Università La Sapienza - Universitat Politècnica de València

Degree Thesis

Aerodynamic investigations on temperature distribution around buildings using CFD

Author:
Eva Peinado Montoya

Supervisor:
Giovanni Paolo Romano

A thesis submitted in fulfillment of the requirements for the degree of Aerospace Engineering

in the

Dipartimento di INGEGNERIA MECCANICA E AEROSPAZIALE

July 15, 2020

UNIVERSITÀ LA SAPIENZA - UNIVERSITAT POLITÈCNICA DE VALÈNCIA

Abstract

Università La Sapienza

Dipartimento di INGEGNERIA MECCANICA E AEROSPAZIALE

Aerospace Engineering

Aerodynamic investigations on temperature distribution around buildings using CFD

by Eva Peinado Montoya

The objective of this thesis is to analyse the heat transfer through a building that is exposed to a stream velocity, with a finite element method software, ANSYS Fluent. The building will be changed in shape, in order to see how the total thermal efficiency is affected, as well as different configurations of several buildings will be analysed.

El objetivo de este trabajo final de grado es analizar el flujo de calor a través de un edificio expuesto a una corriente de aire, con el software ANSYS Fluent basado en el método de elementos finitos. El edificio estará sujeto a diferentes cambios para estudiar la eficiencia térmica, al igual que también se analizarán diferentes configuraciones de grupos de edificios.

L'objectiu d'aquest treball de final de grau és analitzar la transferència de calor a través d'un edifici exposat a una corrent d'aire, amb el software ANSYS Fluent basat en el mètode d'elements finits. L'edifici serà cambiat per poder estudiar la eficiència tèrmica, així com s'estudiaran també configuracions de grups d'edificis.

Acknowledgements

I wish to acknowledge the support and of my family: my mother, my father and my brother. Also I would like to thank the help received from my supervisor, Giampaolo Romano, who has solved any doubt that I could have.

Contents

List of Figures

List of Tables

List of Symbols

Q	heat	J
$\dot{q}$	heat flux per unit area	W/m^2
m	mass	kg
c	specific heat	$J\,kg^{-1}\,K^{-1}$
T	temperature	K
k	thermal conductivity constant	$W/m \cdot K$
h	heat transfer coefficient	$W/m^2 \cdot K$
L_c	characteristic length	m
V	velocity	m/s
ρ	density	kg/m^3
μ	dynamic viscosity	$N \cdot s/m^2$

Chapter 1

Introduction to the thesis topic

Nowadays the use of computational fluid dynamics (CFD) is spread all around the world, and the topic of this thesis is born from the thought of implementing this simulation method, along with knowledge from aerodynamics, in studying the behaviour of wind around buildings. With the full characterisation of the airflow produced by the wind, it is possible to carry out a thermodynamic analysis in order to investigate how the different shapes and positions of buildings can affect the thermal efficiency.

Along the report, specific problems that can be applied to real life cases will be solved with the help of the CFD software Ansys Fluent. Also, the basic principles on heat transfer and computational fluid dynamics software will be explained prior to the exposition of the cases, in order to handle the needed theoretical background.

At the end of this report there will be presented the conclusions on how the shape and location of different buildings affect the thermal efficiency, and consequently the energetic efficiency, as it is the main purpose of this work

Chapter 2

Theoretical background

2.1 Basic principles of heat transfer

From thermodynamics, it is stated that the energy from a system can be transferred by the interactions between the system and its surroundings. These interactions are denominated work and heat, and in this thesis the second one will be explained in order to have the knowledge to make the posterior analysis.

Heat transfer is the energy flowing in a system due to a difference in temperature. That is, every time that a temperature difference occurs in a body or between bodies, there must be heat transfer. The fundamental heat equation establishes the relation between the increment of temperature experimented by a substance (ΔT) and the heat that it exchanges, and it is expressed in 2.1 [1].

$$Q = m \cdot c \cdot \Delta T \tag{2.1}$$

In Equation 2.1, c represents the specific heat ($\mathrm{J\,kg^{-1}\,K^{-1}}$), m is the mass (kg), Q is the heat (J) and ΔT is the temperature increment (K).

However, this heat does not always take place in the same ways: basically there are three different types of heat transfer. The three forms in which heat transfer can be found are: conduction, convection and radiation.

2.1.1 Forms of heat

Conduction

Conduction is considered as the energy transfer from the most energetic particles to the least energetic particles due to their interaction. When having a temperature gradient, the energy transfer due to conduction will occur in direction of the decreasing temperature. This form of heat transfer can take place in solids, liquids or gases, and it can be modelled with Fourier's law, expressed in 2.2

$$\dot{q} = -k\frac{dT}{dx} \tag{2.2}$$

The heat flux by unit area $\dot{q}$ is the velocity in which the heat is being transferred in x direction per unit area, perpendicular to the transfer direction, and it is proportional to the temperature gradient, $\frac{dT}{dx}$ in this direction [1]. The proportionality constant k is the thermal conductivity constant and it is characteristic for each material, measured in $\mathrm{W/m \cdot K}$. A minus sign appears as a consequence of the heat being transferred towards the decreasing temperature. When considering that the distribution of temperature is linear, the equation that models the heat transfer by conduction id 2.3.

$$\dot{q} = -k\frac{\Delta T}{L} \tag{2.3}$$

It should be noted that 2.3 gives the heat flux per unit area. Therefore, the transferred heat per unit time through a plane wall will be the product of the heat flux per unit area times the area perpendicular to the flux direction.

Convection

Heat transfer through convection implies the exchange of heat between a solid surface and a moving adjacent fluid. As a consequence of the interaction between the fluid and the surface is the development of a region in the fluid in which the velocity varies from zero, in the surface, to the outer velocity, associated to the flux. This region is known as hydrodynamic or velocity boundary layer. Moreover, if the temperatures of the surface and the fluid are different, a thermal boundary layer will be formed, and can be identical or different to the one of the velocity [1].

This form of heat transfer can be explained with the random molecular movement and with the volumetric movement of the fluid in the boundary layer. Closer to the zone where the fluid velocity is low, the molecular movement will have a bigger contribution. Furthermore, right in the surface, where the velocity is zero, the heat will only be exchanged by this mechanism. On the other hand, the contribution due to the volumetric movement starts when the boundary layer grows.

The convection heat transfer can be classified in two different forms. Forced convection takes place when the flux is being produced by external means, like a fan or atmospheric winds. On the other hand, free or natural convection takes place when the flux is induced by forces from density differences, caused by temperature variations in the fluid.

Whatever it is the convection procedure in which the heat is being exchanged, it can be modelled with 2.4, which is the Newton's law of cooling.

$$\dot{q} = A \cdot h \cdot \Delta T = A \cdot h \cdot (T_s - T_\infty) \tag{2.4}$$

The heat flux per unit area $\dot{q}$ is proportional to the difference between the surface and the fluid temperatures, respectively. The proportionality constant h is the heat transfer coefficient and depends on the conditions of the boundary layer, which is influenced by the surface geometry, the movement nature of the fluid and thermodynamic properties of the fluid. The thermodynamic studies are reduced to the study and estimation of this coefficient.

As previously said, Equation 2.2 calculates the heat per unit are in convection, and Equation 2.4 expresses the heat per unit area in conduction. Considering both equations and making them equal, a dimensionless number is obtained, commonly known as Biot number. As seen in Equation 2.5, the Biot number represents the relation between the amount of transferred heat through convection and the amount of transferred heat through conduction.

$$\dot{B}i = \frac{h \cdot L}{k} \tag{2.5}$$

Radiation

The thermal radiation is the emitted energy by the bodies which have an infinite temperature. Different from conduction and convection, radiation does not require a physical mean to be propagated [1]. The radiation that a surface emits is originated by the thermal energy limited by the surface, and the velocity at which emits this energy per unit area is denominated the superficial emitted power, and its limit is established by the Stefan-Boltzmann law, in 2.6.

$$E_b = \sigma \cdot T_s^4 \tag{2.6}$$

where the Stefan-Boltzmann constant has a value of $5.67 \times 10^{-8}\,\mathrm{W/m^2 \cdot K^4}$.

In this equation, the temperature refers to the absolute temperature of the surface, which is called ideal radiator or black body. However, the heat flux emitted by a real surface is lower than the one of a black body, and it is determined by the emissivity, ϵ, as shown in 2.7.

$$E_b = \sigma \cdot \epsilon \cdot T_s^4 \tag{2.7}$$

The emissivity takes values between zero and one, and it represents the efficiency of a body when emitting energy with respect to a black body. This property depends on the material and on its surface.

2.1.2 Heat transfer on buildings

Once the heat transfer principles have been explained, they can be applied into buildings. These are considered as bodies which are internally heated and they interact with the outside air exchanging constantly energy. Therefore, a building is considered as a body with air inside at a constant temperature, and its walls have a thickness with an associated conductivity constant according to its material.

For this study, the forms of heat transfer that will be studied will be conduction and convection, since it will be focused on the interaction of the wind at which the building is exposed, and all the temperature heating the building will be considered to come from artificial heating. In this way, radiation will be neglected.

In order to simplify the problem, the temperature inside the building will be considered as constant along the problem. Since temperature in the inside of the building is kept constant, a conduction condition will be considered in the heat exchange through the walls of the building. And finally, convection will take place between the outer surface of the walls of the building and the free-stream air.

2.2 Basic principles on CFD software

Computational Fluid Dynamics (CFD) consists on applying the fundamental laws of mechanics to a fluid, obtaining its governing equations by means of numerical methods. These equations, generally known as Navier-Stokes equations, don't have an analytical solution yet. However, it is possible to obtain an approximate computer-based solutions to these governing equations, and this is the subject matter of Computational Fluid Dynamics. In this case, the CFD software that will be used is ANSYS Fluent.

2.2.1 Working principle of CFD software

The governing equations are analytical equations, so if they must be solved by computer, they will have to be in a discretized form in order to obtain an approximation of the continuous variables.

Therefore, the process of discretization of a continuous fluid must be carried out. The domain is divided in small elements creating a grid, and then the field variables are approximated to a finite number in points called nodes. Then the governing equations are also discretized, in order to go from the integral form (continuous) to an algebraic form (discrete). Finally, the system of algebraic equations is solved and the values of the variables are obtained in each node [2].

2.2.2 Steps in CFD analysis

In order to make a complete analysis in computational fluid dynamics, there are three main steps that should be followed [3].

Previous calculus

This step consists on formulating the problem and defining the governing equations, establishing the boundary conditions and generating a mesh of finite elements. These objectives will vary according to the analysis to be realised and the computational capacity.

Solving equations

This is the main step of the computational fluid dynamics, since it is where the numerical solution of the governing equations takes place.

Results analysis

Once the solution has been performed, the results obtained correspond with the field variables in each point of the mesh, or what is the same, in the nodes. Then, a very important part of the post-processing is to graphically represent the results sin a clear way to make easier the analysis. Here it is also included the comparison with previous results, or even with experimental values.

2.2.3 Spatial discretization

As previously said, the discretization is the working principle of computational fluid dynamics. There are many different ways to discretize the problem and can be classified in three main categories: finite differences, finite volumes and finite element [2]. All of them precise of a previous meshing process in order to perform the discretization of the governing equations, and there are two ways to do it:

- Structured mesh: each point of the mesh is identified by cartesian coordinates and with its corresponding index i, j, k. The cells in 2D are quadrilateral, and hexahedral in 3D.
- Non-structured meshing: the cells and the nodes do not have any particular order. That is, in this case the index won't be useful since the cells and the nodes cannot be identified by it. The elements will be quadrilateral and triangular for 2D, and tetrahedral and hexahedral for 3D.

2.2.4 Equations discretization

As previously said, there are three different approaches to discretize the governing equations of a problem and in this section they will be explained [4].

Finite differences method

This method was the first method applied to obtain the numerical solution of differential equations. It is based on the Taylor series for the discretization of the variable derivatives. It is a first order approximation, since the error is proportional to the higher order term of the remaining series. However, it can also be used to obtain more precise approximations just by taking higher order terms of the Taylor series.

The main advantage of the finite differences method is its theoretical simplicity and its possibility to improve the precision by increasing the order of the derivative approximation. On the other hand, this method requires a structured mesh, which makes it suitable only for simple geometries. Furthermore, this method cannot be used with curvilinear coordinates, so the Navier-Stokes equations would have to be transformed from curvilinear to cartesian coordinates in order to solve the problem. For these reasons, this method is actually only applied to very simple geometries.

Finite volumes method

The finite volumes method applies directly the conservation equations in its integral form. It discretizes the equations in each element of the domain, in which previously a spatial discretization has been done. The surface integral present in the Navier-Stokes equation is approximated to the sum of the fluxes going through each of the element faces.

The main advantage of the finite volumes method is that the spatial discretization is directly carried out in the physical space of the problem, which makes possible a transformation between coordinates systems. Comparing this method with the finite differences method, this one is more flexible since it can be implemented to structured and non-structures meshes, and this is the reason why it is the most commonly used for complex geometries.

Finite element method

The finite element method, as the general method to solve the Navier-Stokes equations, starts with a division of the domain, generating a non-structured mesh. The fineness of the mesh will depend on the element type and the desired precision, and the total number of nodes multiplied by the number of variables gives the degrees of freedom of the problem.

In addition, the shape functions must be defined, and they will represent the solution variation in the inside of the elements. These shape functions are linear distributions of zero value outside the corresponding element of the function. This gives as a result a second order approximation in the results.

The advantage of this method is its flexibility to work with non-structured meshes, as well as with non-newtonian fluids.

2.2.5 Initial and boundary conditions

In order to solve the governing equations, the initial and boundary conditions must be specified. The initial conditions determine the variables in the instant $t = 0$, or in the first step on the integration [2]. In this way, it is trivial to think that as closer the initial condition is to the final solution of the problem, lower will be the time that will take to converge. Furthermore, the possibilities that the problem becomes unstable are reduced.

In every numerical simulation, a part of the fluid domain is considered in order to solve the equations. This leads to an artificial contour where the variables must be specified through boundary conditions. In general, the boundary conditions can be classified as:

- Free-flow boundary conditions: they can be inlet, outlet or inlet/outlet. Specifically, the inlet boundary conditions are specified with the conditions at the infinity.
- Wall boundary conditions: they model the behavior of the fluid in the proximity of the surface. Some examples are the no-slipping condition (null velocity at the contact point) or the tangent condition (normal velocity at the surface).

2.2.6 CFD applications on buildings

CFD software has become a powerful tool in several fields, but it is very useful when it comes to buildings. The air flow around buildings can adopt very complex paths, and separation, recirculation, von Karman vortices in the wake and reattachment must be taken into account [5]. As time goes by, the design on new buildings also becomes more complex, having aerodynamic shapes in order to improve their characteristics, and with CFD software there exists the possibility of testing these new challenging shapes without the need of high investments of money.

This first approach by means of software can be very effective to determine how the temperature will be dissipated along the building according to the wind incidence, as well as the points of maximum loading which will be exposed to failure. By CFD simulations the problem can be analysed in real scale dimensions obtaining the whole set of data, being able to access to each variable of the problem in a simple way, and recognising the points where there might be future problems.

Computational simulations play an important role when calculating the total thermal efficiency of a new building, because it lets evaluating the total exchanged heat flux. Although it is very difficult to model the complete behaviour of the wind around a building and simplified versions are often used, this process still gives a rough approximation of how the building will perform in each environment. However, progressively the wind behaviour is being modelled in a more precise way and a it is studied by the engineering field called Wind Engineering.

The branch of engineering called Wind Engineering deals with the interaction between the wind and any obstacle in the surface of the Earth. Since it takes into account very different fields such as fluid mechanics, structural dynamics or meteorology, CFD is widely used [6].

Chapter 3

Methodology

3.1 Problem definition

The problem that will be analysed consists of a building exposed to a free stream air velocity. In order to perform such analysis, a fluid domain must be defined that allows a proper calculation of the values and it does not interfere with the study object. The same fluid domain will be used for the different cases, so that the dimensions will not affect the results in any of the studies. In this case, the external fluid will be air with temperatures 0ºC, 15ºC and 30ºC, constant in each case.

On the other hand, the building geometry will change for each of the cases, but its properties will be the same. In order to simplify the geometry but still have a good approximation to the real case, the building was designed to be an empty box with concrete walls with a thickness of 30cm. Since the building is empty in the real case, the fluid inside is air at a temperature of 20ºC, simulating nice temperature conditions. This temperature is going to be achieved with a heat generation rate coming from the building simulating the heating. As in the real case, the value of the heat generation rate will be modified in each case, representing the needed power for each air temperature. The air is supposed to be at rest considering that all windows are closed, in order to simplify the problem.

Once the problem that will be dealt with has been presented, the next step is to model it in a computational fluid dynamics software, which in this case is ANSYS Fluent. The first calculation that will be done is the measurements of velocity and the comparison between the different shapes of buildings. Then, the thermodynamic study will be carried out, that is, solving the energy equation and obtaining the temperature profiles for each case.

The main objective of evaluating the problem is to achieve a clear conclusion on how the shape of the building can influence its thermal efficiency and if it is affected by the presence of other buildings nearby.

3.2 Modelling in CFD

3.2.1 Geometry definition

The first step to deal with the problem is to define the building and fluid domain geometry. This procedure has been done with the tool Design Modeler, provided by ANSYS. As previously said, different buildings and configurations will be analysed in order to have a wider view of how the thermal efficiency is affected. However, the fluid domain is the same for all the cases, as well as the dimensions of the building. Therefore, the height of the building has been chosen as a reference measure (H) and has a value of 20 meters. The fluid domain has been defined according to this measure, as shown in Figure 3.1.

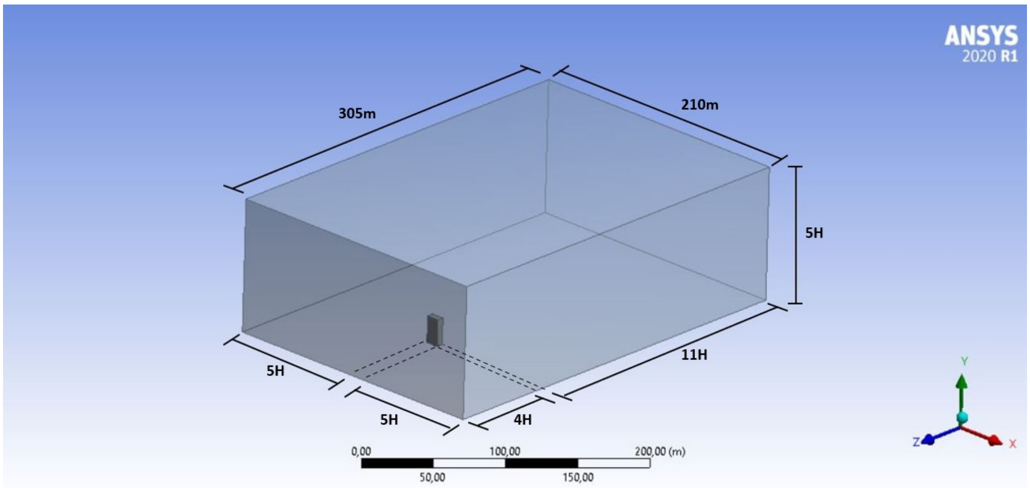


FIGURE 3.1: Geometry of the fluid domain.

So that the fluid domain does not interfere with the flow affected by the building, it has been designed with enough space at each side. For left and right sides a dimension of 5H has been chosen. For the upstream flow, 4H are enough so that the roof of the fluid domain does not affect the results. The space of airflow before the building is set to be 4H since the airflow coming from the inlet is not affected. And finally, for the downstream a dimension of 11H has been chosen, in order to capture the whole behaviour of the wake produced.

In all the four following cases that are going to be presented, the building is located at the zone indicated in dark grey in Figure 3.1, so that they all have the same airflow conditions. In the cases where there is more than one building forming the problem, this black zone indicates the position of the first building of the row.

Single building, basic case

The basic case consists on a building that is aimed to represent the average shape of general buildings. That is, it has a height of 20 meters, 10 meters width and 5 meters depth. In Figure 3.2 the sketch can be found.

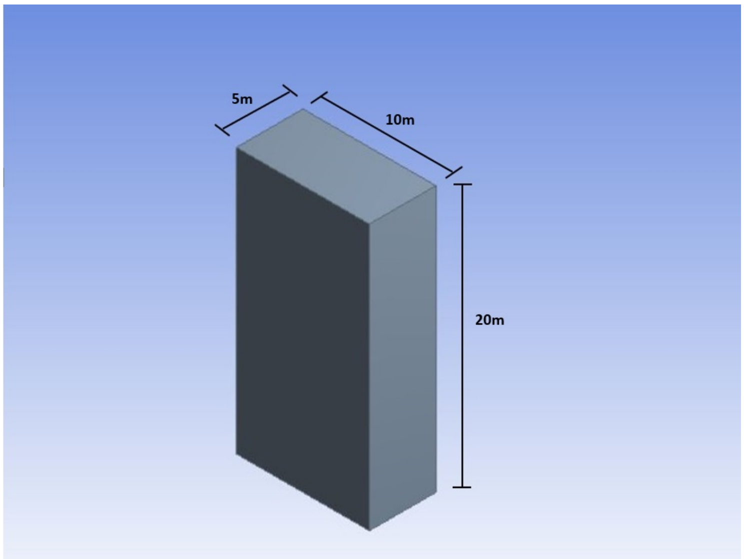


FIGURE 3.2: Geometry of the basic building.

Single building, rounded case

The next building that is going to be analysed is the rounded one, and has been designed from the previous one. As seen in Figure 3.3 the general dimensions are exactly the same as the basic case, but the edges of the walls have been bent with a fixed radius of 1.5 meters. With this design the objective is to make a more aerodynamic shape, in terms of getting the air to adapt in a better way to the walls, creating less turbulence.

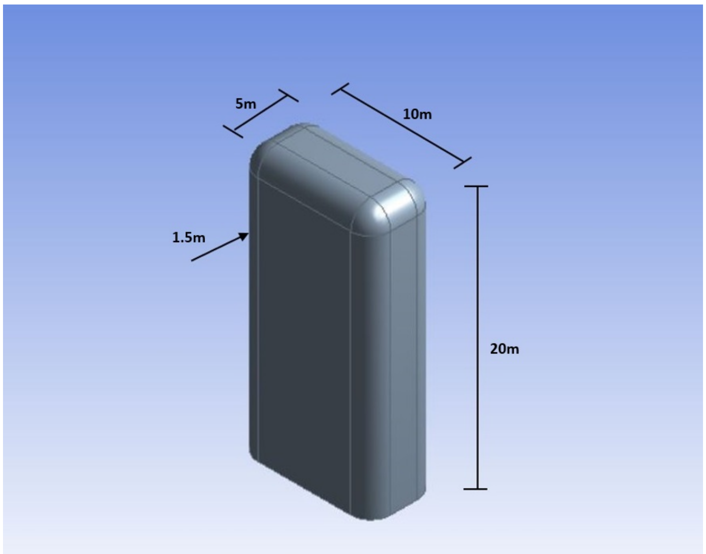


FIGURE 3.3: Geometry of the rounded building.

Group of buildings, basic case

The group of basic buildings consists on a row of three buildings previously explained. The separation between them is of 20 meters. In Figure 3.4 the disposition of the row has been represented.

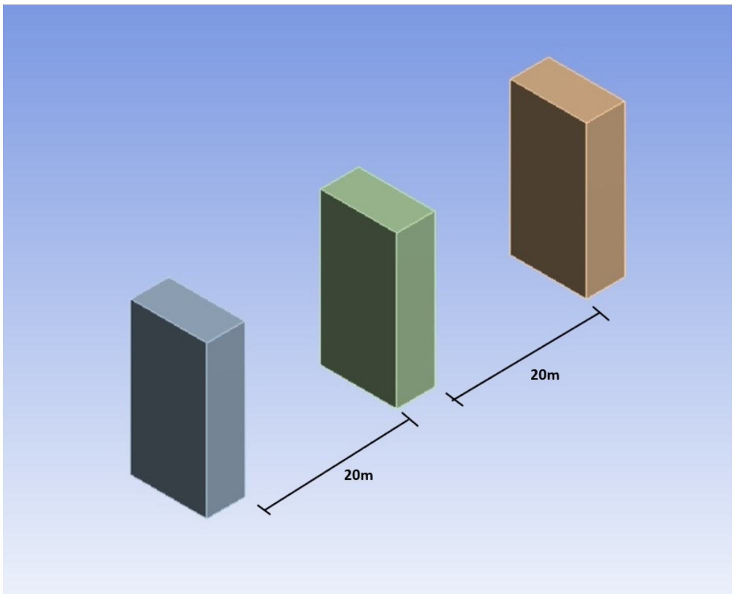


FIGURE 3.4: Geometry of the group of basic buildings.

Group of buildings, rounded case

The last case that will be analysed is the group of rounded buildings, that consists on a row of three buildings with the bent edges. The separation between them is again 20 meters to preserve the similarity with the basic case and make proper comparisons. In Figure 3.5 the three buildings have been represented.

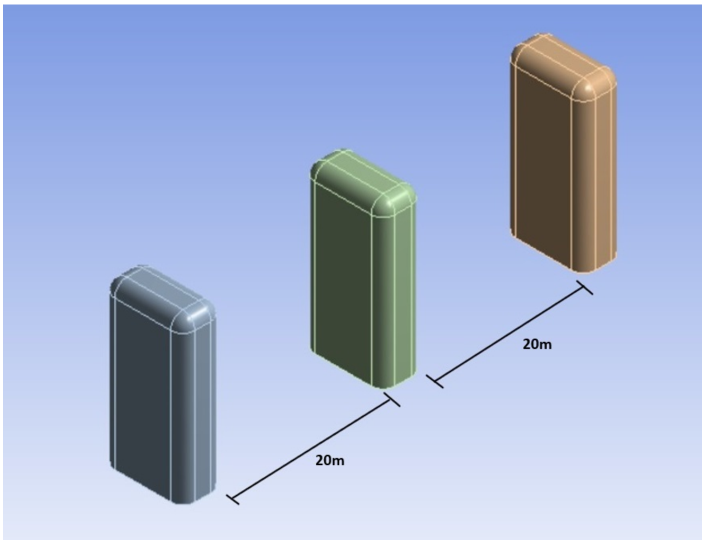


FIGURE 3.5: Geometry of the group of rounded buildings.

3.2.2 Mesh procedure

Once the geometry of the objects to be analysed has been defined, the next step in CFD simulations is the generation of the mesh. The mesh is a very important feature in the pre-processing of a CFD simulation since a bad quality mesh can lead to wrong readings of the variables. Therefore, since there are two different shapes of buildings, throughout this section the two followed methods for each case will be explained.

The part of the fluid domain where the building is placed is going to be defined with tetrahedral elements. They are very simple elements that do not require excessive computational capacity, and are able to adapt to both shapes of buildings. In Figure 3.6 the external look of this mesh is represented, where the bigger elements can be seen since they are the ones more distant from the region of interest, which is the surroundings of the building.

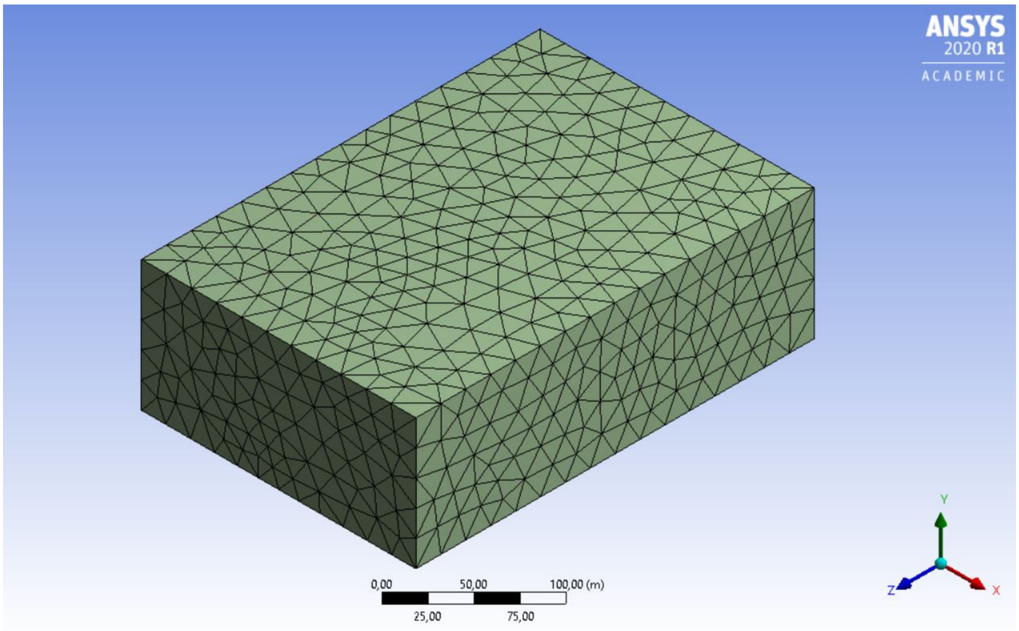


FIGURE 3.6: Mesh of the fluid domain.

Then, the meshing for the building is defined. For the basic case a hexahedral structure is going to be used, since it is possible to divide the building in elements of 50cm in each face of the piece. This method will give 40 elements in the height, 20 elements in the width and 10 elements on the depth of the building. In Figure 3.7 the meshing of the building is represented.

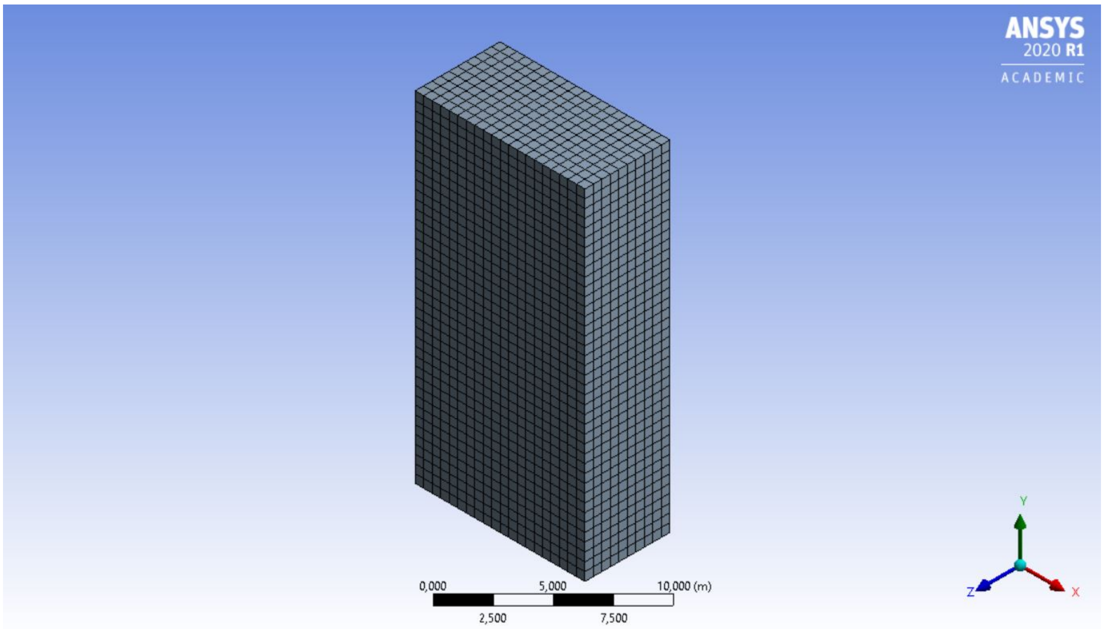


FIGURE 3.7: Mesh of the basic building.

Regarding the interface surface between the fluid domain and the building, the *Proximity* function has been used, so that when the fluid domain mesh approaches the building, it becomes finer until the size of the elements of the building. In Figure 3.8 has been plotted the view when doing a section plane crossing at the middle of the elements. What can be seen here is the transition from big tetrahedral elements to smaller ones, and how the building is composed by hexahedral elements.

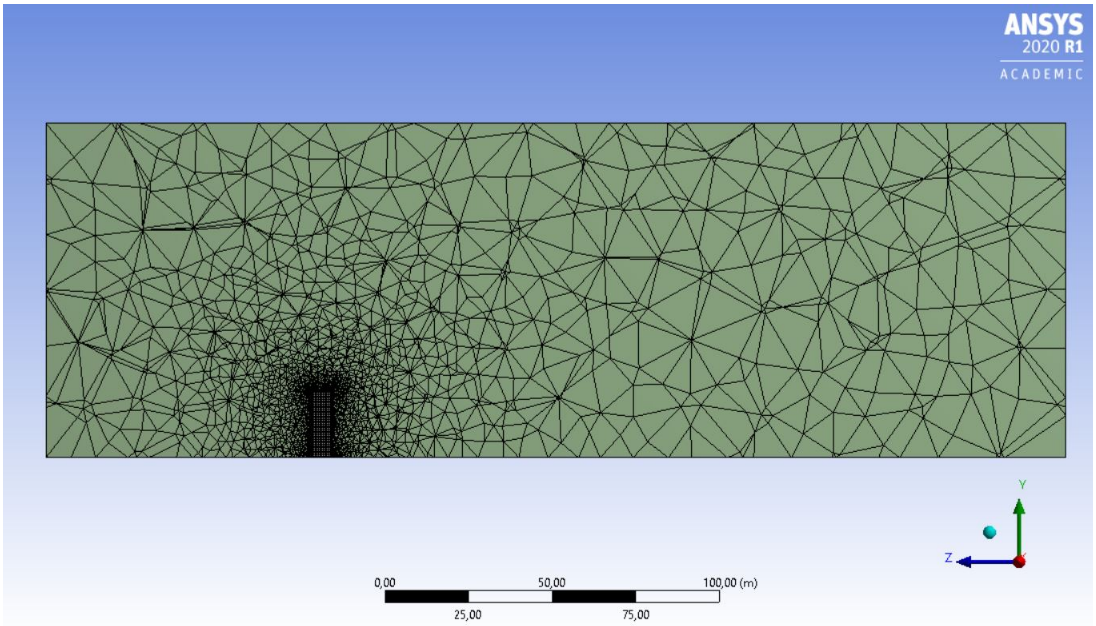


FIGURE 3.8: Mesh in the middle plane of basic building.

It is also possible to zoom the interface zone in order to appreciate the size of the elements which are nearer to the building walls, like in Figure 3.9.

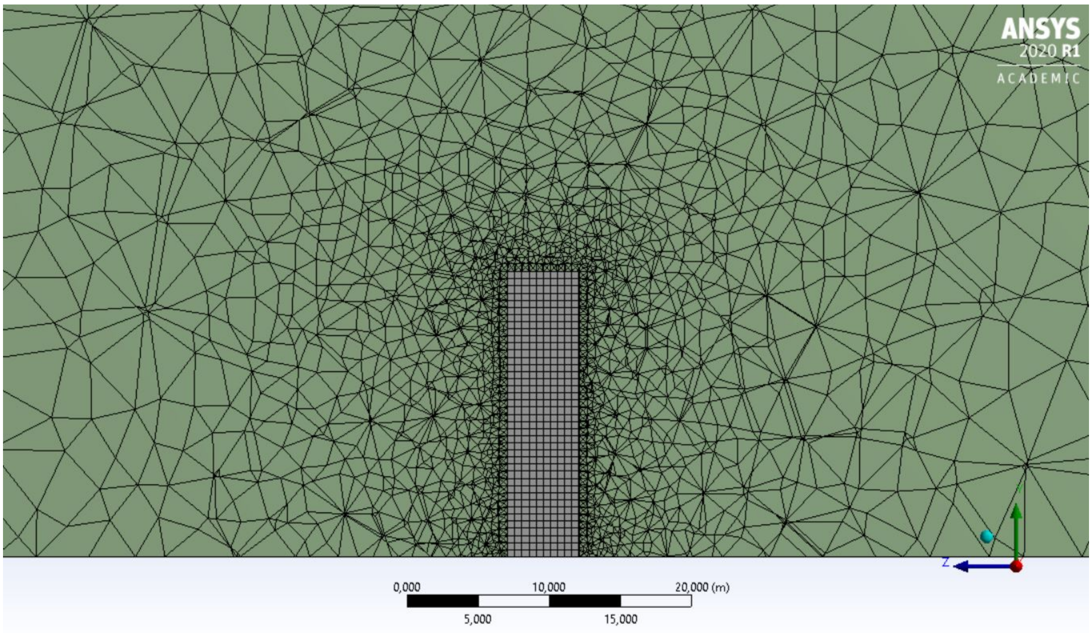


FIGURE 3.9: Zoom on the interface mesh of basic building.

A good way to check if the mesh generated is good enough is with the skewness value. Skewness is defined as the difference between the shape of the element and the shape of an equilateral element of equivalent volume. If the cells have a very high skewness, they can decrease the accuracy and destabilize the solution. A general method to evaluate this factor is that is does not exceed 0.95 for the maximum value, and 0.33 for the average value [7]. In this case, the maximum skewness value is 0.7898 and the average is 0.227. Therefore, it is possible to say that the generated mesh is stable and will not interfere in the convergence of the problem.

The other building is characterised by its rounded shape, so in this case the tetrahedral elements will be used for both the building and the fluid domain. In Figure 3.10 the mesh for the rounded building is represented.

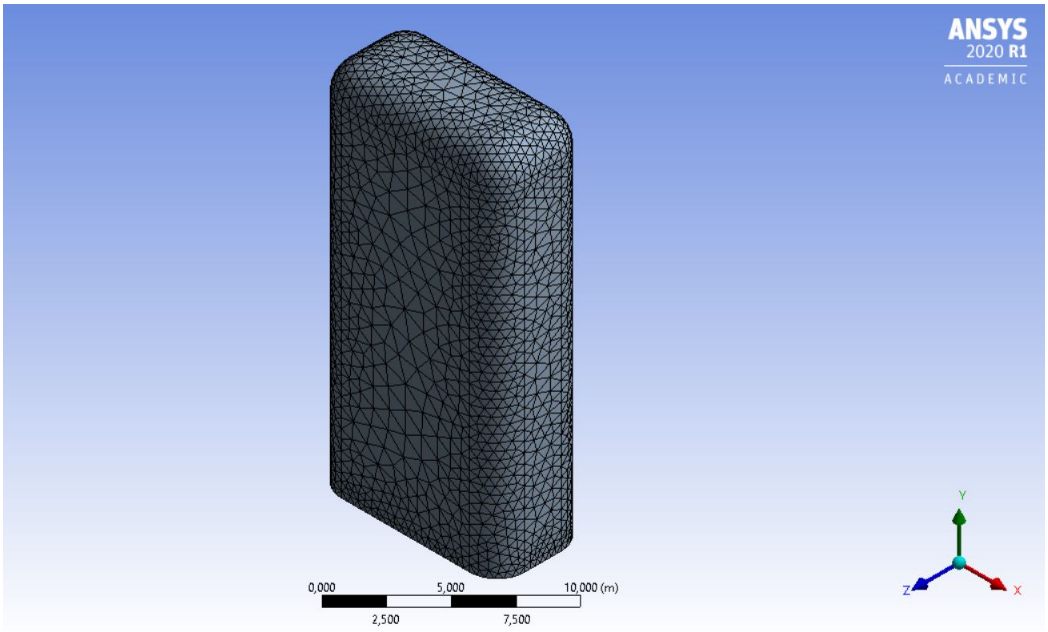


FIGURE 3.10: Mesh of the rounded building.

In this second case, the edges of the building become rounded walls with a fixed radius of 1.5 meters, and it is divided into seven different elements which gives as a result a size of the element of 0.21 meters.

For the fluid domain, the same procedure has been followed, in terms of using the *Proximity* function so that the mesh becomes finer when approaching the building. In Figure 3.11 is represented the shape of the mesh in the interior by means of a section plane in the middle of the building in the YZ plane.

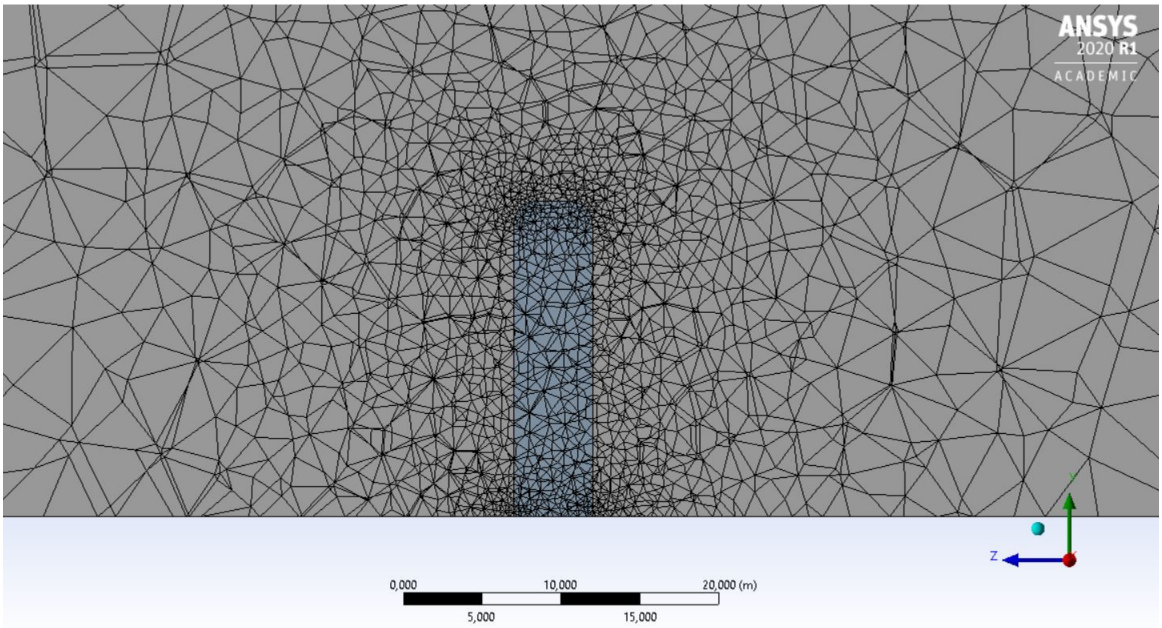


FIGURE 3.11: Mesh in the interface of rounded building.

The maximum skewness value for this mesh is 0.799 and the average value is 0.248, which means that this is a stable mesh and will not interfere with the convergence of the problem.

The next step is to generate the mesh for the different group of buildings. The procedure for the fluid domain meshing will still be the same, as well as the method for each type of building. However, in these cases a refinement between buildings will be needed in order to have all the information of the flow in the gaps left between one building and another.

Starting with the row of three basic buildings, the mesh of the building is conserved as it can be seen in Figure 3.12.

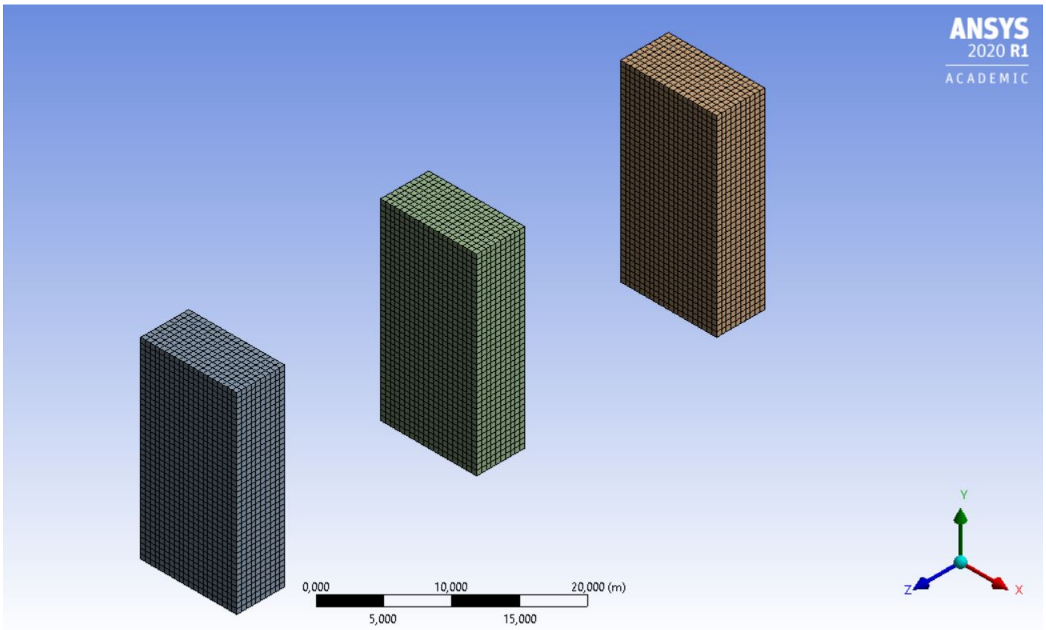


FIGURE 3.12: Mesh of the basic building group.

However, when doing a section in the middle plane, it is observed that the fluid domain has a finer mesh in the space between buildings (Figure 3.13).

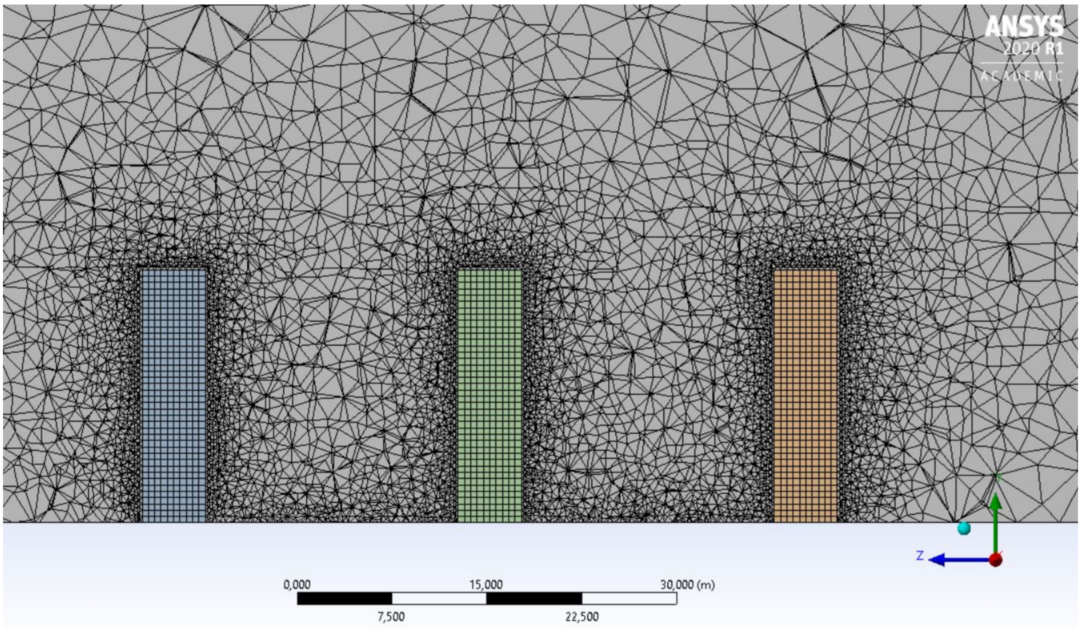


FIGURE 3.13: Mesh in the middle plane section for basic building group.

When generating the mesh for the row of three basic buildings the maximum skewness value is 0.869 and the average value is 0.223. With these values the mesh should not give any problems when evaluating the problem.

The last case to generate the mesh is the row of three rounded buildings. As before, the mesh of the building itself will be done with the same procedure as the single rounded case. The meshing of the row of buildings can be seen in Figure 3.14.

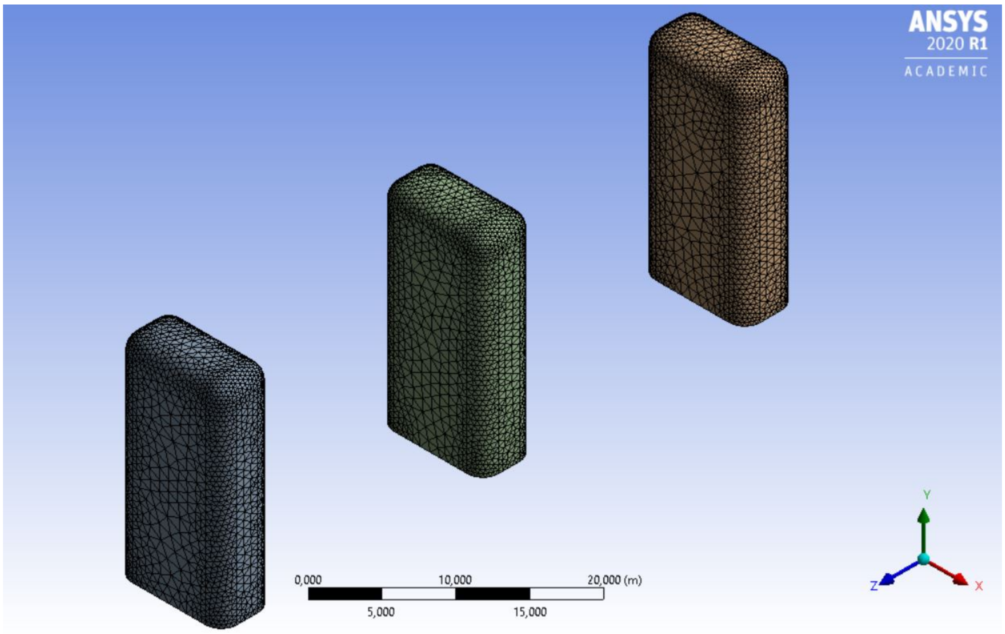


FIGURE 3.14: Mesh of the rounded building group.

Then, for the interface between the building and the fluid domain, the function *Proximity* was used again to create a transition on the size of the cells so that it is refined at the faces and edges and coarser at the walls of the fluid domain. The result of using this function can be seen in Figure 3.15.

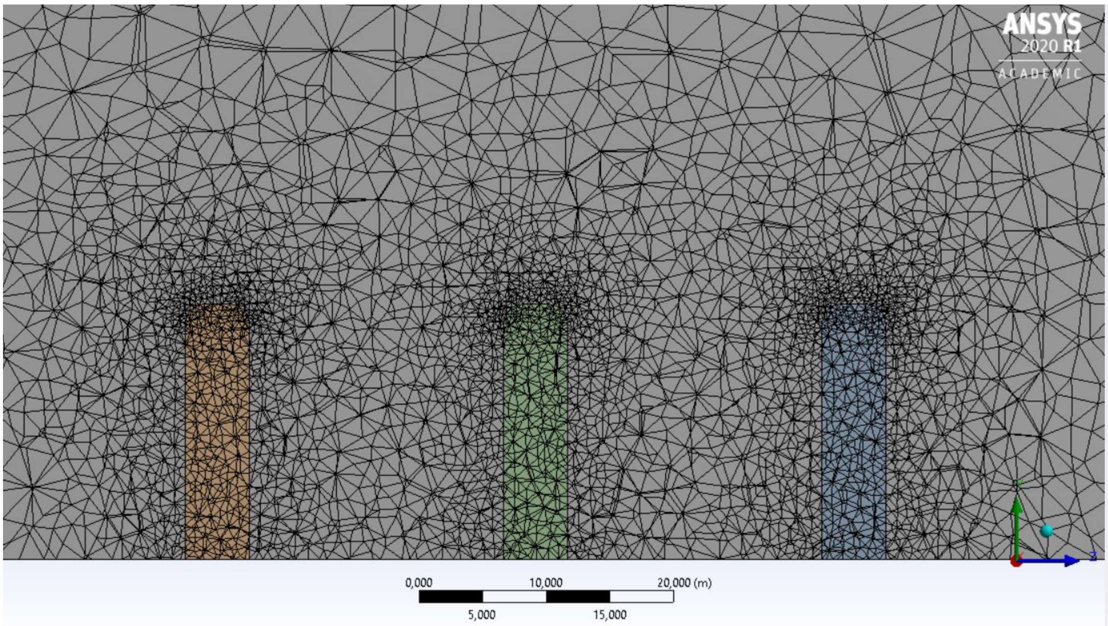


FIGURE 3.15: Mesh in the middle plane section for rounded building group.

In this last case the maximum skewness value is 0.823 and the average value is 0.246, which are both below the critical values. Therefore, this mesh is also stable and it will not give any convergency problems.

Once the meshing procedure has been completed, it is possible to define the setup and the boundary conditions of the problem, which will be discussed in 3.2.4.

3.2.3 Materials

The material properties are going to define the performance of each body participating in the problem. In this problem there are two defined materials: air and concrete.

The air is the material assigned to the fluid domain and the inside of the building. Its properties vary with temperature, so in Table 3.1 all the needed values when carrying out the problem have been gathered [8].

Temperature (ºC)	Density (kgm^3)	Dynamic viscosity ($N \cdot s/m^2$)
0	1.29	$1.72 \cdot 10^{-5}$
10	1.25	$1.76 \cdot 10^{-5}$
20	1.21	$1.81 \cdot 10^{-5}$
30	1.17	$1.86 \cdot 10^{-5}$

TABLE 3.1: Air properties.

Concrete is the material assigned for the building walls so that the similarity to the real case is as real as possible. The properties used by Ansys Fluent are gathered in Table 3.2 [9].

Density (kg/m^3)	Specific heat ($J/kg \cdot K$)	Thermal conductivity ($W/m \cdot K$)
2400	1000	1.13

TABLE 3.2: Concrete properties.

3.2.4 Setup and boundary conditions

The boundary conditions are essential for the good development of the problem, since they will influence the results. In order to avoid possible errors and minimise the possibility of mistakes, the setup has been divided into two different steps. As it has been previously said, the form of heat of convection is basically dominated by the velocity of the flow. Therefore, a cautious way to approach the problem is to make a first analysis of the different geometries without solving the energy equation. That is, in first place the velocity fields will be obtained without any temperature influence. Then, in a second step a thermodynamic analysis will be carried out, giving to the building and the airflow their respective temperatures.

Even if there are two cases to be defined, there are some characteristics on the setup that are common for both of them. In all the cases a double precision has been used in the settings of the solver, since it is recommended for problems with high aspect ratio of the cells, and also for the ones with large scales as these cases. Also, a steady approach will be carried out, since there are no variables that vary with time. The algorithm that has been used in all problems is SIMPLEC because of its good performance on pressure-velocity cases. The last common characteristic is the viscous model, that has been chosen regarding the Reynolds number, calculated in Equation 3.1 with the conditions of the problem defined in Section 3.1.

$$Re = \frac{\rho V L_c}{\mu} = \frac{1.225 \cdot 1 \cdot 5}{1.79 \cdot 10^{-5}} \simeq 3.42 \cdot 10^5 \tag{3.1}$$

The Reynolds number for the problem is one order of magnitude higher than the limit where the fluid can be considered transient. Therefore, even if there are small changes in the density and dynamic viscosity, the Reynolds number will indicate that the regime of the fluid is still turbulent. In order to describe the turbulent regime, the semi-empirical model *k-epsilon* has been chosen, known by its robustness, economy and reasonable accuracy for heat transfer problems.

Once the general conditions have been defined, the setup and boundary conditions for the velocity problem have to be set. In this occasion, the face of the fluid domain which is closer to the building and parallel to its main faces is defined as the *inlet* with a velocity of 1 m/s. The parallel face to this one but further to the building is defined as *outlet*. Then, the walls of the fluid domain and the building are defined as *walls* with no-slip condition.

On the other hand, the thermodynamic case is basically characterised by the energy equation that in this time is turned on, so that the program solves it for each element. The faces corresponding to the inlet and the outlet are defined in the same way as the previous case, with a velocity of 1 m/s, and the temperature of the fluid domain will take values of 0, 10, 20 and 30ºC. However, now a heat generation rate is assigned to the building representing the heating, and it has a defined value for each temperature value.

Chapter 4

Results

4.1 Introduction

Along the following section the results of the CFD simulations will be gathered. This results imply the velocity profiles and temperature gradients for each geometry problem.

4.2 Velocity profiles

In first place, the velocity field has been plotted for all the geometry cases. The point where it has been represented is at the middle plane of the building in the vertical direction, and the middle plane of the building in the transverse direction. In this way, the top quarter part of the building can be seen and it is easier to visualise the velocity field. Also, the profile and the top viiews have been compared between the different cases.

The velocity in z direction, that is, the same direction that the airflow, for the case in which the building has sharp corners, has been represented in Figure 4.1. On the other hand, the velocity in z direction for the rounded building has been represented in Figure 4.2.

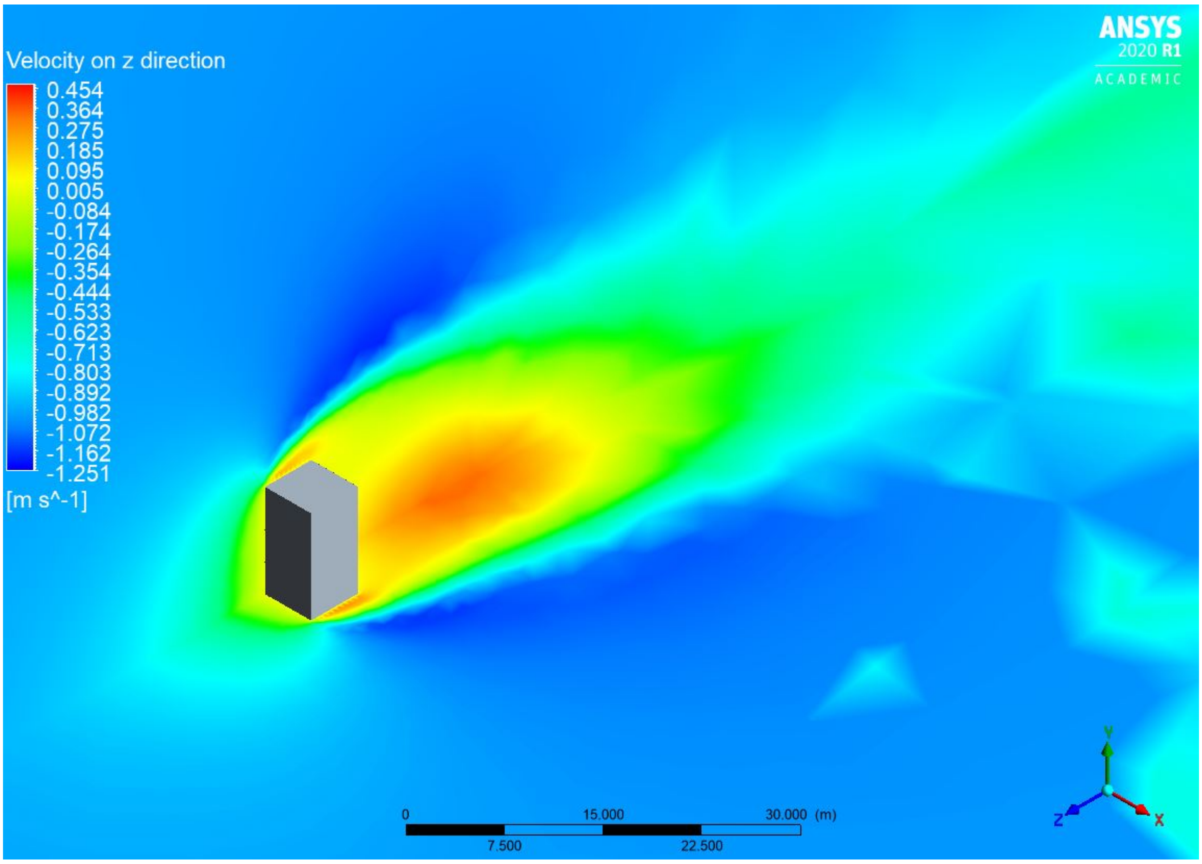


FIGURE 4.1: Velocity gradient for basic building.

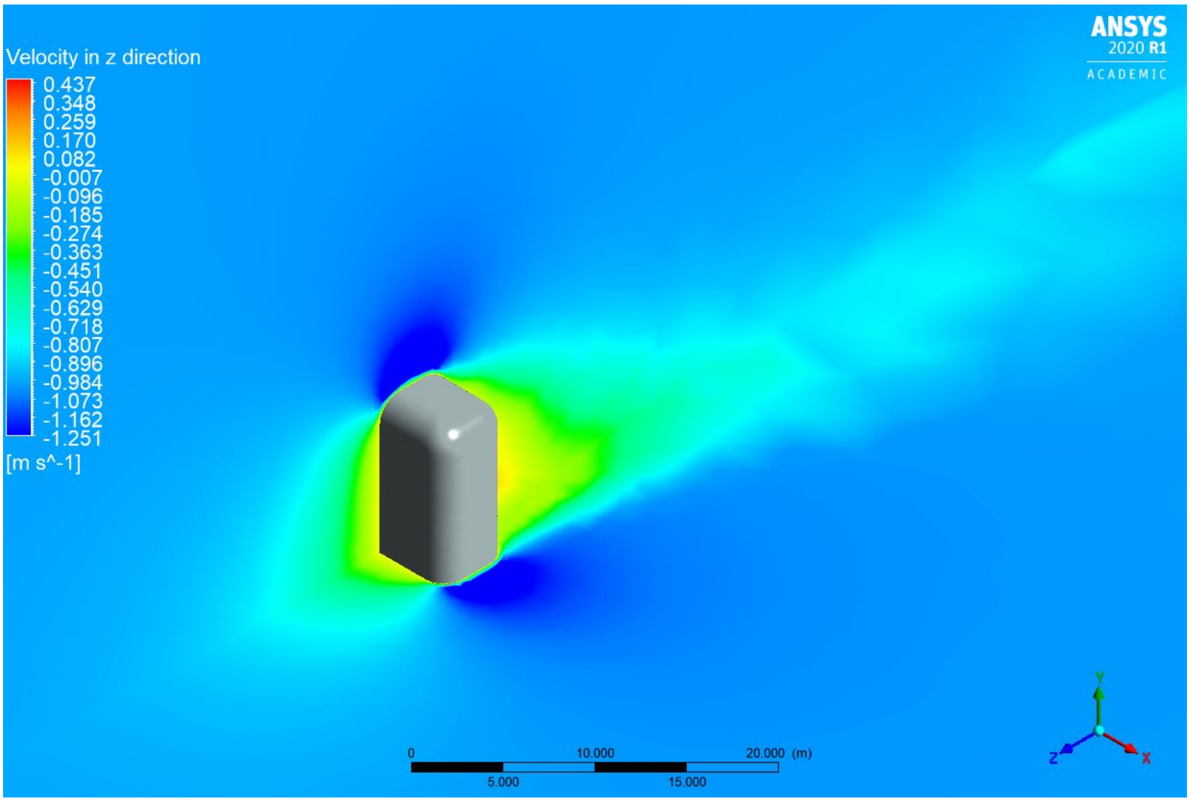


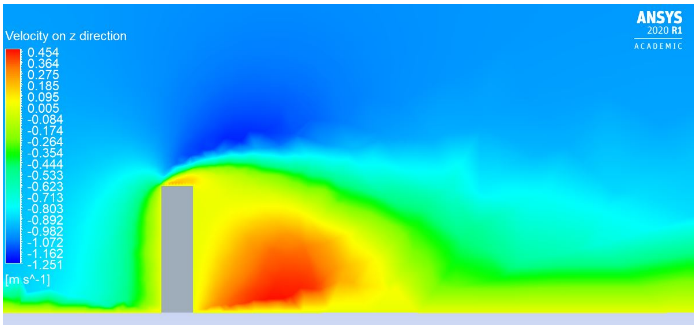
FIGURE 4.2: Velocity gradient for rounded building.

Comparing both Figure 4.1 and Figure 4.2, the wake produced by the building with sharp corners is clearly greater than the one of the building with rounded corners. This is due to the better attachment of the flow when the change of surface is smooth than when it is sharp.

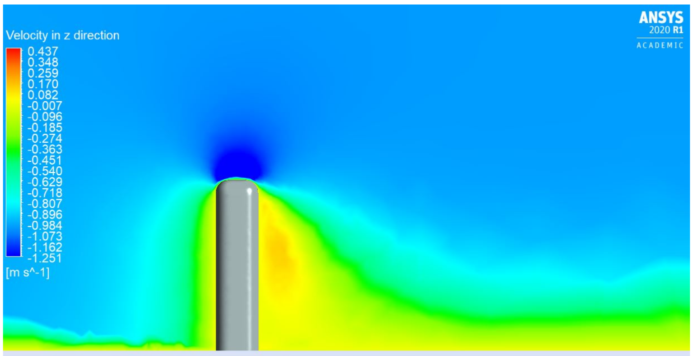
In Figure 4.1, the velocity accelerates at the top and at the sides of the building creating a big area where the flow takes a higher velocity than the one of the airflow, reaching values of 1.25 m/s. Then, at the back of the building it is formed a big zone where the velocity is positive, that is, a recirculating air zone. Here, the velocity of the flow is inverted since it is a low-pressure area and takes values up to 0.45m/s. The points where the velocity is irregular or with opposite sign to the one of the airflow represent losses on energy, since this areas will have vortexes and turbulent flow.

On the other hand, in Figure 4.2 the areas at the top and at the sides of the buildings where the air is accelerated are smaller, as well as the zone of reverted velocity at the back of the building. Due to the smooth surface transition of the walls, the air adapts better generating a smaller wake and a smaller re-circulation zone. From the Figures, the velocity for the accelerated flow at the top and side walls takes the same value as in the basic case, that is, 1.25m/s, but the velocity at the back of the building is significantly lower, being 0.1m/s. Therefore, in the case for rounded walls there are less energy losses and a more regular velocity profile.

The re-circulation zone for each case can be better seen in the profile view of the building. In Figure 4.3 it has been plotted the velocity in z direction from the side view, letting a clearer representation of how bigger the wake is for the basic case.



(a) Velocity profile for basic building.

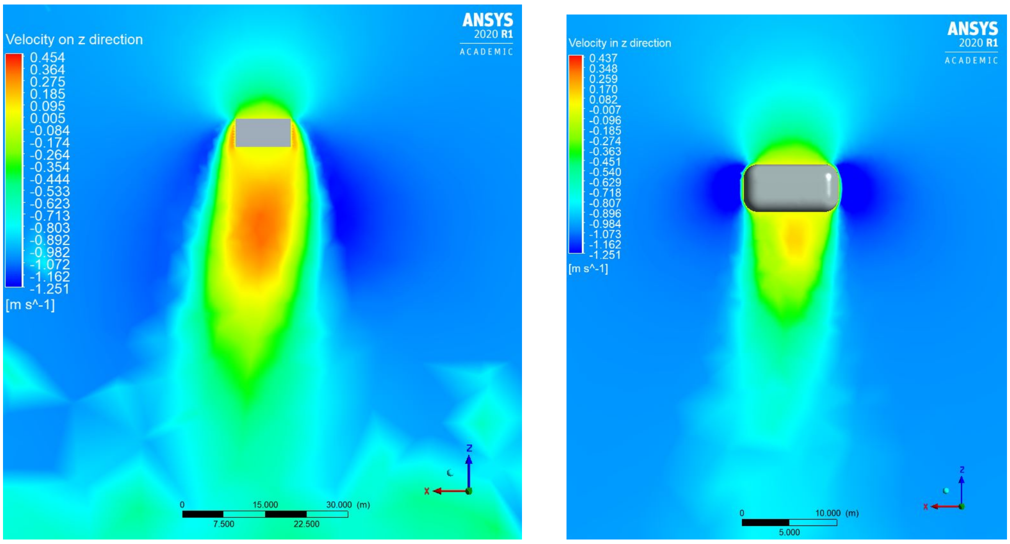


(b) Velocity profile for rounded building.

FIGURE 4.3: Comparison on velocity profiles.

In picture (a) of Figure 4.3 at the back of the building there is a big red zone, representing the flow with velocity opposite to the airflow direction. Due to the poor aerodynamic shape of this building, the flow detaches completely. However, in picture (b) of Figure 4.3 the back of the building has generally a yellow colour, meaning that the velocity in the re-circulation zone takes lower values and therefore, less energy. This is due to the adaptive shape of the building to the airflow.

Another very useful view to characterise the wake of each building is the top view. Representing it, the behaviour of the flow at the sides of the element can be seen, as well as the width of the generated wake. From this view it can be seen that the wake on the basic building is wider than the one of the rounded building, taking values of 15m and 9m respectively. In picture (a) of Figure 4.4 again the big red zone appears representing the re-circulation zone, as well as two big dark blue zones at the sides representing the detachment of the flow also at the lateral walls. On the other hand, in picture (b) of Figure 4.4 the wake is more narrow, and the accelerated flow at the lateral walls does not detach, adapting to the shape of the building.



(a) Velocity profile for basic building. (b) Velocity profile for rounded building.

FIGURE 4.4: Comparison on velocity profiles from top view.

A feature from Figure 4.4 that must also me commented is the low-pressure zone generated at the front of the building. Here, the flow is decelerated since it encounters an element that stops it. If both cases are compared, this zone is practically equal. Therefore, the expected behaviour in the front wall will be the same for both buildings.

It is also important to note that after the wake produced by the building there is still a decrease on the velocity of the airflow. This phenomena happens due to the no-slip condition of the floor of the fluid domain, that is simulating the ground in real life. Therefore, there will be a gradient of velocities where it is zero at the level of the ground.

The next step is to analyse the velocity field for the case of a group of buildings. The followed procedure has been the same, representing the velocity gradient in the two middle planes in x and y axis in order to get a quarter of the buildings and get the trend of the velocity, as well as the profile and top views.

In Figure 4.5 the interaction of the airflow with the first building of the row is very similar to the one of the single case: the velocity accelerates at the top and the sides of the building and a big re-circulation zone is created at the back. However, when observing the second and third buildings there is not an acceleration of the flow and this time at the back of each element the re-circulation zone is smaller. Therefore, when placing numerous buildings in a row, what is achieved is that the first one acts as a barrier protecting the ones behind.

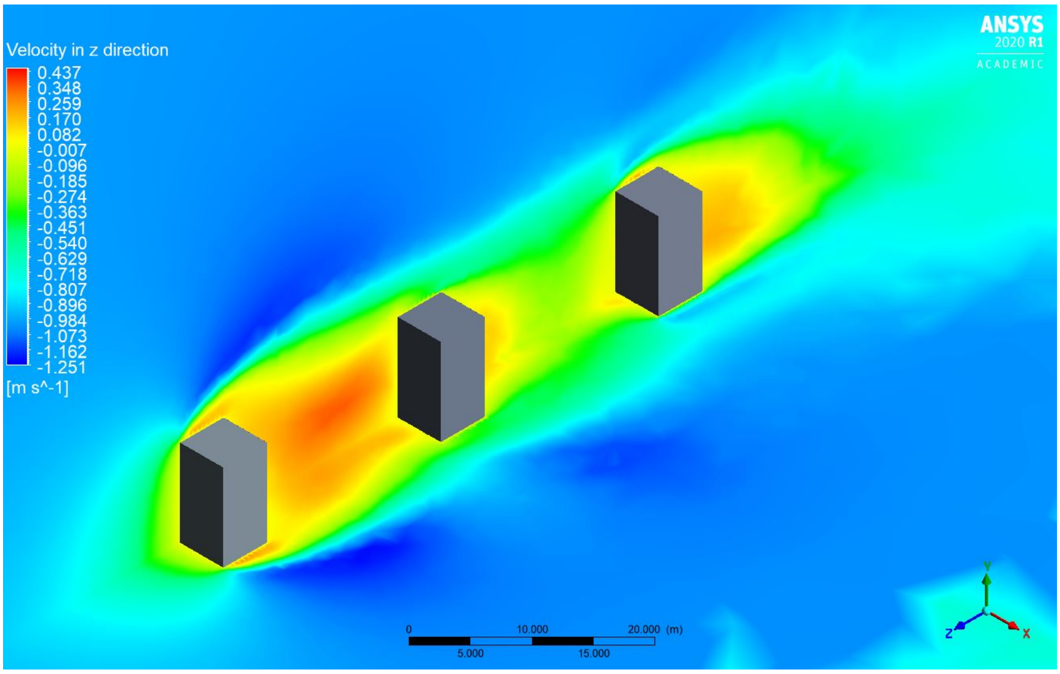


FIGURE 4.5: Velocity gradient for group of basic buildings.

On the other hand, in Figure 4.6 the row of three rounded buildings has been represented. In this case, since the flow does not suffer a very strong detachment in the first building, it has time to recover and have the same effect on the second and third buildings. This fact makes that the flow accelerates and decelerates every time it finds a building. Regarding the re-circulation zone, all three building have again the same behaviour, creating a smaller zone than the one for the basic case.

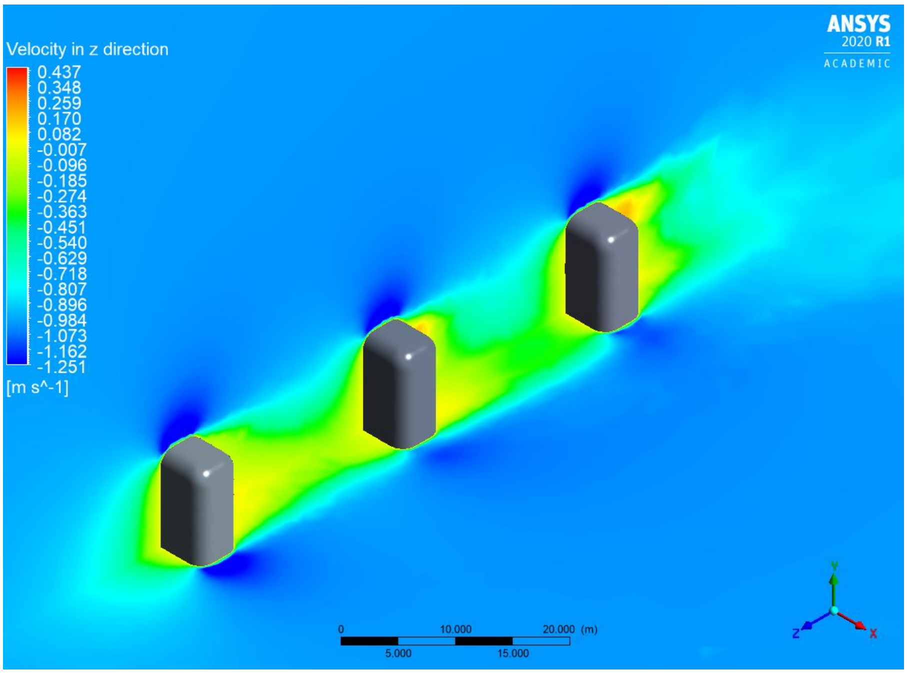
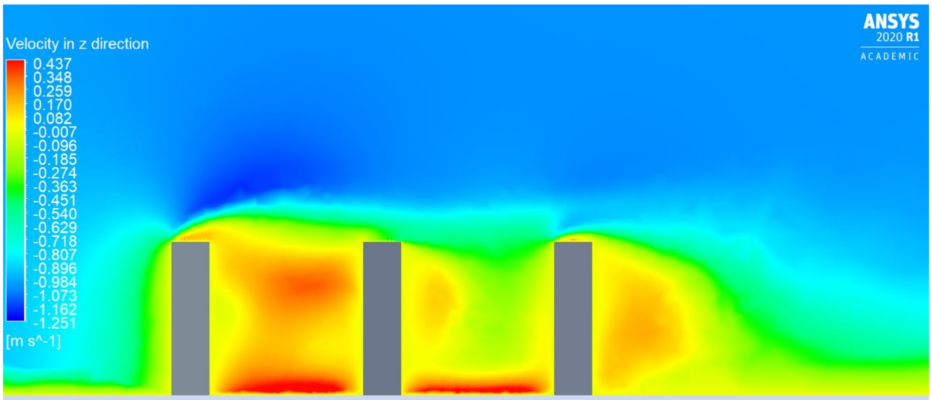
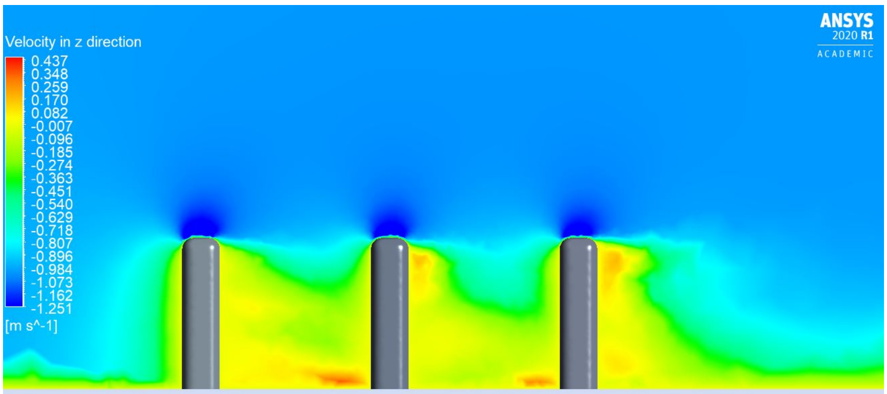


FIGURE 4.6: Velocity gradient for group of rounded buildings.

In order to visualise in a better way the acceleration on the top of the buildings and the re-circulation zone, the profile view has been represented for both cases. In picture (a) of Figure 4.7 the complete detachment of the flow in the top part of the first building can be seen, what makes that all the flow surrounding the second building is decelerated. When it comes to the third building, the flow is recovered and is able to adapt again to the top of the building, but this time with less velocity and therefore, less energy. Regarding the re-circulation zone, the first building is the one which takes the bigger one, and it decreases in the second and the third buildings progressively. On the other hand, in picture (b) of Figure 4.7 a more uniform behaviour of the velocity can be found. Since the flow is more adapted to the building, it does not suffer a big detachment and it is able to behave the same way when encountering all the three buildings. Also, the re-circulation zone has very similar behaviour on the three buildings.



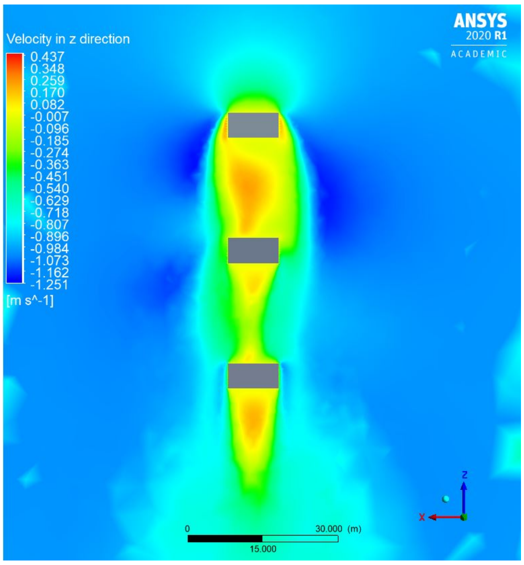
(a) Velocity profile for basic group of buildings.



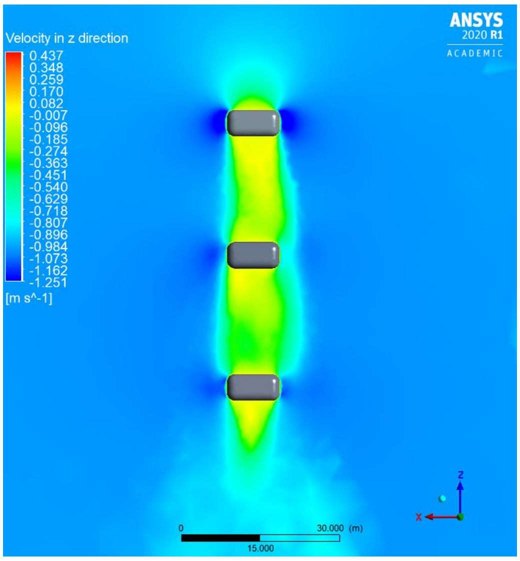
(b) Velocity profile for rounded group of buildings.

FIGURE 4.7: Comparison on velocity profile for group of buildings.

So that the complete analysis of the velocities is done, the top view of the velocity gradient can also be useful in order to characterise the size of the wake. From this view, in the case of three basic buildings in a row a wider wake can be observed due to the detachment of the flow at the lateral walls (picture (a) of Figure 4.8). Then, for the case of three buildings in a row, a more narrow wake is observed because of the attachment of the flow at the sides of the buildings (picture (b) of Figure 4.8). It is also important to note that the re-circulation zones in Figure 4.8 is not that different than the ones compared in Figure 4.7, but this can be explained with the fact that the top view is taken from the middle plane of the building and the profile view shows that the re-circulation zone is lower than the half of the building height.



(a) Velocity profile for basic group of buildings



(b) Velocity profile for rounded group of buildings

FIGURE 4.8: Comparison on velocity profiles from top view for group of buildings.

4.3 Temperature profiles

Once the velocity distribution has been plotted, the next step is to represent the temperature distribution for each case. The main objective is to relate how the velocity gradient is linked to the temperature of the building, and therefore with the shape of the building. Throughout the following section, the temperature gradient between the airflow and the building will be analysed taking values of 20ºC at the building and values of 0ºC, 15ºC and 30ºC for the airflow.

The temperature analysis is going to be based on velocity because the convective term is the one which has more weight in the total heat contribution, since the building is located at an air-stream velocity. Therefore, the zones where the flow is accelerated a loss in temperature is expected. On the other hand, when the flow decelerates a slower gradient of temperatures will be observed. Moreover, zones with turbulent flow and vortexes are also expect to cause a decrease in temperature, since this type of regime requires more energy than the one unaffected by any fluctuation.

4.3.1 Airflow at 0ºC

The first case for the temperature analysis is the one of the basic building. Remembering the shape of the velocity distribution from section 4.2, it is expected to have a dissipation of energy on the top and the sides of the building with the same shape that the one of the accelerated flow, and it is what can be seen in Figure 4.9. Furthermore, following the shape that the one of the velocity, the temperature decreases slower when the flow is detached from the building, and this is the reason why at the roof and lateral walls there is a slow gradient of temperatures. Then, at the back of the building it can be observed a general cold temperature, due to the big re-circulation zone.

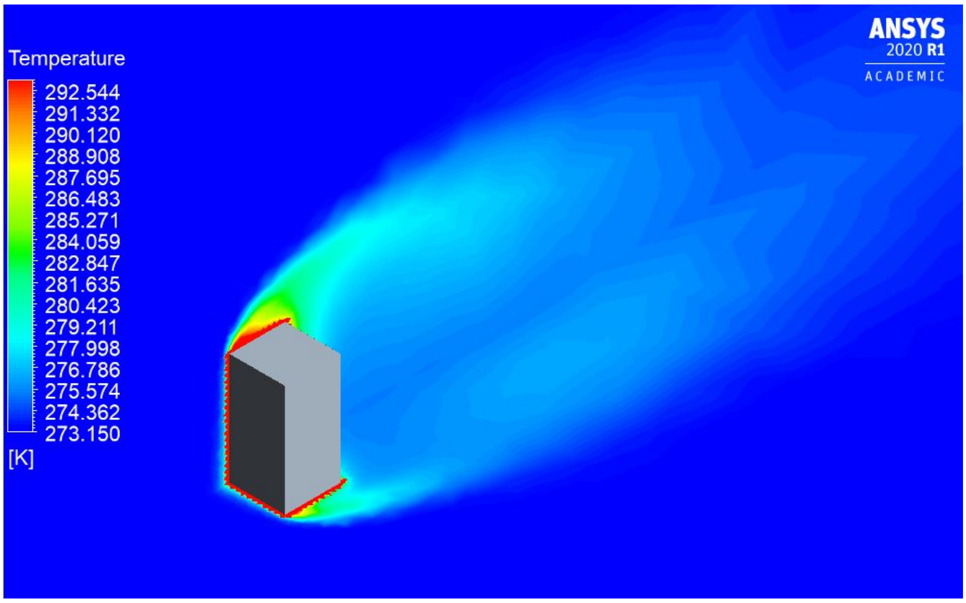


FIGURE 4.9: Temperature gradient for basic building.

The second case to be analysed is the rounded building. In this occasion, at the roof and the lateral walls there is a very small gradient, meaning that the transition from 20ºC to 0ºC takes place in a very quick way. However, at the back of the building the temperature is conserved showed by a slow gradient, due to the lack of a re-circulation zone. Therefore, in this case the building acts as a barrier to the air at the back of it, conserving in a better way the temperature. The commented features of the temperature distribution of the rounded building can be seen in Figure 4.10.

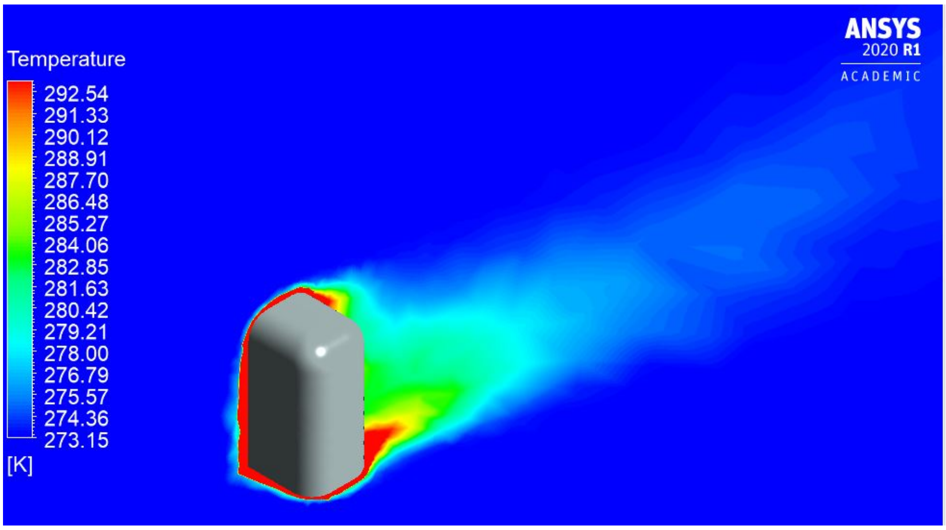
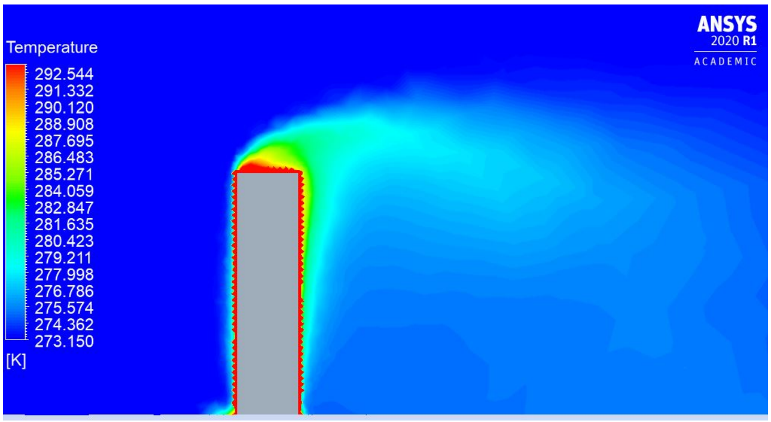
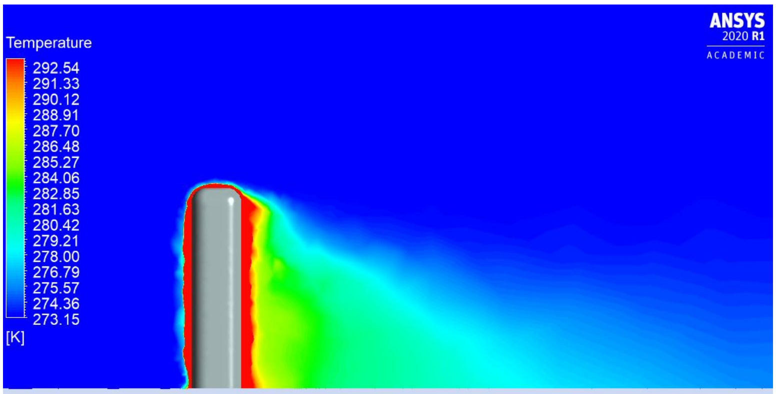


FIGURE 4.10: Temperature gradient for rounded building.

In order to complete the comparison between the two different buildings, it is very helpful to consider the profile distribution of temperatures shown in Figure 4.11. In the case of the basic building, the gradient of temperatures forms a wake between the accelerated flow at the top and the re-circulation flow at the back of the building. On the other hand, in the case of the rounded building the temperature wake is present all along the back of the building but is not present in the roof. Comparing both of them, the wake in the basic building is smaller and covers less surface of the building, while in the rounded case this wake is bigger, covering more square meters of the building. It is also important to note that the region where the hot temperature prevails (red zone around the building) is thicker in the rounded building than in the basic one.



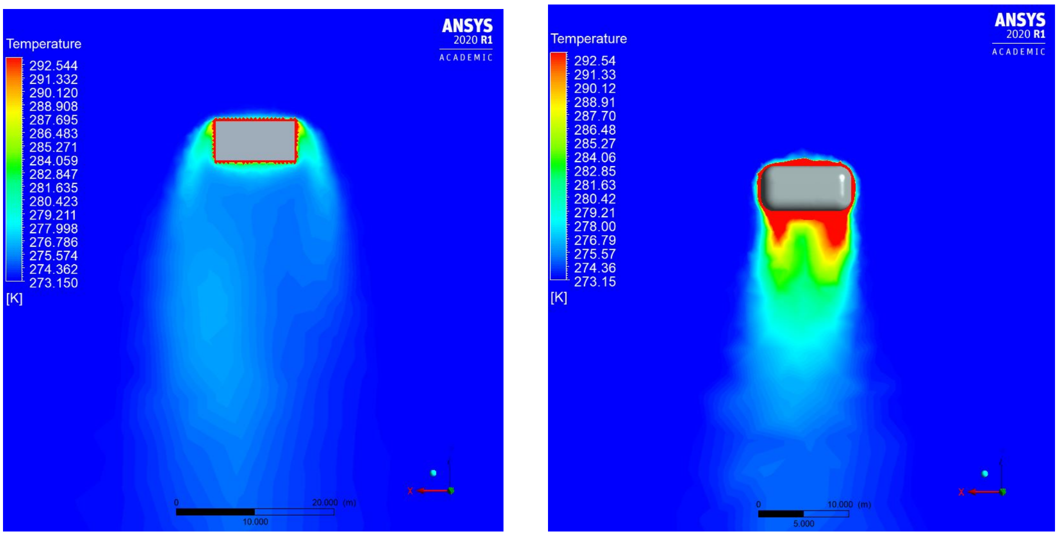
(a) Temperature profile for basic building.



(b) Temperature profile for rounded building.

FIGURE 4.11: Comparison on temperature profiles for single cases.

In order to make a more profound analysis, the top view will be very helpful too. In Figure 4.12 both top views have been plotted, showing the temperature gradient at the middle height of the building. For the basic case, the temperature gradient shows a small wake at both sides of the building, and confirms what has been previously said: at the back of the building the temperature decreases rapidly. In the rounded case, on the other hand, there is not temperature wake at the sides but a slow gradient at the back of the building. Again, the thickness of the red zone, representing hot temperatures, is greater for the rounded case than for the basic one.



(a) Temperature distribution basic building (b) Temperature distribution rounded building

FIGURE 4.12: Comparison on temperature profiles from top view for single cases.

Once the single cases have been analysed, the cases for three building in a row will be commented. Starting with the row of three basic buildings plotted in Figure 4.13, the first noticeable feature is that the gradient of temperatures at the back of the buildings has a slower transition towards colder temperatures. Regarding the top and side walls, the behaviour is very similar to the one from the single case, following the shape of the velocity of the flow.

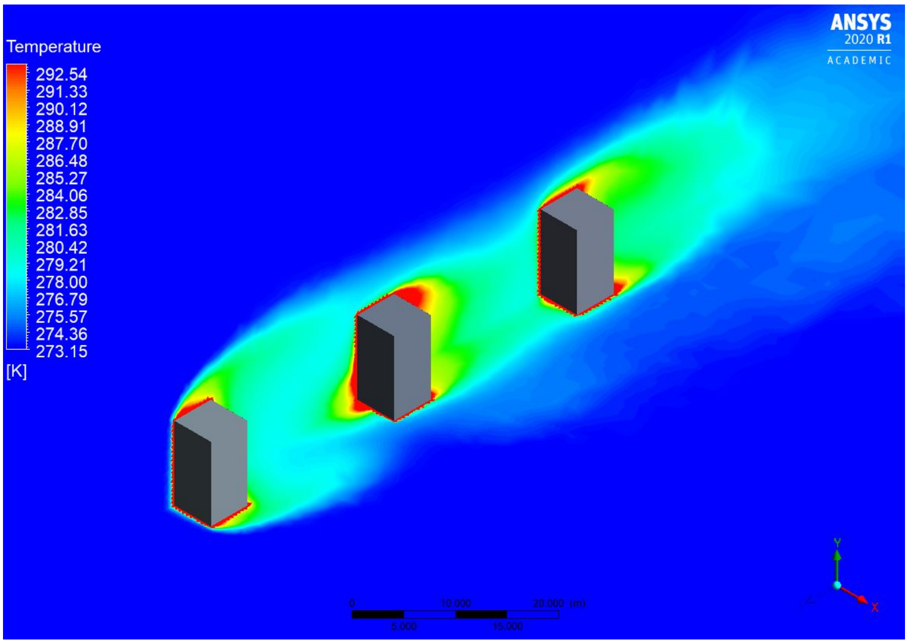


FIGURE 4.13: Temperature distribution on basic group of buildings.

When it comes to the row of three rounded buildings, the most noticeable feature in Figure 4.14 different from the single case is that the hot temperature at the back of the buildings reaches bigger distances than the single case. Regarding the top and side walls of the buildings the behaviour is the same, since recalling the velocity distribution the flow acts in the same way in all the three buildings.

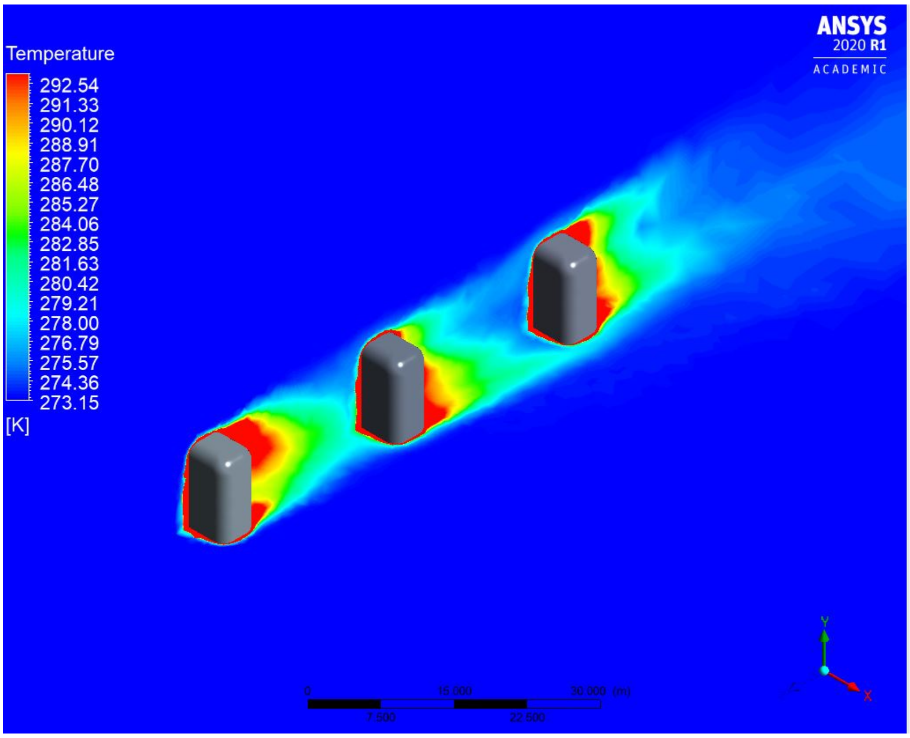
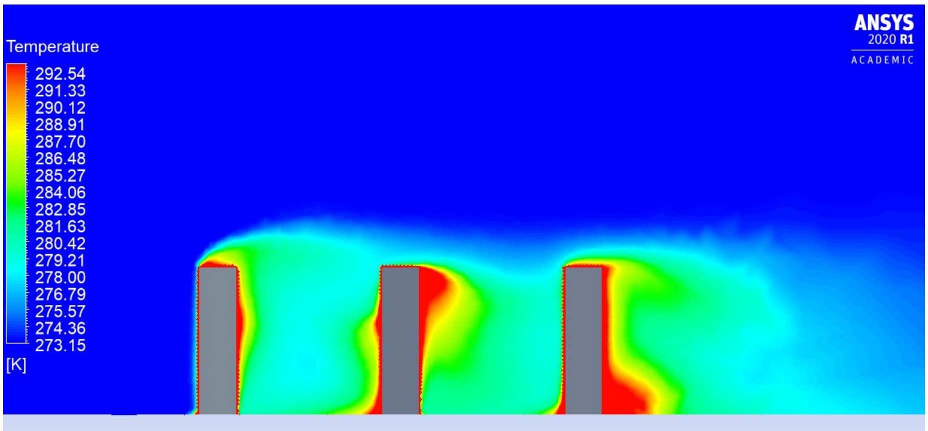
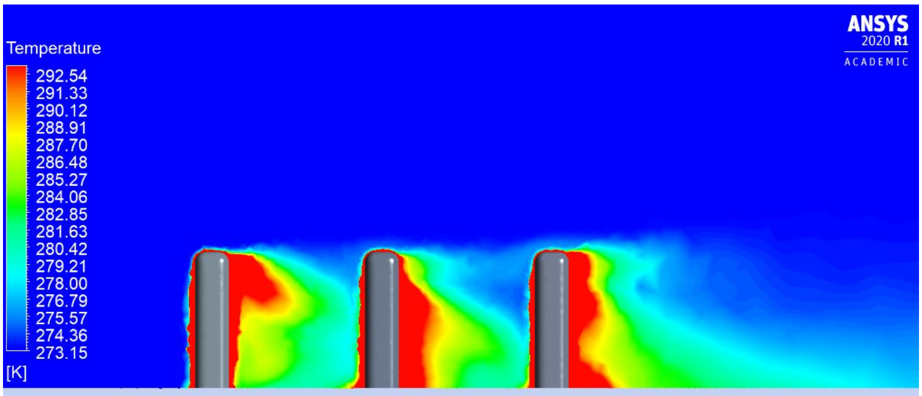


FIGURE 4.14: Temperature distribution on rounded group of buildings.

As done for the previous cases, the profile view for the temperature distribution has been plotted in Figure 4.16 because of its utility in the analysis. From this view it can be seen that the three basic buildings have improved their performance at the back when built together, since now a big red zone has appeared representing hot temperatures. On the other hand, the row of three rounded buildings has also increased the red zone but in a more efficient way, since this zone is bigger than the one for the basic buildings.



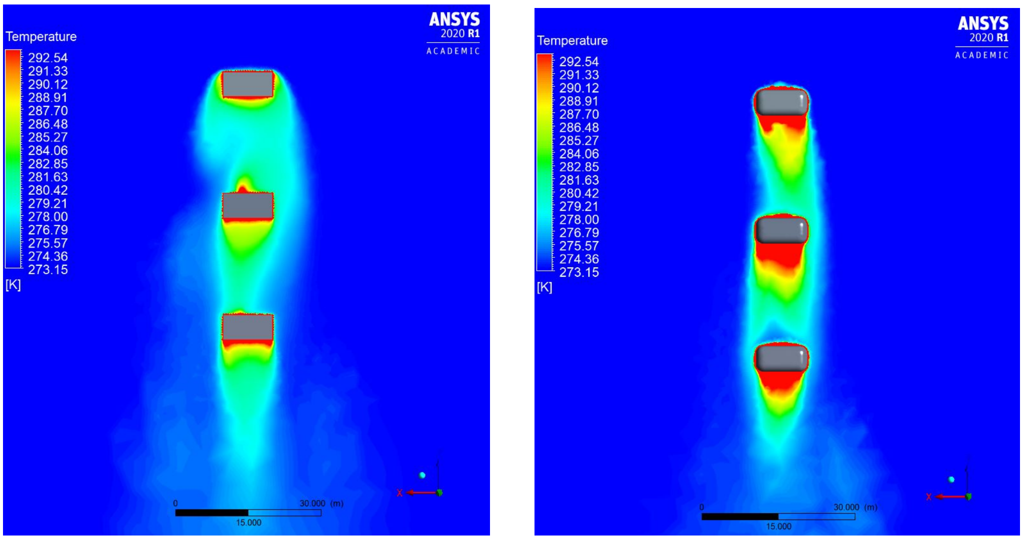
(a) Temperature profile on basic group of buildings.



(b) Temperature profile on rounded group of buildings.

FIGURE 4.15: Comparison on temperature profiles on group of buildings.

Finally, the case of group of buildings is analysed with help of the top view of the temperature distributions represented in Figure 4.16. In this case, for the group of basic buildings a wider wake can be observed, but the gradient this decreases in a slower manner compared to the single case. On the other hand, for the group of rounded buildings, the wake is more narrow but the high temperatures are better conserved next to the building walls.



(a) Temperature profile on basic group of buildings.

(b) Temperature profile on rounded group of buildings.

FIGURE 4.16: Comparison on temperature from top view for group of buildings.

As a general conclusion, when applying a temperature of 20ºC between the internal and external flow, rounded buildings are able to maintain a thicker zone around them where the temperatures are higher compared to squared ones. Also, when placing a row of buildings one behind another, an immediate improvement on the temperatures is achieved since it helps avoiding re-circulations of air that require energy. However, more and deeper conclusions will be done in Chapter 5, considering all the cases.

4.3.2 Airflow at 15ºC

In this occasion the case that is going to be analysed is the one when the airflow takes a temperature of 15ºC. The aim of this analysis is to study the behaviour of the building temperature when a smaller difference of temperatures is imposed between the external and internal flows.

When applying an airflow with speed of 1 m/s and temperature 15ºC to a squared building that is heated to have a constant temperature of 20ºC, a temperature wake is observed. The shape of the temperature distribution follows the velocity field shape as expected, but this time the wake is smaller due to the smaller temperature difference, which makes easier the transition to colder temperature for the internal flow. Another important feature is the gradient at the roof of the building, which in this case is parallel to the wall but in the previous case a bubble was formed. In this case a regular gradient is formed due to the lack of energy, since there is only a difference of 5ºC. In Figure 4.17 the plot of the temperature distribution for the simple building can be found.

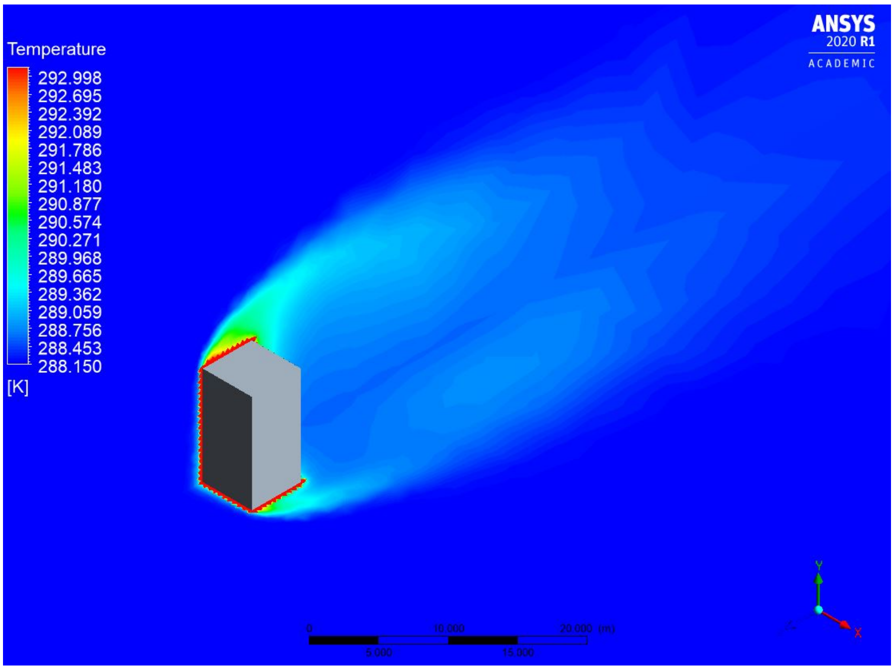


FIGURE 4.17: Temperature gradient for basic building.

The second case to be analysed is the building with rounded corners. In Figure 4.18 the temperature gradient can be observed to be smaller than the one in the previous section, meaning that the transition towards the outer temperature is faster. This increase in the transition velocity can be seen in the temperature wake at the surroundings of the building.

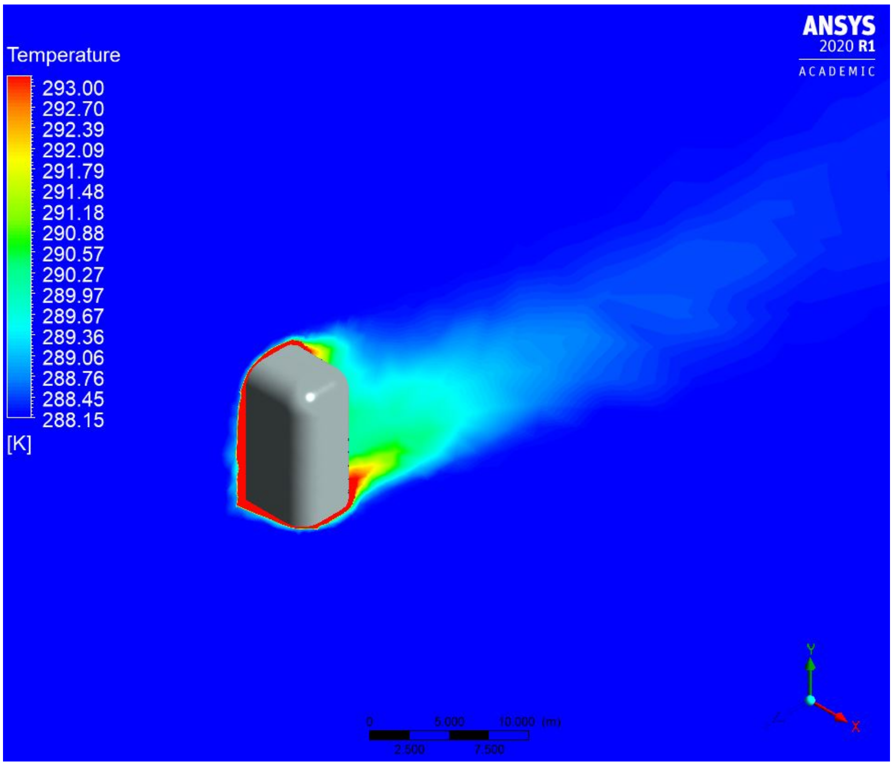
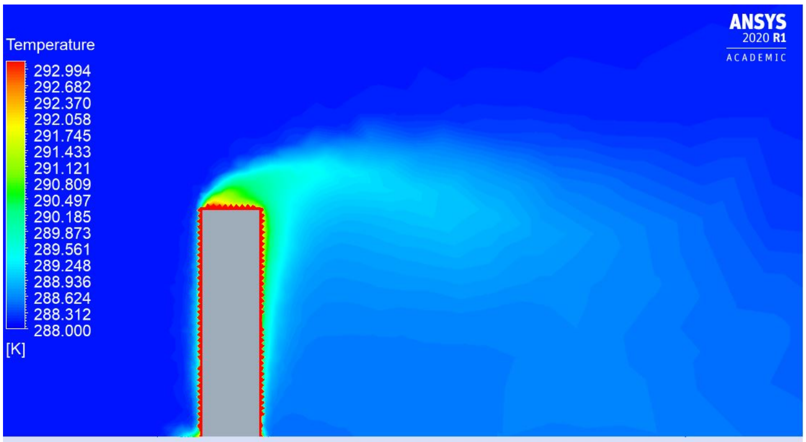
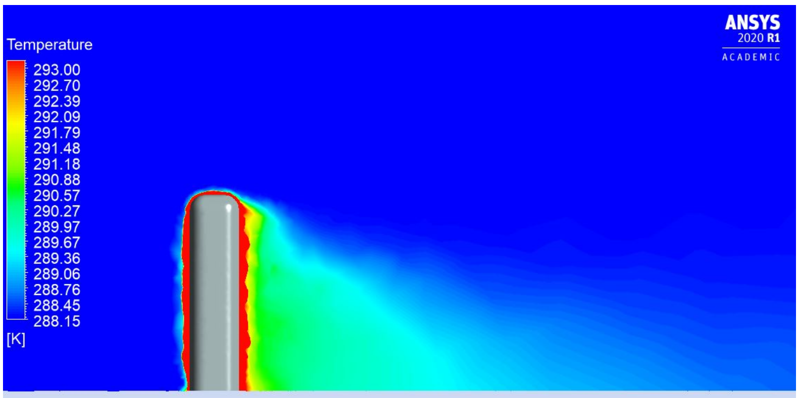


FIGURE 4.18: Temperature gradient for rounded building.

In order to complete the study of this case, it is very useful to plot the side and top views so that it is possible to observe the shape of the temperature wake. In first place, in Figure 4.19 it can be found the side view, which shows a very clear temperature wake for both cases, basic and rounded building. For the basic building, the wake is spread towards the superior zone of the back of the element because this is the zone where there is less re-circulation and velocity. Also, it is noted that the red zone throughout all the profile of the building, representing the temperature of 20ºC, is very uniform and with constant thickness. On the other hand, for the rounded building the temperature wake is spread in the lower zone closer to the ground and is bigger in comparison with the one of the basic building. Also, the thickness of the red zone is greater, which means that the high temperatures prevail more time in this case.



(a) Temperature profile for basic building.

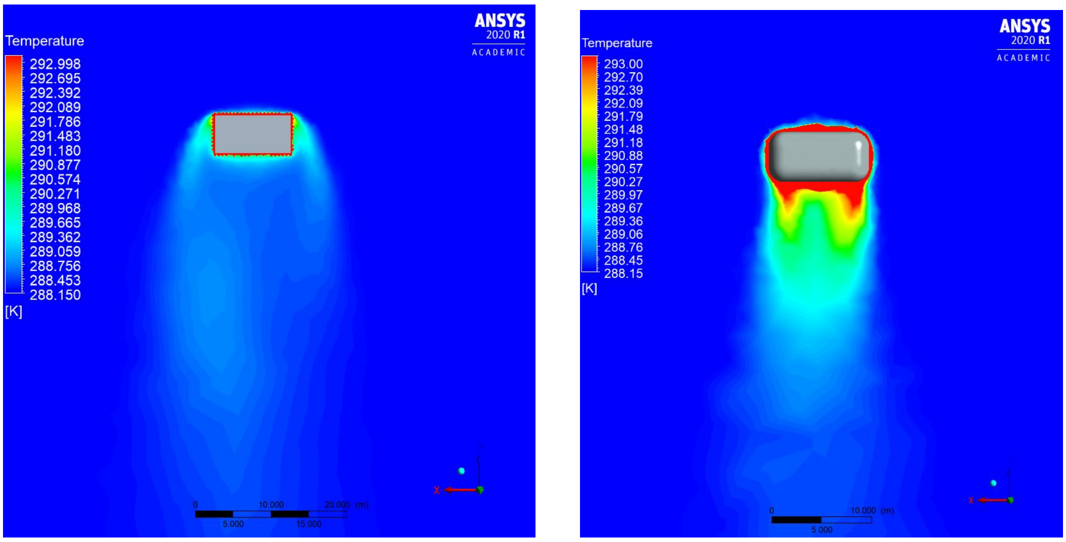


(b) Temperature profile for rounded building.

FIGURE 4.19: Comparison on temperature profiles for single cases.

In second place, the top view of the temperature wake at middle height for both buildings has been represented in 4.20. Comparing both cases there is a clear difference on the size of the temperature gradient: on the basic building two small wakes are formed at both sides but fade rapidly and almost no wake can be seen at the back, but for the rounded building a big wake is formed at the back of the building.

From this view it is also clear the difference on the thickness of the red zone, which for the basic case is about 0.5m and for the rounded case is around 0.7m (increase of 40% with respect to the basic building)



(a) Temperature distribution basic building.

(b) Temperature distribution rounded building.

FIGURE 4.20: Comparison on temperature profiles from top viewfor single cases.

The same temperature study has been carried out for the case of a row of three buildings. In Figure 4.21 the temperature distribution for the case of three basic buildings has been plotted. It represents the transition from 20ºC in the building towards 15ºC in the external flow. This time the wake generated is bigger, meaning that the zone where the hot temperatures prevail is bigger. The fact that the buildings are disposed in a row helps maintaining the temperature since the re-circulation zones are moderated and the first building acts as a barrier for the other two, letting the red zone increase specially at the back.

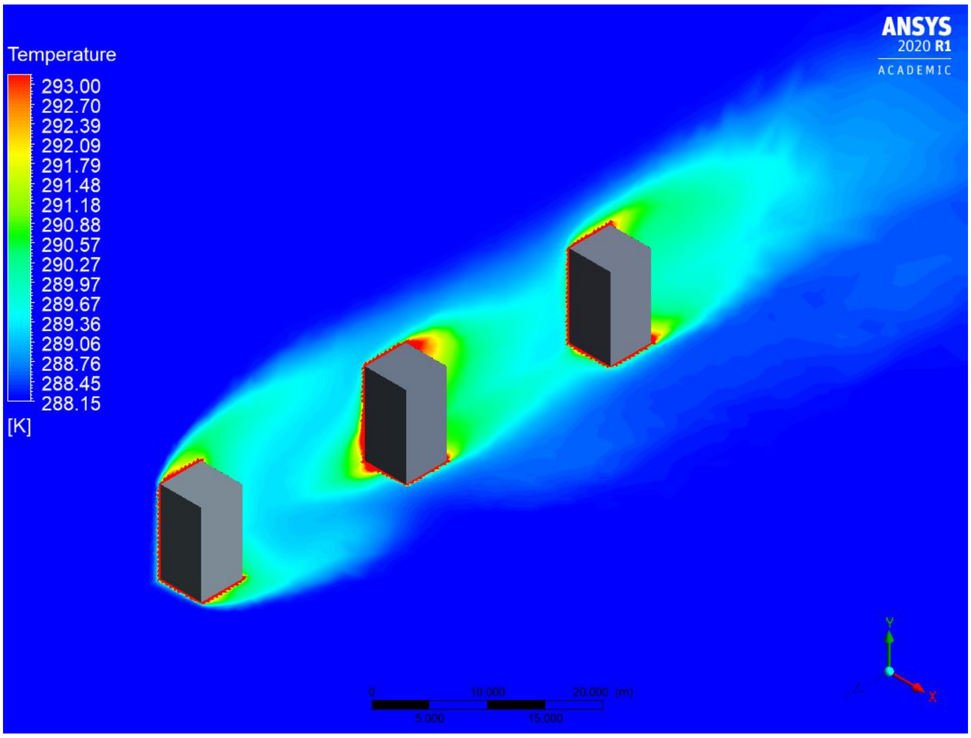


FIGURE 4.21: Temperature distribution on basic group of buildings.

When plotting the case of three rounded buildings in a row in Figure 4.22 it can be observed that the temperature gradient does not change significantly in comparison with the single case. This fact is due to the good attachment of the flow when having a disposition of rounded buildings in a row, avoiding big re-circulation zones and detachment at the top of the buildings. However, these assumptions must be confirmed by the top and side views of the temperature profiles.

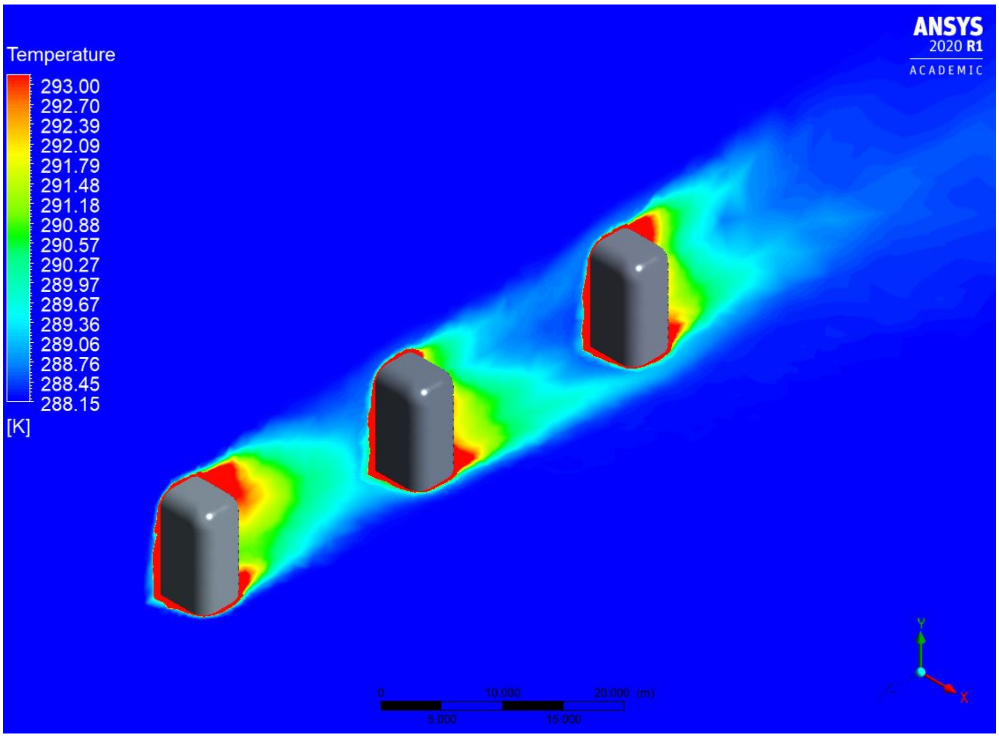
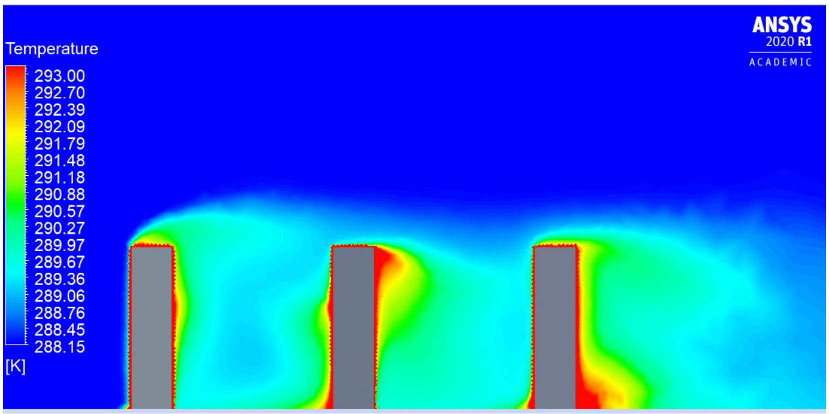
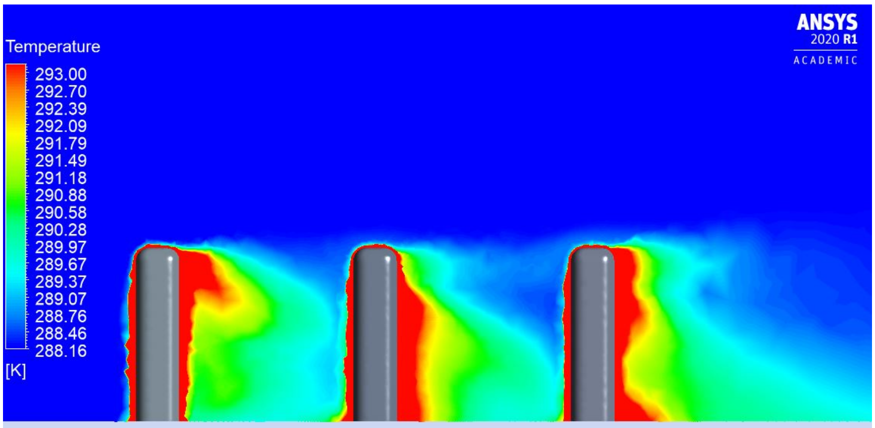


FIGURE 4.22: Temperature distribution on rounded group of buildings.

Again, in Figure 4.23 and 4.24 the side and top views of the temperature gradients have been represented in order to know the complete behaviour of the wake. In the basic case, the temperature gradient between the buildings has become slower (temperature does not drop down suddenly) and in some points the red layer takes greater thickness. On the other hand, the rounded case has the same shape of the wake but the red zone is increased, making bigger the zone with hot temperatures and being possible to have the same temperature distribution with less heat generation rate. The main difference between the basic and rounded case is that in the first one there is a wake originated from the roof of the building, when in the second case there is not any wake coming from here.



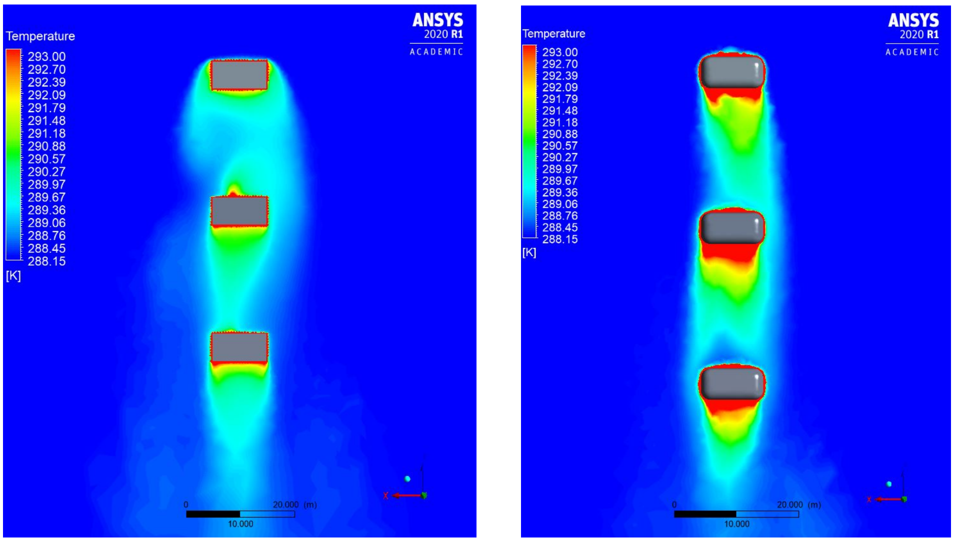
(a) Temperature profile on basic group of buildings.



(b) Temperature profile on rounded group of buildings.

FIGURE 4.23: Comparison on temperature profiles on group of buildings.

Finally, in the top view on the basic case the phenomena where the first building acts as a barrier can be clearly seen, since the wake of this element is wider to the ones of the second and third buildings. However, the temperature experiences a slower transition for all the buildings, thus improving the overall thermal efficiency. Even more, in the last building an increase on the red layer can be found. In the rounded case the same behaviour is observed: the effect of the airflow is uniform in each building, having a straight temperature wake and an increase of the red layer in the three elements.



(a) Temperature profile on basic group of buildings.

(b) Temperature profile on rounded group of buildings.

FIGURE 4.24: Comparison on temperature from top view for group of buildings.

4.3.3 Airflow at 30ºC

The last case to study consists on having an external temperature of 30ºC, while the internal flow is supposed to have a constant temperature of 20ºC. In this occasion, the performance of each type of building will be analysed in order to characterise its behaviour when the hot temperatures are outside and the objective is to keep cold temperatures inside the building.

The basic building with squared corners has been exposed to this airflow with a velocity of 1 m/s and the behaviour is very similar to the one when having cold temperatures for the external flow. This time the wake goes from a blue zone representing the cold temperature, adjacent to the walls of the building, towards a red colour representing the hot temperatures of the airflow. In between these two colours, the temperature takes place, and it follows the shape of the velocity profile. This temperature gradient is extended from the top and side walls towards the direction of the airflow, but it is not present right at the back of the building due to the big recirculation zone. The plot of this temperature distribution can be found in Figure 4.25.

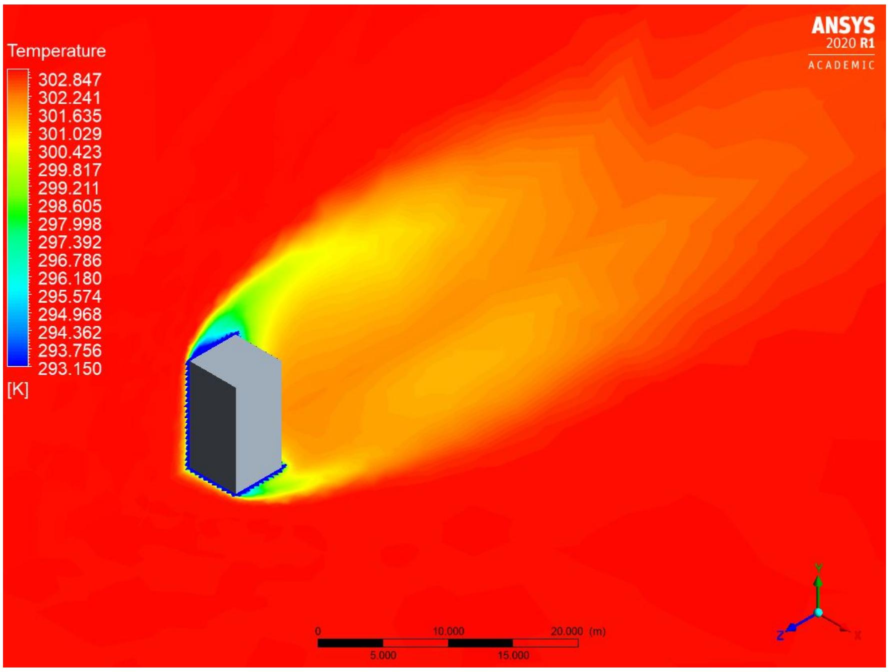


FIGURE 4.25: Temperature gradient for basic building.

For the rounded case it has also been represented the temperature distribution for the external temperature of 30ºC and it can be found in Figure 4.26. Here, the temperature wake is extended at the back of the building and with lower temperatures. Also, the blue layer surrounding the building is thicker, meaning that the 20ºC are kept in an easier way.

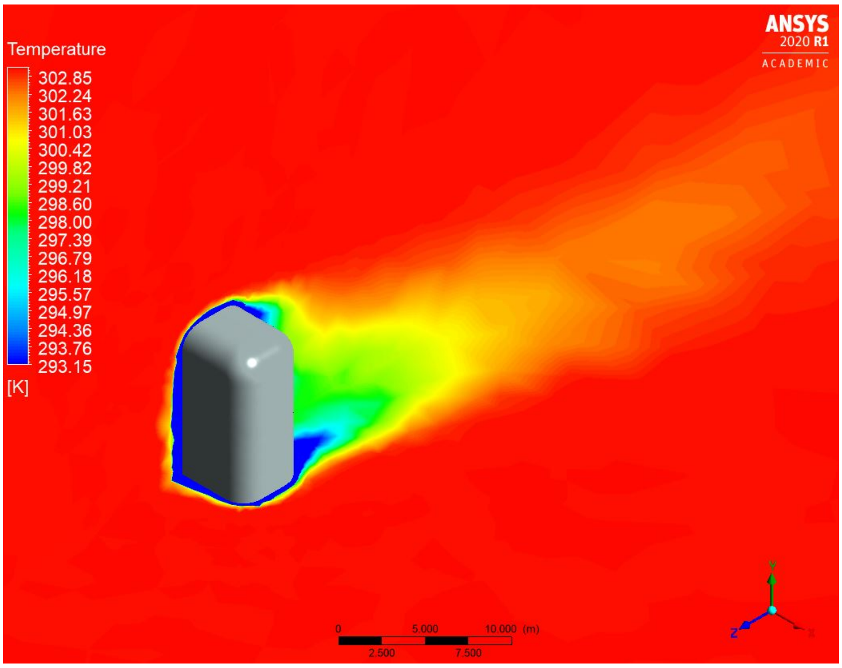
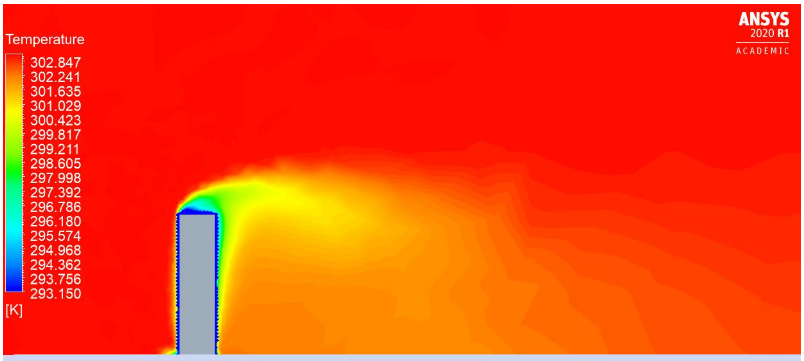
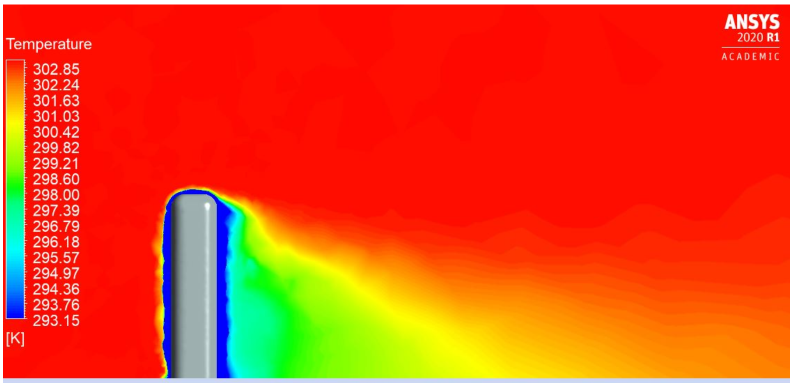


FIGURE 4.26: Temperature gradient for rounded building.

In order to study the complete behaviour, in Figure 4.27 the side view of the temperature wake of the two cases has been plotted. With this view it is observed the size difference of the temperature gradient between the basic and rounded building. In the basic building the wake generated is extended up to 25m downstream, but for the rounded building the wake is extended up to 42m. The other main difference is that in the basic building the wake follows the path of the zones where the velocity is lower and there are no vortexes and this is in the top of the building, where the flow detachment takes place, but in the rounded building this zone is at the back of the building, since at the top the flow is accelerated and at the back it does not experience a big re-circulation zone.



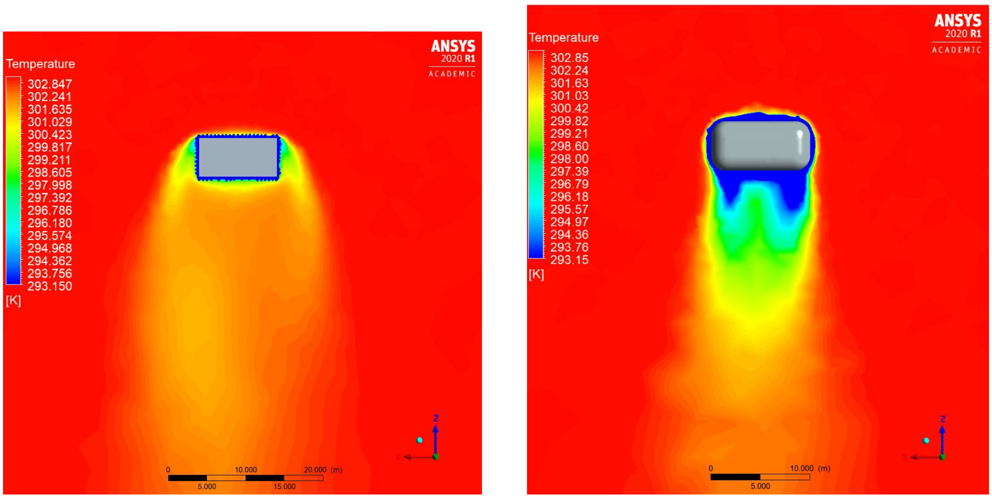
(a) Temperature profile for basic building.



(b) Temperature profile for rounded building.

FIGURE 4.27: Comparison on temperature profiles for single cases.

The top view also helps to characterise the temperature wake, and it is shown in Figure 4.28. From this view, the feature that at the back of the basic building there is no wake can be clearly seen, and in the rounded one all the wake happens right at the back. Also, looking at both cases, the difference on the width of the wake can be spotted: the wake of the basic building is extended in the sides creating a wake of 21m, while in the rounded building the wake has the same width as the building itself, 10m.



(a) Temperature distribution basic building

(b) Temperature distribution rounded building

FIGURE 4.28: Comparison on temperature profiles from top view for single cases.

In all the Figures analysed, the thickness of the blue layer in the rounded building is greater than the basic one, indicating that with a decrease on the heat generation rate the same effect would be achieved. Therefore, the rounded building needs less energy than the basic one.

The last step to complete the study is to plot the temperature gradient for the cases of three buildings in a row. In Figure 4.29 the row of three basic buildings is exposed to the external flow with temperature of 30ºC, and an increase in size of the wake is observed: the first building protects the other two avoiding the dissipation of heat and the re-circulation zone is softened, so there are no significant temperature losses in the gaps between buildings. Also, due to the protection of the second and third buildings, the temperature gradient in the front of the building is not as affected as for the first one, which is directly exposed to the external flow.

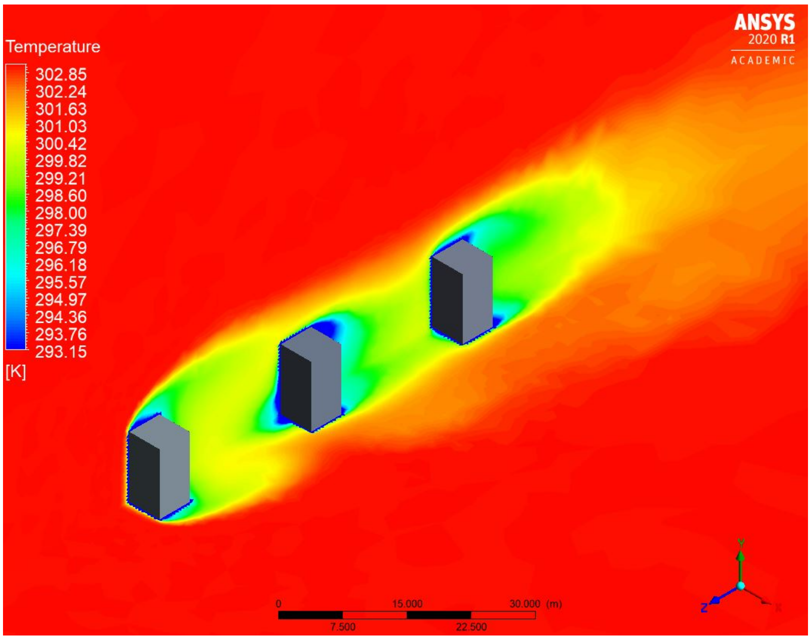


FIGURE 4.29: Temperature distribution on basic group of buildings.

In the case of the rounded building, the re-circulation behind each building is almost the same because the flow attaches in a better way, so the temperature distribution is very similar to the one of the single case: at the top of the building the temperature is dissipated due to the acceleration of the external flow and at the back the temperature is slowly dissipated. In this case, the front walls of the second and third buildings are more protected, thus increasing the blue layer thickness. In Figure 4.30 all these features can be observed.

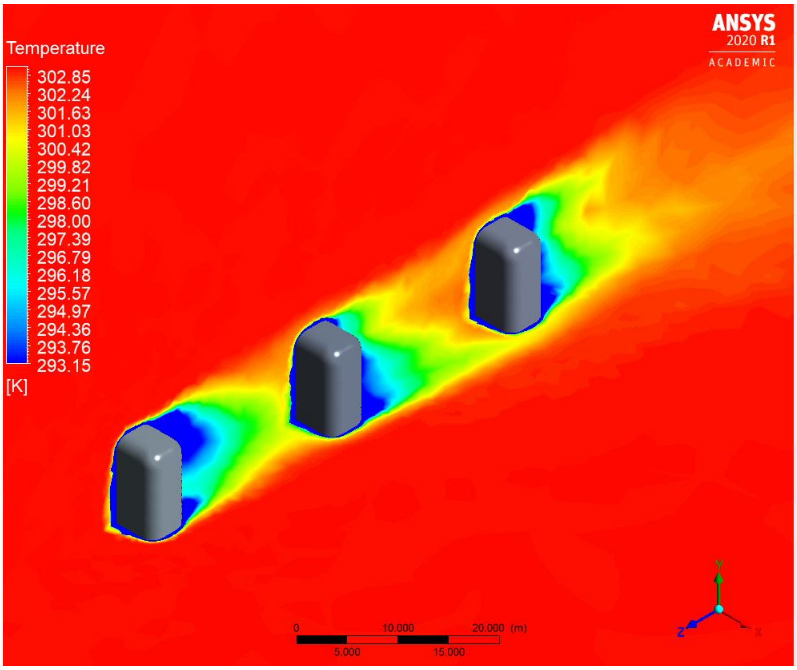
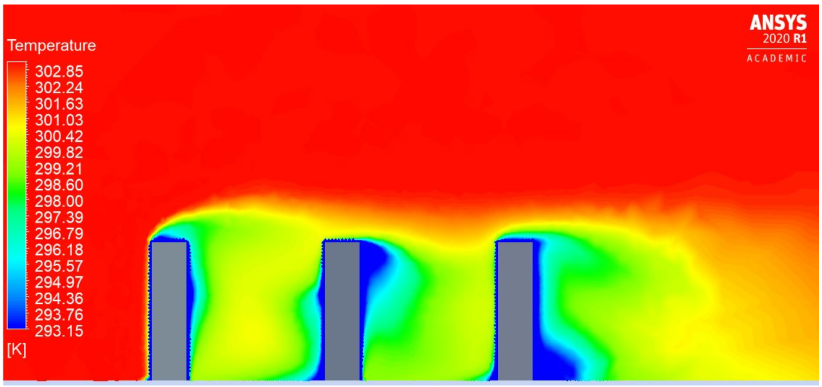
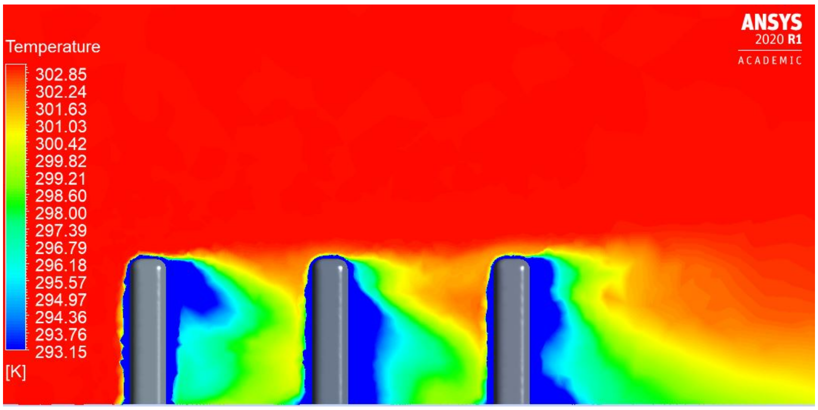


FIGURE 4.30: Temperature distribution on rounded group of buildings.

Plotting the side view of the different groups of buildings is very useful in order to know the shape of the wake behind the buildings. In Figure 4.32 it can be seen how the wake of the basic buildings has increased in size and the re-circulation zones that previously dissipated all the heat now have less kinetic energy, so the temperatures behind each building are lower, meaning that the cold temperatures are better preserved. On the other hand, in the group of rounded buildings the most important feature is that the blue layer is greater than the one of the basic case, and it is also greater than the one of the single case. However, the wake has still the same size and cold temperatures are completely dissipated from the top of the buildings.



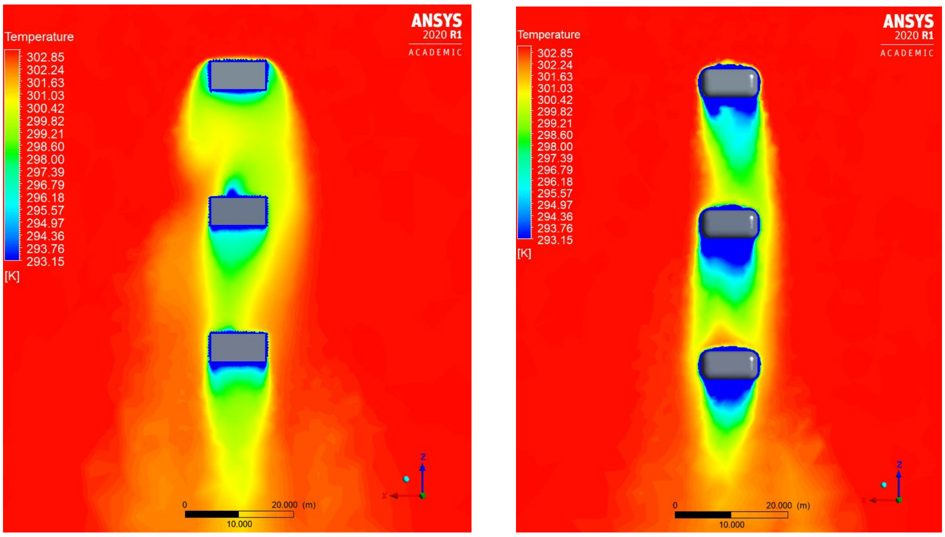
(a) Temperature profile on basic group of buildings.



(b) Temperature profile on rounded group of buildings.

FIGURE 4.31: Comparison on temperature profiles group of buildings.

Finally, the top view of the temperature distribution has been plotted in Figure 4.32 with the aim of showing how the wake is generated in this plane. In the basic case it can be observed that the first building produces a wider wake than the ones of the second and third buildings, that is explained by the fact that in the first element the flow detaches and then the flow velocity in the second and third elements is lower, making possible the adaptation of the flow in the walls. On the other hand, the rounded buildings present a wake which width is equal to the width of the building. Moreover, when interacting the buildings with each other what is achieved is an increase on the blue zone, having in this way a greater layer where the temperature of 20ºC is preserved.



(a) Temperature profile on basic group of buildings.

(b) Temperature profile on rounded group of buildings.

FIGURE 4.32: Comparison on temperature from top view for group of buildings.

Chapter 5

Conclusions

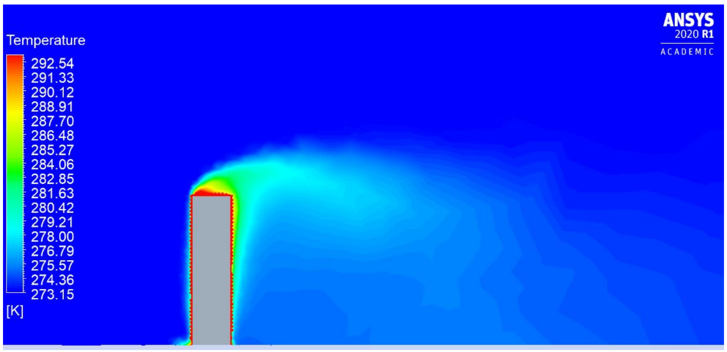
Throughout this report it has been presented a specific problem to be solved and the results have already been shown. In this section, the conclusions of this results as well as the main objective of the thesis will be gathered, with the aim of determining the performance of each type of building or configuration.

5.0.1 Comparisons

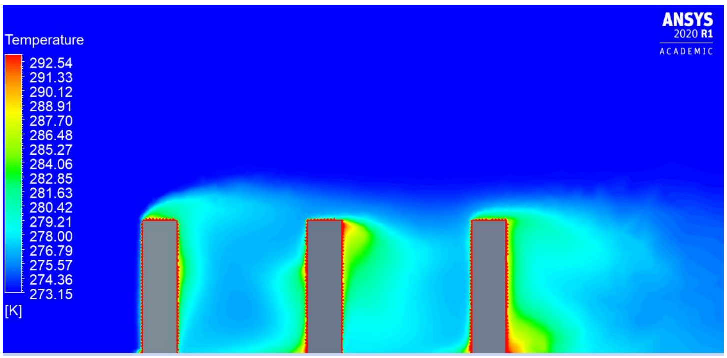
Single building and group of buildings

The first comparison that is going to be done is the one on the performance between a single building and a group of buildings. In all the results exposed in Chapter 4 the cases of a group of three buildings in a row have shown a more consistent distribution of the temperature in terms that they are able to maintain the desired temperature. This is due to the fact that when two or more elements are placed in a row they interact with each other and reduce the re-circulation zones and vortexes. In both cases, basic and rounded, the first building is the one that is most affected by the airflow, being the one that is exposed directly to the temperatures of the flow. This creates a barrier for the other two buildings, making possible a good distribution of temperatures. Moreover, the presence of three buildings with a gap between them makes possible that the re-circulation is decreased, so there are less heat losses at the back of the buildings.

In order to prove that the buildings are more thermally efficient when placed in groups than single, in Figure 5.1 has been represented the temperature distribution when the single building has a heat generation rate of 50 W/m^3 and each building forming the group of buildings has a heat generation rate of 30 W/m^3.



(a) Temperature distribution for basic building.



(b) Temperature distribution for basic group of buildings.

FIGURE 5.1: Comparison on temperature profiles with different heat generation rates.

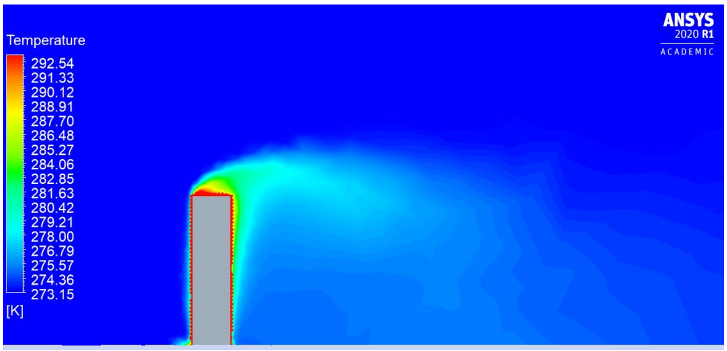
In Figure 5.1 it can be seen that the red layer has the same thickness in the single case and in the group case, but in this last one the heat generation rate is lower. Therefore, it has been shown that with a row configuration the energy needed to heat the entire building can be reduced up to 40%.

Basic building and rounded building

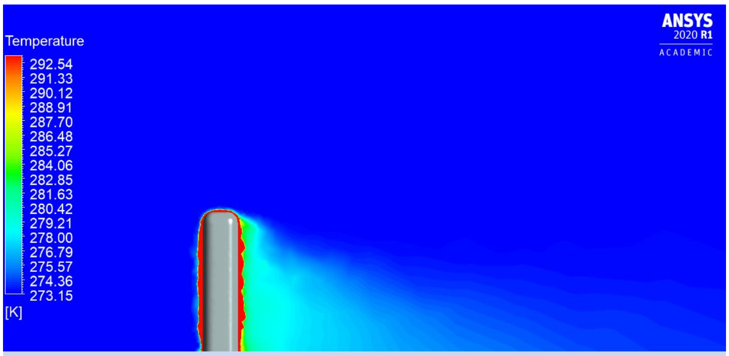
The second comparison that is going to be carried out is the one between the basic building and the rounded building. With the results obtained in Chapter 4 it has been shown that in all cases the basic building favoured the dissipation of heat, while the rounded building is able to keep a uniform temperature in its surroundings.

Recalling the velocity distributions for each building in their single case, the basic building causes a detachment of the flow at the top and the side walls, and due to its poor aerodynamic shape a high-energetic re-circulation zone is created at the back of the building, and the heat loss due to convection increases. Therefore, in a squared building there will always be higher heat losses at the lower part and the front face. However, the rounded building has presented in all the studied cases a very consistent temperature wake at the back of the building, while at the top and the side walls do not present any kind of wake due to the good attachment of the flow, that accelerates all along the walls. Therefore, in the rounded case the heat losses will occur in the roof and the side walls, while at the back the temperature is conserved. Also, it has been noticed that the thickness of the layer that represents the desired temperature is up to 40% greater than in basic buildings.

In conclusion, rounded buildings are more efficient thermally speaking because they will need less power on the heating than basic ones in order to reach the desired temperature. Also, due to their more aerodynamic shape, zones with high kinetic energy are avoided (such as vortexes or re-circulation zones) and this means that there will be less heat losses. In order to prove this, the same simulation as in the previous section has been carried out, but this time comparing the two single buildings, squared and rounded.



(a) Temperature distribution for basic building.

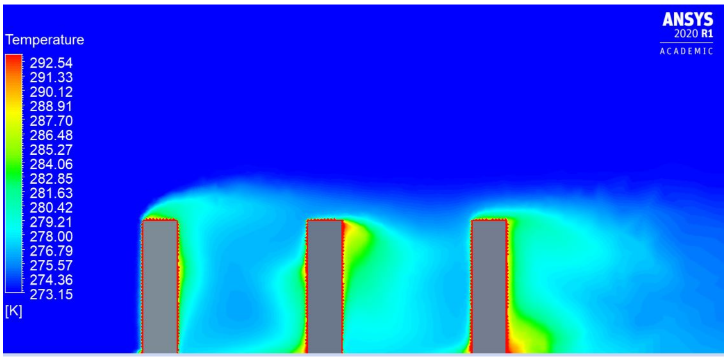


(b) Temperature distribution for rounded building.

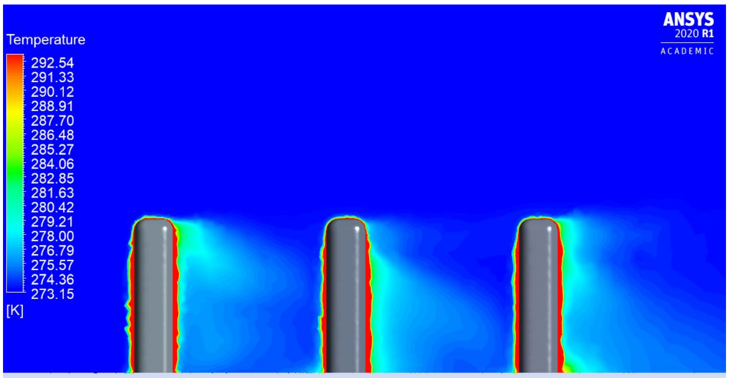
FIGURE 5.2: Comparison on temperature profiles with different heat generation rates for single cases.

In Figure 5.2 the basic building has a heat generation rate of 50 W/m^3, while the rounded building has a heat generation rate of 25 W/m^3. They both have similar red layers around the walls of the building, what means that when using rounded buildings the energy needed on heating or cooling can be reduced up to a 50% without changing the temperature inside the building.

It is also possible to compare both groups of buildings. In Figure 5.3 the group of basic buildings and the group of rounded buildings have been exposed to an airflow and each one has a different minimum generation rate in order to maintain the temperature of 20ºC in the inside.



(a) Temperature distribution for basic group of buildings.



(b) Temperature distribution for rounded group of buildings.

FIGURE 5.3: Comparison on temperature profiles with different heat generation rates for groups of buildings.

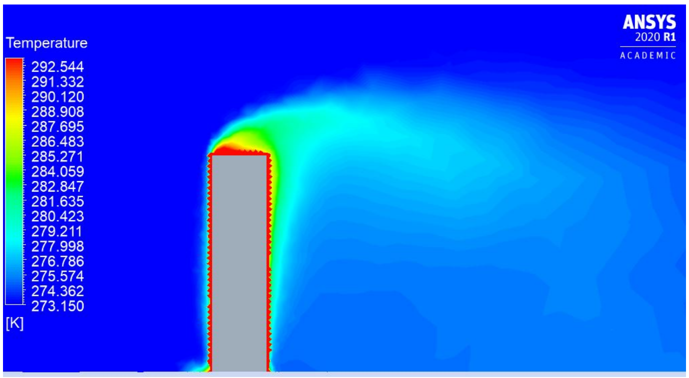
The heat generation rate needed in the group of basic buildings is, as previously said, 30 W/m^3, and the heat generation rate for the group of rounded buildings is 10 W/m^3. In group, the rounded buildings can save up to 66% in energy with respect to the squared ones.

External temperatures of 0ºC, 15ºC and 30ºC

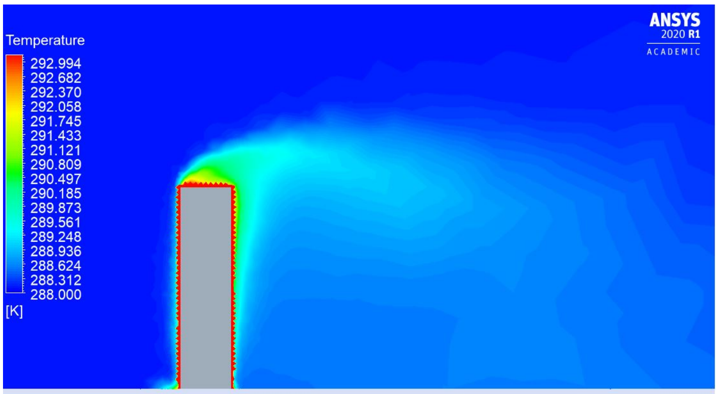
This last comparison has the objective to evaluate if the performance of each type of building when having different external temperatures is the same. In order to do so, the different plots on temperature distribution for each case will be gathered and compared.

In the case of the basic single building, the behaviour of the temperature wake is the same in the three cases with very small differences in the size of the wake downstream. These differences are almost not noticeable, but comparing the wakes on (a), (b) and (c), the first one is a bit more bigger than the other two, and the one of (b) is the shortest one. This makes sense, since the temperature differences are

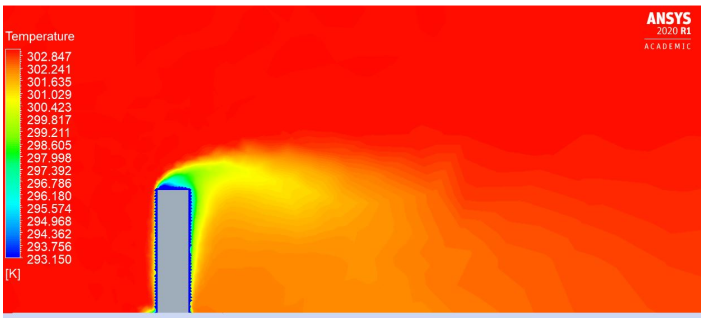
20ºC for (a), which has the biggest wake, 5ºC for (b), that has the smallest wake, and 10ºC for (c), which size is in the middle of the other two. Therefore, when having a higher temperature difference the transition towards the external temperature will need more energy and there will create a bigger wake.



(a) Temperature distribution with external flow at 0ºC.



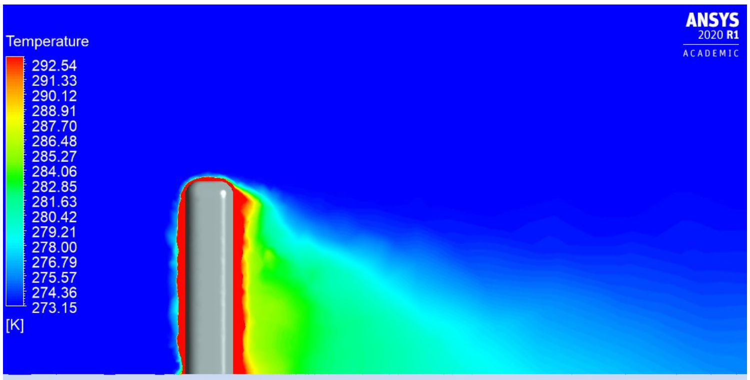
(b) Temperature distribution with external flow at 15ºC.



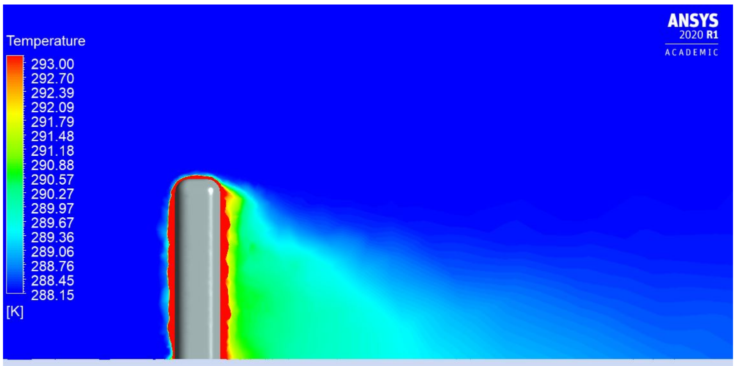
(c) Temperature distribution with external flow at 30ºC.

FIGURE 5.4: Comparison on temperature profiles with different external temperatures for basic building.

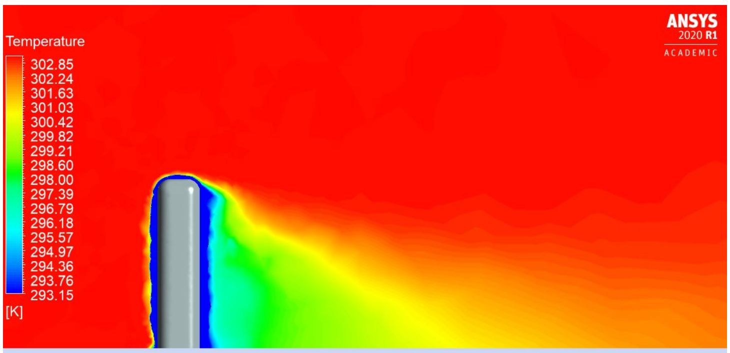
The same analysis has been carried out in Figure 5.5 for the rounded building. The wake in (a) happens to be the one with more length, then (b) is the shortest one and (c) has a middle length between the other two.



(a) Temperature distribution with external flow at 0°C.



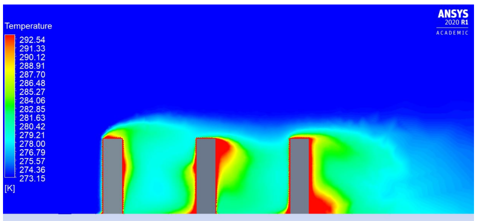
(b) Temperature distribution with external flow at 15°C.



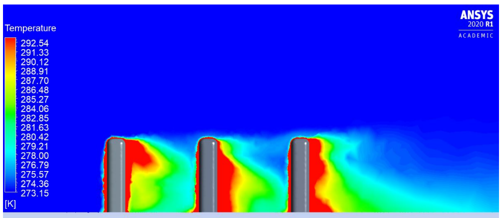
(c) Temperature distribution with external flow at 30°C.

FIGURE 5.5: Comparison on temperature profiles with different external temperatures for rounded building.

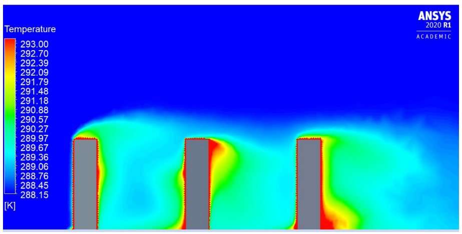
Let's try to make the same analysis for the different groups of buildings. In Figure 5.6 the pictures on the left represent the group of basic buildings (a, c and e) and the pictures on the right represent the group of rounded buildings (b, d and f).



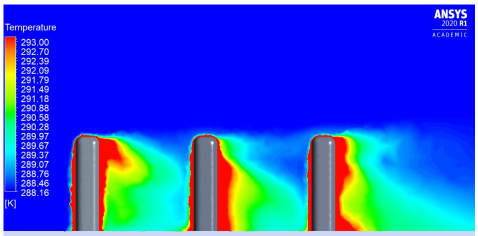
(a) Temperature distribution with external flow at 0ºC.



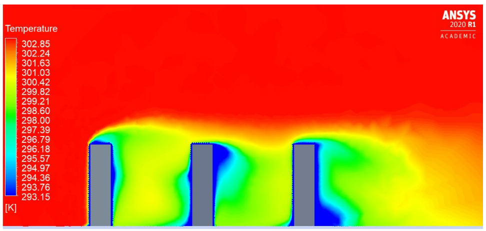
(b) Temperature distribution with external flow at 0ºC.



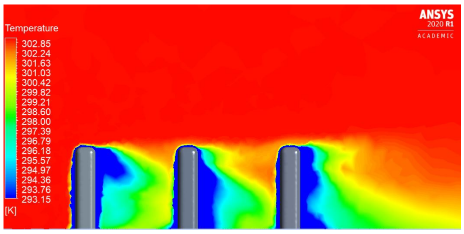
(c) Temperature distribution with external flow at 15ºC.



(d) Temperature distribution with external flow at 15ºC.



(e) Temperature distribution with external flow at 30ºC.



(f) Temperature distribution with external flow at 30ºC.

FIGURE 5.6: Comparison on temperature profiles with different external temperatures for groups of buildings.

In the case of basic buildings (pictures on the left) there is not a clear differentiated wake for each building, so the main differences that can be spotted are in the gaps between the buildings. In the first case, the temperature in the middle point of the gap is predominantly 6ºC in the first space and 9ºC in the second space, having a temperature difference with respect to the external flow of 6ºC and 9ºC respectively. In the second case, the predominant temperature in the first space is 16ºC and 17ºC in the second space, having a temperature difference with respect to the external flow of 1ºC and 2ºC respectively. And in the third case, the temperature in the middle point of the first space is 28ºC and in the second space is 25ºC, having a temperature difference with respect to the external flow of 2ºC and 5ºC respectively. Therefore, in this case the same conclusion can be made: as the temperature difference between the external and internal flows is higher, the temperature transition will be slower but also more energy will be needed to heat the building.

Looking at the case of rounded buildings the same behaviour can be observed: in picture (a) the red zone at the back of the buildings is the bigger one since the temperature difference is greater, in picture (b) the smallest red zone is observed because the temperature difference is the lowest, and in picture (c) an intermediate case can be found.

In conclusion, these images confirm what was expected: when having high temperature differences between the external and internal flows, more energy will be needed to heat or cool the building.

5.0.2 Future applications

This project has been very useful in order to know the behaviour of the temperature around buildings exposed to an airflow, and it has a lot of potential to be used in real life cases. CFD makes possible the analysis of temperature in order to have a rough idea of how the energy consumption and thermal efficiency will be in a building and lets the engineer study different possible designs according to the generated temperature gradients.

Although in this report the studied buildings are very simple designs, future applications can take into account more complex shapes that present new challenges. The possibility of using the wind as an energy source is also very interesting and could be studied with CFD, since the generation of clean energies could also help to decrease the cost of the heating or cooling of a building.

Bibliography

[1] Frank, P. and Dewitt, David P. *Fundamentals of heat and mass transfer*. México: Prentice Hall, 1999.

[2] Blazek, J. *Computational Fluid Dynamics*. Germany: Butterworth-Heinemann, 2015.

[3] Stolarski, T., Nakasone, Y. and Yoshimoto, S. *Engineering analysis with Ansys software*. 2018.

[4] Bhaskaran, R. and Collins, L. "*Introduction to CFD basics*". In: *Cornell University* (2015).

[5] Montazeri, H. and Blocken, B. "*New generalized expressions for forced convective heat transfer coefficients at building facades and roofs*". In: *Elsevier science* (2017).

[6] Stathopoulos, T. and Baniotopoulos, C.C. *Wind effects on buildings and design of wind-sensitive structures*. Italy: CISM, 2007.

[7] Ansys. *Ansys guide*. https://www.afs.enea.it/project/neptunius/docs/fluent/html/ug/node167.htm. Accessed on 2020-07-01. Jan. 2009.

[8] Engineering ToolBox. *Air - Density, Specific Weight and Thermal Expansion Coefficient at Varying Temperature and Constant Pressures*. https://www.engineeringtoolbox.com/air-density-specific-weight-d_600.html. Accessed on 2020-07-01. 2003.

[9] Engineering ToolBox. *Thermal Conductivity of selected Materials and Gases*. https://www.engineeringtoolbox.com/thermal-conductivity-d_429.html. Accessed on 2020-07-01. 2003.