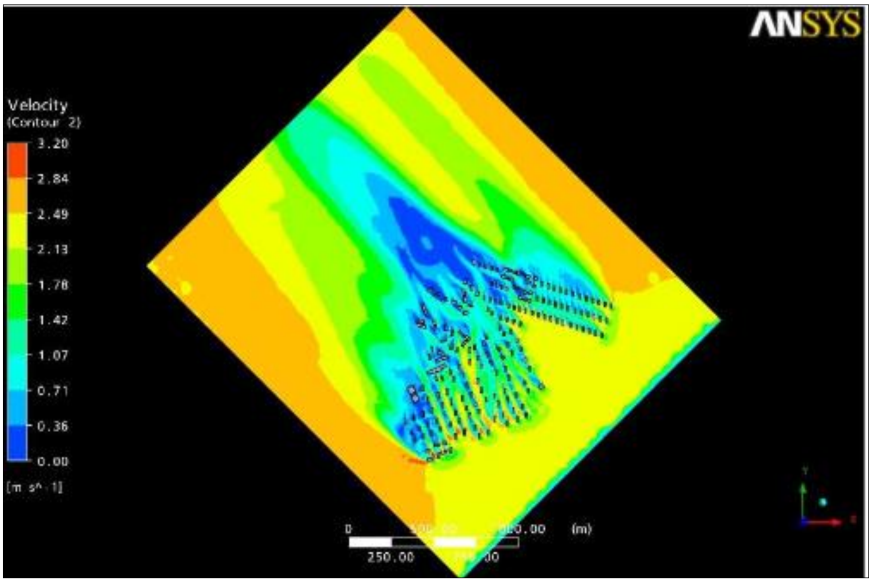

Figure 5-33: Velocity contours over PH2 (10 m height) in intermediate density model (SE-wind) . [Renan, 2011]

Eventually, after a simulation process, it was observed that the wind pattern and the airflow at 36 meters above ground level in three models were as follows: the stagnant flow area appears to be stretched and narrower in the maximum density model. It also appears to be concentrated form in the intermediate density model. [Renan, 2011]

The few stretches with speeds below 1.0m/s are observed at the center of the intermediate model, the generation of small vortices was also observed within this model due to the increased rate of land use. [Renan, 2011]

The area with wind velocities below 0.6 m/s is located far from the buildings, and located in the lee wind side devoid of obstacles in the maximum density model, and it is located right next to the buildings in the intermediate density model. However, its aspects seem to be diluted, generating smaller areas of stagnant wind flow. See figures 5-34, 5-35. [Renan, 2011]

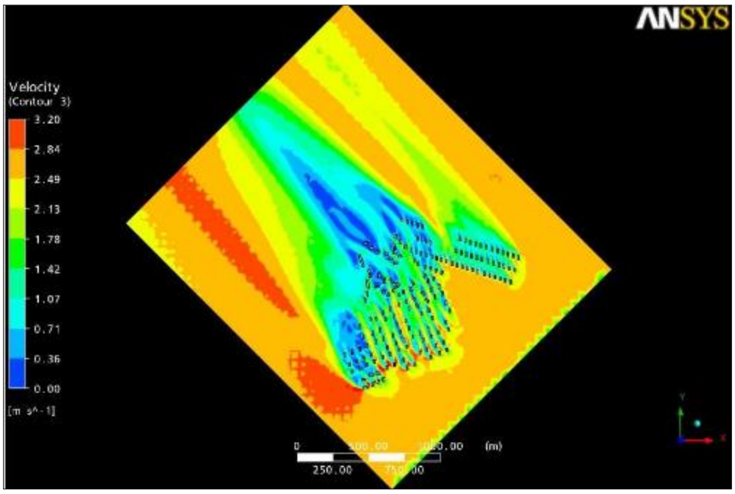

Figure 5-34: Velocity contours over PH (36 m height) in maximum density model (SE-wind). [Renan, 2011]

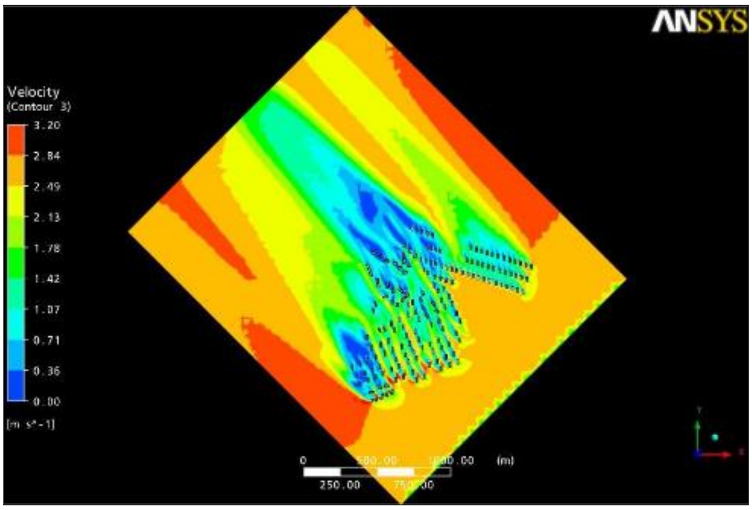

Figure5-35: Velocity contours over PH3 (36 m height) in intermediate density model (SE-wind). [Renan, 2011]

As occurred in the simulation with the east wind orientated models, the streamlines identified the increase of the flow velocity at higher levels of the maximum density model due to the height of buildings in this model. Once the wind passes through 16-storey buildings and before deviating the 24-storey buildings in this model, the wind gradient will elevate as shown in figures 5-36. [Renan, 2011]

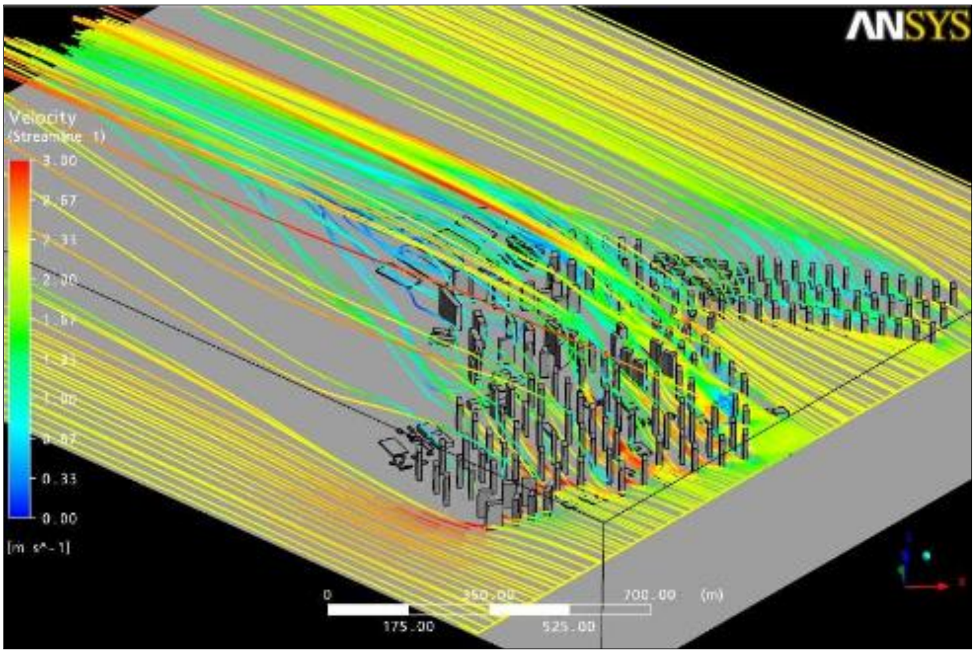

Figure 5-36: Using streamlines to visualize the airflow over the maximum density model (SE-wind). . [Renan, 2011]

In the intermediate model of density the higher velocity streamlines occur when entering the area between the buildings. Analogously to the model of maximum density, the vortices are characterized as areas in which the turbulent streamlines undergo major deviations and revolutions. [Renan, 2011]

Unlike the maximum density model, in this intermediate density model, an elevation in the wind flows is not that evident. General wind flow remains, in general, slightly above the top of the buildings as shown in figure 5-37. [Renan, 2011]

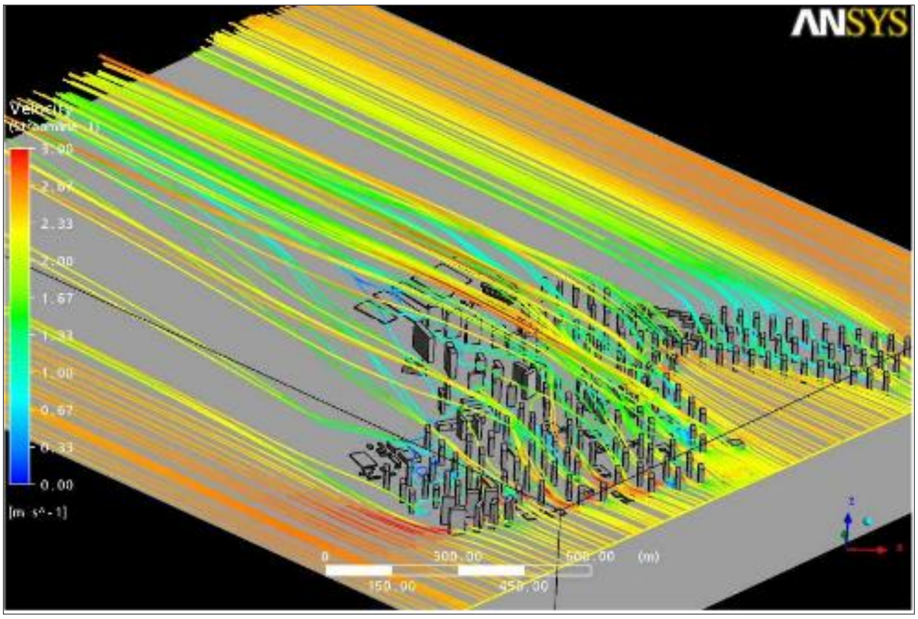

Figure5.37 Using streamlines to visualize the airflow over the intermediate density model (SE-wind). [Renan, 2011]

3- The Use of Comparison Points for Airflow Analysis

Wind velocity profiles were measured at a comparison points, located in different heights on streets and at the center of blocks to compare the impacts of the different roughness with the wind flow in both models of maximum and intermediate density and existing situation. See figure 5-38. [Renan, 2011]

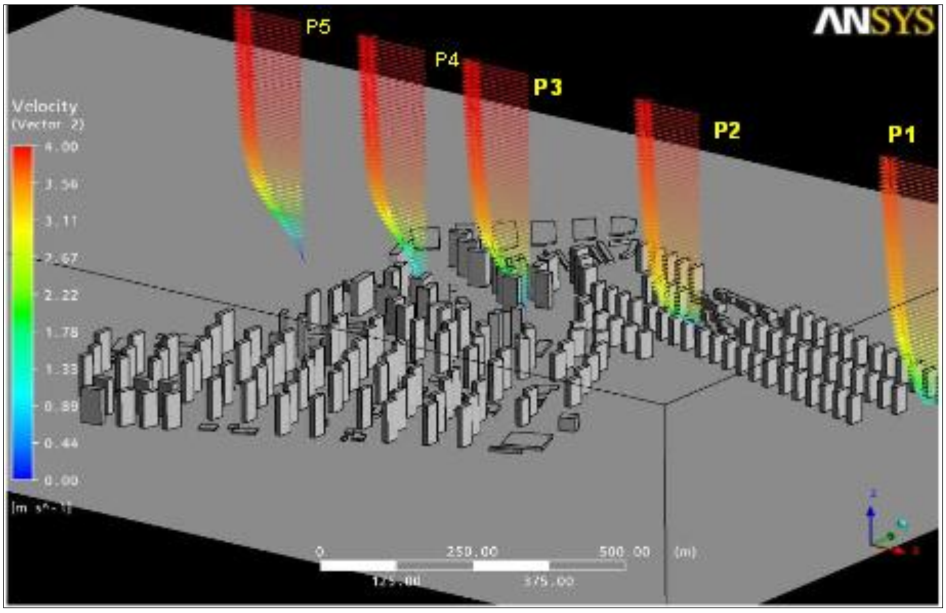

Figure 5-38 locations of comparison points 1, 2, 3, 4 and 5 in the maximum density model (E-wind). [Renan, 2011]

The same characteristics of the wind flow at the height of 10 m were observed in points 1 and 2 of the both maximum and intermediate densities model, but there is a reduction of 30% in points 1 and 2 compared to the existing situation. See figure 5-39. [Renan, 2011]

Wind velocities registered at the points of comparison 1.2m, 3m, 4m and 5m at different heights are shown in table 3 below. [Renan, 2011]

Table 5-2: Wind velocities registered in the points of comparison 1, 2, 3, 4 and 5 at different heights (Renan, 2011)

POINT OF COMPARISON 1			
MODEL	PH 1	PH 2	PH 3
Existing situation	1,81 m/s	2,02 m/s	2,33 m/s
Maximum occupation	1,09 m/s	1,34 m/s	1,90 m/s
Intermediate occupation	1,32m/s	1,58 m/s	1,92 m/s
POINT OF COMPARISON 2			
MODEL	PH 1	PH 2	PH 3
Existing situation	1,67 m/s	1,86 m/s	2,27 m/s
Maximum occupation	0,47 m/s	0,44 m/s	0,33 m/s
Intermediate occupation	0,56 m/s	0,62 m/s	0,44 m/s
POINT OF COMPARISON 3			
MODEL	PH 1	PH 2	PH 3
Existing situation	1,37 m/s	1,50 m/s	1,89 m/ s
Maximum occupation	0,45 m/s	0,50 m/s	0,85 m/s
Intermediate occupation	0,20 m/s	0,31 m/s	0,70 m/s
POINT OF COMPARISON 4			
MODEL	PH 1	PH 2	PH 3
Existing situation	0,62 m/s	0,85 m/s	0,56m/s
Maximum occupation	0,69 m/s	0,71 m/s	0,61 m/s
Intermediate occupation	1,31 m/s	0,98 m/s	0,87 m/s
POINT OF COMPARISON 5			
MODEL	PH 1	PH 2	PH 3
Existing situation	0,69 m/s	0,62 m/s	0,40 m/s
Maximum occupation	0,30 m/s	0,19 m/s	0,44 m/s
Intermediate occupation	0,18 m/s	0,12 m/s	0,46 m/s

Roughness has impacts on the velocity profile generated in point 7 at extended areas. Other points 8 and 9 are located in a high constructive density area. Point

10 is used to assess the impacts on wind movement in the downwind area. Figure 5-39. [Renan, 2011].

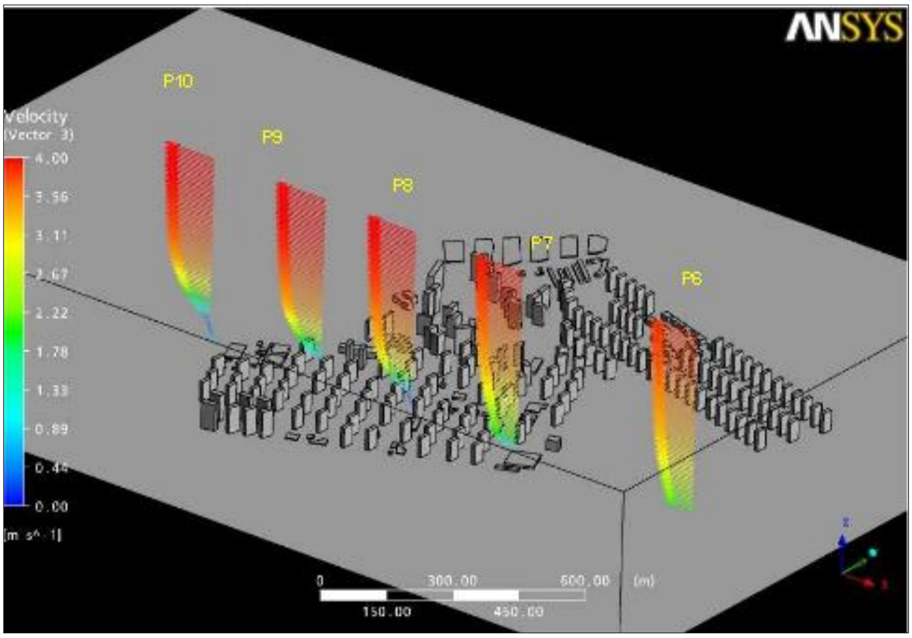

Figure 5-39: Localization of comparison points 6, 7, 8, 9 and 10 in the intermediate occupancy model (E-wind). [Renan, 2011]

Compared to existing model, there is a great reduction in the extended area. The reduction is caused by the decrease of the wind velocity at the points 6, 7, 8, 9, and 10. The wind flow is slower in the lower layers in this extended area. It ranges between 65% and 95%. [Renan, 2011]

From the point 7 and beyond the reduction in the wind intensity in both models of a dense land occupation is quite evident compared with the existing situation. More specifically, the significant decrease of wind velocities at lower levels of maximum density model shows that the increase in buildings dimensions to 72 m has a major impact on the local wind patterns. Figure 5-39. [Renan, 2011]

Wind velocity registered in the points of comparison 6, 7, 8, 9, and 10 at different heights is illustrated in table 5.3 below. [Renan, 2011]

Table 5-3: Wind velocities registered in the points of comparison 6, 7, 8, 9 and 10 at different heights. [Renan, 2011]

POINT OF COMPARISON 6			
MODEL	PH 1	PH 2	PH 3
Existing situation	2,01 m/s	2,16 m/s	2,37 m/ s
Maximum occupation	1,55 m/s	1,86 m/s	2,23 m/s
Intermediate occupation	1,91m/s	2,23 m/s	2,30 m/s

POINT OF COMPARISON 7			
MODEL	PH 1	PH 2	PH 3
Existing situation	1,75 m/s	2,05 m/s	2,42 m/s
Maximum occupation	0,48 m/s	0,79 m/s	1,82 m/s
Intermediate occupation	0,92 m/s	1,35 m/s	2,09 m/s
POINT OF COMPARISON 8			
MODEL	PH 1	PH 2	PH 3
Existing situation	1,00 m/s	1,67 m/s	2,39 m/s
Maximum occupation	0,15 m/s	0,20 m/s	0,14 m/s
Intermediate occupation	0,74 m/s	0,58 m/s	0,16 m/s
POINT OF COMPARISON 9			
MODEL	PH 1	PH 2	PH 3
Existing situation	1,25 m/s	1,51 m/s	2,30m/s
Maximum occupation	0,40 m/s	0,42 m/s	0,33 m/s
Intermediate occupation	0,81 m/s	0,91m/s	0, 72 m/s
POINT OF COMPARISON 10			
MODEL	PH 1	PH 2	PH 3
Existing situation	1,50 m/s	1,71 m/s	2,17 m/s
Maximum occupation	0,92 m/s	0,92 m/s	0,69 m/s
Intermediate occupation	0,71 m/s	0,63 m/s	0,30 m/s

According to the measurement of velocity profiles in the existing model, the wind velocity remains close to the values recorded in present model at points P11 and P12. There is a reduction of 10% in these locations. It can also be noted that velocity was slower compared with the maximum density model. [Renan, 2011]

However, the reduction became higher when the wind flow reaches a higher roughness at the points 3 and 4. [Renan, 2011]

Wind velocities registered in the points of comparison 11 - 19 at different heights (E – wind) is shown in table 5.4 below. [Renan, 2011]

Table 5-4: Wind velocities registered in the points of comparison 11 - 19 at different heights (E – wind). [Renan, 2011]

POINT OF COMPARISON 11			
MODEL	PH 1	PH 2	PH 3
Existing situation	2,12 m/s	2,16 m/s	2,37 m/s
Maximum occupation	1,60 m/ s	1,85 m/s	2,18 m/s

Intermediate occupation	1,85m/s	2,12 m/s	2,31 m/s
POINT OF COMPARISON 12			
MODEL	PH 1	PH 2	PH 3
Existing situation	1,73 m/s	2,14 m/s	2,40 m/s
Maximum occupation	1,80 m/s	2,10 m/s	1,72 m/s
Intermediate occupation	1,40 m/s	1,72 m/s	2,17 m/s
POINT OF COMPARISON 13			
MODEL	PH 1	PH 2	PH 3
Existing situation	1,12 m/s	1,88m/s	2,47 m/s
Maximum occupation	0,56 m/s	0,60 m/s	0,45 m/s
Intermediate occupation	0,86 m/s	0,89 m/s	0,51 m/s
POINT OF COMPARISON 14			
MODEL	PH 1	PH 2	PH 3
Existing situation	0,83 m/s	1,31 m/s	2,31 m/s
Maximum occupation	0,04 m/s	0,03 m/s	0,16 m/s
Intermediate occupation	0,44 m/s	0,52 m/s	0,55 m/s
POINT OF COMPARISON 15			
MODEL	PH 1	PH 2	PH 3
Existing situation	0,89 m/s	1,32 m/s	2,30 m/s
Maximum occupation	0,98 m/ s	0,99 m/s	0,94 m/s
Intermediate occupation	0,78 m/s	0,79 m/s	0,50 m/s
POINT OF COMPARISON 16			
MODEL	PH 1	PH 2	PH 3
Existing situation	2,00 m/s	2,05 m/s	2,27 m/s
Maximum occupation	0,65 m/s	0,61 m/s	0,25 m/s
Intermediate occupation	0,23 m/s	0,25 m/s	0,44 m/s
POINT OF COMPARISON 17			
MODEL	PH 1	PH 2	PH 3
Existing situation	2,01 m/s	2,06 m/s	2,29 m/s
Maximum occupation	0,25 m/s	0,23 m/s	0,35 m/s
Intermediate occupation	0,66 m/s	0,72 m/s	0,96 m/s
POINT OF COMPARISON 18			
MODEL	PH 1	PH 2	PH 3
Existing situation	2,02 m/s	2,07 m/s	2,31 m/s
Maximum occupation	0,45 m/s	0,50 m/s	0,68 m/s
Intermediate occupation	0,92 m/s	1,00 m/s	1,19 m/s
POINT OF COMPARISON 19			
MODEL	PH 1	PH 2	PH 3
Existing situation	1,91 m/s	2,07 m/s	2,30m/s
Maximum occupation	0,73 m/s	0,78 m/s	0,91 m/s
Intermediate occupation	2,30 m/s	0,91m/s	1,30 m/s

Simulating southeast wind direction to evaluate the wind flow at the points of comparison, in both maximum and intermediate models, the results were indeed, the points 32, 33 and 34 are set in high-density areas. There is a reduction of 50% to 70% below 50 m height in the wind velocities at those points when compared to the existing situation. [Renan, 2011]

Evaluation of velocity profiles in the intermediate model shows acceleration in wind velocity when compared it to the maximum density scenario. [Renan, 2011]

Compared to the existing situation, there is a little reduction in the intensity of flow at the entire lee wind area in points (35 -39) in the intermediate model.[Renan, 2011]

Wind speed reaches values of 5% to 15% lower than those registered in the existing situation do. [Renan, 2011]

In the maximum occupancy model, on the contrary reductions of 18% to 35% are registered. Figure 5-40. [Renan, 2011]

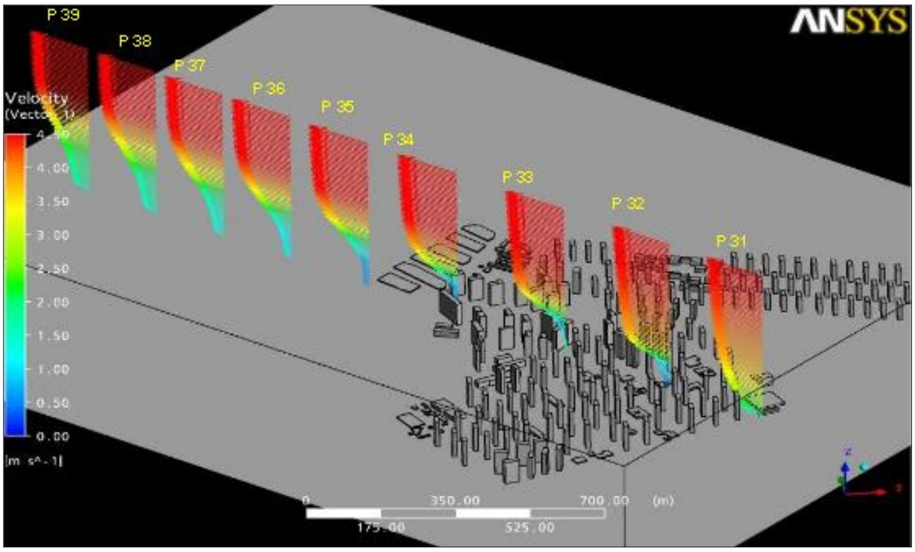

Figure 5-40: Localization of comparison points 31 to 39 in the maximum occupancy model (SE-wind). [Renan, 2011]

5.6. Conclusion

1. Wind pattern studies in many urban areas over the world have treated CFD as accepted, reliable, accurate, precise, and trusted tool. It is observed from the

first study (Compinas City) that the study was done at a residential area and all houses with height of 1 to 3 storey only, It was observed that in four residences, the wind flow reduced, in two the flow was about the same and in five; there was an increase of wind flow inside the terrain. [Oliviera, et al, 2009].

2. It was also observed that the position of the houses in the block affects the wind velocities. Some houses that were not modified had their wind flow changed due to the other houses and the pattern of the surroundings. [Oliviera, et al, 2009].

3. It was also observed that the position of the houses in the block affects the wind velocities. Some houses that were not modified had their wind flow changed due to the other houses and pattern of the surroundings. [Oliviera, et al, 2009].

4. In the second study (Singapore city), the residential buildings with height of 6 to 36 m, the CFD was used to predict how much of the wind follow is altered by buildings and to determine the wind pattern around buildings and to specify the area with bad wind circulation. [Chung, 2010]

5. In the third study, which the buildings with height between 6m to 72 m, It observed that, the largest reductions of air velocity were verified in the maximum density model. [Renan, 2011]. Compared to the intermediate density model results, all the points of comparison in the downwind area of the buildings registered lower speeds, with wind velocity values from 30% to 90% are slower than the existing situation model. The simulation results show that the verticality of the city and lower density rate allow better wind circulation within the city fabric since this urban form is more permeable to the wind. On the other hand, the elevation of the urban canopy layer and the reduction of the wind velocity at the back areas are the main effects of this urban form. . [Renan, 2011]

6. The study of natural ventilation conditions should be considered according to the conditions of each locality, since the conditions of wind circulation are easily changed by the construction parameters. [Renan, 2011]

7. This research might help urban design process specifically in parts of the city in constructive densification process. It indicated that despite the increase in height, the higher land use rates may cause the stagnation of the wind flow even if it is associated with lower building's height.

8. After studying the three cases of residential areas with buildings of various heights it could be concluded that the wind tunnels and full-scale measurements can be both time consuming and costly. As a result, CFD has become more widely accepted as an alternative tool for wind studies. CFD methods are convenient to access for design practice and simulate the flow field about a building and predict parameters of interest such as velocity, pressure, and temperature fields. The CFD therefore is adopted to analyze wind movement in this research.