INSTITUTO POLITÉCNICO DE BRAGANÇA
Escola Superior de Tecnologia e Gestão

Evaluation of the wind action in a single-family house of light steel ground floor

Imed Dkhili

Final report of the thesis presented to the

Polytechnic Institute of Bragança – IPB

For the fulfilment of the requirements for a master's degree in

Construction Engineering

May 2021

Evaluation of the wind action in a single-family house of light steel ground floor

Imed Dkhili

Final report of the thesis presented to the

Polytechnic Institute of Bragança – IPB

For the fulfilment of the requirements for a master's degree in

Construction Engineering

Supervised by:

PhD Carlos Alberto Rodrigues Andrade

PhD Luís Costa

May 2021

Acknowledgments

I would like to show my sincere appreciation to those who have contributed to this dissertation and supported me in one way or the other during this amazing journey for without the generous support of them, this work would not have been possible.

First, I would like to express my pride to be supervised by widely known and highly experienced scientific researchers.

Professor **Carlos Andrade**, I am very thankful to have such a wonderful, professional and perfectionist persons as my thesis supervisors. I am deeply thankful for your continuous support and patience in this learning path, for your persistent help, and for the constant motivation to do more and better. You offered continuous advice and encouragement and I benefitted from your knowledge and scientific experience. I appreciate your effort and I want to thank you for your wise guidance, kindness, and continuous availability.

I am deeply grateful to all members of the jury who kindly honored me by their presence to participate in the defense of this thesis.

My big appreciation to my special and wonderful friends, who helped me in every possible way to achieve my dreams and successfully complete this research work. I will never forget all their support and effort. You are the best.

Keeping the best for last, I would like to thank my parents Tawfik Dkhili, Alia Mezrigui, my brother Marwen and my sister Hana whose help made this journey possible from the beginning they believe in me and supported my studies abroad both morally and financially. They have been my inspiration and my unfailing source of passion and energy. Thank you for your irreplaceable support, for all your love, patience and kindness, for always encouraging me to pursuit my education, for believing in me and for everything that you have done to make me the person I am now. Without you none of this would have been possible. I love you!!

ABSTRACT

A building must resist the actions to which it is likely to be subjected during its life.

This thesis indicates how to determine the natural wind actions to be considered in the calculation of the building structures, as well as their constituent elements for each of the zones concerned by these actions. The local wind speed depends on the location, the height above the ground level, the type of terrain it encounters on its way. On its side, a building is an obstacle to the progress of the wind. The wind exerts a pressure on the walls which generates forces perpendicular to the surface of the building. The actions on the building depend on the shape, dimensions, rigidity, openings (doors, windows, leaks) made in the building, as well as its immediate environment.

Our work studies the evaluation of the wind' action in a single-family house on the second floor in light steel. With the first part we started the software of climatological advice to make a detailed study of the wind rose of the site of installation to have all the directions and speeds of the wind and we chose 3 maximum values of different direction.

we focus on a multidirectional numerical study by using software (design building) to get the results of the pressure and wind speed on the house and we used the three maximum values to find the wind rose.

Keywords: CFD analyses; Drag coefficient; Drag force; Fluid dynamics; DesignBuilder; Light Steel frame

RESUMO

Um edifício deve resistir às ações a que é suscetível de ser sujeito durante a sua vida.

Esta tese mostra como determinar as ações naturais do vento a ter em conta no cálculo das estruturas dos edifícios, bem como os seus elementos constituintes para cada uma das zonas afetadas por estas ações. A velocidade do vento depende do local, da altura acima do solo, do tipo de terreno que encontra no seu caminho. Um edifício é um obstáculo para o progresso do vento. O vento exerce uma pressão sobre as paredes que gera forças perpendiculares à superfície do edifício. As ações no edifício dependem da forma, dimensões, rigidez, aberturas (portas, janelas, fugas) feitas no edifício, bem como do seu ambiente imediato.

Este trabalho estuda a ação do vento numa casa unifamiliar no primeiro andar em aço leve. Na primeira parte começamos com o *software* de aconselhamento climatológico para fazer um estudo detalhado da rosa do vento do local de instalação, para ter todas as direções e velocidades do vento e assim escolhemos 3 valores máximos de diferentes direções.

E depois fizemos um estudo numérico multidirecional pelo *software (design building)* para obter os resultados da pressão e velocidade do vento e utilizámos os três valores máximos para encontrar a rosa do vento.

Palavras-chave: Análise CFD; Coeficiente de arrasto; Força de arrasto; Dinâmica de fluidos; Aço leve; Designbuilder

Index

List of Figure

List of Tables

Chapter 1:

Introduction

In the scenario of looking for alternatives with lower impacts on the environment, electrical energy production methods are the goal of studies that open room for the use of renewable energy sources with low to no emissions and pollutant generation, minimizing environmental impacts [1].

The wind is one of the oldest energy sources exploited by humans and today is the most established and efficient renewable energy source. The wind is source of available energy and the means of exploring this ability. It takes the focus of many research lines. Eolic technology, presents in a wide range of potential applications such as buildings to meet the energy needs of these locations, is among the solutions developed [2].

Wind in short, heavy buildings with thick walls is not a concern, but in slender structures, one of the most critical steps in the design of structures is to be decided. In accordance with NBR 6123/1988 "Forces due to wind in buildings," considerations for evaluating the forces due to the wind are controlled and measured [3].

Wind power consists of transforming the energy produced by wind turbine blade movement into electrical energy. Moreover, wind power is a 'native' energy because it is available almost anywhere in the plant, leading to the reduction of energy imports and the creation of wealth and local jobs. For these reasons, the production and efficient use of electricity via wind energy contributes to sustainable development. Wind energy does not release into the air any toxic substances or toxins that can be very harmful to the environment and to humans [4].

The wind is produced by temperature differences of air masses in the atmosphere, the easiest case to identify is when a cold front arrives in the area and collides with the hot air producing wind, this type of phenomenon can be observed before the start of rain.

The present work aims to determine certain properties of fluid dynamics, focusing on the drag coefficient and drag force of an autonomous wind created by building in a wind tunnel using experimental methods and analysis of Computational Fluid Dynamics (CFD).

1.1. Justification

In fluid mechanics, because of the existence of the phenomena involved, purely theoretical methods of study raise a high difficulty to be overcome. This complexity causes the amount of data that can be collected by these methods to be reduced.

This complexity inspired the development of new analytical methods and turned physical and computational experimental methods into the most useful tool for obtaining fluid dynamic properties. The drag coefficient is between these characteristics, the correct determination of which is necessary to make a properly built part that can be implemented in the lowest possible way in operation low risk of failure.

I.2 Objectives

The main objectives of this work are:

- To study the evaluation of the action of the wind in a single-family house on the first floor in light steel frame and to evaluate the drag coefficient of a house obtained by CFD simulations using the design builder software.

- To achieve the general objective, the following specific objectives have been defined:

a) we used the climatological consulting software to make a detailed study of the wind rose of the installation site to have all the directions and speeds of the wind.

b) Modeling of a three-dimensional component using technical drawings provided with real dimensions. This process is carried out using computer aided design (CAD) software.

c) Conversion of the CAD file using the actual dimensions of the component into a compatible format and import into the CFD software used.

d) Simulation of the component with real dimensions aimed at determining the force acting on the body and calculating the drag coefficient for three different positions.

d-1) obtained for wind tunnel simulations and CFD analysis.

D-2) Carry out a detailed study of the wind rose of the implantation site.

D-3) Digital study (design building) multidirectional box1.3 Light Steel Framing (LSF)

The prescriptive light steel construction system, or Light Steel Framing (LSF), constitutes an expeditious design process for the cold-formed steel design of simple structures (one or two floors). The method was developed in conjunction with current structural regulations by AISI (American Iron and Steel Institute) and published for the first time in 1997[5].

Moreover, with the cooperation of LSF researchers, manufacturers, developers and building commissions, this method was developed. This document encourages the use of modern materials for non-structural uses by encouraging the construction of homes with a light steel frame and provides manufacturers and users of cold-formed steel with uniform and clear specifications in LSF [5].

The construction of structures in LSF (“Light Steel Framing”), “Light Steel” or “Cold-Formed Steel”, originates from its designation in the fact that the structural elements are manufactured from folded steel sheet which, being thin, gives the steel structure a light appearance - see Figure 1.1. Although the appearance may suggest the opposite due to the large number of structural elements, it is found in practice that an LSF structure has a much lower final weight (several orders of magnitude) than solutions in concrete or masonry.[5].

Figure 0.1: LSF structure of a house with about 400 m2 and two floors

The demand for the construction of cold-formed steel structures has been gradually rising over the last decade and is now emerging as a viable and successful alternative to other construction solutions.[5]

Among other factors, LSF construction is often correlated with shorter building times, good thermal and acoustic quality, and superior environmental performance. If LSF construction is a relatively recent development in Portugal, it is a fact that LSF construction in the United States of America reflects this form of conventional construction (USA), where it was originally proposed, and in Australia [6].

Recognizing the high cost of steel relative to other structural materials, it is reasonable that any solution that enables the weight of steel to be minimized will often be an economical metal solution. In cold bending (i.e., at room temperature or "mild steel") of reduced-thickness steel sheets, the principle behind "cold-formed" is. All the manufacturing and assembly activities for the profiles are carried out at room temperature, except for the production of the steel sheet itself [6].

The folded sheet profiles used in the LSF construction are considerably lighter than the standard hot-rolled steel profiles for resistant stress values of the same order of magnitude (I, H, T, U, and L). "That is why a natural evolution of metal construction is "mild steel [7].

1.3 Presentation of the used software's

1.3.1 Designer builder

Design Builder is a dynamic simulation software, with a graphical interface offering many features not available simultaneously in existing software [8].

- Calculation of envelope thermal losses / benefits in winter / summer
- Dimensioning heating
- Dimensioning of refrigeration by natural ventilation and/or air conditioning
- Dynamic simulation (STD) that restores comfort data, heat balance, ventilation, etc.
- Realistic 3D building with drop shadow vision (BIM model)
- Modeler for construction, including window creation wizards, composition of construction, automatic detection
- Energy saving: free cooling, energy recovery from harvested air, night ventilation, brightness dimming, demand-based control of supply air temperatures, variable air volume... Within a few taps, you are already available.
- Map of natural illumination in FLJ and autonomy of light

- Module for optimization to determine the building parameters that give the best compromise between cost, comfort and GHG

Design Builder, including 3D design, day lighting and simulation, is a fast and accurate modeling program. On the Design Builder website, you will find out more.

1.3.2 Climate Consultant

A software for designing energy efficient building.

In this study, the concept of Climate Consultant Software for making a building energy efficient is represented. This is a modern software which is very helpful now-a-days in designing the buildings. Climate Consultant helps us in designing climatic data like temperature, humidity, dew point, sky cover range, wind velocity, psychrometric chart etc.

1.4 Building location and presentation:

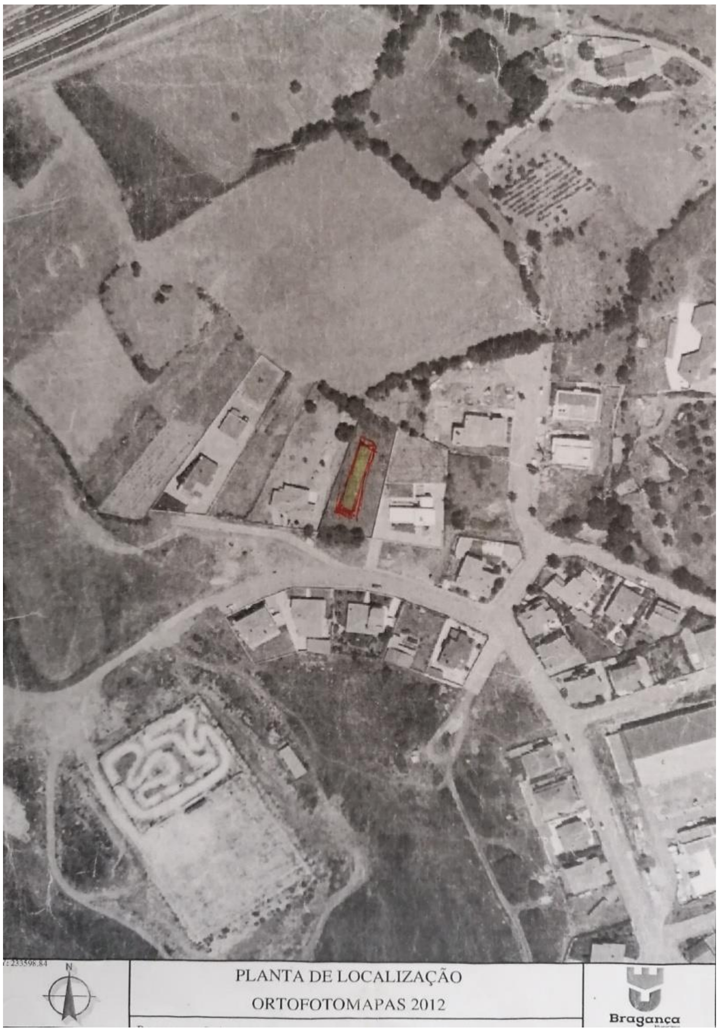

Figure1.2: Location Plan

My project's building is in the center of a ground that has the form of a quadrilateral. There are some medium trees in its north and south. In the Northeast we find an empty space however in the south-east there is another building. That corresponds to the right side. On the left side, there is some other trees in the Northwest. In the west, there is another empty space. In the Southwest there is another medium building.

This work is achieved using an autonomous wind [9].

1.5. Work structure

To present the theoretical contents necessary and to demonstrate the experimental method correctly, this work is divided into the following sections.

chapter 2: listed the essential previous works that use similar concepts, seek for fluid dynamic properties, or use the same methods. And presented the basic concepts that allow the correct understanding of the phenomenon that occurs during the proposed analyses. In this chapter is also presented a historical content that permits the understanding of the importance of experimental methods.

chapter 3: presented the experimental procedures used in the analyses.

chapter 4: tackled the obtained results and commentaries about then

chapter 5: presented the conclusions that can be made after the experiments performed.

Chapter 2:

Fluid dynamics theory and computational fluid dynamics

The two-dimensional external airflow are exposed to the studied portion, where two dimensions of the exposed body are used during the analysis process [10].

Airflow, like other fluid flows, poses a large number of fluid mechanics-related phenomena, as well as, properties that depend on the characteristics of the fluid and the form, size and orientation of the body immersed in that flow[10, 11].

Some fundamental concepts important for the correct realization of the proposed analyses and for interpreting the results obtained are presented in this chapter. Historical material is also provided in this chapter.

2.1. Introduction to fluid mechanics

Fluid Mechanics is the study of fluid that explains its properties and interactions with other materials, whether it is in motion or stationery. This branch of mechanics can be used in various fields, such as the construction of pumps and pipes and the air conditioning of residential buildings [12, 13].

A liquid can be found in a liquid or gaseous state and the continuous deformation when a shear or tangential stress is applied that occurs independently of the stress module is the main characteristic. It is normal to say that a fluid will flow if it is injected into it when subjected to some form of contact with the environment [12, 13].

The movement of fluid particles is represented in fluid mechanics using current lines that allow us to see what happens to a fluid when it encounters a solid body immersed in the flow and allows certain points of interest to be identified [12].

The first point is called the point of stagnation in which any fluid particle is at rest. The current lines appear to open around the body, reflecting that the velocities are low near the body surface and, therefore, the pressures have a high value [12].

The no-slip condition makes the velocity of the fluid on the surface of the body equal to zero when a fluid in motion interacts with a fixed rigid body. The creation of a velocity profile and the formation of a boundary layer are responsible for this phenomenon, as shown in Figure 2 [13].

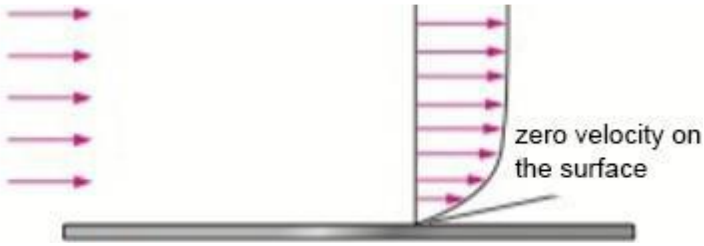

Figure 2.1: Boundary layer and velocity profile in a fluid flow over a solid surface [13].

As the flow develops, the velocity tends to increase causing a pressure decrease. Due to the formation of the boundary layer and the adverse pressure gradient existing over the solid body, in a certain point, known as the separation point, the fluid particles begin to detach from the body surface causing the formation of a region known as wake turbulence [12]. In Figure 3, point A represents the stagnation point, point B is a low-pressure point and point D represents the separation point.

Inside the wake turbulence region, the flow is turbulent and the pressure as a low value but in contrast, the region that surrounds the wake turbulence region presents a higher-pressure level. This adverse gradient is responsible for the formation of the drag phenomenon [12].

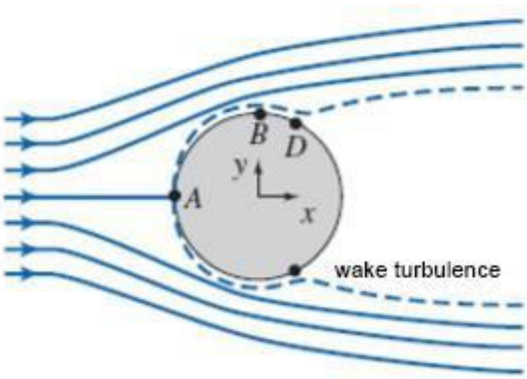

Figure 2.2: Behavior of the current lines over a solid body immersed in the flow [12].

2.2. Drag force

It is subjected to the action of a set of forces when a solid body is submerged in the flow of some fluid. The drag force, which describes a force acting parallel to the direction of the fluid flow but in the opposite direction, is among these forces [6, 12].

The drag force is a composition of pressure and shear forces and consists of the loss of flow that must be resolved to enable the body submerged in a flow to travel. This phenomenon is undesirable and must be reduced to ensure greater protection and increase the useful life of structures exposed to fluid flows [6, 13].

Forces of fluid dynamics present a complex origin that for direct analytical determination makes their determination difficult and for the drag force it is no different. Its drag coefficient (CD) can be used by the Eq to assist the determination of the drag force acting on a body. Eq.1 of [12].

$$\mathbf{CD} = \frac{\mathbf{D}}{\frac{1}{2}\boldsymbol{\rho}\mathbf{U2A}} \qquad \text{Eq.1}$$

Where:

CD: is the drag coefficient.

D: is the drag force measure in N;

ρ: is the specific weight of the fluid according to the application temperature;

U: is the flow velocity measured in m/s;

A: is the frontal area measured in m^2.

In the Eq.1, it is possible to see. that, once its frontal region is part of this equation, CD depends on the geometry of the body and may classify a body as aerodynamic or blunt according to the direction of flow. CD is also dependent on the velocity of the flow, and consequently flows to low velocities of the Reynolds number (Re) where its values begin to increase once the boundary layer becomes turbulent.

This effect arises because shear forces derive the main contribution of the drag force and these are stronger in turbulent flows [10, 11].

CD values for a set of simple geometries can be found in tables, although other approaches need to be calculated for complex geometries.

Determine the CD by using Eq.1, the exact weight of the fluid needs to be known according to its temperature as well. For a set of products, this property can be found in tables. For air, its basic weight is shown in Table 1 with an ambient pressure of 1 atm and the action of this property with a change in temperature is shown in Figure 3.

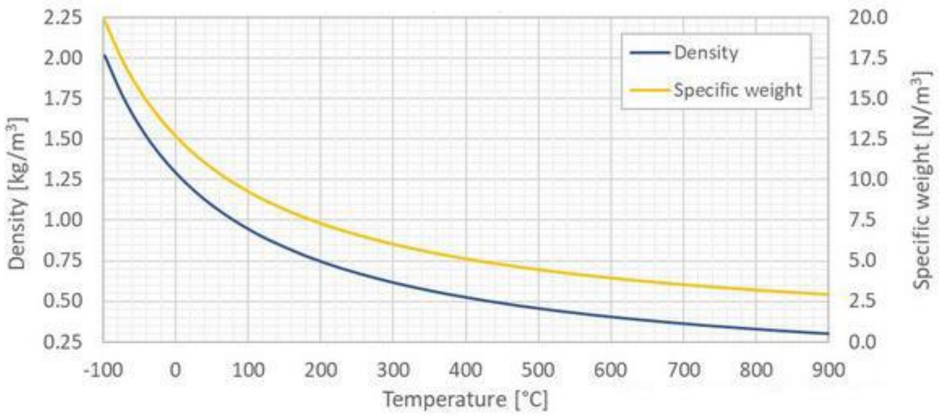

Figure 2.3: Air density and specific weight at atmospheric pressure [14].

Table 2.1: Specific weight o fair for 1 atm. [14]

Temperature (^{0}C)	**Density (kg/m^3)**
30	1.164
25	1.184
20	1.204
10	1.246
5	1.268
0	1.292
-5	1.316

Once the drag effect is associated with wake turbulence, it can be managed to create an aerodynamic structure by adjusting in the geometry of the body such a lower pressure differential is achieved at the rear of the body.

The drip shape shown in Figure 4 is an example of an aerodynamic shape used as a guide for designing several components, especially aeronautical components.

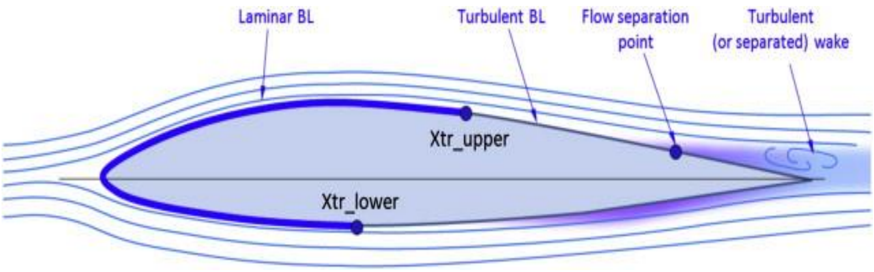

Figure 2.4: Current lines over a drip shaped body [11].

2.3. Mechanical action of the wind

The wind works in the form of pressure directly on the body and may differ over time. A simplified representation using equivalent forces in a Cartesian plane can be rendered for turbulent flows [15].

The wind speed can be split into two parts, known as average velocity and float velocity. More relevant is the average velocity (vm) which can be measured by using Eq. 2 [15].

$$v_m = c_r(z) * c_o(z) * v_b \qquad \text{Eq.2}$$

where:

V_m: is the average velocity of the wind measured in m/s in certain high (z) above the ground.

C_0: is the orography coefficient ,usually defined as 1.0;

C_r: is the roughness coefficient and can be defined as

$0.19 * (\frac{z0}{0.05})^{0.07} * \ln\left(\frac{z}{z0}\right)$ for $z_{min} \leq z \leq 200$ m

$C_r = C_r(z_{min})$ for $z \leq z_{min}$

The values for z0 and zmin, which indicate the roughness length and minimum height, respectively, must be calculated based on the ground form. Table 3.2 lists the potential values, which are shown in Figure 3.5

Table 2.2: Values for z_O and Z_{min} according to the ground type [15].

Ground type		Z0 (m)	Zmin (m)
0	Sea or coast zone exposed to winds coming from the sea	0.003	1
I	Lakes or plane zones with almost no vegetation and free of obstacles	0.01	1
II	Zone with ground vegetation and isolated obstacles (trees or builds) with a distance of, at least, 20 times its highness between them	0.05	2
III	Zones with regular vegetation or buildings coverage with isolated obstacles with a maximum of 20 times its highness between them (villages, permanent forests)	0.3	5
IV	Zone with at least 15% of the surface is covered by builds with at least 15 m high.	1	10

The roughness coefficient can also be calculated using the curves in Figure 3.6, which demonstrate the relationship between the ground form and the ground's high decrease.Eq. 3 [17] can be used to calculate the reference value of the wind velocity (vb):

$$V_b = c_{dir} * c_{season} * V_{b,0} \qquad \text{Eq.3}$$

Where:

V_b is the reference value of the wind velocity.

$V_{b,0}$ is the basic value of the wind velocity; 14

c_{dir} is the direction coefficient usually adopted as 1,0;

c_{season} is the seasonal coefficient usually adopted as 1,0.

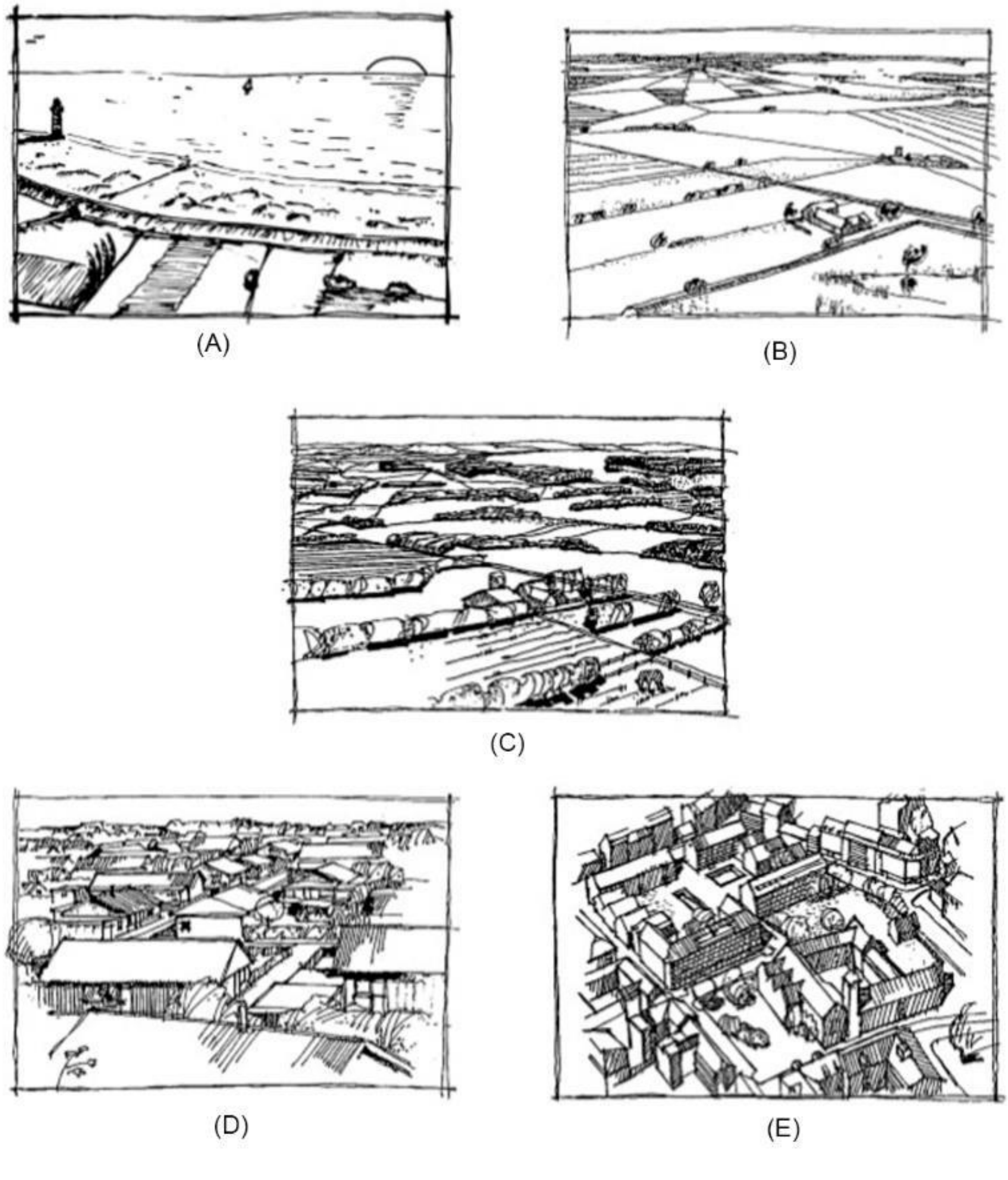

Figure 2.5: Representation of the groud types where (A) is type 0, (B) is type I, (C) is type II, (D) is type III e (E) is type IV [15].

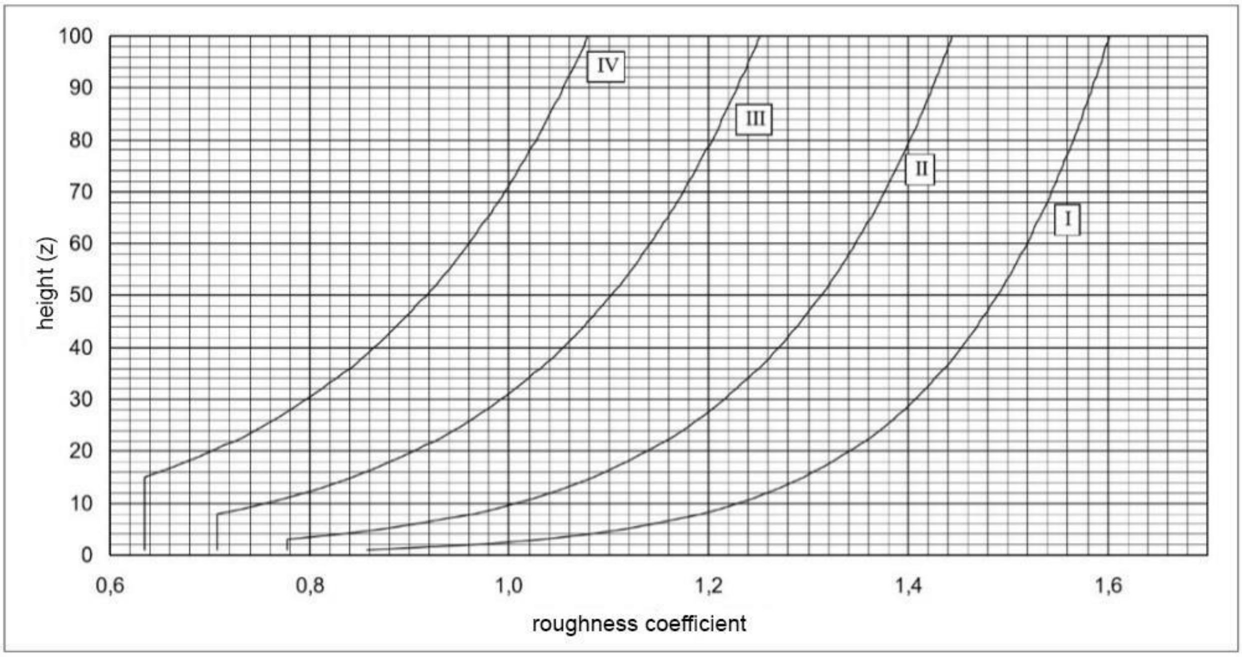

Figure 2.6: Roughness coefficient curves according to the ground type and high in relation to the grounds [15].

The basic value of the wind velocity can be determined according to the ground type as shown in Table 2.3

Table 2.3: Basic value of the wind velocity according to the ground type [15].

Zone	$V_{b,0}$(m/s)
(A)General ground	27
(B)Açores archipelago and Madeira; continental regions located on the coast with at least 5km length or high of at least 600 m	30

2.4.Introduction to computational fluid dynamics simulation

In order to reduce the project time and the number of tests necessary, Computational Fluid Dynamics (CFD) has become increasingly common, allowing global properties such as lift and drag force to be determined more quickly and also allowing the flow field characteristics to be determined [13].8

The study of structures containing systems involving computational fluid dynamics or CFD, which is a way of computer-based modeling, fluid flow, heat transfer and related processes such as chemical reaction. The approach is very effective and covers a wide variety of fields of industrial and non-industrial use. There are some examples:

- Aerodynamics of aircraft and vehicles: lift and drag
- Hydrodynamics of ships
- external and internal environment of buildings: wind loading and heating/ventilation

The computational algorithms that can be discussed are structured around CFD codes.

Fluid-flow questions. To have easy access to their solving strength, in order to sophisticated user interfaces for feedback include all commercial CFD packages. Parameters of the dilemma and to analyze the effects. Therefore, the three key components of both codes are: I a pre-processor, (ii) a solver, and (iii) a post-processor.

Simply stated, from the iterative solution of Eq.4 and Eq.5, a CFD analysis runs called the continuity equation and the Navier-Stokes equation respectively, initially chooses a domain and generates a mesh containing small elements in which the equations are resolved [5].

Once the mesh is correctly defined, the boundary conditions can be determined according to the problem studied, the fluid type and its properties defined, and the solution algorithms present in the program used to perform the simulations chosen.

$$\vec{\nabla} * \vec{V} = 0 \qquad \textbf{Eq.4}$$

where

$\vec{V}$ is the flow velocity. $$(\vec{V} * \vec{\nabla})\vec{V} = -1\rho\vec{\nabla} P' + \upsilon\nabla 2V \qquad \textbf{Eq.5}$$

where

$\vec{V}$: is the flow velocity;

ρ : is the specific weight of the fluid according to the application temperature; is the kinematic viscosity of the fluid defined as $\upsilon = \mu/\rho$;

P': is the modified pressure due to the absence of free surface effects.

2.5 Mesh generation

The planning of the mesh that will be used is one of the most critical steps in a CFD simulation, taking as long as it takes to ensure that the mesh will show the best possible characteristics [13].

For complex geometries, while it is more difficult to achieve, preferably aim to obtain a standardized mesh in which its components can be counted according to an i, j, k coordinate system. This form of mesh contains fewer components and thus facilitates better resolution in areas of higher importance, i.e., findings are more reliable [13].

Its consistency is another point to remember in mesh construction. The lowest possible orientation of the elements must always be assured and abrupt changes in their scale must be minimized, meaning that the integration of CFD codes is not impeded.

It is also more recommended to apply an unstructured mesh, i.e., the elements of which do not conform with the coordinate system I j and k but have a better consistency than a standardized mesh [13].

2.6 Boundary conditions

To obtain a solution through CFD analysis, it is necessary to model the flow correctly using the correct boundary conditions. The first and simplest of these boundaries is called a "wall" in which a fluid cannot pass through and both the normal and tangential components of fluid velocity are equal to zero. It is also possible to model a wall in which the fluid can slip along. This type of wall is called a non-viscous wall and can be used for free surface cases or when the model boundary represents an open environment [13].

It is also necessary the specification of the flow inlet and outlet conditions and it can be done in various ways. For flow inlet, it can be defined as a velocity or pressure input where the flow inlet velocity or the total pressure along the inlet must be defined respectively. In the case of the pressure output, the pressure along the surface outside the computational domain, whose value is normally assumed to be the atmospheric pressure value shall be provided. Due to the coupling of velocity and pressure equations to avoid mathematical over speeding, pressure values should not be provided in the velocity input boundary conditions and the same is true for situations where pressure inputs or outputs are used where velocities values should not be provided [13].

Another usual condition for domain output is to define it as an outflow boundary condition. This type of condition is used for the case where the flow is fully developed, i.e., the velocity profile does not change with the flow evolution, in which case no flow property must be specified so that they are forced to have a null gradient in the normal direction to the exit [13].

Chapter 3:

Methodology

A building must withstand the actions to which it is likely to be subjected during its lifetime. This thesis indicates how to determine the actions of the natural wind to be considered to calculate the structures of the buildings, as well as their constituent elements for the zone most affected by the wind by these actions. The local wind speed depends on the location, the height above the ground, the type of terrain that it encounters on its path. For its part, a building presents itself as an obstacle to wind progression. The latter exerts pressure on the walls which generate forces perpendicular to the surface of the construction. The actions on the construction depend on the shape, dimensions, rigidity, openings (doors, windows, leaks) made in the building, as well as its immediate environment.

In order to obtain the desired results, design building software is used for this computational analysis the steps necessary to perform the modeling and the CFD analyzes are described.

3.1 The Climate Consultant 6.0 software

The purpose of Climate Consultant 6.0 is not simply to plot climate data, but rather to organize and represent this information in easy-to-understand new ways that reveals the subtle attributes of the climate and its impact on built form. The goal is to help users create more energy efficient, more sustainable buildings, each of which is uniquely designed for its spot on our planet. The basic plots are the usual temperature, radiation, and humidity bar graphs, while beyond these are more sophisticated graphic analysis options.

3.2 The different steps for using the Climate Consultant 6.0 software

Weather data in Energy Plus Weather (EPW) file format can be downloaded from the US Department of Energy's EnergyPlus site while running Climate Consultant. You can download weather files for any of the thousands of places around the world. You also have the possibility to choose data for one of the climatic zones of Portugal by selecting a zone on a map of PORTUGAL or by entering the postal code of the zone which interests you.

Project data is automatically saved to a file upon exiting Climate Consultant and can be recalled at the start of the next Climate Consultant session by selecting Continue previous session. I also name and save the project data into files to be opened in subsequent sessions.

Another new option allows me to create custom plots of the original EPW data and / or statistics calculated by Climate Consultant. I printed the complete Climate Consultant window (graphic frame, legend frame and title bar), also, I select one of the four definitions of comfort. Each has its own Criteria screen and changes the way comfort is displayed on the psychrometric chart and other graphs. The shapes of the comfort zone and other passive design strategy zones have been updated to meet the latest comfort criteria as well as any user-modified criteria. I chose the Adaptive Comfort Model in ASHRAE 55-2010 as a comfort model. The climate consultant will determine the best set of passive design strategies displayed on the psychrometric map for the selected weather site. This is the smallest set of strategies that maximizes the number of hours of comfort without using conventional heating or cooling.

Based on the climate data and the set of passive design strategies selected on the psychometric map, the climate consultant will produce a list of design guidelines. If you are designing a residential building, each guideline is accompanied by a graphic sketch to help illustrate the application of this guideline. If you are designing a small non-residential building, many guidelines contain links to the 2030 palette (2030palette.org) for examples and supporting information.

A feature added to psychometric chart analysis displays the times of a specific day as dots in order. This gives us a fine-grained view of the daily changes in temperature and humidity. Additions to the sun shading diagram allow you to click and drag shading masks for fins and overhangs and enter data on objects (trees, buildings, chimneys) that shade windows or sensors solar. The number of unshaded hours that shading is needed, solar gain is needed, and when solar gain is not useful, as changed by shading from distant objects, fins and overhangs is calculated and displayed. A radiation trace on an inclined surface is displayed on the radiation range graph. Slanted surface radiation is also a plot option on the timetable plot and 3D graphics.

3.3 The results of the Climate Consultant 6.0

4.3.1 Monthly Wind Velocity Range

Wind Velocity Range: Wind speed is caused by air moving from high pressure to low pressure, usually due to changes in temperature. Wind Direction, from which the wind blows, is defined as the number of degrees from North, measured clockwise. For calm winds, wind direction equals zero. The Monthly Mode displayed in the Weather Data Summary is the wind direction that occurs most often. It is measured in 10-degree increments. The recorded wind speed is 6 m/s as a minimum value and 29 m/s as a maximum value, but in the simulation, we need the average wind value is 10 m/s. This can be illustrated in Fig:

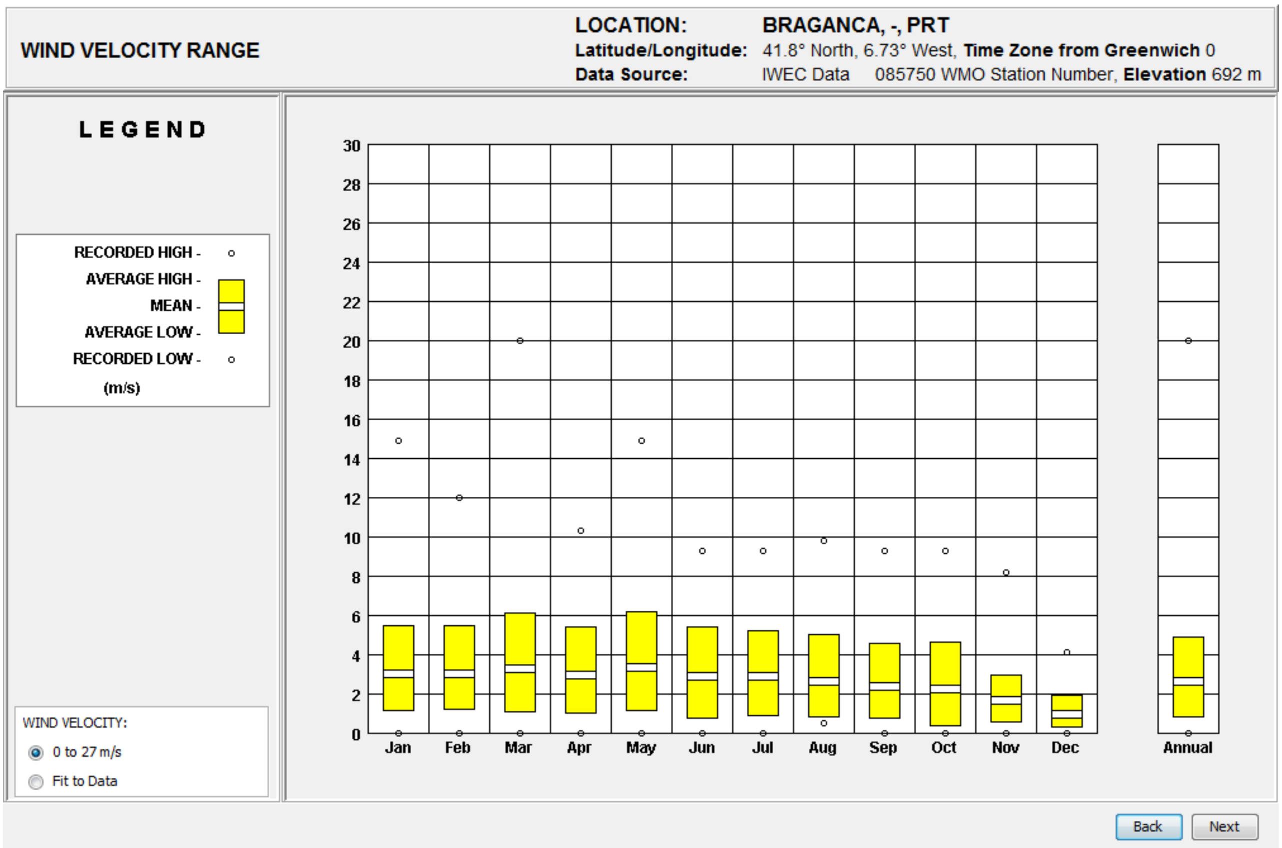

Figure 3.7: Monthly Wind Velocity.

3.3.2 **Monthly Dry Bulb and Dew Point**

Dry Bulb and Dew Point: is shown in 12 Charts in fig 7. These 12 charts are the average for each hour, and of each month of the Dry Bulb Temperature (yellow dot) and the concurrent Dew Point (green dot). Also shown on each monthly chart which is a grey bar for the Comfort Zone. Dry Bulb temperature increases sharply at sunrise and peaks around 2 or 3 in the afternoon, but that Dew Point temperature is relatively stable throughout the day. As shown in Fig 3.2

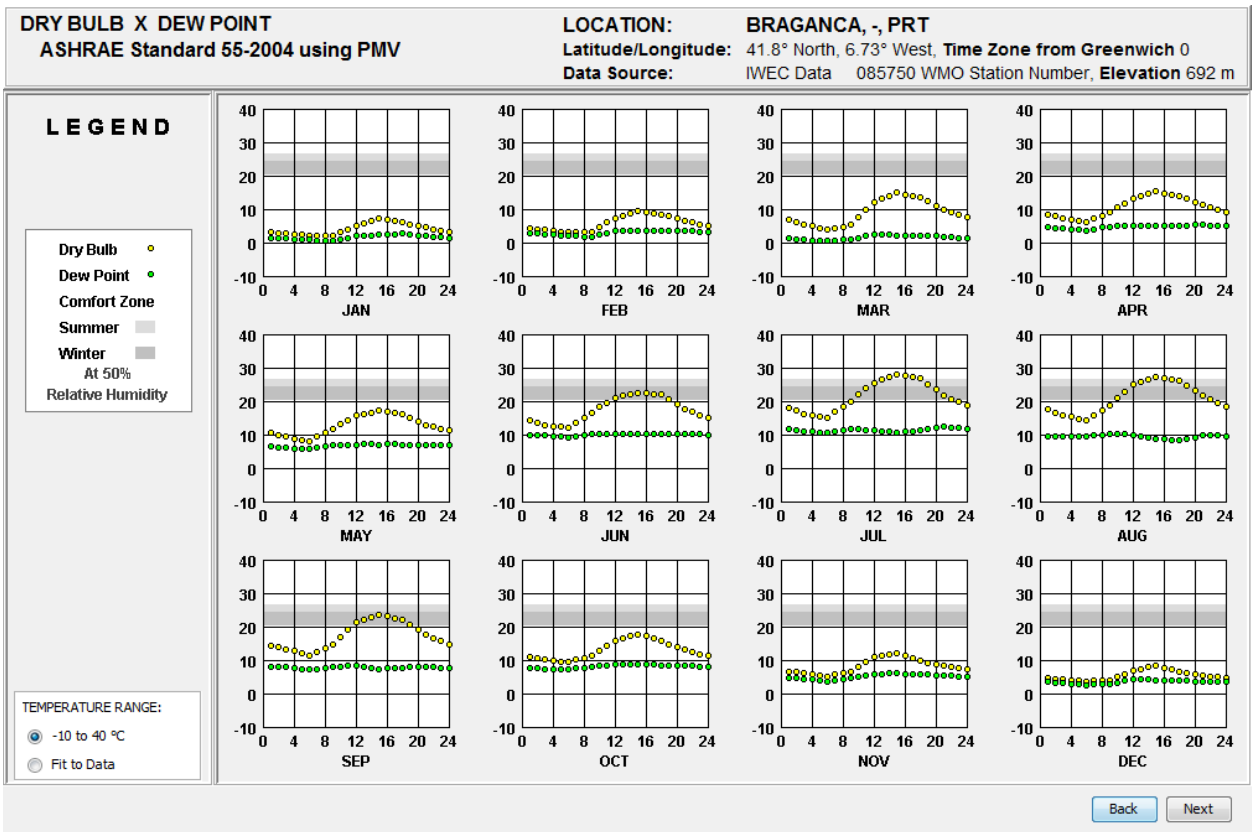

Figure3.2: Monthly Dry Bulb and Dew Point.

3.3.3 **Ground temperature (Monthly Average)**

Ground Temperature: The Average Monthly Temperature of the soil at various depths 1.64, 6.56, 13.12 feet in yellow, parrot and green colors respectively are shown on the Ground Temperature chart (Fig 8). The top and the bottom of the bar charts on the right expresses the highest monthly temperature and lowest monthly temperature, while the average monthly temperature is shown in the centre of each bar. Depth is given in feet (or meters) and the temperatures are in degrees F (or degrees C). As the depth increases the thermal mass of.

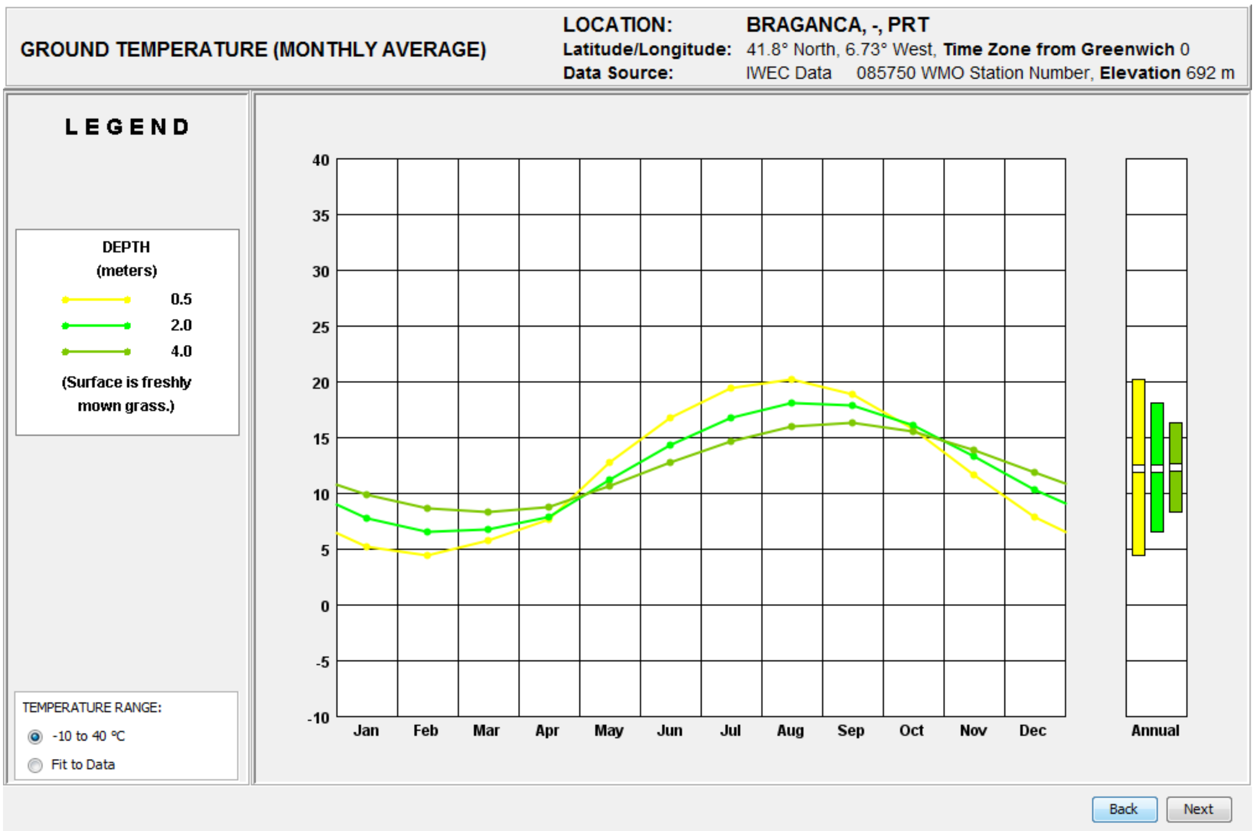

Figure03: Ground temperature (Monthly Average).

3.3.4 **Monthly Dry Bulb and Relative Humidity**

Dry Bulb and Relative Humidity: is the sensible temperature typically measured by a thermometer with a dry bulb while Relative Humidity is the ratio of the amount of moisture in the air compared to the total amount it could hold at the same dry bulb temperature. Relative Humidity is measured as a percent while Dry Bulb are either in degrees C or F.

These 12 charts (Fig 9) are the average for each hour of each month of the Dry Bulb Temperature (yellow dot) and the concurrent Relative Humidity (green dot). Also shown on each monthly chart is a grey bar for the Comfort Zone as defined on the Criteria screen. Dry bulb temperature is almost exactly the inverse of relative humidity.

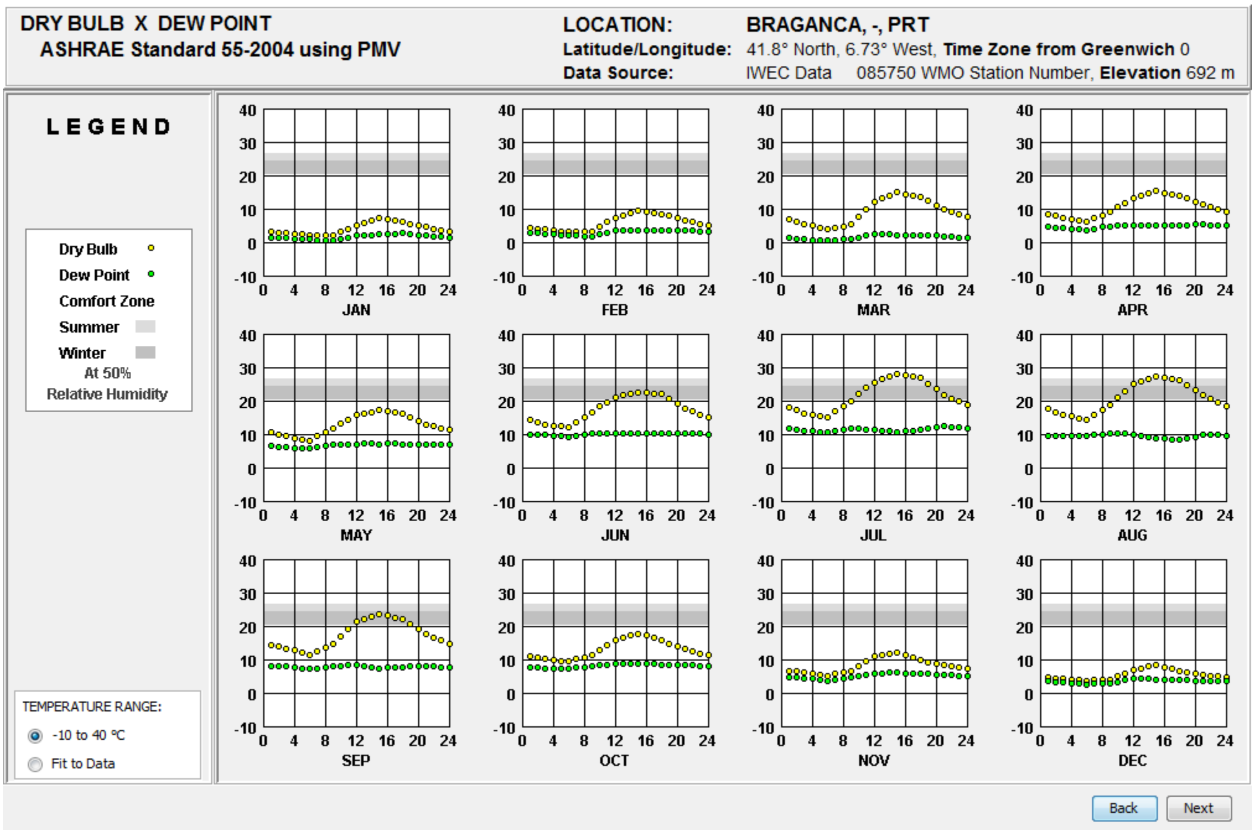

Figure:04: Monthly Dry Bulb and Relative Humidity

3.3.5 **Psychometric Chart**

Psychometric Chart as shown in Chart (Fig 10), it is one of Climate Consultant's most powerful design tools. It shows the temperature of the dry bulb at the bottom and the moisture content of the air at the side. This vertical scale is also called absolute humidity and can be represented as the humidity ratio in pounds of water per pound of dry air (or grams of water per kilogram of dry air), or as the pressure of steam. The curved line on the far left is the saturation line (100% relative humidity line) which represents the fact that at lower temperatures the air may hold less moisture than at higher temperatures. Each hour in the EPW climate data file is represented by a dot on this graph. Some points may represent more than an hour, for example when a given temperature and humidity occurs more than once a month. The point of a given hour may meet the criteria of more than one policy zone, in which case it is counted in the percentage of hours for both zones, therefore the percentages add up to more than 100%.

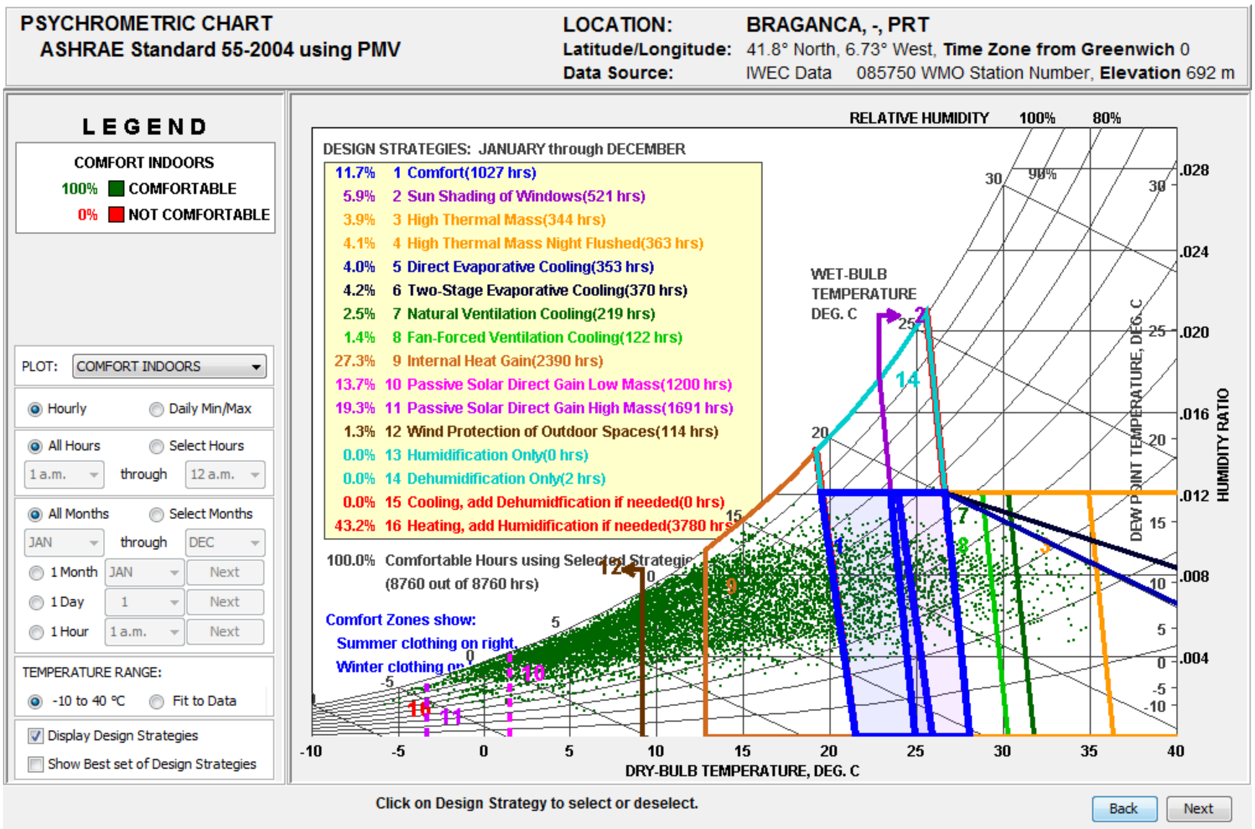

Figure05: Psychometric Chart

3.3.6 The Wind Rose of Hissar 1

The wind rose describing the predominant direction and frequency and velocity wind in the region is shown as in Fig 6: Here, the Wind Rose for the month of June is conspired .Likewise, we get for the other months also . Red shade displays the temperature range between 81-100 °F and dark red shade displays temperature above 100 °F. Light green shade displays relative humidity between 30-70 %. Pencil nib type triangles in light orange shade depicts low wind speed while in dark orange shade depicts high speed

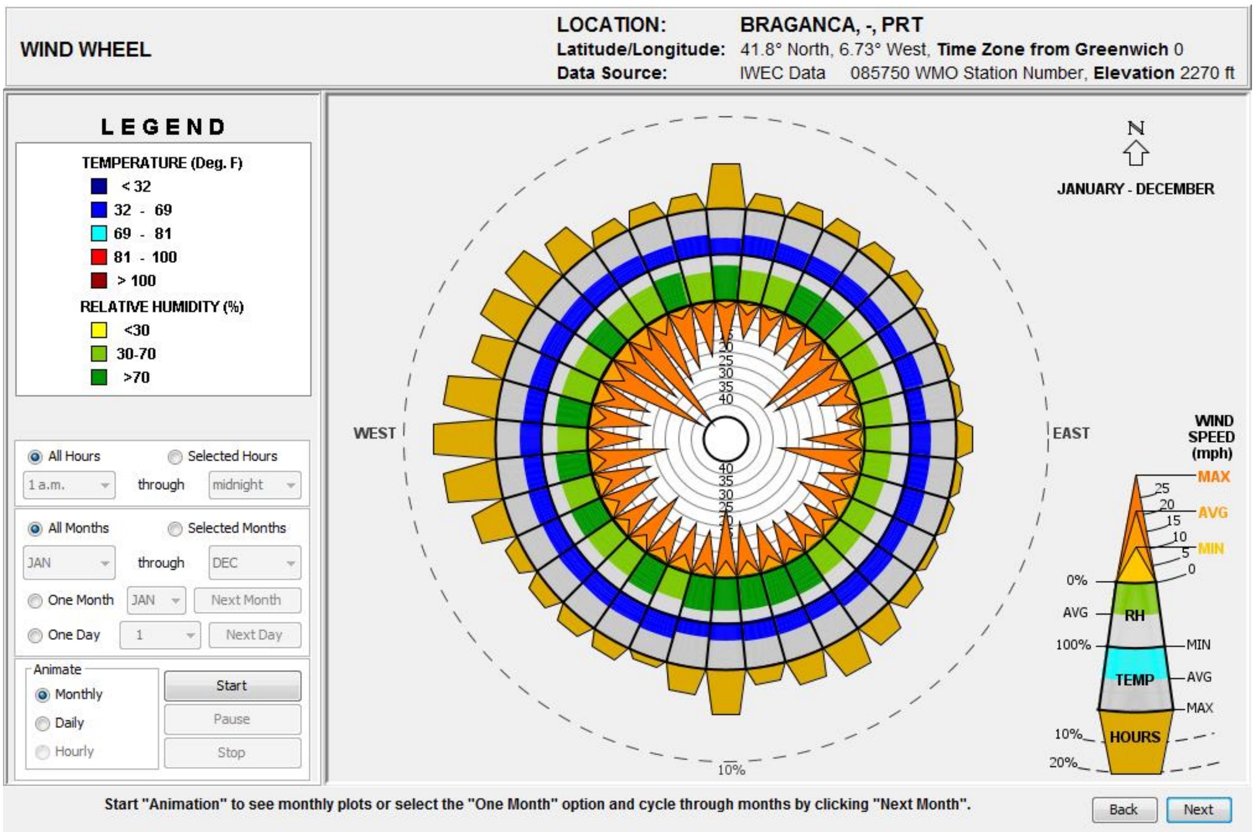

Figure 3.6: Wind Rose of Hissar

In the calculation of the wind action we used the average wind speed values and we chose 3 interesting values to find good results in the north west side the wind speed equal 6m / s and in the north side - equals 5.9m / s and in the south equals 5.7m / s.

Chapter 4:

The CFD simulation

4.1 Design builder software

Design Builder is a user-friendly modeling environment where you can work (and play) with virtual building models. It provides a range of environmental performance data such as: energy consumption, carbon emissions, comfort conditions, daylight illuminance, maximum summertime temperatures and HVAC component sizes. I have used the Detailed Design of HVAC and Natural Ventilation Systems including the impact of supply air distribution on temperature and speed distribution in a room using CFD to perform the simulation

4.1.1 **Integrated CFD simulation coupled with thermal simulation**

CFD is used to calculate air flows and temperature distribution in a room or around buildings. Design Builder automates technical CFD initiation tasks so that you can focus on studying phenomena. You can enter objects (free drawing), blowing, extraction, radiators, radiant, and panels …. etc. The CFD module can use the boundary conditions (wall temperatures, windows, outside air, air flow through the sash) previously calculated by Energy Plus to analyze a very specific situation. [21]

4.2 The Different steps for CFD simulation

The New CFD Analysis dialog is displayed which allows you to name the analysis, define grid generation variables, wind data and the extents of the domain to be included in the analysis:

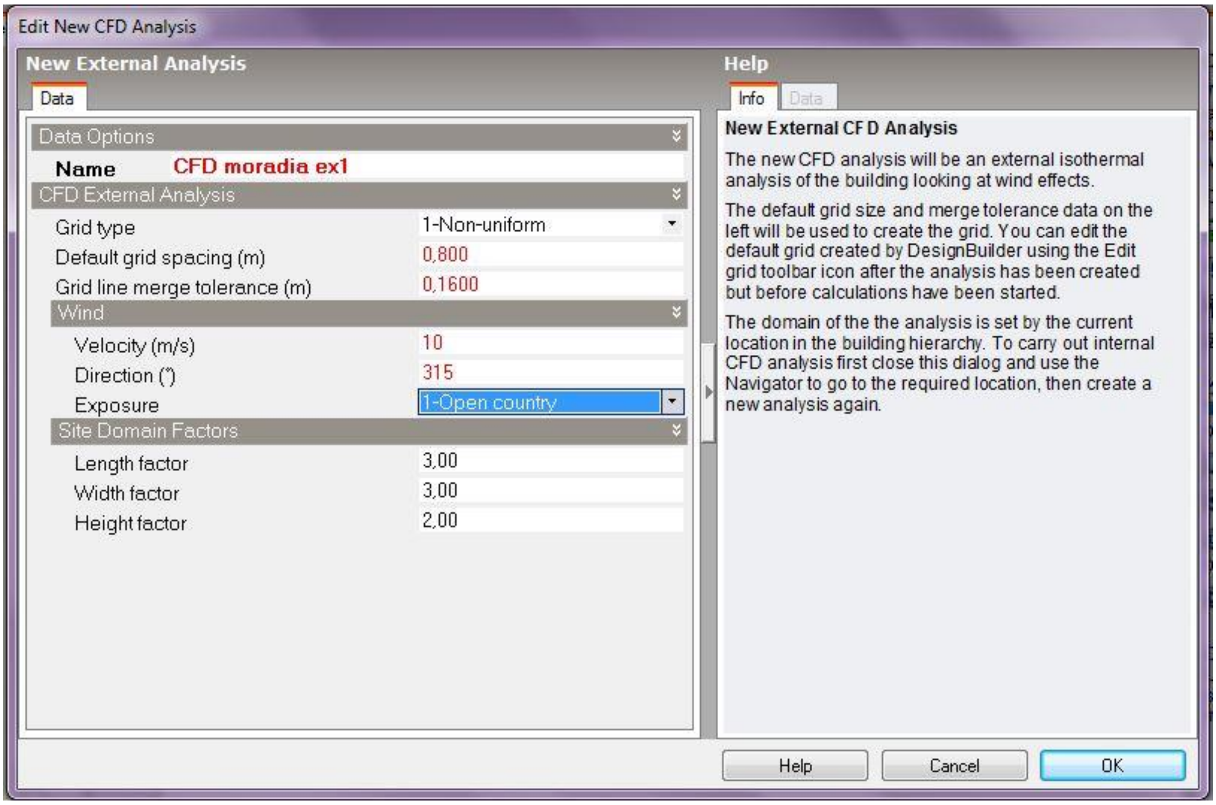

Figure 4.8: Edit new CFD Analysis

4.2.1 **New External CFD Analysis**

4.2.1.1 Grid type

I choose the grid non-uniform performing external CFD analyses:

The non-uniform grid used for external analyses is the same as the grid used for internal analyses, in which a list of key coordinates is found from the geometry and these key coordinates are used to set up grid areas that are then spaced uniformly using the default grid spacing. Tolerance for merging grid lines has been set.

4.2.1.2 Default grid spacing

The CFD grid is automatically constructed throughout the total external domain extents when you enter the CFD screen after completing and closing the new CFD analysis dialog. The grid is built by first determining key locations along the principal axes that are acquired from constituent model blocks, with the distance between each of these key points known as regions. Each major axis of each CFD grid region I. The figure xx illustrate the CFD grid .

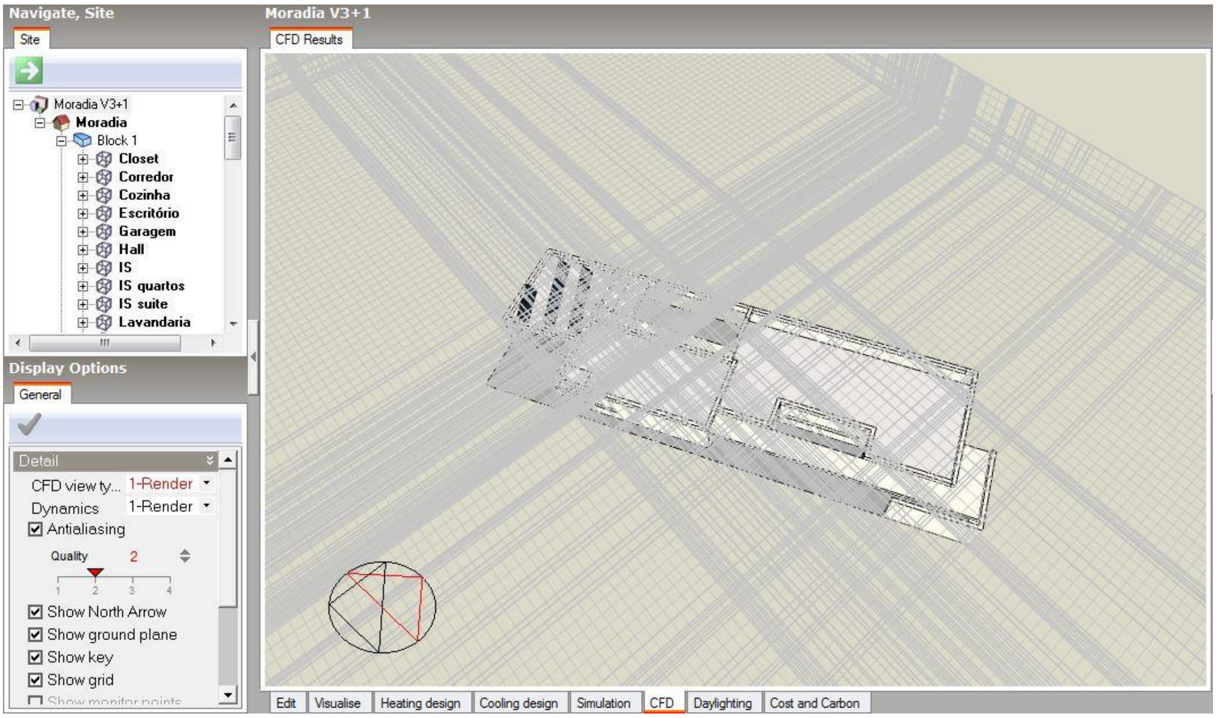

Figure4.9: illustrate the CFD grid

4.2.1.3 Grid line merge tolerance

When you reach the CFD screen after completing and closing the new CFD analysis dialog, the CFD grid is automatically built throughout the whole external domain extents. The grid is created by first establishing key locations along the primary axes, which are obtained from constituent model blocks, and then measuring the distance between each of these key points, which is known as regions. Each CFD g has a primary axis.

4.2.2 Wind

4.2.2.1 Wind velocity

Enter the required free stream wind velocity in m/s (measured at 10m above ground).

4.2.2.2 Wind direction

The wind direction is defined clockwise from North. The default direction is 270°, i.e., Westerly.

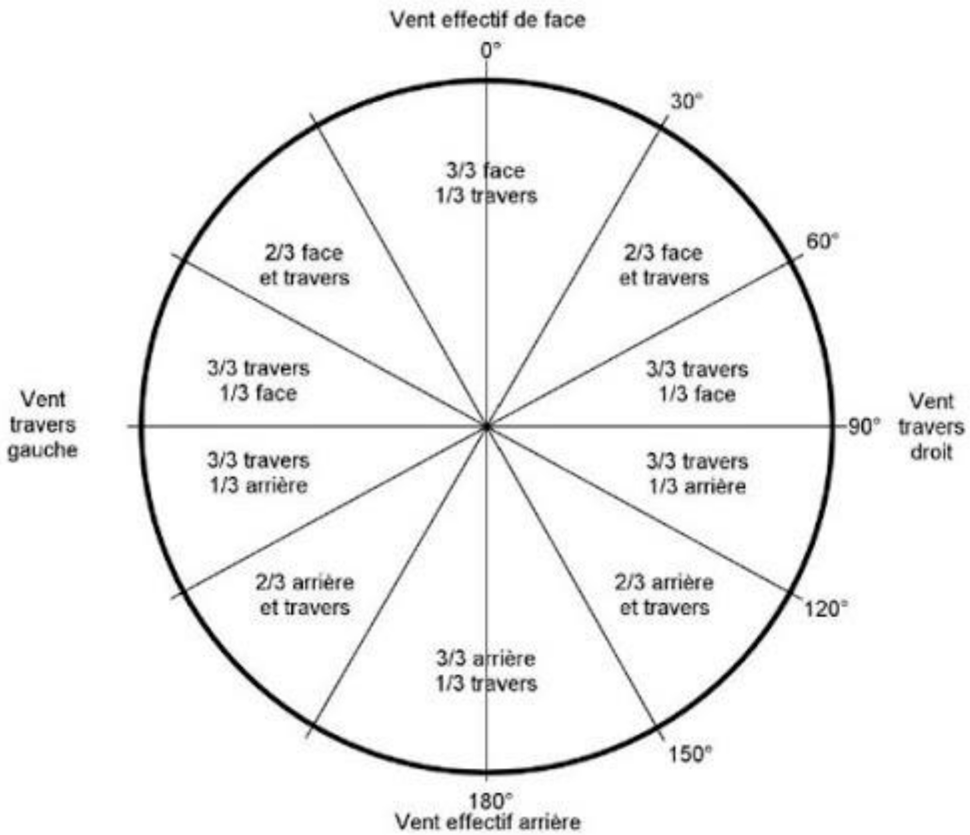

Figure4.3: The wind direction (clockwise from North)

4.2.2.3 Wind exposure

The free stream wind velocity is corrected for height above ground and surrounding terrain using an empirical relationship. The following exposures can be selected:

- Urban,
- Suburban and
- Open country.

Wind speed profile used by Design Builder (obtained from ASHRAE Fundamentals "Airflow around buildings")

$$U_{(h)} = U_{met} \left(\frac{\delta_{met}}{h_{met}}\right)^{a_{met}} \left(\frac{h}{\delta}\right)^{a}$$

Where:

U(h) = wind speed (m/s) at height h (m)

Umet = wind speed (m/s) measured at 10m above ground level

δmet = meteorological site air layer thickness (270m– 'Country' exposure)

amet = exponent for meteorological site (0.14 – 'Country' exposure)

hmet = measurement height for meteorological (10m)

h = height above ground level (m)

Table4.1:Air layer thickness and exponent values for various exposures

Exposure	**a**	**δ (m)**
Country	0.14	270
Urban	0.22	370
City	0.33	460

4.2.2.4 Site Domain Factors

The length, width, and height site domain factors are multipliers that are used to the total dimensions of the building model to produce an exterior volume or domain for the study. The default factors apply a factor of 3.0 to the length and width, and a factor of 2.0 to the model height.

The CFD screen appears after hitting the OK button:

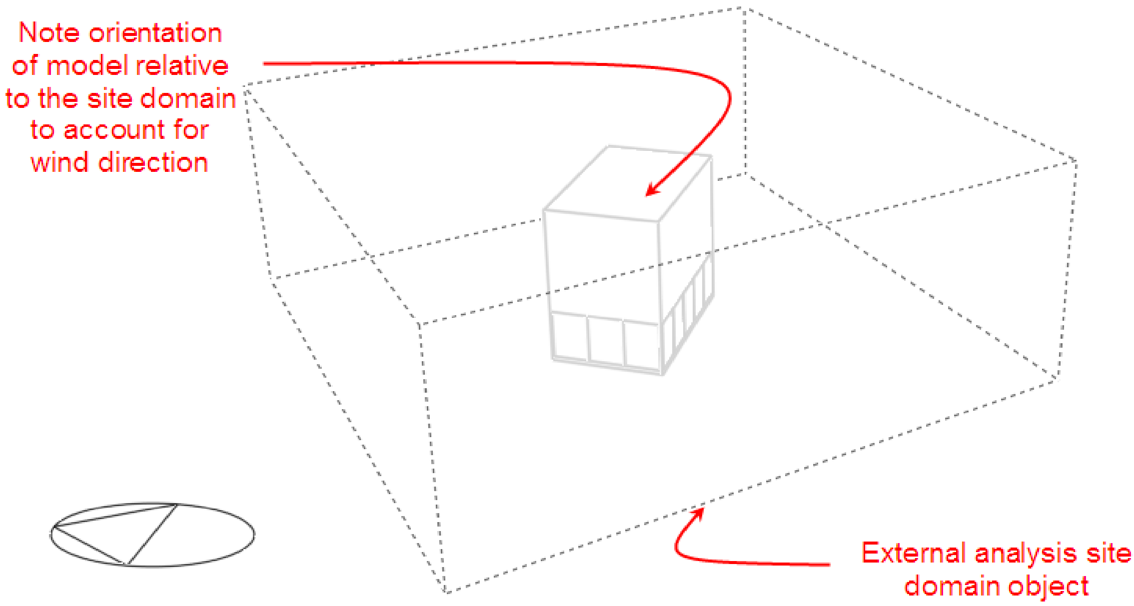

Figure4.4: Site Domain Factors

Note: **N**ow we can start the CFD calculation

4.2.3. CFD Calculation Options

If there are no errors when building the grid, the 'Edit Calculation Options' dialog will appear:

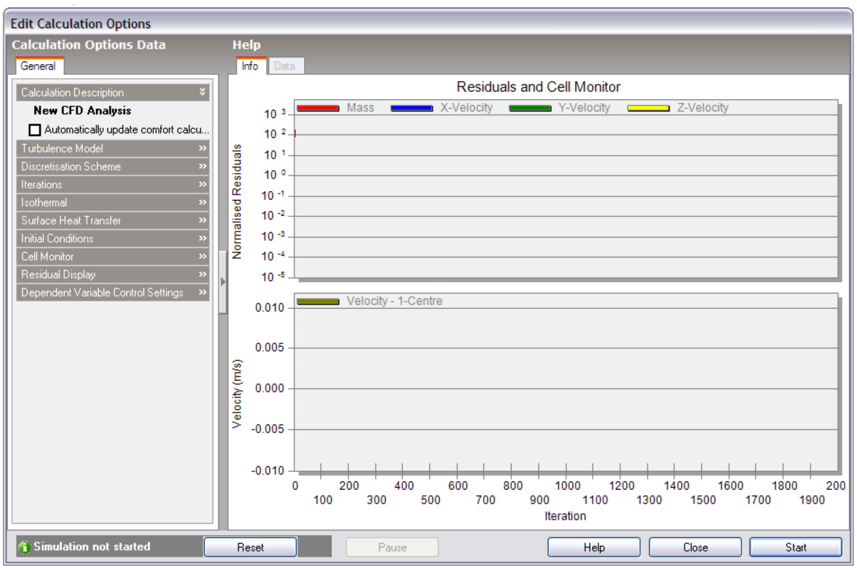

Figure4.5: CFD Calculation Options

The Edit Calculation Options dialog is divided into two main sections, the Residuals and Cell Monitor graphs and the Calculation Options Data panel.

4.2.3.1 Turbulence Model

- Constant effective viscosity: The constant effective viscosity model is a fairly simple approach that involves substituting a constant effective viscosity for the molecular viscosity in the Navier-Stokes equations (typically in the order of 100-1000). Despite the fact that this model is incapable of modeling local turbulence or turbulence transfer, it is computationally significantly less expensive.

- k-e: This model is one of the most widely used and tested of all turbulence models, belonging to the so-called RANS (Reynolds Averaged Navier-Stokes) family of models. These models involve replacing the instantaneous velocity in the Navier-Stokes and energy equations with a mean and fluctuating component. The resulting equations give rise to additional terms known as Reynolds stresses and turbulent heat flux components. Reynolds stresses are replaced with terms involving instantaneous velocities where molecular viscosities are substituted for effective viscosities and a similar substitution is conducted for the energy equation. The effective viscosity is the sum of the molecular viscosity and a turbulent viscosity, which is derived from the turbulence kinetic energy and the dissipation rate of turbulence kinetic energy:

4.2.3.2 Discretization scheme

-Upwind: The calculation process involves replacing the defining set of partial differential equations with a set of finite difference equations. The conventional approach to this is to use a Taylor's series formulation, which leads to a set of central difference equations. However, although this approach is physically realistic for diffusion, it is not found to be realistic for convection because of its one-way nature, i.e. upwind conditions affect downwind conditions but not the reverse. The upwind scheme allows the convective term to be calculated assuming that the value of the dependent variable at a cell interface is equal to the value at the cell on the upwind side of the interface.

–Hybrid: a more computationally expensive approach than the upwind scheme but it reduces numerical diffusion at high values of Pe number.

–Power-Law: The power-law scheme is arguably more accurate than the hybrid scheme but is more computationally expensive.

4.2.3.3 Iterations

This is the maximum number of iterations that the outer iterative calculation loop can perform (see the section on CFD Calculations and Convergence). Regardless of whether or not the solution has converged, the calculations will end when the number of iterations reaches this value.

4.2.3.4 Initial conditions

In some circumstances, setting the start conditions closer to the ultimate expected conditions can result in a faster solution. Enter the start velocities in the x, y, and z axes, as well as the domain's initial temperature.

4.2.3.5 Cell monitor

Select any defined cell monitor point and associated dependent variable to be displayed on the cell monitor (see Setting Up CFD Cell Monitor Points section).

4.2.3.6 Residual Display

Residuals give an indication of the extent to which the calculations have converged to the final solution. They are a measure of the total 'error' still left on the solution. You can select the dependent variables and/or mass for which residuals are to be displayed in the residuals monitor. The mass residual is like the dependent variable residuals but is extracted from a continuity equation mass balance for each cell and is the most significant residual in terms of indicating a successfully converged solution.

The residuals have no units and are normalized by scaling to enable similar termination residual magnitudes for each variable. See also the Termination residual section below.

It is also important to use monitor points (above) in conjunction with termination residuals because they can provide a more meaningful indication of convergence (i.e., when the values of the dependent variables at the monitor points level out).

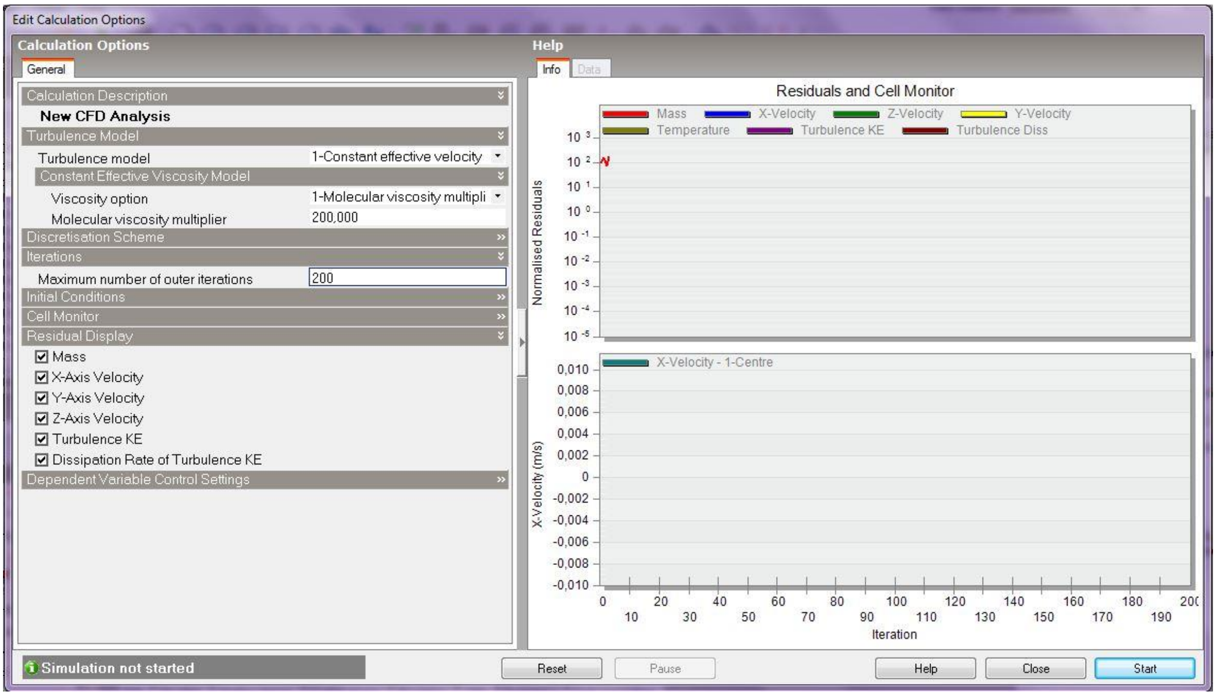

Figure 4.6: New CFD Calculation

4.3. The results of the Design Builder software

4.3.1 Definition of Design Builder CFD

Conventional CFD software requires expertise and extreme attention to detail to set up the correct geometry and boundary conditions. Design Builder streamlines the process by automatically creating precise 3D geometry with options for CFD boundary conditions such as surface temperatures, heat or air flows that can come from EnergyPlus.

4.3.2 Obtained results for wind velocity simulations

The parametric analysis in the practical part first of all I made the choice of 3 maximum values of wind speed the north west which equals 10 m / s, north is equal 8 m / h and the south 7 m / s.

4.3.2.1 First simulation (10m/s)

in the first simulation one chose in the model of turbulence the constant effective viscosity and numerically much more stable. And we chose up-wind for Discretization scheme because The upstream scheme allows the convective term to be calculated by assuming that the value of the dependent variable at a cell interface is equal to the value at the cell on the upstream side of the interface.

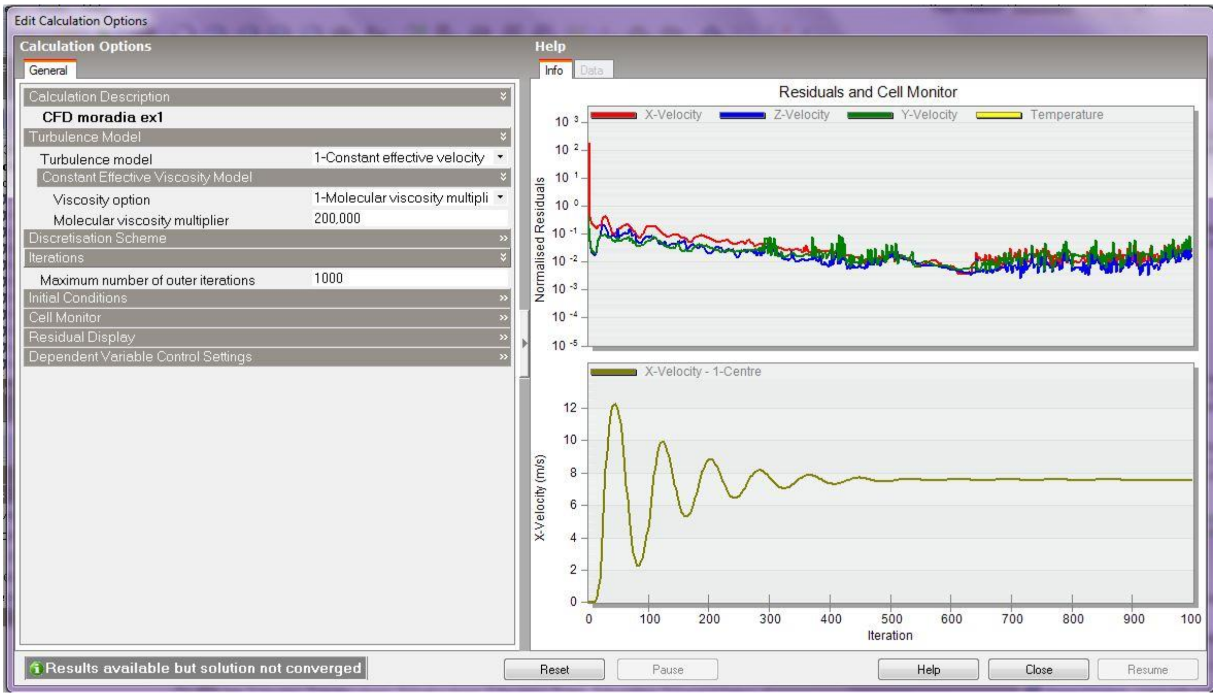

Figure4.7: 1st residuals and cell monitor for 10 m/s

The first result found by the CFD simulation is to estimate on two diagrams. the first diagram is the residuals and the cells it is divided on several velocity curves for the 3 dimensions (x, y, z) we notice on the first diagram that the velocity does not reach 10^{-4}

and on the second diagram, we find that the speed curve on x is a damped sine curve because when the iteration reaches 500 the stroke will be stable.

4.3.2.2 Second simulation (10m/s)

In the second simulation we chose the k-e turbulence model because This model consists in replacing the instantaneous speed in the Navier-Stokes and energy equations by an average and fluctuating component and we chose the Hybrid for the diagram discretization because it reduces digital scattering to high values of the number Pe.

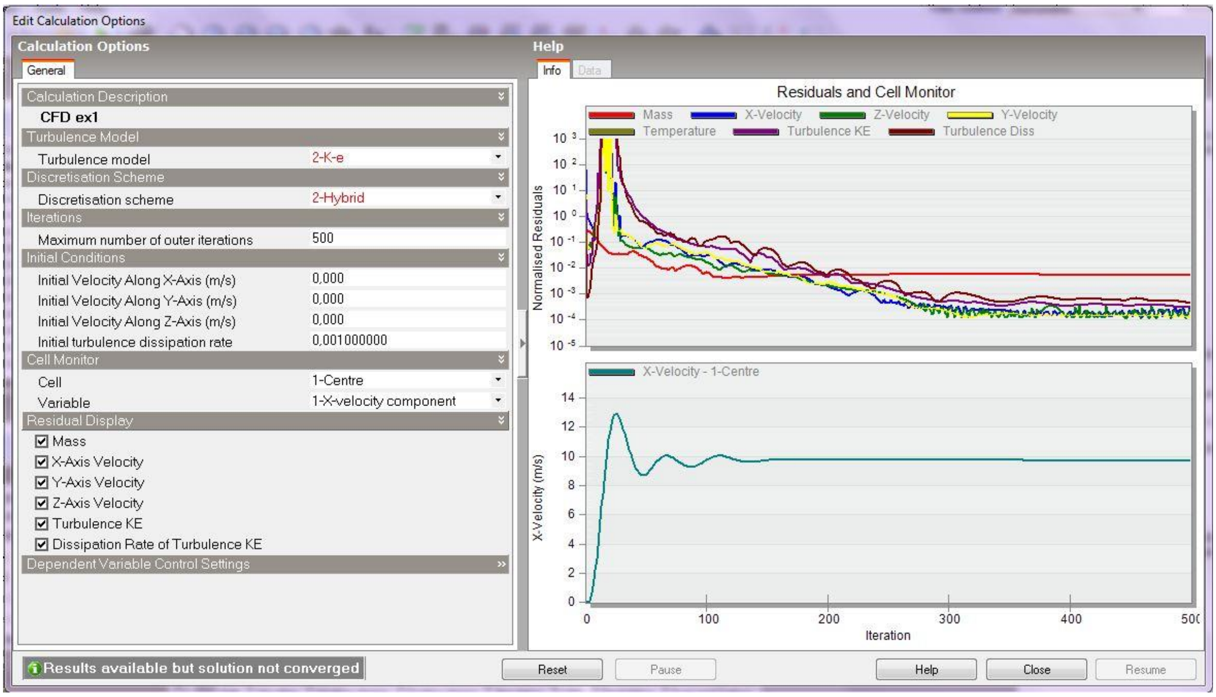

Figure4.8: 2nd residuals and cell monitor for 10 m/s

We determine the value of φ at the face by linear interpolation between the cell centered values.

This is more accurate than the first order upwind scheme, but it leads to oscillations in the solution or divergence if the local Peclet number is larger than 2. The Peclet number is the ratio between convective and diffusive transport:

It is common to then switch to first order upwind in cells where Pe>2. Such an approach is called a hybrid scheme.

We notice that on the second simulation that the velocity reaches the 10-4

We are going to make a 3rd simulation with the k-e turbulence model and with the up wind for the discretization diagram.

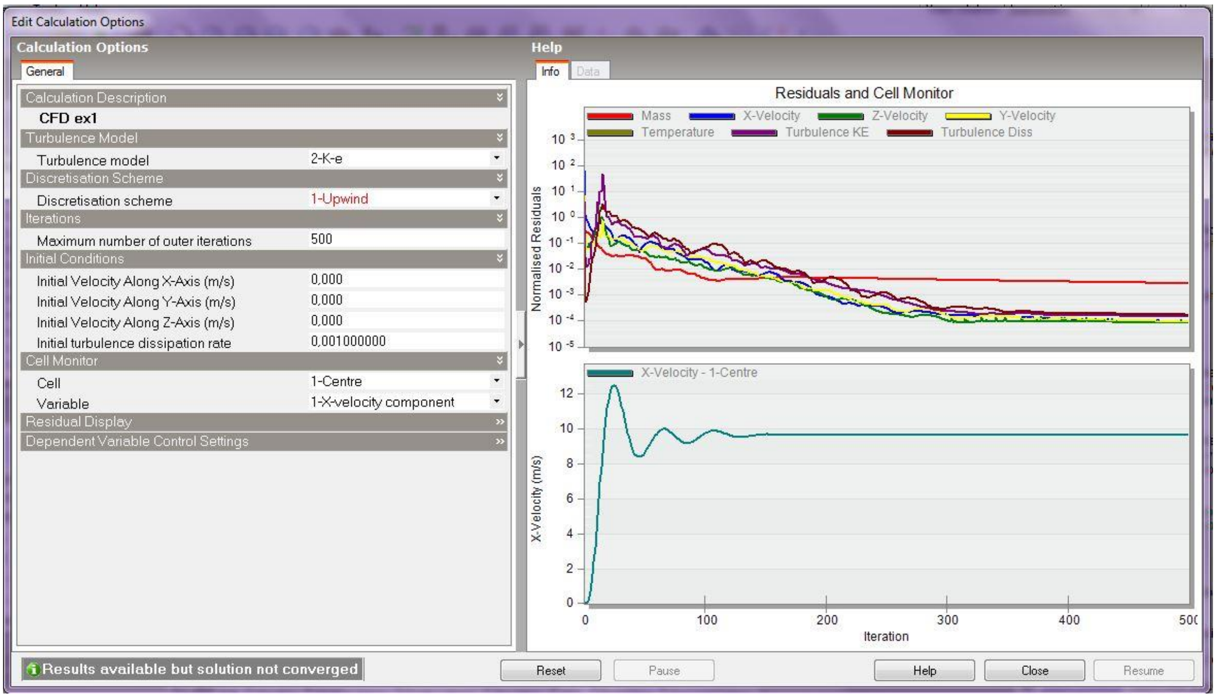

Figure4.9: 3rd residuals and cell monitor for 10 m/s

This is the simplest numeral scheme. It's the method we used in the discretization example before.

The value of at the face is assumed to be the same as the cell centered value in the cell upstream of the face.

The main advantages are that it is easy to implement and that it results in very stable calculations, but it also very diffusive. Gradients in the flow field tend to be smeared out, as we will show later.

This is often the best scheme to start calculations with.

We notice that with up wind in the discretization schema we reach the 10-4 are good results and the same results so we will complete the other simulation with the model of turbulence k-e and discretization schema up wind, because we don't have a very big problem with the wind speed value and the upwind scheme is a more economic approach in terms of calculation than the hybrid scheme.

We move now on to the influence of velocity on the building from these results we can see that there is a large velocity on the north corner of the building which equals 9.63 m / s .

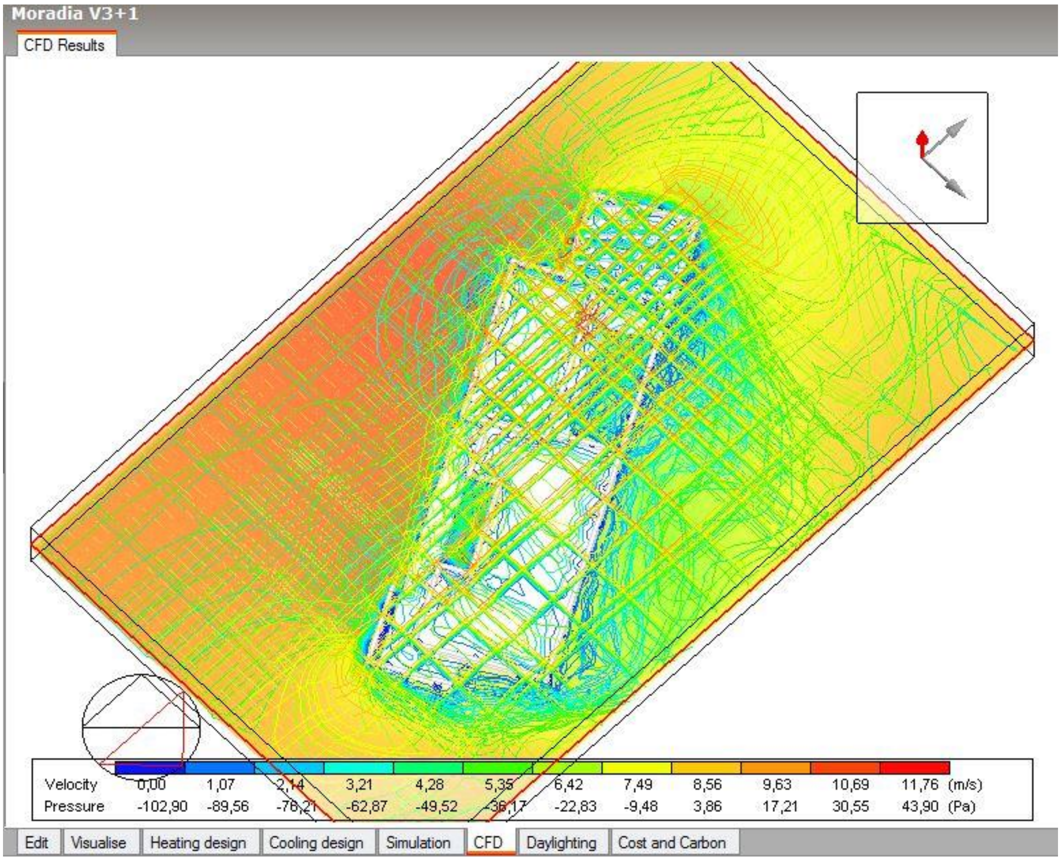

Figure4.10: The wind velocity

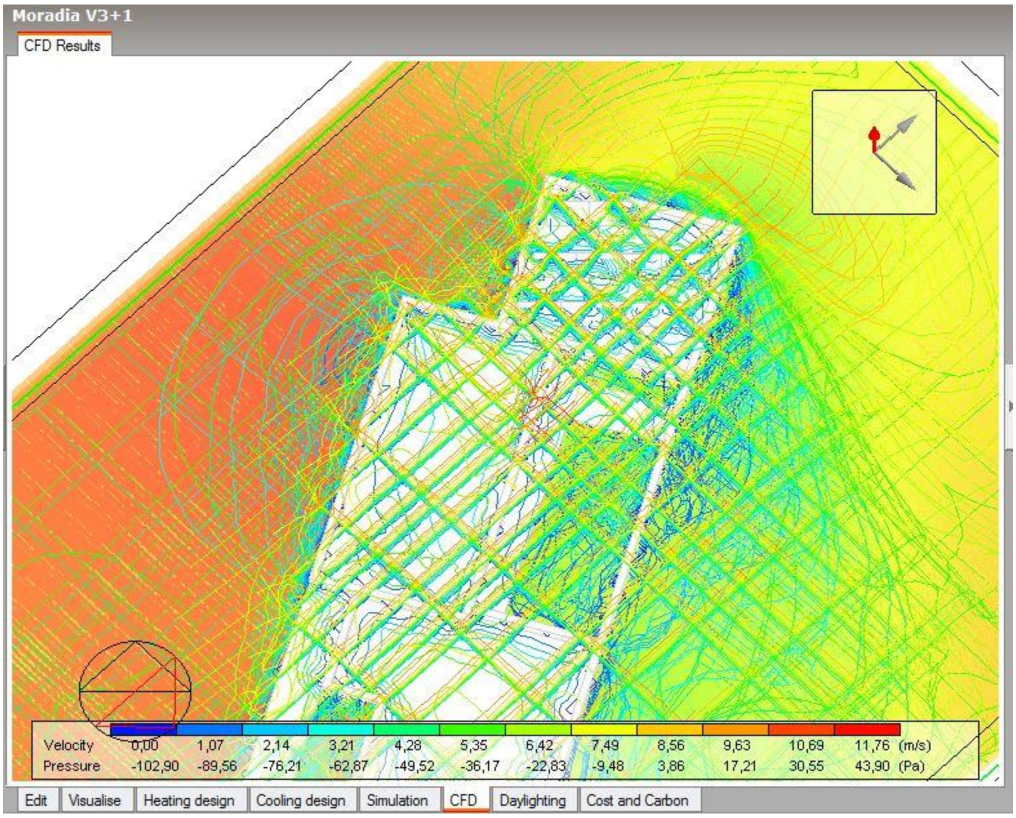

Figure 4.11: the wind velocity in the north corner

In the figures 5.9 and 5.10 we notice that the part which contains more pressure are the northwest dimension of value 30.55 pa and s the north of value -36.17 Pa this value indicates that there is a tourbillion in this dimension. on the other hand, on the southeast side there is a great compression of value - 22.83 pa

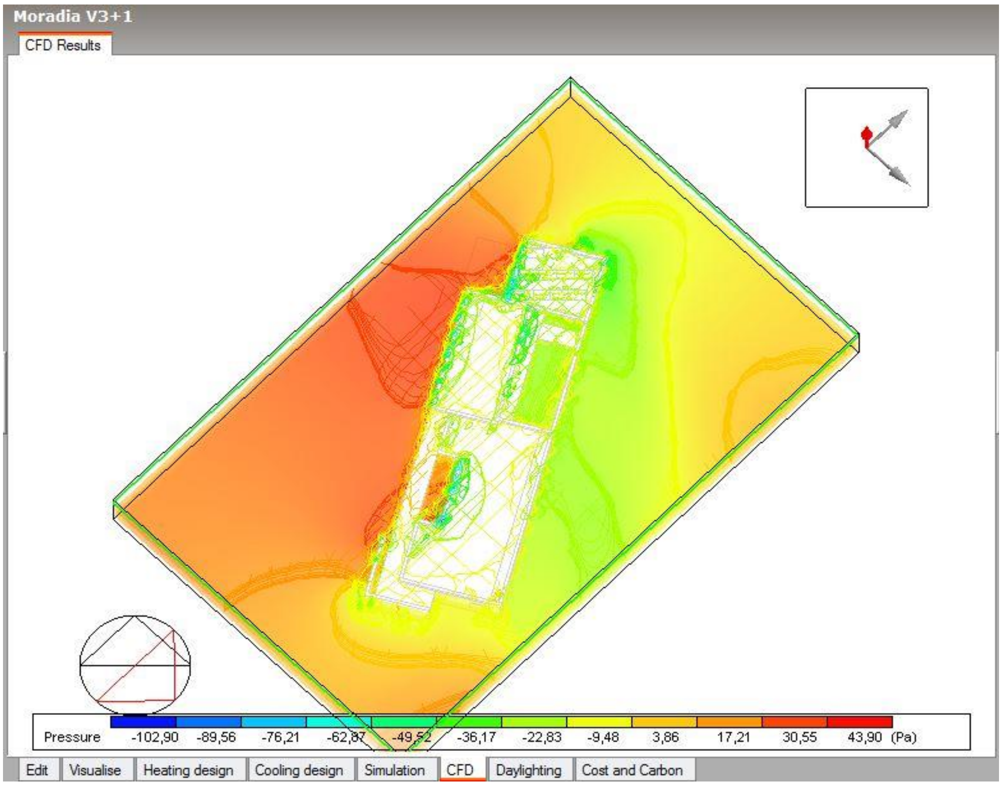

Figure 4.12: The wind pressure

The influence of the vortex influence on the building imply a significant compression on the slab which is equal -36.17Pa

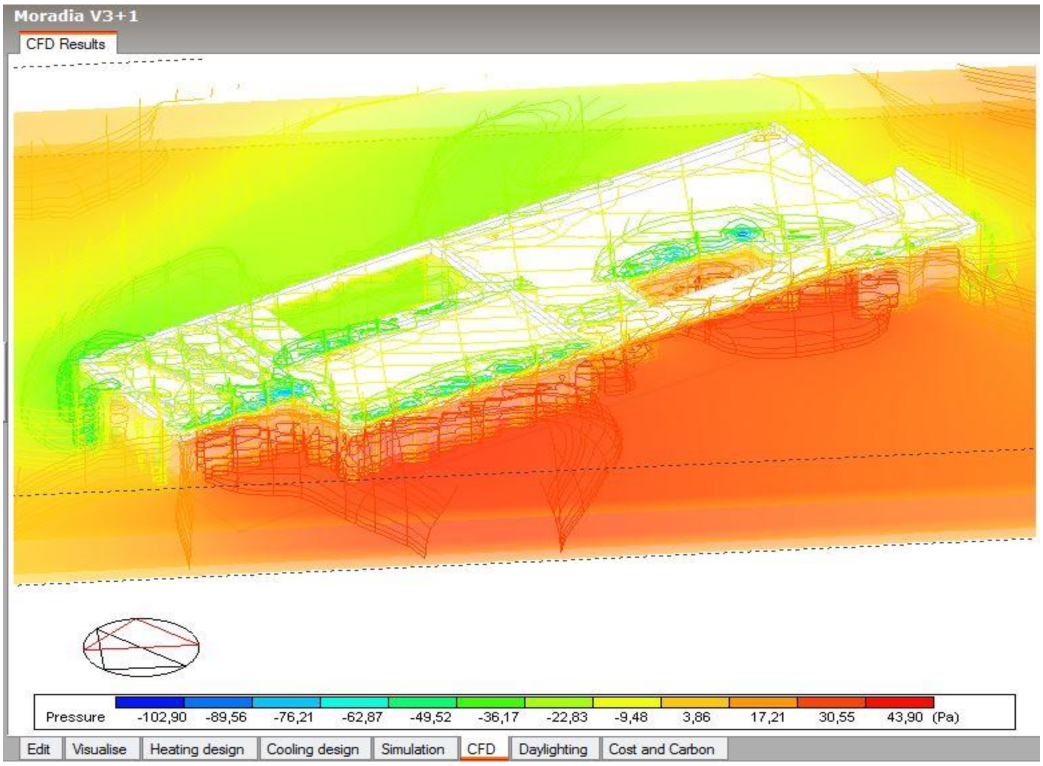

Figure 4.13: The wind pressure in the north corne

4.3.2.3 Second simulation (8 m/s)

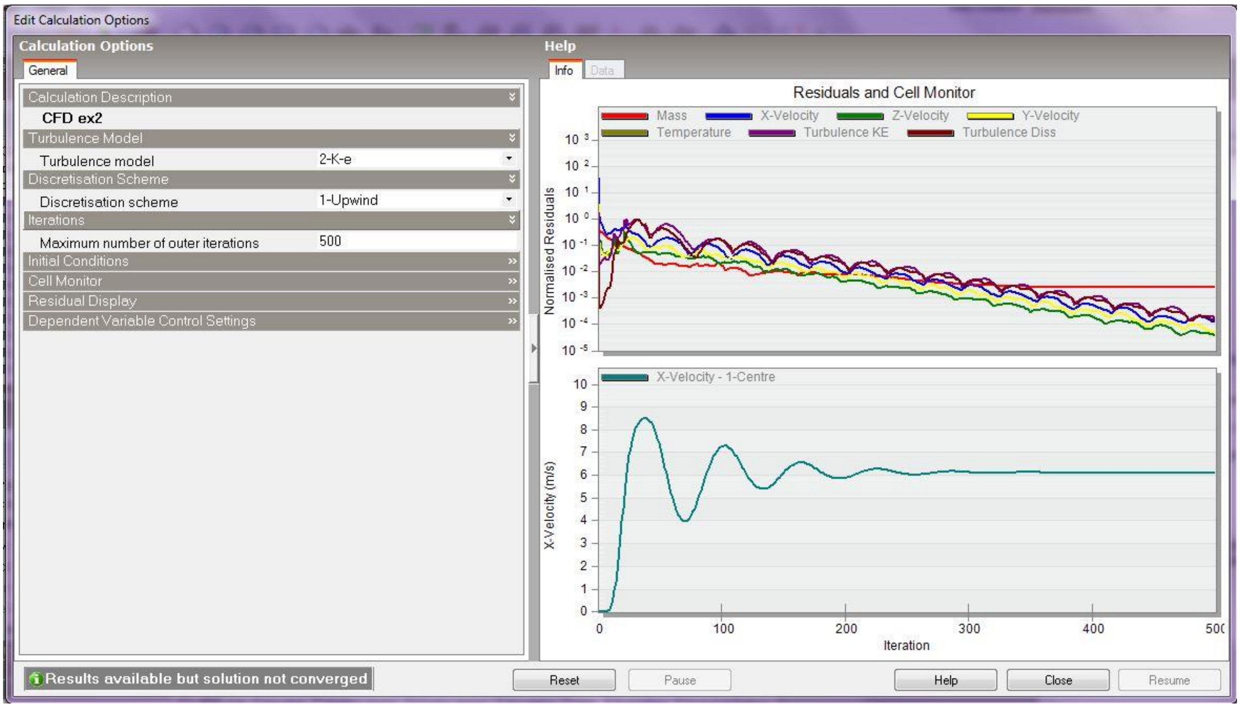

Figure 4.14: residuals and cell monitor for 8m/s

We move now on to the influence of velocity on the building from these results we can see that there is a large velocity on the north corner of the building which equals 9.8m / s .

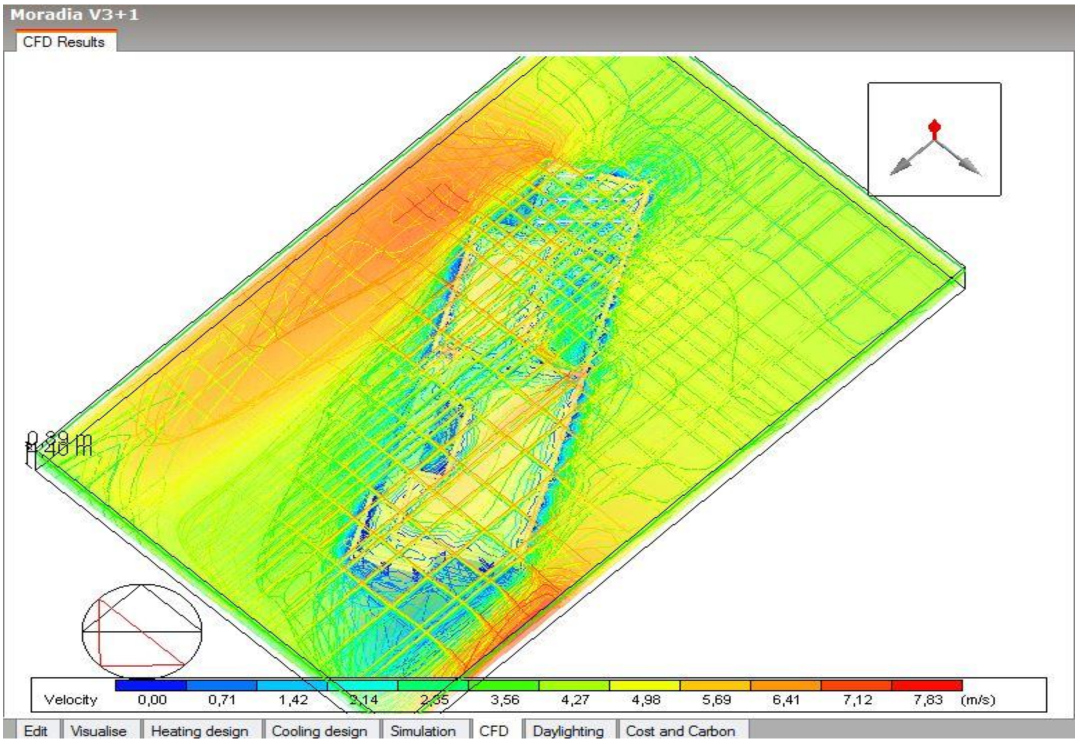

Figure 4.15: The wind velocity

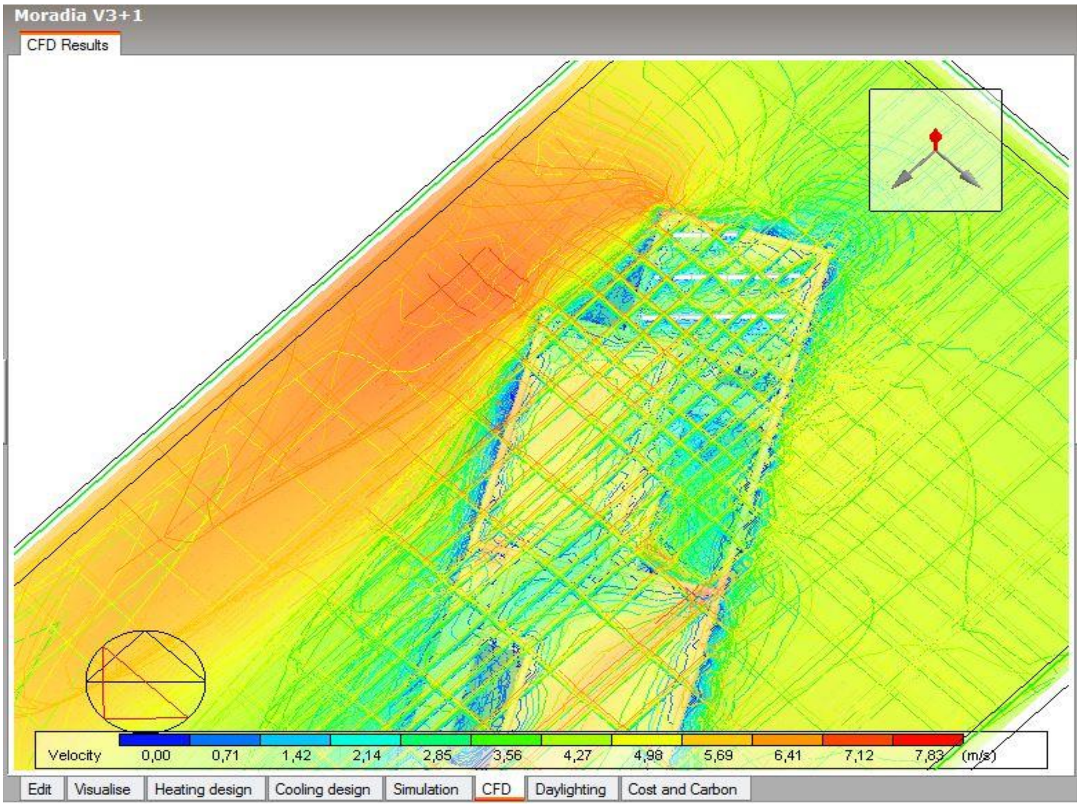

Figure 4.16: The wind velocity in the nord corner

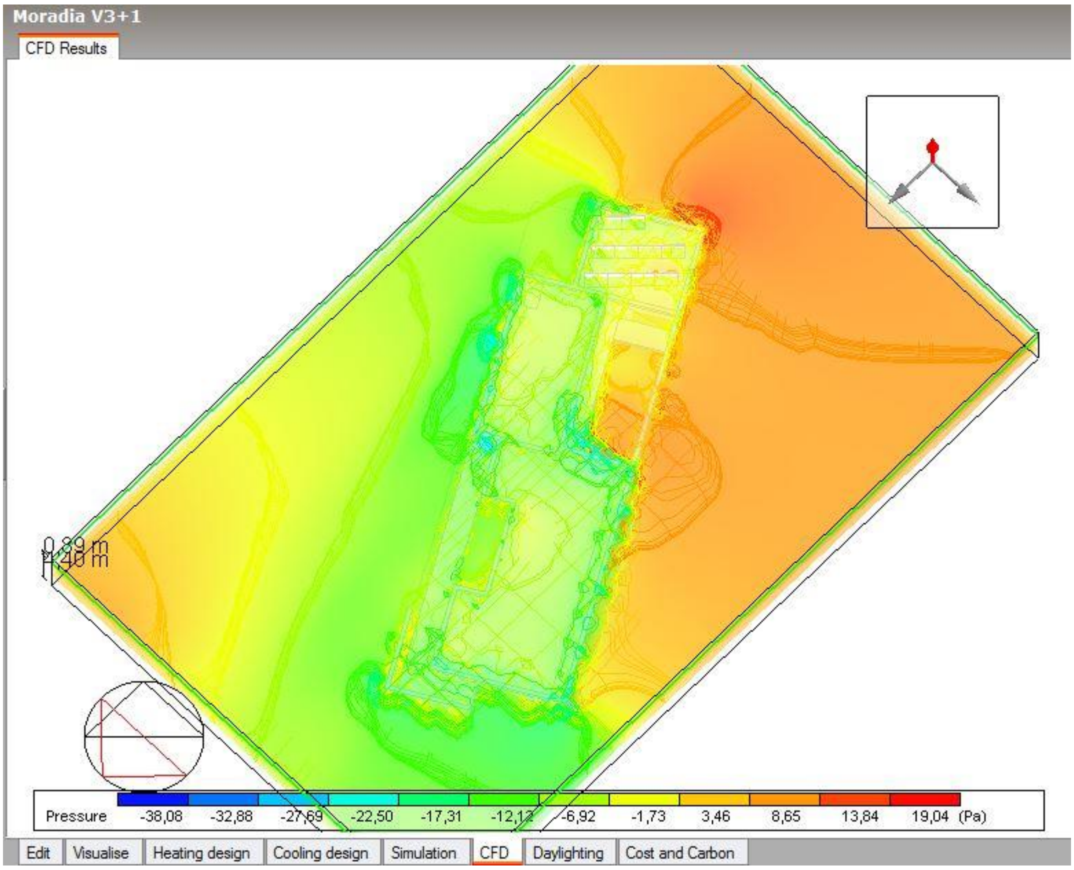

Figure 4.17: The wind pressure

It can be seen that the influence of the vortices on the building implies a low compression of the northwest side with a value equal to -22.50 Pa and an average pressure on the north east side equal to 19.04 Pa.

4.3.2.4 Third simulation (7 m/s)

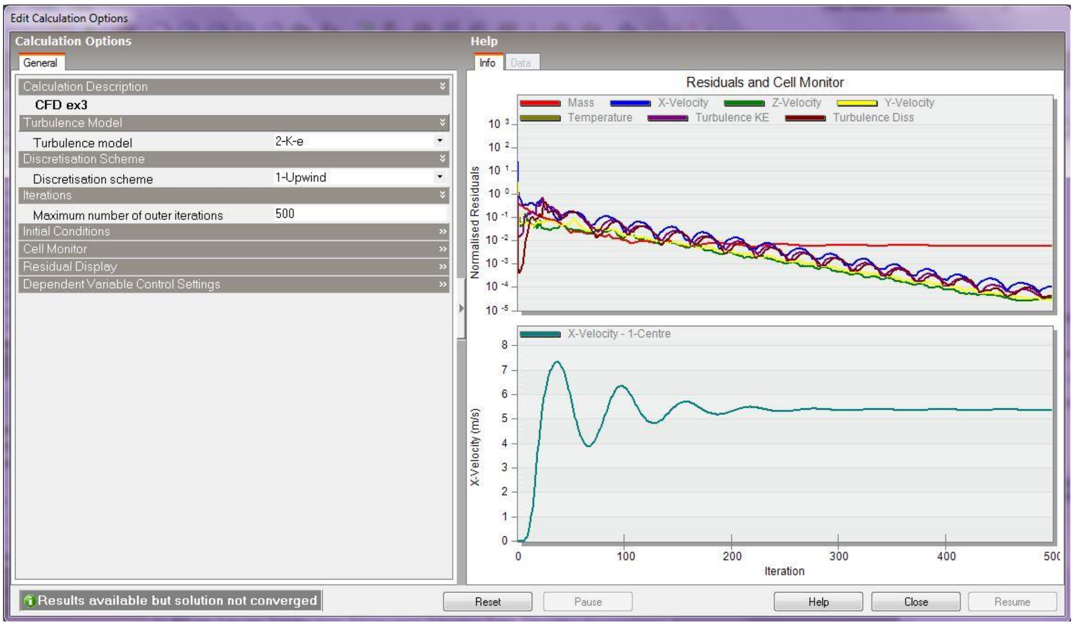

Figure 4.18: residuals and cell monitor for 7m/s

We move now on to the influence of velocity on the building from these results we can see that there is a large velocity on the north corner of the building which equals 9.8m / s .

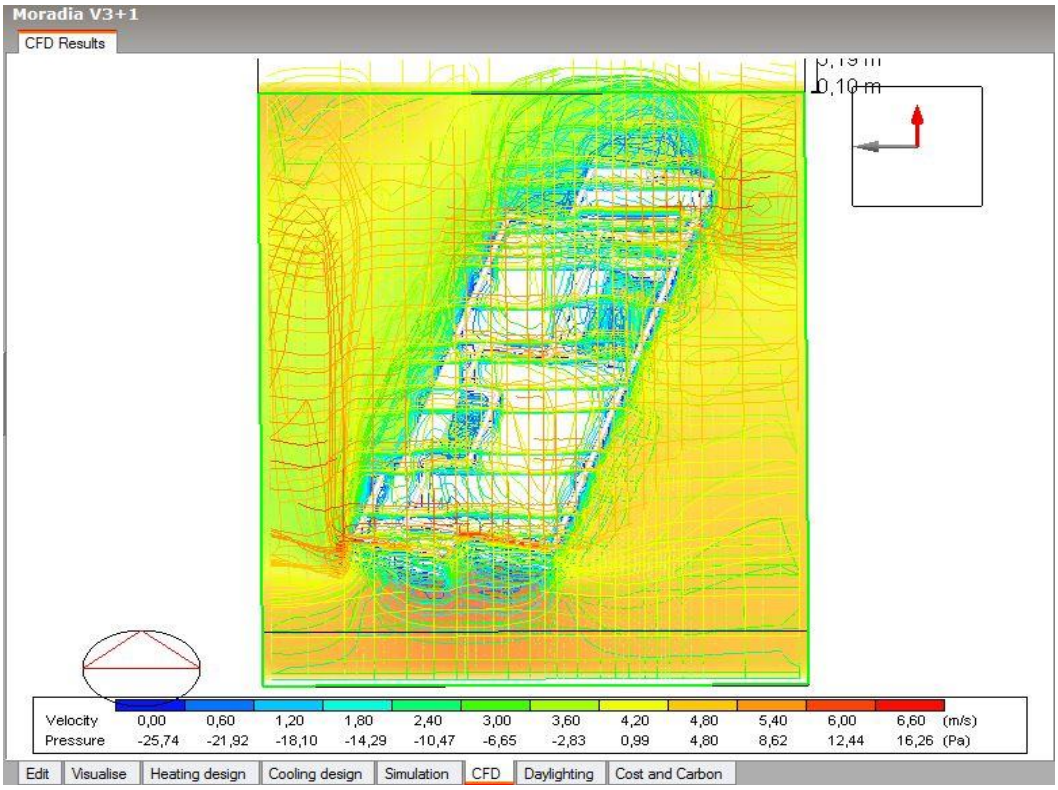

Figure 4.19: The wind velocity

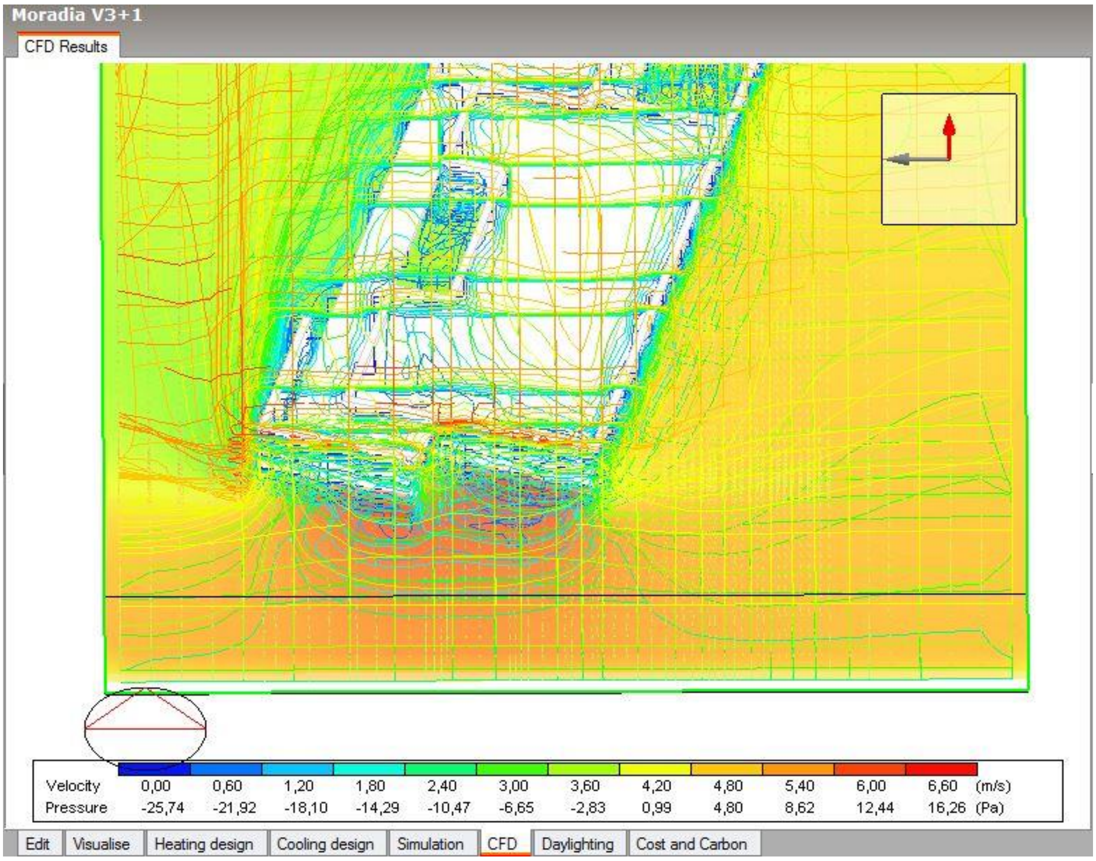

Figure 4.20: The wind velocity in the south

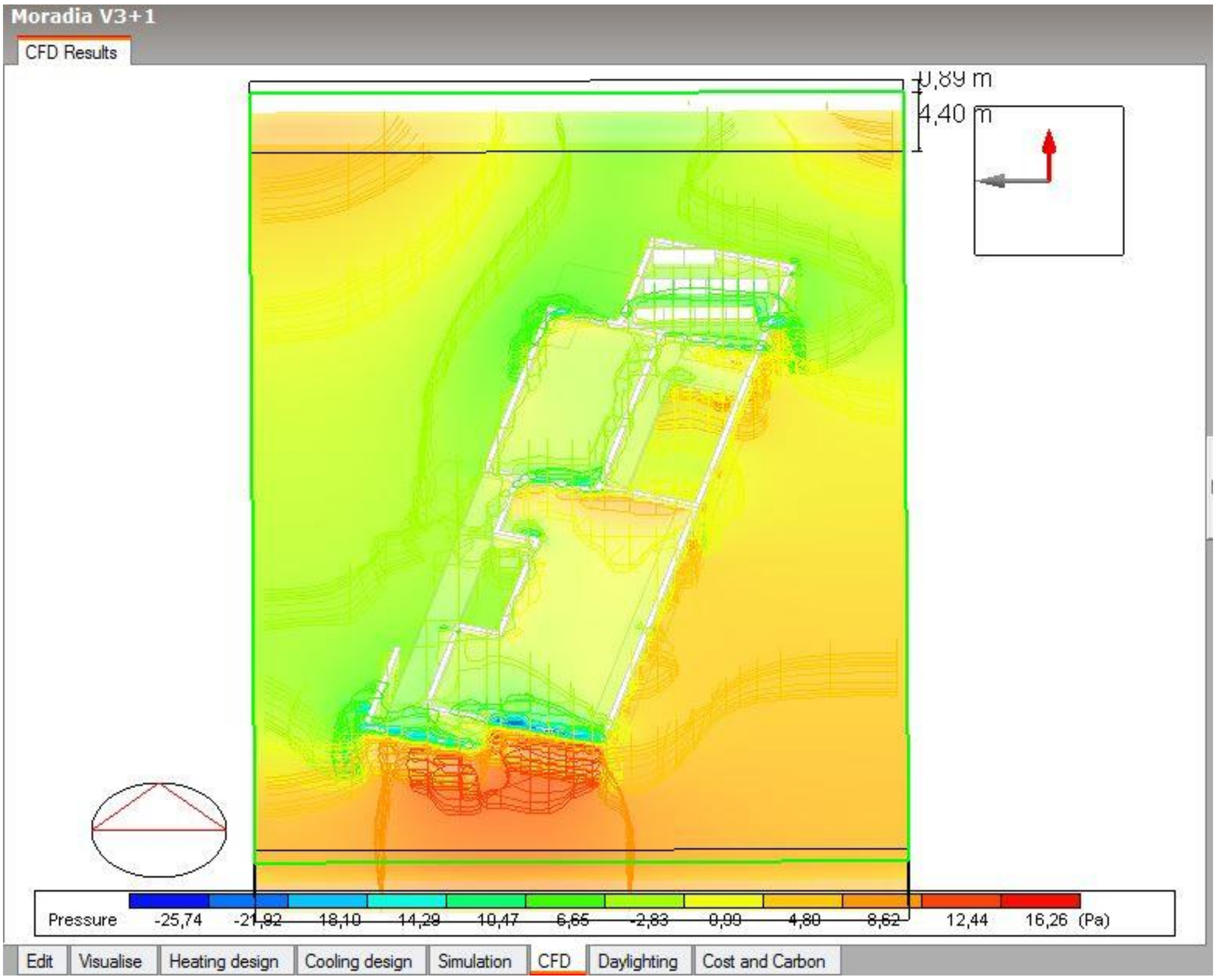

Figure 4.21: The wind pressure

4.3.2.5 Third simulation (30 m/s)

We notice in figure 4.6 of wind rose that we have 3 maximum, average and minimum wind speeds. In our work we do the simulations with the average speed, and we will do an example with the maximum speed given by the wind rose which is equal to 45 m/s, and its value is very high so we will choose the maximum value given by the Eurocode 4 [15] which equals 30m / s

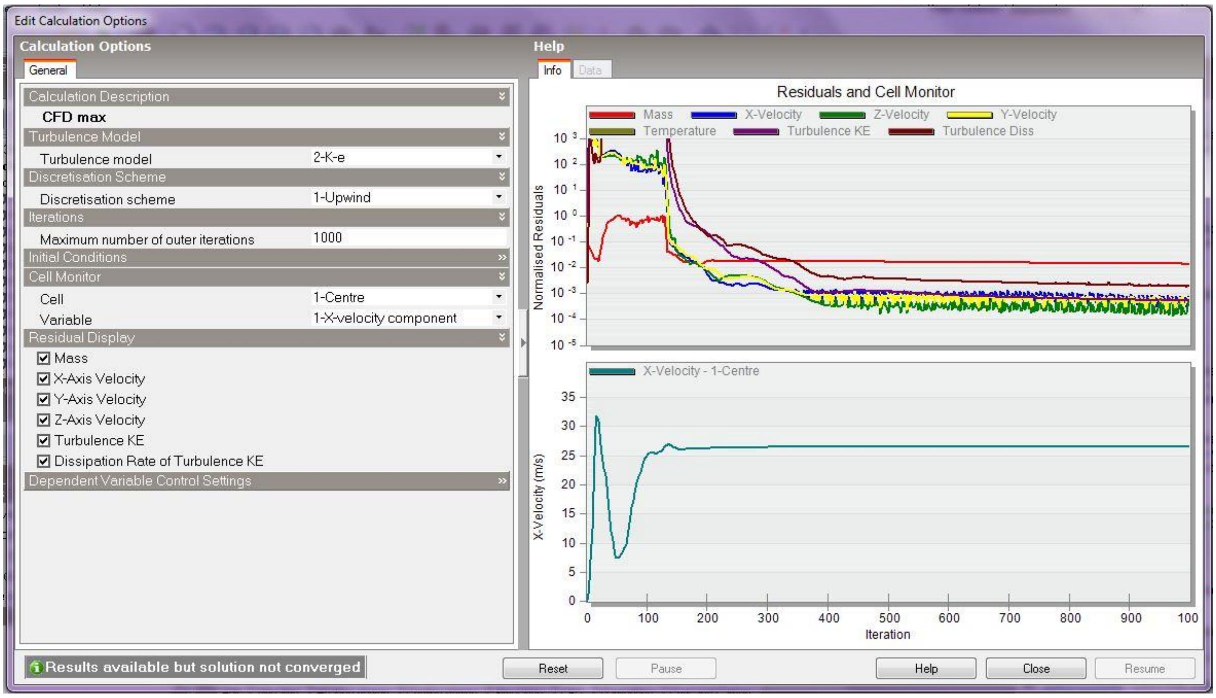

Figure 4.22: residuals and cell monitor for 30m/s

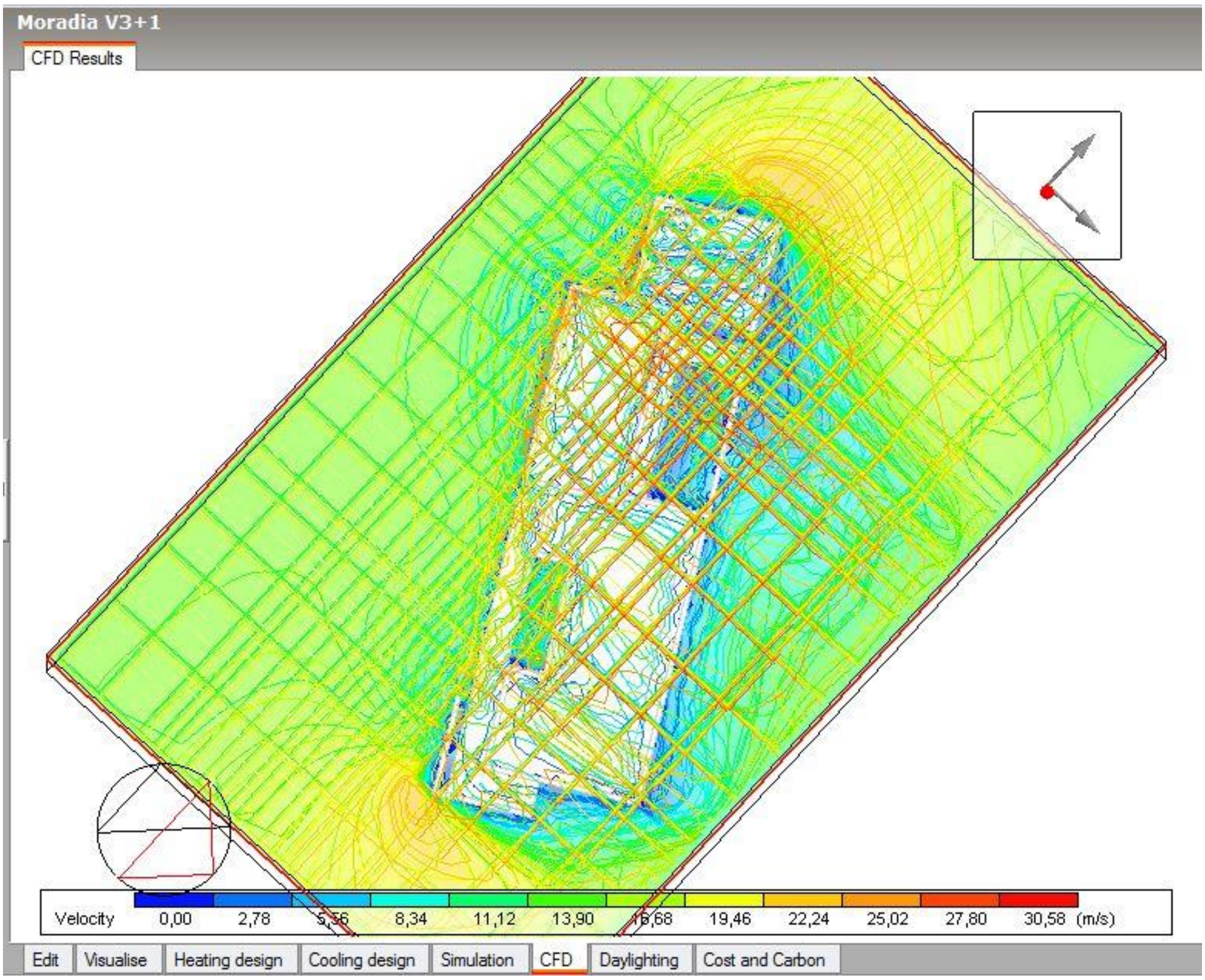

Figure 4.23: The wind velocity

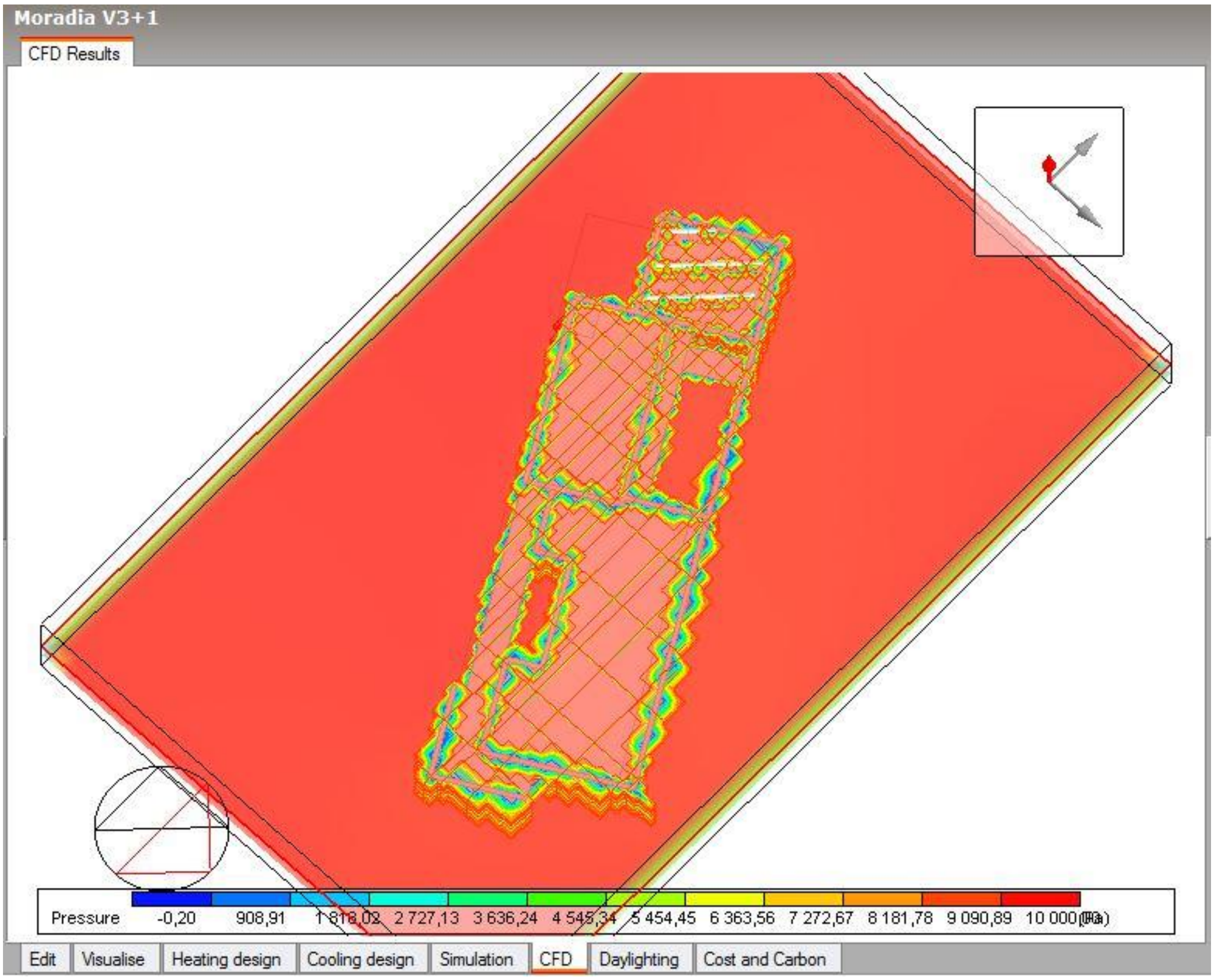

Figure 4.24: The wind pressure

Chapter 5:

Conclusion

Fluid mechanics is a field of study that seeks to determine the behavior of a fluid in flow condition when it encounters a solid body, determination a series of properties for both fluid and body, including the drag coefficient of the body and the drag force imposed

by the fluid on it.

In the present work, the determination of the drag force and the drag coefficient of a

north side - equals 5.9m / s and in the south

Table5.1:The results of the drag force

	Drag force simulation(10m/s)	**Drag force simulation(8m/s)**	**Drag force simulation(7m/s)**
North	-36.17 Pa	-17.31 Pa	-10.47 Pa
South	-9.48 Pa	-12.12 Pa	16.26 Pa
East	-22.83 Pa	8.65 Pa	-2.83 Pa
west	30.55 Pa	-17.31 Pa	-10.47 Pa
North-East	-49.52 Pa	19.04 Pa	-10.47 Pa
North-west	43.9 Pa	-17.31 Pa	-6.65
South-East	-22.83 Pa	-22.50 Pa	0.00 Pa
South-west	-9.48 Pa	-22.50 Pa	-6.65 Pa
“The big value in roof “	-62.87 Pa	-27.69 Pa	-10.47 Pa

single-family house of first floor in light steel, then the object of numerical tests in a wind action of type subsonic suction CFD analyses using the two software clima consultant and design building

The CFD simulations presented an easier execution without any interference. The objective of this work was the creation of a CFD analysis model that can be used during the design process of a component.

The objective of this work was to create a CFD analysis model that can be used during the design process of a component and to do so in the simplest way, the transition effects were not taken into consideration. Considering the results obtained in the simulation of wind action as standard values, the which is not acceptable for the purpose of the project. The tool can be used as a support tool to be used during the design to assist in non-critical parts of the project. A better determination can be made if the same simulations were done using another material in the model that has better mechanical properties that will produce a better approximation of the results obtained in the CFD analysis, probably to that performed in the in the design building software. To improve the results obtained with CFD analysis, transitions effects can be taken into account.

5.Bibliografy

[1] VILLALVA, M. G. Energia Solar Fotovoltaica: Conceitos e Aplicações. 2 ed. São Paulo: Editora Saraiva, 2015

[2] WORLD ENERGY RESOURCES: 2016. World Energy Council. Avaliable at: < https://www.worldenergy.org/assets/images/imported/2016/10/ World-EnergyResources-Full-report-2016.10.03.pdf >. Access in: 20 out. 2019.

[3] Gaiato, J. M. C. Projeto de Colunas de Iluminação Autonomas Solares. 2017. 155p. Project Course Report. Instituto Politécnico de Bragança, Bragança, 2017.

[4] GAZELA: POSTE D'ILUMINAÇÃO AUTÓNOMO FOTOVOLTAICO VALLED. Avaliable at:Access in: 20 out. 2019.

[5] Stuart A.A. (1928). It's here - All steel buildings, Popular Science Monthly, November, pp. 33-34. (Disponível em Google Books e http://en.wikipedia.org/wiki/Steel_frame).
[6] Department of the Navy, Bureau of Yards and Docks (1947). Bases in South America and the Caribbean Area, Including Bermuda, Building the Navy's Bases in World War II: History of the Bureau of Yards and Docks and the Civil Engineer Corps, 1940-1946, Part III Advanced Bases, Capítulo 18, pp. 1-46. (http://www.ibiblio.org/hyperwar/USN/Building_Bases/bases-18.html).
[7] History of light steel framing in steel homes and metal homes, Stud Chopper Metal Homes .(http://www.studchopper.com/steel_homes.html).

[8] Batisim – design builder 2018. (https://www.batisim.net/designbuilder.html)

[9](https://googleearth.gosur.com/?gclid=CjwKCAjwqcKFBhAhEiwAfEr7zeiGgRFq5HHVOhau2Evz0tE4Xw_Vb4LdI_vpCTA7XgMLCzKJX5ruYRoCYNEQAvD_BwE&ll=41.8026568398777,-6.736017696227009&z=11.570194261974924&t=satellite)

[10] Arnoldi, P. H. C., Estudo de Parâmetros Aerodinâmicos de Perfis em Túnel de Vento. 2011. 49p. Final Report. Universidade Regional do Noroeste do Estado do Rio Grande do Sul, Panambi, 2011.
[11] Sarmento, C. V. da S.; Souza Junior, M. A.; Ribeiro, P. M. V. Estimativa do Coeficiente de Arrasto com a Aplicação da Dinâmica dos Fluidos Computacionais: Estudo de Caso de um Aqueduto da Transposição do Rio São Francisco. 2014. 21p. Proceedings of the XXXV Iberian Latin-American Congress on Computational Methods in Engineering. CILAMCE, 2014.
[12] Fox, R. W.; Mcdonald, A. T.; Pritchard, P. J. Introdução à Mecânica dos Fluidos: 8 ed. Gen/LTC, 2014.
[13] Çengel, Y. A.; Cimbala, J. M. Mecânica dos Fluidos: Fundamentos e Aplicações. Porto Alegre: AMGH, 2007.
[14] AIR - DENSITY, SPECIFIC WEIGHT AND THERMAL EXPANSION COEFFICIENT AT VARYING TEMPERATURE AND CONSTANT PRESSURES. Avaliable at: < https://www.engineeringtoolbox.com/air-density-specific-weightd_600.html >. Access in: 20 out. 2019.

[15] NP EN 1991-1-4:2009. Eurocódigo 1 – Ações em estruturas. Parte 1-4: Ações gerais. Ações do vento. Insituto Português de Qualidade, 2009.

[16] Maia, V. M. J. M.. Análise e Dimensionamento de Torre Eólica Offshore: Estudo paramétrico. 2009. 135p. Masters Dissertation. Faculdade de Engenharia da Universidade do Porto, Porto, 2009.

[17] Escusa, G. G. Análise do Comportamento Dinâmico da Estrutura de Suporte de um Túnel de Vento. 2014. 137p. Masters Dissertation. Faculdade de Engenharia da Universidade do Porto, Porto, 2014.

[18] Barlow, J. B.; Rae, W. H.; Pope, A. Low-speed Wind Tunnel Testing: 3 ed. Wiley, 1999

[19] Antón, P. S. Wind Tunnel and Propulsion Test Facilities: An Assessment of NASA's Capatibilities to Serve National Needs. Rand, 2004.

[20] OPEN RETURN WIND TUNNEL. Avaliable at: < https://www.grc.nasa.gov/WWW/k-12/airplane/tunoret.html>. Access in: 20 out. 2019.

[21] https://www.batisim.net/designbuilder.html eful for comfort studies, air distribution optimization, high volume stratification issues.