Tutors

Dra. Alexandra E. Bonet Ruiz

Dr. Ricardo Torres Castillo

Secció Departamental d'Enginyeria Química

Grau en Enginyeria Química

Treball Final de Grau

ANSYS Fluent simulation of a solar chimney

Simulació d'una xemeneia solar en ANSYS Fluent

Ricardo Moya Chamizo

June, 2017

UNIVERSITAT DE BARCELONA

En primer lloc, agrair als meus pares, i companys de classe tota la confiança, ànims i suport absoluts que m’han prestat, sense ells tot hauria sigut molt complex.

En segon lloc, dono les gracies als meus tutors, Ricard Torres i Alexandra Bonet, per les ajudes indiscutibles rebudes per part seva, tant referides al treball com consells aleatoris de tots els camps d’enginyeria, i sobretot pel seu comportament humà.

Finalment dono les gracies a la resta del professorat, per tot el que he après gracies a les seves classes i per conseqüència la confiança personal que he guanyat en vers a la vida laboral.

REPORT

CONTENTS

1. SUMMARY

This final degree project aims to model a chimney to incorporate it in a future work, which is a whole solar chimney model, also to compare temperature data obtained from an experimental pilot plant (literature) with temperature data obtained from two ANSYS® simulated models. To make a comparison of the results, there will be a study of temperature at different heights marked as 10#, 11# and 12#, matching the available experimental points. The simulation used models are: model with atmosphere, to obtain the indoor and outdoor data as accurate as possible, and a model without atmosphere. Once the results are obtained and compared to experimental data, the both simulated models would be compared between them. Finally, there will be studies of temperature, density, velocity and turbulence for both.

There is detailed explanation of each ANSYS® module. The explained modules are the Geometry design, Mesh Generation and Model Setup of Fluent, being this last one to quantify and export results, emphasizing in distributions, profiles, contours and values of temperature, density and velocity throughout the system. In the Geometry Design, the chosen geometry was 2D, in order to make it easier to compute.

The solar chimney dimensions are: a cylinder with, 8 m high and 30 cm diameter for both models. In the atmosphere model, the atmosphere dimensions surrounding chimney cylinder are 160 meters high and 3 meters in width with respect to the center of the chimney. For both models, the outdoor temperature and pressure are 305.5 K and 101,325 Pa respectively. The chimney inlet temperature is 322.4 K.

The pictures were obtained as contour and vector formats. The distributions of temperature, density, velocity and turbulence throughout the system are shown. The graphical profiles of temperature, density and speed at different heights and, the numerical data tables of temperature, density, velocity and Reynolds, are shown for both models too. The obtained results of the simulation, have a similarity for the atmosphere model and the model without atmosphere of 99.7% and 99.5% respectively between them and the experimental data.

The results of temperature in each of the predetermined heights are nearly identical to the experimental data in both simulations.

The atmosphere model reproduces more realistic results than the model without atmosphere.

Given that the goal of the study is to compare the results with a real case, confirms the fact that the two models give acceptable and similar values.

If there is an interest in designing a new chimney, it is recommended the implementation of the atmosphere model.

Given the results of comparing models, it is verified that all the results as density, speed, Reynolds and heat transfer coefficients are correct.

2. RESUM

Aquest treball final de grau pretén modelitzar una xemeneia per adaptar-la posteriorment a un model complet d'una xemeneia solar i també comparar les dades de temperatura obtingudes d'un cas experimental (bibliogràfic) amb les dades de temperatura obtingudes en els dos models simulats amb el programa ANSYS®. Per tal de comparar els resultats, es realitzarà l'estudi de temperatura a les diferents altures predeterminades i senyalitzades com a 10#, 11# i 12#, utilitzant un model amb atmosfera inclosa, per aconseguir les dades de l'interior i l'exterior de la forma més acurada possible i, un model sense incloure l'atmosfera envoltant. Un cop obtinguts i comparats els resultats amb les dades experimentals, es compararan els dos models simulats entre ells, i finalment, es realitzaran estudis de temperatura, densitat, velocitat i turbulència per ambdós.

S'ha realitzat una explicació detallada de cada mòdul de l'ANSYS®. Els mòduls explicats han sigut el Geometry design, el Mesh generation i el Model Setup del Fluent, havent sigut aquest últim programat per quantificar i exportar resultats, emfatitzant en les distribucions, perfils, contorns y valors de temperatura, densitat i velocitat en tot el sistema d'estudi. En el Geometry Design es va optar per realitzar l'estudi per pura geometria en 2D, per facilitar la velocitat de computació. En el Mesh generation es va optar per fer la malla de la forma mes complexa, manualment. En el Model Setup del Fluent s'han trobat dificultats amb els "under-relaxation factors", per motiu del temps de computació requerit.

Les característiques de la xemeneia solar son: un cilindre de 8 m d'altura i 30 cm de diàmetre per ambdós models, les dimensions de la atmosfera circumdant en el cas del model amb atmosfera son 160 m d'altura i 3 m d'amplada respecte del centre de la xemeneia. Per als dos models, la temperatura exterior és de 305,5 K, la pressió atmosfèrica és de 101.325 Pa i, a l'entrada de la xemeneia la temperatura d'entrada és de 322,4 K.

S'han obtingut imatges en format "contours" i vectorials de les distribucions de temperatura, densitat, velocitat i turbulència en tot el sistema, gràfics dels perfils de temperatura, densitat i velocitat a diferents altures preestablertes, i taules de dades d'integrals realitzades a les diferents altures preestablertes, amb resultats numèrics de temperatura, densitat, velocitat i Reynolds, per ambdós models. Els resultats obtinguts per la simulació tenen una similitud amb les dades experimentals, pels models amb atmosfera i sense atmosfera de 99,7% i 99,5% respectivament.

Els resultats de temperatura en cadascuna de les altures predeterminades son pràcticament idèntics a les dades experimentals en les dues simulacions. Es pot observar com el model amb atmosfera reprodueix resultats més realistes que els del model sense atmosfera.

En el cas de que el que interessa sigui dissenyar una nova xemeneia, es recomana la realització del model amb atmosfera. Tenint en compte els resultats de temperatura y pressió obtinguts en la comparació de models, es verifica que tots els resultats obtinguts, com la densitat, la velocitat, el Reynolds i els coeficients de transmissió de calor son reproduïbles i de confiança.

3. INTRODUCTION

The concept of Solar chimney power technology was first conceived in 1931 by Hanns Gunther, and was proven with the successful operation of a pilot plant constructed in Manzanares (Spain) in the early 1980s. The natural ventilation is the best way to improve the indoor thermal and breeze comfort and to reduce the energy consumption that is used in artificial ventilation systems. A solar chimney system uses solar radiation to heat the air inside a control volume, thereby converting the thermal energy into kinetic energy.

The increase in world population and the improvement of living standards led to a growing demand for electricity. The limited availability of fossil fuels and environmental pollution caused by them, push the development of innovative technologies to produce electricity from renewable energy sources. With a view to sustainable development, therefore, the energy future must have as protagonists renewable sources (solar, geothermal, wind, etc.), which are not exhausted and have no environmental impact because they do not produce greenhouse gases.

In building constructions after the international energy crisis in 1973, the energy required for heating and cooling of buildings was approximately 40% of the total world energy consumption. The indoor environment for summer is normally obtained by air conditioning or ventilation including mechanical ventilation and natural ventilation (N. Pasumarthi, 1998). Natural ventilation not only can save energy and life cycle costs, but also can alleviate the environmental charge from the products used in energy production. There is not only one purpose, but the first one is to replace the air conditioning systems in certain regions. Modern society, is characterized by high civilization level and advanced accommodations, involving heating supply for the severe winter and air conditioning for warm summer. The annual summary report of International Energy Agency (IEA) shows that for the well-insulated office buildings, with a well-controlled natural ventilation system can reduce more than 50% of energy requirement.

A solar chimney is essentially divided into two parts, one is the solar air heater (collector) and the second one is the chimney, as shown in *Figure 1 and 2*. Two configurations usually used are: vertical solar chimney with vertical absorber geometry, and roof solar chimney. For the vertical configuration, vertical glass or steel is used to gain solar heat (there are also mechanical issues because of the wind, therefore the material must be resistant). The temperature difference between vertical glass duct and interior room produces a pressure difference. An interior air will go out through inlet because of this pressure difference. The temperature difference is a determining factor of performance of solar chimney. Designing a solar chimney includes height, width and depth of cavity, type of glazing, type of absorber, etc. Besides these system parameters, other factors such as the location, climate, and orientation can also affect its performance.

The present study considered the same structural parameters as (Zhou et al. 2007) did use in their real and physic Solar chimney. Their project which was published as experimental and theoretical results, is going to be used in order to make a comparison between their results and the ones obtained by (ANSYS Fluent User's Guide) Fluent simulator. The Schematic diagram of the proposed model with the collector drown for a better understand is shown in *Figure 2*.

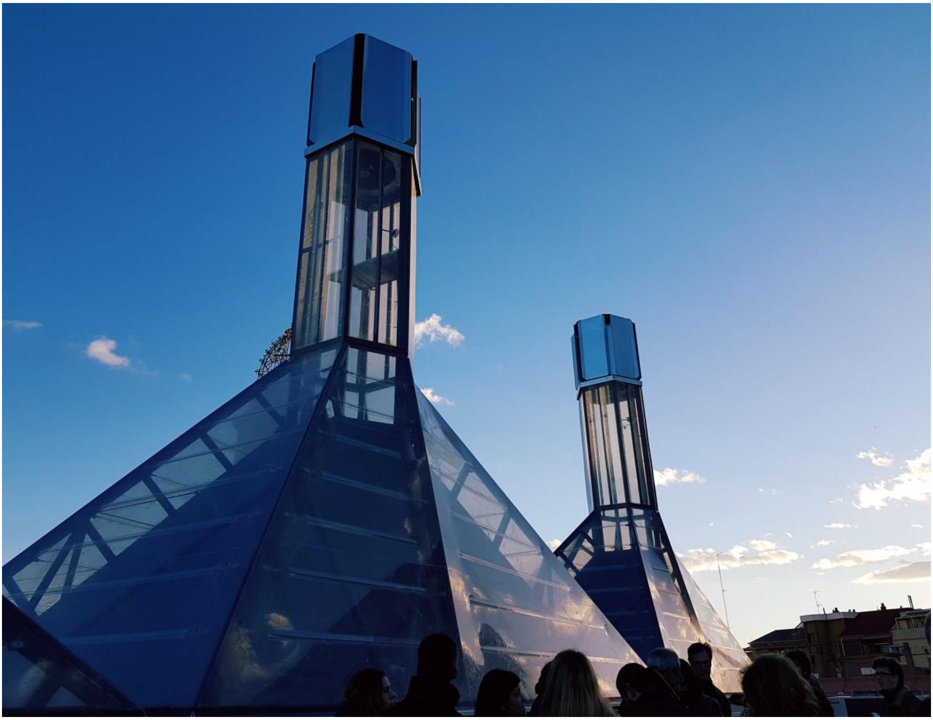

Figure 1. Solar chimney (Centre Cívic Cristalleries Planell, Barcelona)

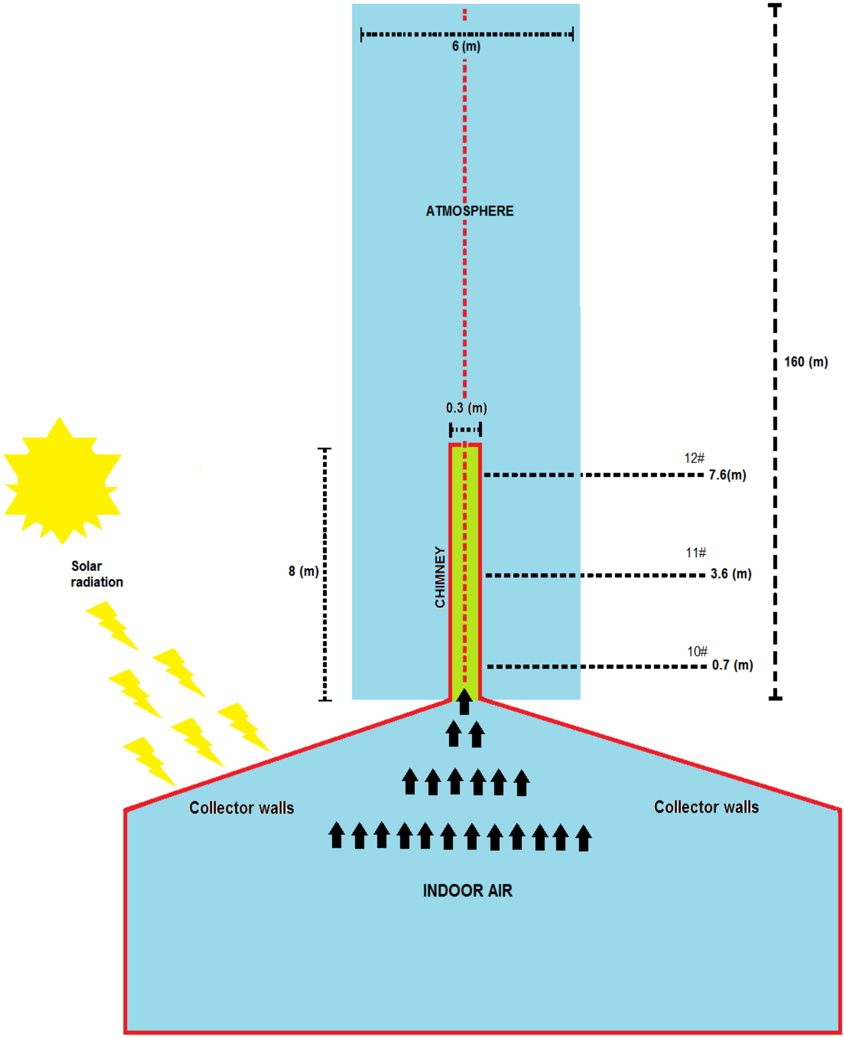

Figure 2. Diagram of solar chimney proposed model (with collector and red axis-symmetric centerline)

3.1. STATE OF THE ART

Other interesting simulations can be found when searching for information about solar chimneys using CFD (Computational Fluid Dynamics). The one that Islamuddin et al. (2013) did is not different from the one that is analyzed in this project. Although the base of the chimney pipe is not simulated in this final project, the cylindrical pipe is the same geometry and form, and the main ideas are quite similar. A model to implement the exhaust gases effect into the classical solar chimney power plant was needed. The proposed model consisted of hollow rectangular channels beneath the collector cover. The idea was to pass the exhaust gases through the channels by means of pipes that exits into the chimney base. Therefore, the idea was the same as the one in this project, because there was a requirement in warming the walls of the channel to let a heat transfer mechanism take place between the walls and the air flowing beneath the collector cover. In consequence, the air would gain more kinetic energy increasing the power output through the turbine. There is a need to say that in this project there is not a turbine. To finish, in their study, exhaust gases could escape through upper part of the chimney. Therefore, a simulation study using ANSYS® Fluent was performed, to see how could this model work in a real case.

Although the simulation was in 3D, no precision or accuracy is lost, both their solar chimney and the one used in this project have a complete symmetry that allows to work in 2D with axisymmetric tools because of the constant values in the radial coordinate. This technique is widely used, as it reduces the computational time and cost and does not affect the results.

Another simulation that has been found is a prototypal solar chimney system integrated in a south facade of a building. It was performed by Bernardo et al. (2014). Although the analysis was carried out in 3D, and the geometry is far different from the one used in this project, the main ideas and the boundary conditions used were the same ones. The air flow and the governing equations were given in terms of k-ε turbulence model as in some cases in this final project. Their problem was solved by ANSYS® Fluent and the results were performed for a uniform wall heat flux on the vertical wall. All the results, as in this project were given in terms of wall temperature distributions, air velocity, temperature fields and transversal profiles. The main difference is that Bernardo et al. (2014) modeled this to evaluate the differences between two base configurations.

The following Table 1, shows a bibliographic summary about ANSYS® Fluent solar chimney simulations.

Table 1. Summary table of ANSYS® simulations with properties and generalities. h_s (heath transfer coefficient), E_{ext} (external emissivity), K-∈ (turbulent method). In green, this project.

References	**Real format**	**Format and model**	**Outlet boundary conditions**	**Inlet boundary conditions**	**Chimney walls boundary conditions**	**Collector boundary conditions**	**Atmosphere walls boundary conditions**	**Atmosphere Interior**
Islamuddin et al. 2013	3D cylinder + Conic Collector	3D, unavailable	Top chimney zone (circular). Pressure outlet	Bottom chimney zone (circular). Mass inlet	unavailable	unavailable	none	none
Bernardo et al. 2014	3D squared	3D, Turbulent K-∈	Top chimney zone (square). Pressure outlet	Bottom chimney zone (square). Pressure inlet	Heat flux	none	none	none
Sudprasert et al. 2016	3D cylinder	2D, Turbulent K-∈	Top left chimney zone (circular). Pressure outlet	Bottom left chimney zone (circular). Pressure inlet	Constant temperature (∞ heat transfer coefficient)	none	none	none
Hu et al. 2017	3D cone + Conic Collector	2D, Turbulent K-∈	Top chimney zone (circular). Pressure outlet	Bottom chimney zone (circular). Pressure inlet	Adiabatic wall	Mixed wall: T, h_s, E_{ext}	none	none
Degree final project	3D cylinder	2D, Turbulent K-∈	Top chimney zone (circular). Pressure outlet	Bottom chimney zone (circular). Velocity inlet	Convection/ conduction/ convection	none	Constant temperature	Real gas
Degree final project	3D cylinder	2D, Turbulent K-∈	Top chimney zone (circular). Pressure outlet	Bottom chimney zone (circular). Velocity inlet	Constant temperature (∞ heat transfer coefficient)	none	none	none

In the simulation of Bernardo et al. (2014), all the thermophilically fluid properties were assumed to be invariant except the density, which is different from this project where the density and the viscosity are both variant. The compression work, viscous dissipation and radiative transport were negligibly small, in this project all those are used as not negligible to make the study more realistic. They considered the steady, turbulent, 3D natural convection flow in a solar chimney as shown in *Figures 3 and 4.*

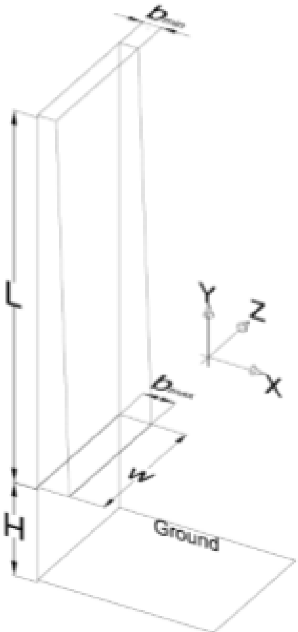

Figure 3. Geometry configuration

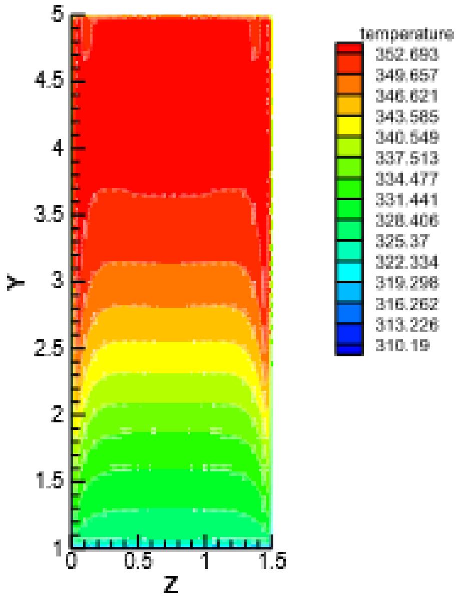

Figure 4. Temperature field on vertical surface

3.2. PROJECT WHICH THE SIMULATION IS BASED ON

The previously mentioned work of Zhou et al. (2007), is an experimental study of temperature field in a solar chimney power setup. This final project is based in their work. Therefore, is important to be said that all the parameters that are used in this project are extracted from the mentioned work in order to check if the ANSYS® Fluent is capable to make the simulation and deliver the same results.

In order to perform a detailed investigation into the measured temperature field in solar chimney power system, a pilot experimental setup was built as shown in *Figure 5*, consisted in a 10 m diameter air collector, 8 m chimney height and 30 cm chimney diameter, it was built in HUST, China. It is why in this project the 8 m tall chimney is defined as the cylinder, but there is no collector defined, because all the experimental and numeric information given is at the chimney inlet and outlet and there is not anything explained about materials, thickness and composition of the base collector. The chimney was built on the roof of a building, to avoid the shadows of buildings on the collector.

Figure 5. Pilot experimental setup (Zhou et al. 2007)

The cylinder of the schematic diagram shown in *Figure 6*, is going to be the pattern for this one, in fact the tags shown in *Figure 6* (#9, 10#, 11#, 12#) are the nomenclature used in the present final project. The #9 is going to be the fluid flow entrance (inlet) of the pipe, and the 10#, 11#, 12# are going to be diferent heighs where the wall temperature, air temperature, density and velocity distributions inside and outside the chimney, as well as temperature, density and velocity fields and transversal profiles are going to be shown. Also, another experimental performance with a prototype solar chimney was checked (Kulunk, 1985), this was done to check the similarity between them in order to understand better the results.

To check if the values of velocity are the correct ones, and therefore the simulation is well done, the temperature surface integral in each height must have the same numeric value as (Zhou et al. 2007), with the minimum error possible. (see *Figure 7*).

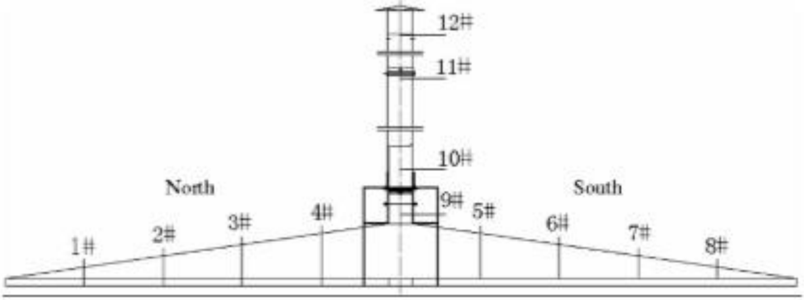

Figure 6. Schematic diagram of proposed model

The boundary conditions that have been extracted from the Zhou et al. (2007) project would be the following ones: ambient air 305.5 K, velocity inlet 2.13 m/s, and pressure outlet 1 atm.

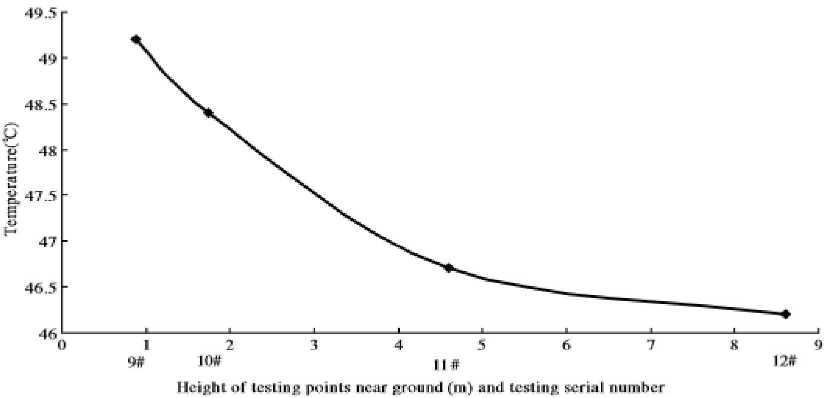

Figure 7. Distribution of air temperatures at different heights in solar chimney on a typical warm day (ambient temperature; 305.5 K, updraft velocity in the chimney; 2.13 m/s)

4. OBJECTIVES

The aim of this project is to model a chimney to incorporate it in a future work into a whole solar chimney model and also to check the ANSYS® Fluent modelling comparing its results with experimental data.

First of all, the learning on how to perform a simulation in ANSYS® Fluent, more than a goal is a necessity. And, once learned how to perform simulations with the tool and taking advantage of it, the following objectives have been planned:

- To study the temperature surface integral at different heights, using a model with a steady atmosphere around the chimney pipe. And then, checking if the simulation model fits the experimental data (Zhou et al. 2007).
- To study the temperature surface integral at different heights, using a model without an atmosphere around the chimney pipe. And then, checking if the simulation model fits the experimental data (Zhou et al. 2007).
- To compare the two previous models to check which one of them is the most representative.
- To study the temperature, velocity, density and turbulence distributions in the whole system in each model.

It must be said that in this project there is something that nobody did before to simulate a solar chimney, there is an atmosphere created where the distributions of temperature, velocity, density and turbulence will be obtained. It is important to know what happens in the atmosphere when something that can cause an environmental impact is created, to be able to regulate or fix the situation if necessary. Therefore, another model without the atmosphere is created, in order to compare the results and check if the first approximation is well enough.

5. MATERIALS AND METHODS: USING ANSYS®

This simulation is going to be done by computational fluid dynamics modelling, using ANSYS® Fluent commercial software.

Computational fluid dynamics (CFD), is the cheaper way in designing and running simulations and experiments, and there is no need to create or build the pilot plant. The cost of pilot plant building and repeating the process until the desired result is quite large.

Selecting the appropriate models is the way that is going to give the correct results, because there is no precision or accuracy if the global model is not well done. In the following subsections, the chosen models are explained, and reasons why they are used are given.

The ANSYS® commercial software provides access to several modules that allow to simulate a high number of scenarios focusing in many different engineering cases. The Fluent is one of those ANSYS® modules, and its characteristics and sub-models are suitable for the solar chimney that is going to be simulated.

A brief description of how does the software program work is going to be explained in order to understand better the work, also it is going to be explained what does the user need to perform a simulation.

Each step is going to be explained to perform a successfully simulation. Focusing on the mathematical models and on the boundary conditions formation setup.

The previous steps that must be completed to perform a simulation are the following:

1- Geometry design
2- Mesh generation
3- Models set up

5.1 Geometry design

When a new simulation is carried on, the first thing to do is to set the geometry that is wanted to be analyzed. First there is a need in check if the geometry can be ruled in 2D symmetry, because it would be faster to obtain the results after. The whole scheme calculation domain with axis-symmetry is shown in *Figure 8,* in order to show how this project is going to be designed. To do it, ANSYS® Fluent "DesignModeller" offers different tools that allow to draw any kind of geometry, it is important to draw the minimum geometry possible to make the project more optimized. Moreover, files of another software such as ABAQUS, CFX, GAMBIT and GeoMesh, NASTRAN, PATRAN Neutral, HYPERMESH ASCII, AutoCAD, Solidworks or Unigraphic etc. can be imported/uploaded to ANSYS® Fluent workbench which is a very useful feature.

The solar chimney used in this degree final project is represented in two different models. The first one represents the geometry of a cylinder inside an atmosphere control volume, the whole study area is 960 m^2 because of the atmosphere. The atmosphere is 20 times taller than the chimney and 20 times wider than it too, this size will make some future problems in the Fluent set up because of the limited mesh, but that problems can be solved by using the URF (under-relaxation factors). The second one is just to perform the same but without an atmosphere, the aim to do this is to reduce the complex system area to 2.4 m^2 and to compare the results with the first one.

The system in study has a high symmetry, and it is why allows to perform the project in 2D. The model with atmosphere will be a rectangle inside another one, being the last one the assumed atmosphere. This feature drastically reduces the computational time to calculate the solution. The calculation domain is shown in *Figure 9*. Thanks to the symmetry the area is half of 960 m^2, but there is a need to say that the picture is quite small because this area is of 480 m^2. The atmosphere must be oversized to perform a more realistic model and to avoid wrong calculations. The model without an atmosphere will have only a chimney pipe and thanks to the symmetry the area is 1.2 m^2.

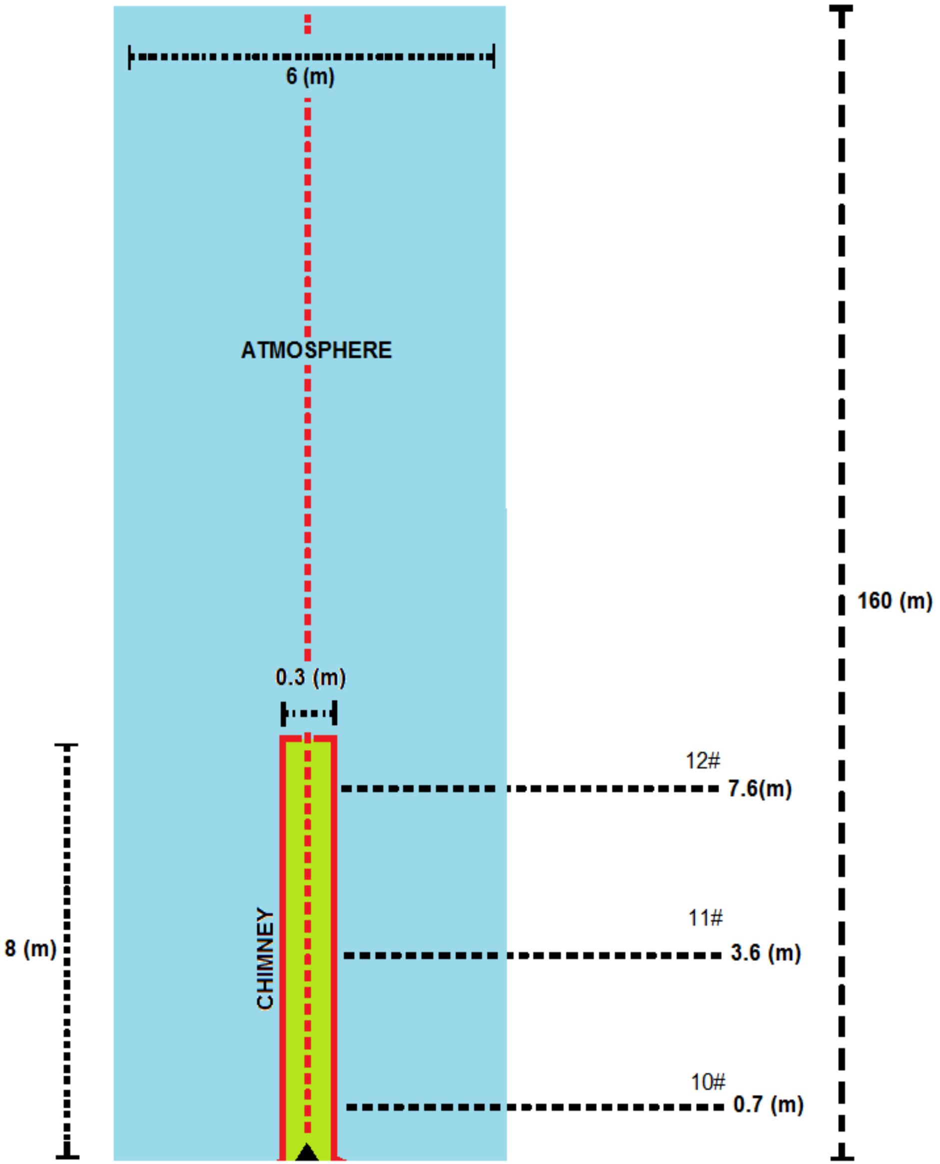

Figure 8. Whole scheme calculation domain with axis-symmetry in red

Figure 9 shows the two models used: A) represents the case with atmosphere, B) represents the case without atmosphere. The chimney is not at the same scale in both pictures due to the atmosphere surrounding the pipe in A), but the true size is the same one.

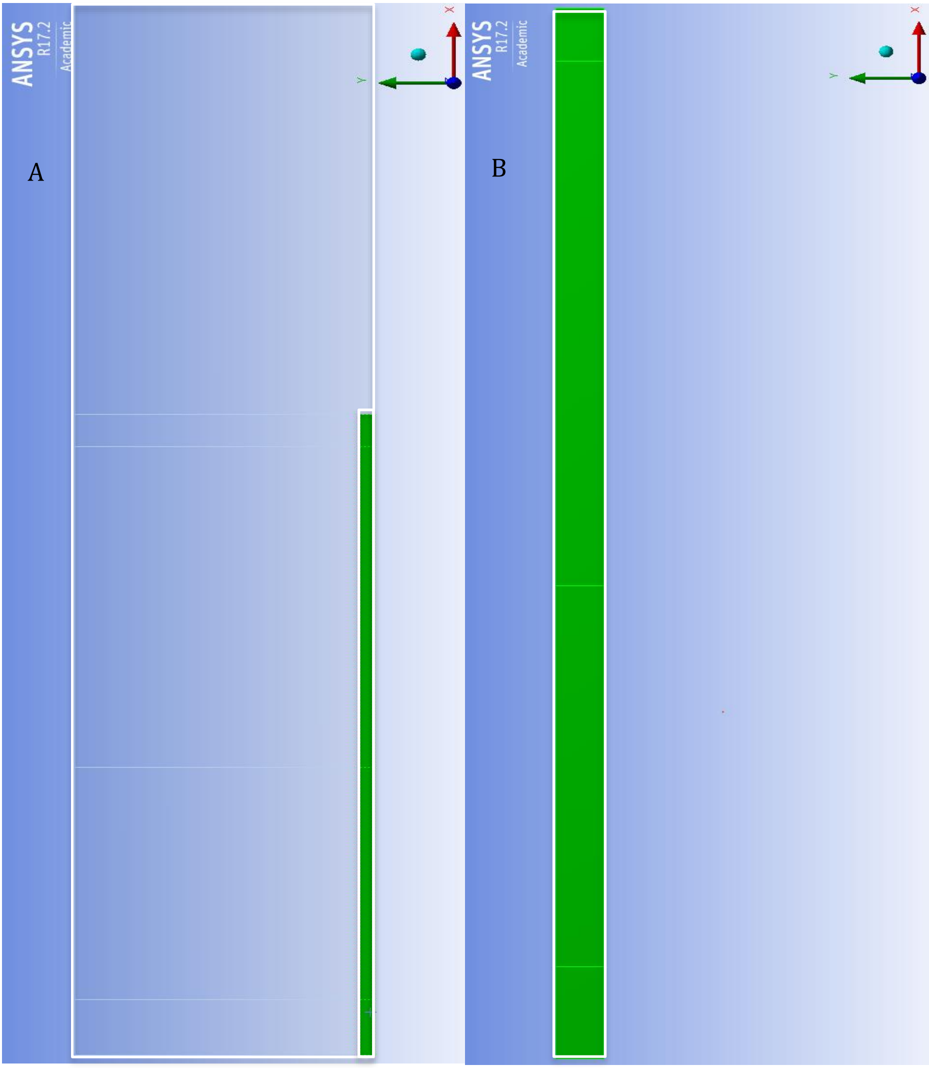

Figure 9. A) Model with atmosphere, Inner chimney fluid calculation domain (in green), the blue part inside the white framework is the atmosphere. B) Model without atmosphere, Inner chimney fluid calculation domain (in green) inside the white framework. No axis-symmetry applied

The model without atmosphere is so small, thereby all the model can be shown in *Figure 9,* but the model with atmosphere is oversized and it is why this one will need more pictures to understand and represent it better. *Figure 10* shows the solid wall of the chimney surrounded by the fluid atmosphere and inner chimney fluid. To make more space in the project sheets, all the next chimney pictures will be shown in horizontal.

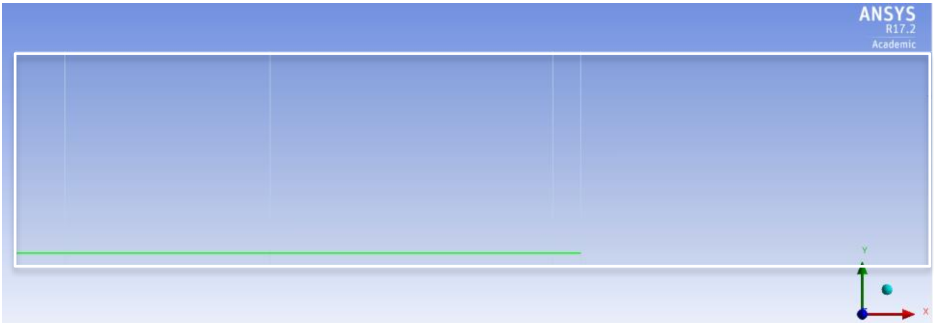

Figure 10. Chimney wall calculation domain (in green). The blue part surrounded by the white framework is the atmosphere and the inner chimney fluid. No axis-symmetry applied

To finish, *Figure 11*, shows the most relevant part of the whole atmosphere, a zoom is used because of the oversizing to let the concept be more understandable.

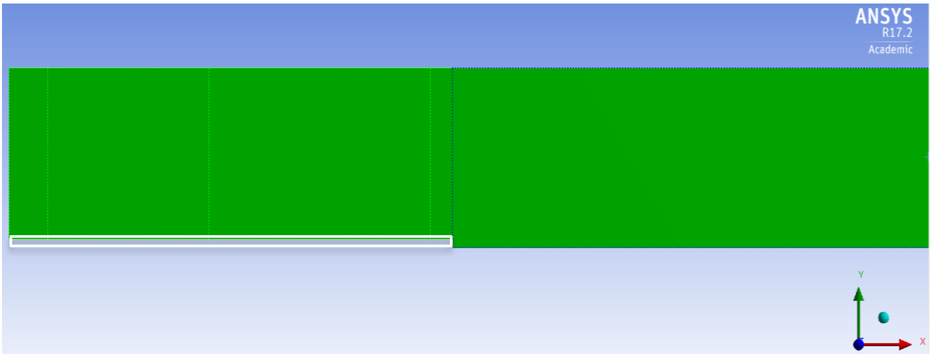

Figure 11. Atmosphere calculation domain (in green). The blue part surrounded by the white framework is the inner chimney fluid. White framework represents the chimney wall. No axis-symmetry applied

To lay out the dimensions of the atmosphere model, without the symmetry applied, another schematic *Figure 12* is shown but this one in vertical. The model without atmosphere would be the same chimney but without the surrounding atmosphere.

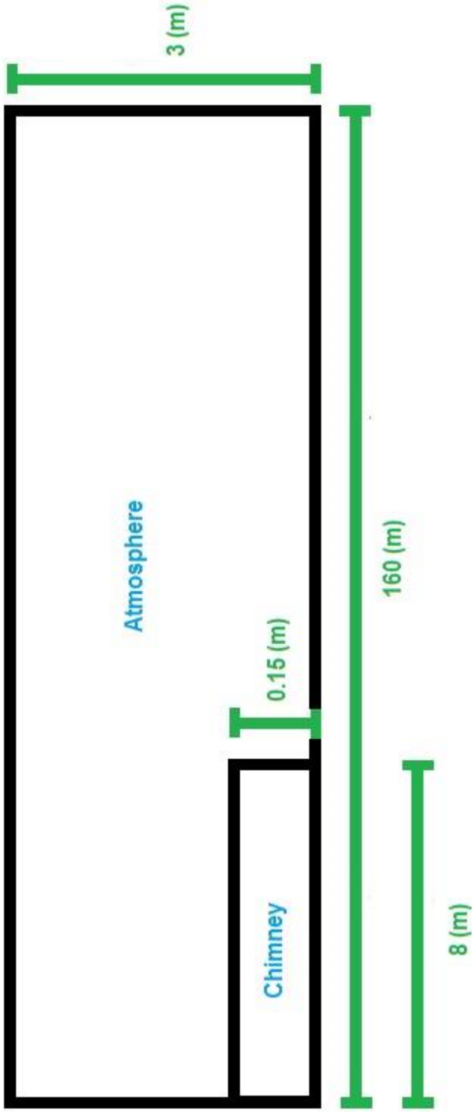

Figure 12. Atmosphere model, whole domain scheme without symmetry applied

5.2 Mesh generation

Once the geometry has been drawn or imported from other program, the system must be discretized to solve the mathematical model with the finite elements method. To solve the project without discretizing it, microscopic balances should be solved, which are differential equations. But if the project is discretized, several finite elements are created, and the size of that thousands of finite elements are such small that those differential equations from the microscopic balances can be transformed to algebraic equations, which require less effort to be calculated and thereby optimizes the process.

The solution is never 100% exact, but it is near to it, because the equations transformation is just an approximation. This approximation is going to be better and closer to 99.99% if finite elements size is reduced. Nevertheless, as smaller the finite elements size, the higher number of nodes should be calculated. Therefore, the mesh cannot be too thin, otherwise the computational time to perform the simulation would be too high. Moreover, in the ANSYS student license there is a maximum number of nodes that can be used, in this case there are 500,000.

In order to minimize the error, a very common strategy is to create inflation zones or just make manual meshing and therefore increasing the number of nodes in the sites that is predictable that the numbers are going to change more (that last technique is the most complex one and the one that has been used in this project), and then reducing the number of them in the other parts of the study. To sum up, both techniques consist of mesh refining only in the zones where a lot of changes are expected to happen, such as inlets, outlets, walls, and the places where the profiles or distributions plots are needed. By doing this, the accuracy is increased in the most important zones while the computational time is still reasonable.

In the solar chimney pipe used in this study, the zones that need an inflation or a fine meshing are the air inlet, air outlet, chimney walls (where is going to be heat conduction), chimney interior near the wall (where is going to be heat convection) and in the atmosphere model, the atmosphere near the chimney wall (where is going to be heat convection too), moreover the other parts must have a good mesh, because the velocity and density distribution and profiles are wanted.

It is important to say that the mesh must be full squared in order to predict a good result and a good transmission of the mesh into the Fluent calculation software. In the following pictures is going to be shown the mesh.

To make more space in the project sheets, all the following chimney pictures will be shown in horizontal as happened in the geometry section.

In order to make it shorter to explain, only the atmosphere model is going to be shown, due that this one includes the chimney that is equal meshed to the model without atmosphere. *Figure 13 and 14* (a zoom is used because of system size) show the difference between the high meshed parts and the others that are not as relevant but necessary too. The atmosphere bordering the chimney wall is high-meshed in comparison with the atmosphere that is far away from the chimney. Also, the inlet, outlet and the chimney wall are with a high-mesh and moreover the fluid that goes outside trough the outlet is high-meshed because there are a lot of changes here and is necessary for the program to make a good convergence.

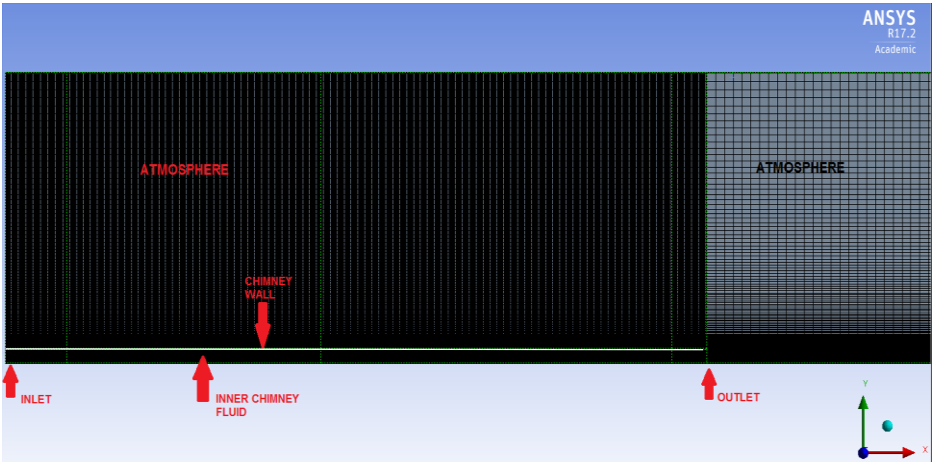

Figure 13. Difference between high meshed parts and others. White line is not a meshing line but represents the chimney wall to make it more understandable. No axis-symmetry applied

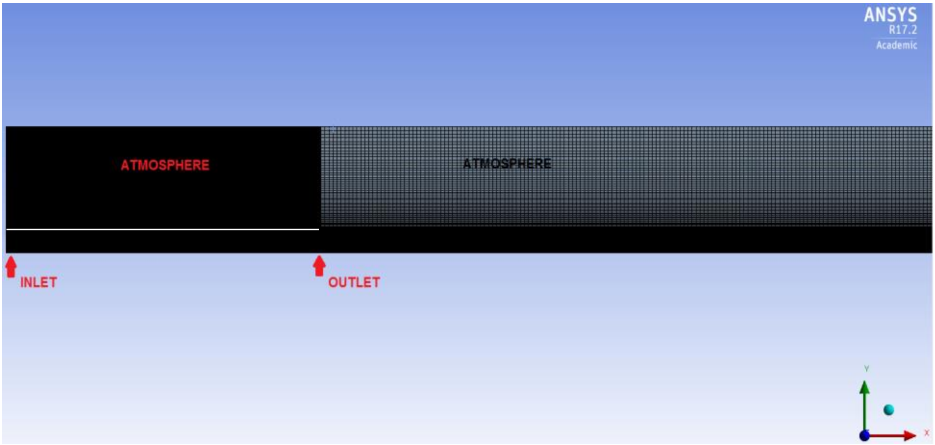

Figure 14. Difference between high meshed parts and others. White line is not a meshing line but represents the chimney wall to make it more understandable. No axis-symmetry applied

Figure 15, with a high zoom, shows the mesh used in the fluid surrounding the chimney wall. Moreover, the chimney wall that persists in black color even when a big zoom has been done. Also *Figure 16* with a higher zoom than *Figure 15* is shown, to make the reader be able to see the mesh in the chimney wall.

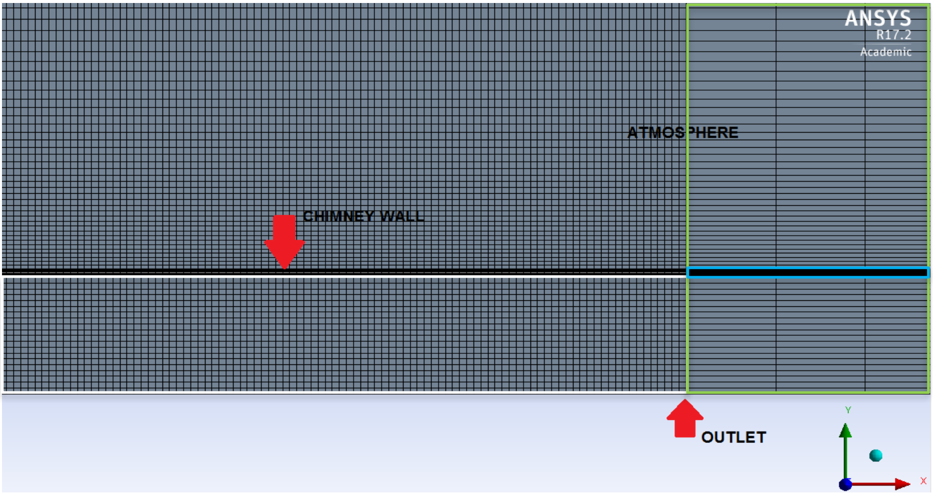

Figure 15. Difference between high meshed parts and the others. The white framework surrounds the inner chimney fluid, and the green framework surrounds the atmosphere, the zone that contacts both frameworks is the called chimney outlet. The blue framework is surrounding an atmosphere zone. No axis-symmetry applied

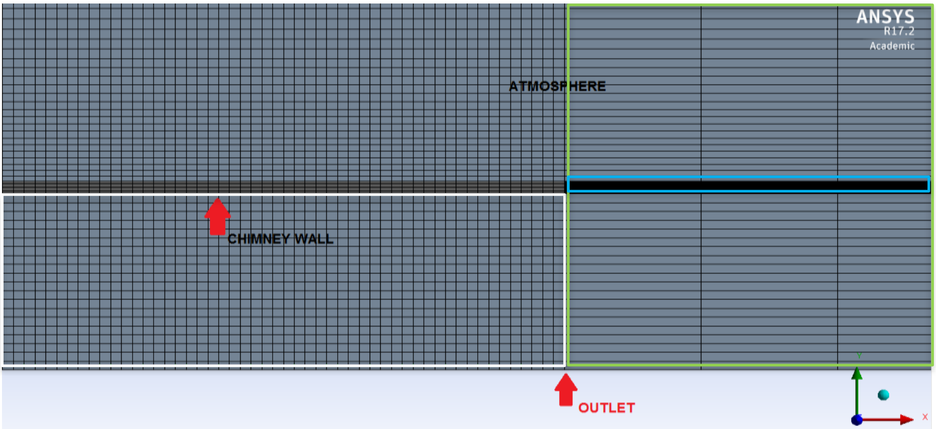

Figure 16. Chimney wall meshing focused. The white framework surrounds the inner chimney fluid, and the green framework surrounds the atmosphere, the zone that contacts both frameworks is the called chimney outlet. The blue framework is surrounding an atmosphere zone. No axis-symmetry applied

There is a need to say that a black meshing zone, persists in *Figure 15 and 16*. This zone surrounded by the blue framework, represents a part of the atmosphere. This location is important to be high meshed, because there will be high changes in the temperature, the density, the velocity and the turbulence, due to the pure convection heat transfer outside the chimney.

5.3 Simulation setup

ANSYS® Fluent module is going to be used in this step. The two previous steps are general for all ANSYS® simulation such as Static Structural, Transient Structural, Thermal-Electric, etc. And once completed, the appropriate module is chosen to simulate the system behavior.

In this step the correct models, materials, boundary conditions, cell zone conditions, mesh interfaces, reference values, etc. should be chosen. Is crucial to obtain a reliable solution. The resolution methods and controls are established.

In the model definition section (multiphase, energy, viscous model, acoustics, etc.), where the equations that will be used to calculate the process will be ready to check, one must consider the mathematical model that governs the process to analyze. In Fluent case, there are several mathematical models, the ones that are going to be used are the Turbulent k-∈ viscous model and the energy model. There are composed of fluid flow equations, which are series of balances and fluid properties, including the overall mass balance, known as the continuity equation, the momentum balance, the energy balance, which is decomposed in different equations, and partial mass balances. The information about the energy equations is extracted from (Krisst, 1983).

Those equations are:

Continuity

$$\frac{\delta\rho}{\delta t} + \nabla.(\rho U) = Sm \qquad (1)$$

Momentum

$$\frac{\delta\rho U}{\delta t} + (\nabla.\rho UU) = -\nabla p + \nabla.\tau + \rho g \qquad (2)$$

$$\tau = \mu\,[\,(\nabla\, U + \,\nabla\, U^T) - \frac{2}{3}\,\nabla\cdot U\, I] \qquad (3)$$

Enthalpy

$$\frac{\delta\rho E}{\delta t} + \nabla.\big(U(\rho E + \rho)\big) = \nabla.(keff\;\nabla T - [\textstyle\sum_l hj\,Jj + (\hat{z}eff\,\cdot U)] + S_h \qquad (4)$$

$$E = h - \frac{P}{\rho} + \frac{U^2}{2} \tag{5}$$

Temperature

$$P = \frac{RT}{V-b} - \frac{\alpha}{V(V+b)Tr^{0.5}} \tag{6}$$

$$Tr = \frac{T}{Tc} \tag{7}$$

To solve the problem, Fluent discretizes these governing equations for each node generated in the meshing step.

Fluent also have different additional models that can be activated depending on the system. In this case, one more additional model must be activated to perform the simulation as accurately as possible. This one is the Turbulent k-∈ viscous model, it must be activated because there is a natural fluid flow through the system. Moreover, Sudprasert et al. (2016), Hu et al. (2017) and Bernardo et al. (2014) used it in their project and that was why it seemed to be the best one, also due to the high natural convection velocity in the inlet, that is around 2.13 (m/s).

This Turbulent k-∈ viscous model, consists of two equations, one for the turbulent kinetic energy (k) and one for the turbulent dissipation rate (∈). These standard high-Reynolds number equations (Md. Mujibur Rahman 2017) are:

Turbulent kinetic energy equation (k):

$$\frac{\delta(k)}{\delta t} = P_k - \epsilon + \nabla \cdot \left(\frac{\mathrm{v_T}}{\sigma_K} \nabla k\right) \tag{8}$$

Turbulent dissipation rate equation (∈):

$$\frac{\delta(\in)}{\delta t} = \frac{1}{T}(C_{1\in}\, P_k - C_{2\in} \in) + \nabla \cdot \left(\frac{\mathrm{v_T}}{\sigma_{\in}} + \nabla_{\in}\right) \tag{9}$$

The equation for mean turbulent time scale T= k/∈ can be expressed as:

$$\frac{\delta(T)}{\delta t} = \frac{\delta(k)}{\in \delta t} - \frac{k\ \delta(\in)}{\in^2 \delta t} \tag{10}$$

In Equations (8) and (9), the turbulent production term P_k can be expressed as:

$$P_k \ = -\bar{u}_i \bar{u}_j (\frac{\partial u_i}{\partial x_j}) \tag{11}$$

Where the Reynolds stresses $\bar{u}_i \bar{u}_J$ can be related to the mean strain-rate tensor S_{ij} as:

$$-\bar{u}_i \bar{u}_j = 2\ v_T\ (S_{ij} - \frac{1}{3} S_k \delta_{ij}) - \frac{2}{3} k \delta_{ij} \tag{12}$$

Using equation (8) and (10) and the above relations, and the turbulent Prandtl number (σ) (which connects the diffusive of k and ∈ to the eddy-viscosity) relations for (k) and (∈) as $\sigma_k = \sigma_\in = \sigma$ (due to the near-wall approximations) the final equation is obtained:

$$\frac{\delta(T)}{\delta t} = (1 - C_{1\in}) \frac{P_k}{\in} + (C_{2\in} - 1) + \frac{2\ v_T}{\sigma\ k}\ \nabla k \cdot \nabla T - \frac{2\ v_T}{\sigma\ T}\ \nabla T \cdot \nabla T + \nabla (\frac{v_T}{\sigma} \nabla T) \tag{13}$$

Also, when the model and solver are checked, the most important part of the simulation (the correct convergence of the calculations) must be checked too. Sometimes the project simulated in ANSYS® Fluent have converge difficulties due to the residual errors or meshing and geometry problems.

There are other kind of problems more difficult to correct. One of them is the non-convergence between the equations and the nodes that have been created. To correct this, the URF (under-relaxation factors) will be used. The pressure-based solver as this one, uses under-relaxation of equations to control the update of computed variables at each iteration. This means that all equations solved using the pressure-based solver, will have under-relaxation factors associated with them.

In ANSYS® Fluent, the default under-relaxation parameters for all variables are set to values that are near optimal for the largest possible number of cases. These values are suitable for many problems, but for some particularly nonlinear problems (e.g., some turbulent flows or natural-convection problems, as this one) it is prudent to reduce the under-relaxation factors initially.

It is a good practice to begin a calculation using the default under-relaxation factors. If the residuals continue to increase after the first 4 or 5 iterations, the best idea should be to reduce the under-relaxation factors. Occasionally, it is important to make changes in the under-relaxation factors and resume calculations, only to find that the residuals begin to increase. This often results from increasing the under-relaxation factors too much.

The viscosity and density are under-relaxed from iteration to iteration (if they are variable as in present work). Also, if the enthalpy equation is solved directly instead of the temperature equation, the update of temperature based on enthalpy will be under-relaxed.

For most flows, the default under-relaxation factors do not usually require modification. If unstable or divergent behavior is observed, however, a great reduction of the under-relaxation factors for pressure, momentum, k, and $\in$ from their default values to about 0.2, 0.5, 0.5 and 0.5, respectively, would be fine if the simulation is like this one. In problems where density is strongly coupled with temperature, as in very high Rayleigh number natural convection as this project or in mixed convection flows, it is wise to also under-relax the temperature equation and/or density.

For other scalar equations (e.g., swirl, species, mixture fraction and variance) the default under-relaxation factors may be too aggressive for some problems, especially at the beginning of the calculation. Those should be reduced to 0.8 to facilitate convergence.

6. RESULTS AND DISCUSSION

As described before, the most important variables to analyze in a solar chimney with natural convection are temperature, density and velocity in each point of the system. Therefore, their profiles, contours and distributions will be discussed.

Furthermore, another important values and longitudinal image distributions will be exposed in order to understand better the simulation, as for example, the surface heat transfer coefficients for the chimney walls in contact with indoor and outdoor air. Also, the turbulent distributions are shown inside and outside the chimney, and the Reynolds values in each height.

Moreover, there is a need to say that the graphics will be exported from ANSYS® Fluent setup, because there are so much nodes, and it is impossible to transfer all the data in to Excel or any other program that allows to build graphics.

The distribution contour pictures, the graphics and the non-graphical results that are going to be shown will be exposed in the following order:

- Temperature contours, graphics and integrals
- Density contours, graphics and integrals
- Velocity contours, graphics and integrals

6.1. Temperature contours, graphics and integrals

The temperature contour pictures, the non-comparative graphics of each height 10#, 11# and 12# and a comparative graphic with the three heights for both models are shown for both models. In those graphics, the radial temperature profiles inside the chimney pipe, in the chimney wall and in the atmosphere for each height are going to be shown too.

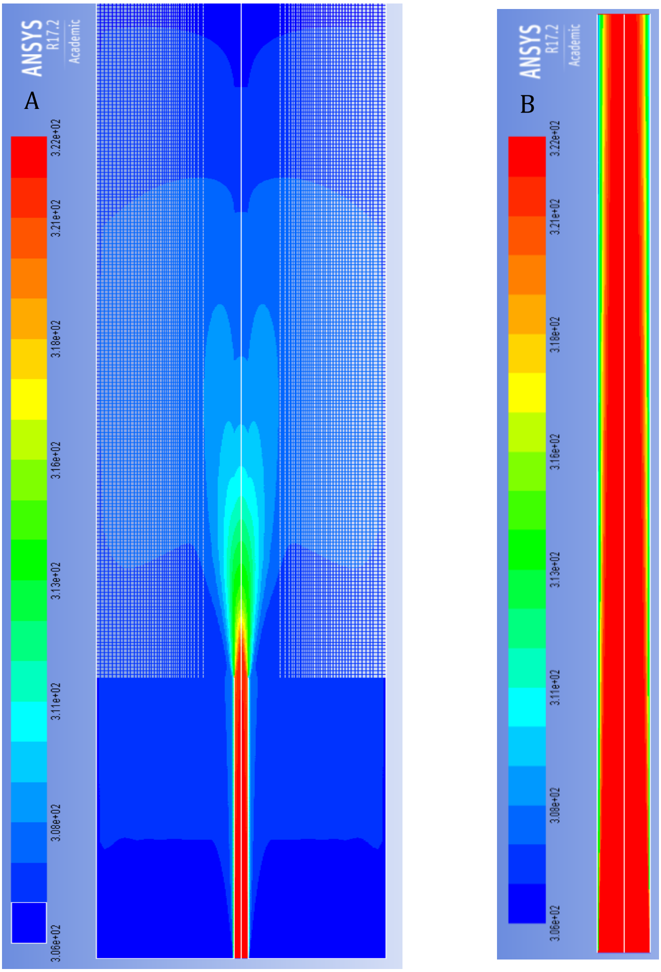

Figure 17. A) Model with atmosphere, temperature distribution inside and outside the chimney, zoom applied (the most important part of the system). B) Model without atmosphere, the temperature distribution inside the chimney, (all the system). Axis-symmetry applied

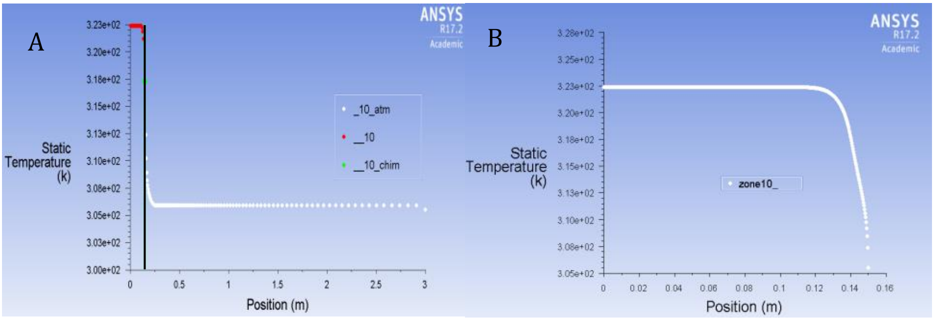

Figure 18. A) Model with atmosphere, 10#, 10# chim and 10# atm radial temperature profiles inside and outside the chimney. The vertical black line references the chimney wall. B) Model without atmosphere, radial temperature profile of 10#. (These are not comparative graphics)

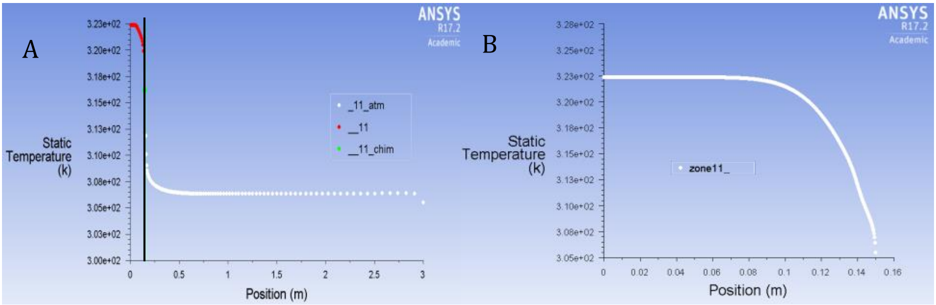

Figure 19. A) Model with atmosphere, 11#, 11# chim and 11# atm radial temperature profiles inside and outside the chimney. The vertical black line references the chimney wall. B) Model without atmosphere, radial temperature profile of 11#. (These are not comparative graphics)

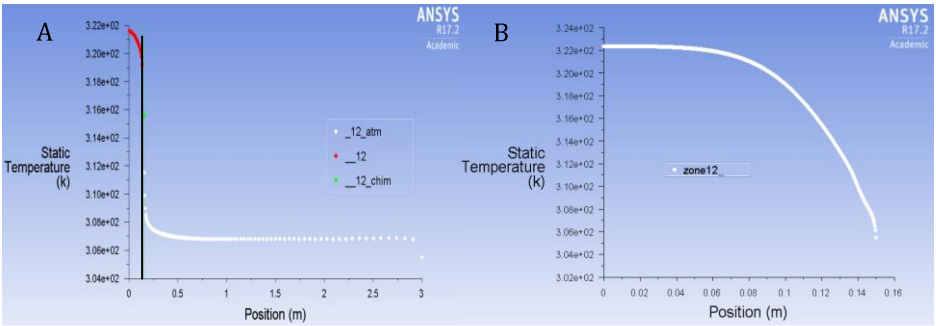

Figure 20. A) Model with atmosphere, 12#, 12# chim and 12# atm radial temperature profiles inside and outside the chimney. The vertical black line references the chimney wall. B) Model without atmosphere, radial temperature profile of 12#. (These are not comparative graphics)

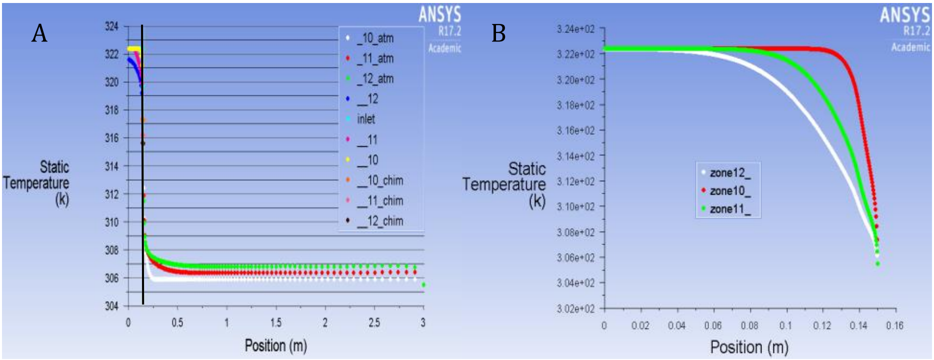

Figure 21. A) Model with atmosphere, inlet, 10#, 11#, 12#, 10# chim, 11# chim, 12# chim, 10# atm, 11# atm and 12# atm radial temperature profiles inside and outside the chimney. The vertical black line references the chimney wall. B) Model without atmosphere, 10#, 11#, 12# radial temperature profiles. (These graphics are not comparative between them)

The surface temperature integrals in each height are shown in Table 2 for both models, to make the comparison between the experimental data (Zhou et al. 2007) and those obtained one from this simulation.

Table 2. Temperature surface integrals in each height. *Values obtained with model without atmosphere and their comparison with those obtained by* Zhou et al. (2007) *are shown in green*

Height Tag	(1), Atmosphere model. Temperature (K)	(2), Non-atmosphere model. Temperature (K)	(3), Experimental data. Temperature (K)	Similarity % between (1) and (3)	Similarity % between (2) and (3)
Outlet	318.9	316.8	---	---	---
12# atm	306.8	---	---	---	---
11# atm	306.4	---	---	---	---
10# atm	305.9	---	---	---	---
12# chim	315.6	---	---	---	---
11#chim	316.2	---	---	---	---
10# chim	317.3	---	---	---	---
12#	320.4	316.9	319.4	99.7	99.2
11#	321.3	318.7	319.9	99.5	99.6
10#	322.1	320.9	321.5	99.8	99.8
Inlet	322.4	322.4	322.4	100	100.0

The longitudinal surface heat transfer coefficients values that have been obtained from the external chimney wall layer in contact with the atmosphere layer, and from the interior chimney wall layer in contact with the interior fluid layer, are going to be shown in a table 3.

Table 3. Surface heat transfer coefficients that have been obtained from a longitudinal integral in the corresponding layers

Tag	Atmosphere model. Surface heat transfer coefficient [W/(m^2·K)]	Non-atmosphere model. Surface heat transfer coefficient [W/(m^2·K)]
Atmosphere – chimney wall	1.30	---
Interior fluid – chimney wall	1.40	6.84

In *Figure 17* where the temperature distribution is shown, it is important to consider the chimney inlet. The inlet temperature is 322.4 K for both models, with this temperature at the beginning of the simulation, the contours complete the energy equations showing the temperature distributions. The heat flow goes to the pipe wall by convection heat transfer through the fluid and through the pipe wall by conduction heat transfer. Only in the atmosphere model it goes with convection heat transfer trough the atmosphere.

In the atmosphere model, the temperature is smaller in the upper parts of the chimney, because the wall absorbs heat and it leaks to the surrounding atmosphere. The same happens in the non-atmosphere model but the heat goes nowhere in the simulation because there is no layer there.

In the same *Figure 17* and focusing on the atmosphere model, near the chimney outlet, in the outdoors, a “tongue” can be seen, this one is created because there is only convection heat transfer due that there is not a wall. This causes a faster heat exchange and the temperature of the air increases faster. But the temperature goes down to the atmosphere one 305.5 K at approximately 30 m tall.

In *Figures 18, 19, 20 and 21* the radial temperature profiles can be seen for both models. Focusing in *Figure 21*, the B) that corresponds to the non-atmosphere model, only shows the temperature inside the pipe at different heights. It has the same behavior as the atmosphere model, A). In A) case, the temperature in 10# (_10) is higher that the temperature in 11# (_11) and 12# (_12), because in the inlet the temperature is 322.4 K and the heat is liberated from the pipe to the atmosphere reducing the temperature inside the chimney. The same happens in the wall, in 10# chim (_10_chim) the temperature is higher than in 11# chim (_11_chim) and in 12# chim (_12_chim). But something different happens outside the chimney, were the temperature before running the simulation is 305.5 K. The temperature in 12# atm (_12_atm) is higher than in

11# atm (_11_atm) and this one is higher than the temperature in 10# atm (_10_atm). This happens because the temperature in the atmosphere grows up when the chimney inner temperature goes down due to the heat flow released to the outside.

In *table 2,* it is important to see that the temperature is not the same, but it is close for both models. In the atmosphere model, the simulation was carried without air convection in the atmosphere. This means that in the simulation it was a static fluid before the temperature started changing its density and the natural advection started. This convection would directly take part in the temperature of the wall, cooling it and reducing the inner chimney temperature to the wanted one. In order to check this, the values of the non-atmosphere model, which has a constant temperature boundary condition at the chimney wall, are shown in *table 2.*

In the atmosphere model, the heat transfer coefficients that can be seen in *table 3* are very low values. This is caused because the air started the simulation as static fluid, and all the convection that was created in the simulation was due to the natural advection of the fluid.

In the non-atmosphere model, the heat transfer coefficient at the outdoor is supposed to be very high because of the constant temperature boundary condition, (cannot obtain the value because there is not a layer there to obtain it). Due to this, the indoor heat transfer coefficient in the non-atmosphere model is higher than in the atmosphere model.

In this case an additional *Figure 22* is build, to make the reader have an idea about the temperature fluctuations in the longitudinal axis inside the chimney wall for the atmosphere model.

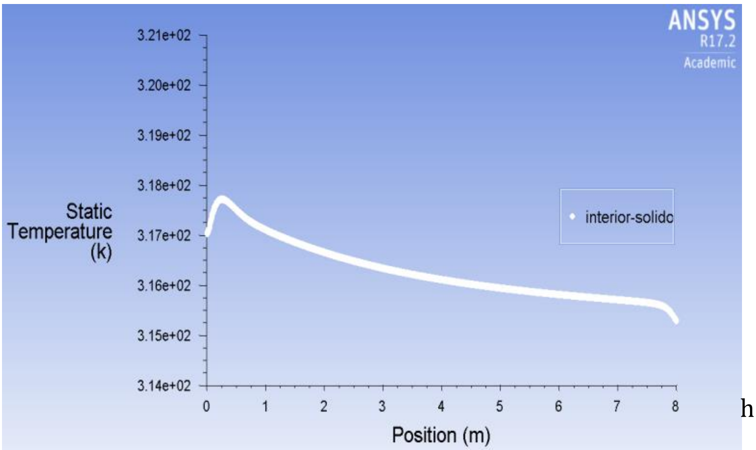

Figure 22. Temperature profile inside the chimney wall in the longitudinal axis (h = height) (interior-solid = interior wall, which means the interior part of the chimney wall)

The *Figure 22* corresponds to the atmosphere model. There is no chance to obtain a graphic like this one for the non-atmosphere model, because the boundary condition in the chimney wall, is a constant temperature. This boundary condition simulates a high convection. In order to make the reader understand better the difference between the 2 models, a new two graphics *(Figure 23)* with the temperature profiles of 10#, 11# and 12# heights inside the chimney pipe are going to be shown.

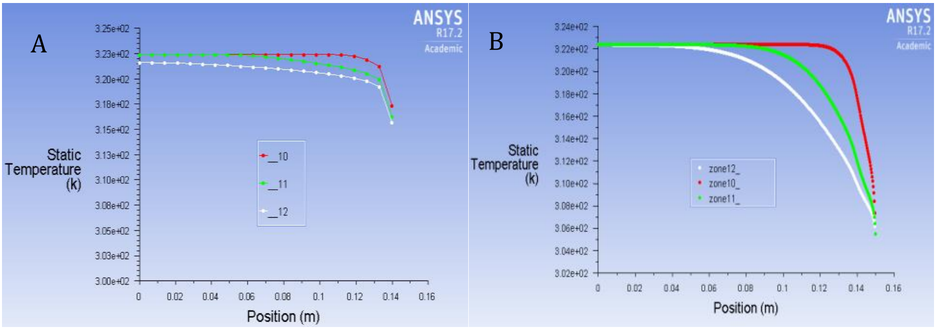

Figure 23. A) Model with atmosphere, 10#, 11# and 12# radial temperature profiles inside the chimney. B) Model without atmosphere, 10#, 11#, 12# radial temperature profiles. (These graphics are comparative between them)

6.2. Density contours, graphics and integrals

The density contour pictures, the graphics of each height 10#, 11# and 12#, a comparative graphic with the three heights and a table will show in this section for both models. In those graphics, the radial density profiles inside the chimney pipe and in the atmosphere for each height will also show.

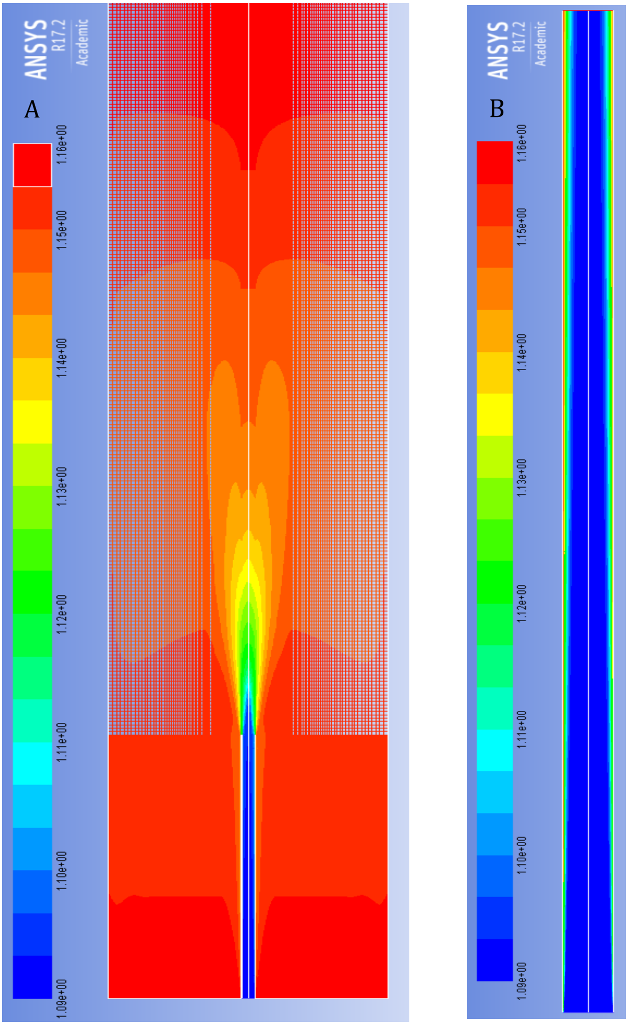

Figure 24. A) Model with atmosphere, density distribution inside and outside the chimney, zoom applied (the most important part of the system). B) Model without atmosphere, density distribution inside the chimney, (all the system). Axis-symmetry applied

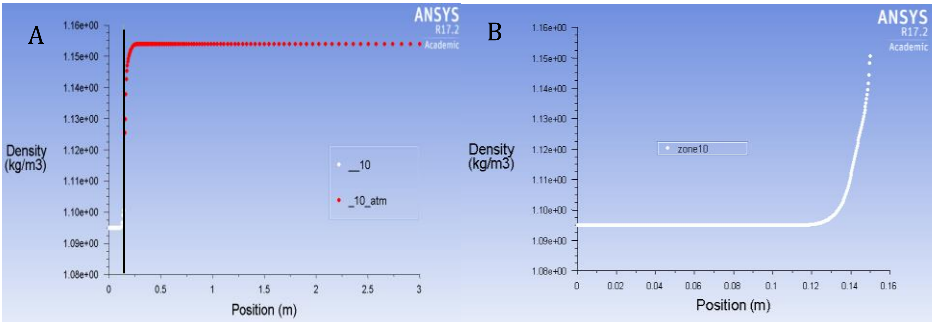

Figure 25. A) Model with atmosphere, 10#, and 10# atm radial density profiles inside and outside the chimney. The vertical black line references the chimney wall. B) Model without atmosphere, radial density profile of 10#. (These are not comparative graphics)

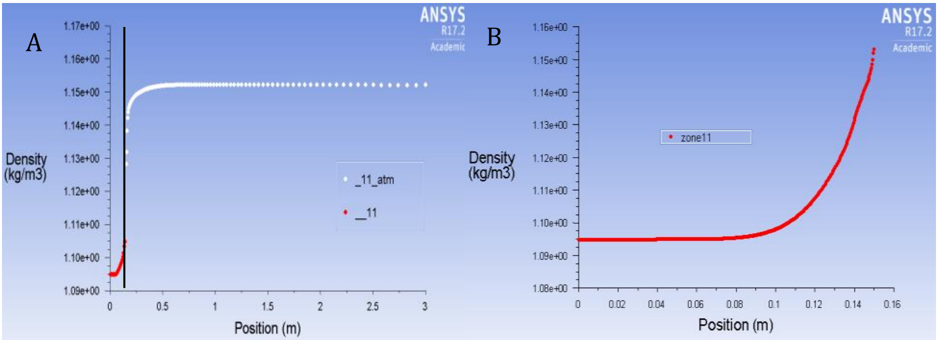

Figure 26. A) Model with atmosphere, 11#, and 11# atm radial density profiles inside and outside the chimney. The vertical black line references the chimney wall. B) Model without atmosphere, radial density profile of 11#. (These are not comparative graphics)

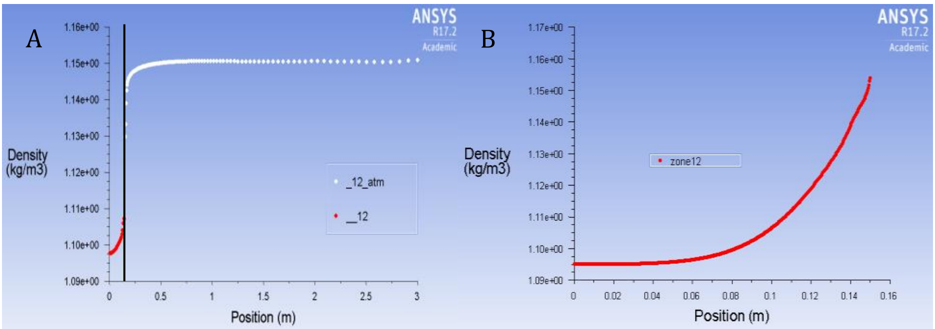

Figure 27. A) Model with atmosphere, 12#, and 12# atm radial density profiles inside and outside the chimney. The vertical black line references the chimney wall. B) Model without atmosphere, radial density profile of 12#. (These are not comparative graphics)

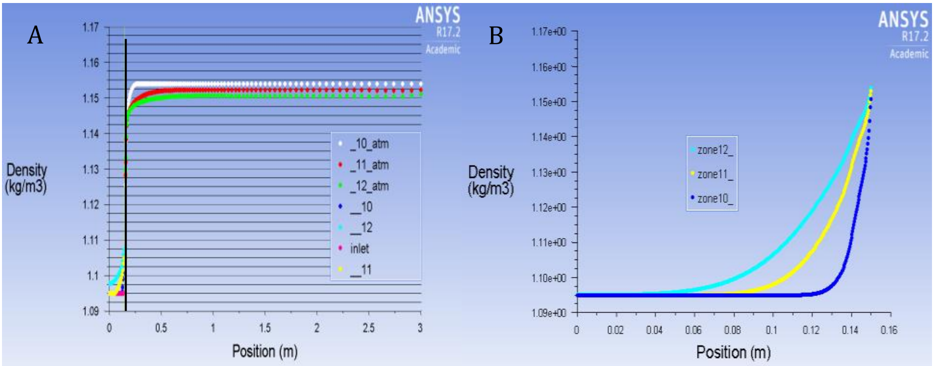

Figure 28. A) Model with atmosphere, inlet, 10#, 11#, 12#,10# atm, 11# atm and 12# atm radial density profiles inside and outside the chimney. The vertical black line references the chimney wall. B) Model without atmosphere, 10#, 11#, 12# radial density profiles. (These graphics are not comparative between them)

Table 4: Density surface integrals in each height. In this case, there is no comparison between the experimental data (Zhouet al. 2007) and the simulation data, because experimental density is unknown

Height Tag	(1), Atmosphere model. Density (kg/m^3)	(2), Non-atmosphere model. Density (kg/m^3)	Similarity % between (1) and (2)
Outlet	1.106	1.116	99.10
12# atm	1.151	---	---
11# atm	1.152	---	---
10# atm	1.154	---	---
12#	1.105	1.114	99.19
11#	1.102	1.107	99.54
10#	1.097	1.100	99.72
Inlet	1.095	1.095	100.00

Figure 24 shows the density distribution for both models. The picture is like the temperature one, but with reversed colors. This effect is caused due that the density is a temperature function. When the temperature changes, the density inversely changes.

The density is the cause of the fluid buoyancy. It is the induction force that will make the fluid to move up through the chimney. It is important to say that the weather depends directly on the results. The atmosphere temperature is around 305.5 K in the atmosphere model, and in the non-atmosphere model the chimney wall is always at 305.5 K. This means that the difference between the density inside the chimney pipe and outside it, is not as different as it could be in winter temperatures. Moreover, in the atmosphere model, the heat flow that goes outside the chimney, is the one that makes the interior chimney fluid to be colder near the walls than in the center of the pipe. Due to this, the density in this chimney part is higher than in the middle of it, where the temperature is maintained and the density is nearly to be constant. The same happens in the non-atmosphere model, the program knows that the heat flow must go out, but it cannot be seen due there is no layer there.

This higher density near the chimney wall, which can be seen in both models, is such an important characteristic of the system. The fluid doesn't go up at the same velocity in those parts. That lower velocity near the walls makes the fluid to become a wall in comparison with the fluid that is in the middle of the pipe. Due to this, the effective diameter where the fluid is circulating through is being reduced. This diameter reduction caused by the high-density fluid near the walls is such a section reduction. Therefore, this effect is creating a velocity increment in the middle part of the chimney where the fluid will go faster.

In the atmosphere model, concretely in the outlet of the chimney, a "tongue" can be seen, this is created because there is only convection heat transfer due that there is not a wall. This means that, the heat exchange is faster through the atmosphere and the temperature increases faster reducing the density in each point. The temperature goes down to the atmosphere one 305.5 K at approximately 30 m tall, which means that the density will be higher there.

Figures 25, 26, 27 and 28 show the radial density profiles for both models. Focusing on Figure *28*, the case B), that corresponds to the non-atmosphere model, shows the density inside the pipe at different heights. In B) case, the density has the same behavior as in the atmosphere model A) inside the pipe. The density in 12# (_12) is higher that the density in 11# (_11) and 10# (_10), because in the inlet the temperature is 322.4 K and the heat is liberated from the pipe reducing the temperature inside the chimney and thereby increasing the density near the walls. In the atmosphere model, something different happens outside the chimney, where the temperature before running the simulation was 305.5 K. The density in 10# atm (_10_atm) is higher than in 11# atm (_11_atm) and this one is higher than the density in 12# atm (_12_atm), this happens because the air temperature increases when the chimney inner temperature decreases due to the heat flux that is liberated to the atmosphere. This effect will decrease the density in the higher parts of the chimney. In the A) case, concretely in the atmosphere, a constant density zone can be seen, because the heat flux is not too high to change the temperature of the fluid that is far away from the chimney wall. This happens in all the heights where there is a chimney wall domain. When the chimney pipe finishes the above mentioned "tongue", changes the temperature of that atmosphere easily, due that there is not a wall. Therefore, the density is changing there too.

In *table 4*, it is important to note that there is no comparison between the experimental data (Zhou et al. 2007) and present simulation data.

When a simulation is performed, all the results as temperature, density and velocity will be obtained at once. If the temperature values are correct, the density values will be correct too.

6.3. VELOCITY CONTOURS, GRAPHICS AND INTEGRALS

The velocity contour pictures and the graphics of each height 10#, 11# and 12# and a comparative graphic with the three heights will be shown in the present section for both models. In those graphics, the radial velocity profiles inside the chimney pipe and in the atmosphere for each height will be shown too.

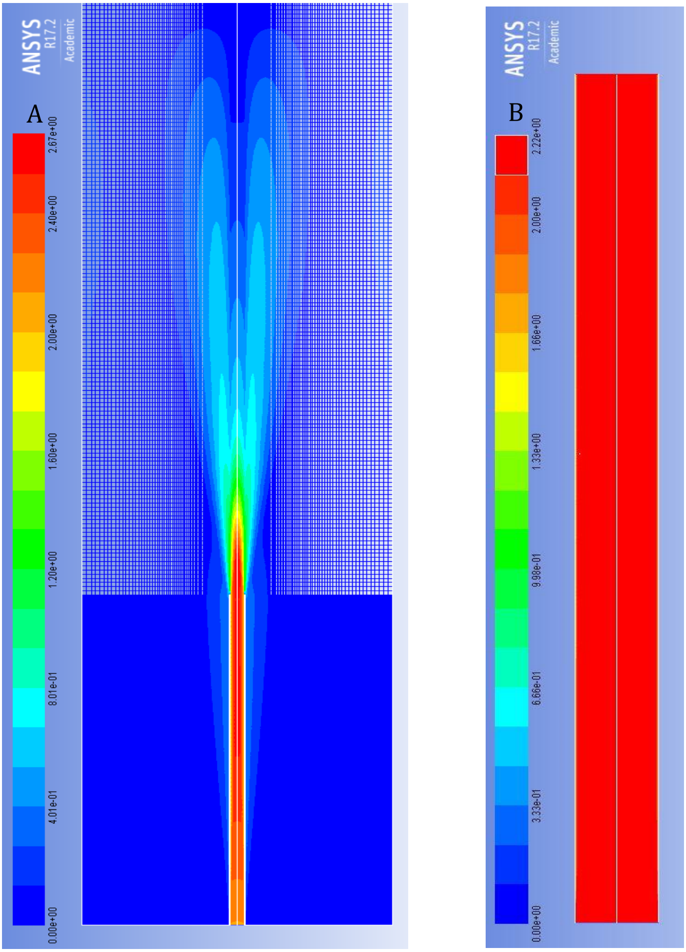

Figure 29. A) Model with atmosphere, velocity distribution inside and outside the chimney, zoom applied (the most important part of the system). B) Model without atmosphere, velocity distribution inside the chimney, (all the system). Axis-symmetry applied

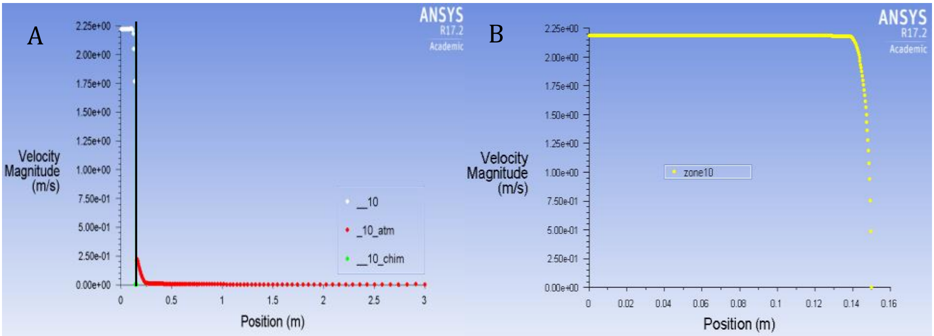

Figure 30. A) Model with atmosphere, 10#, and 10# atm radial velocity profiles inside and outside the chimney. The vertical black line references the chimney wall. B) Model without atmosphere, radial velocity profile of 10#. (These are not comparative graphics)

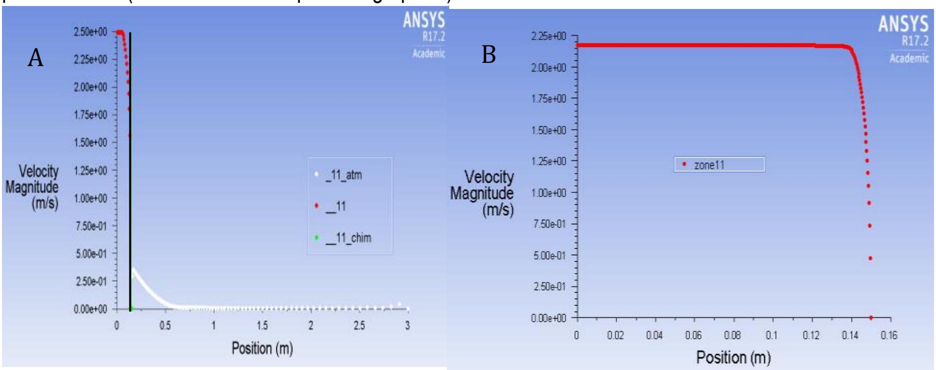

Figure 31. A) Model with atmosphere, 11#, and 11# atm radial velocity profiles inside and outside the chimney, the vertical black line references the chimney wall. B) Model without atmosphere, radial velocity profile of 11#. (These are not comparative graphics

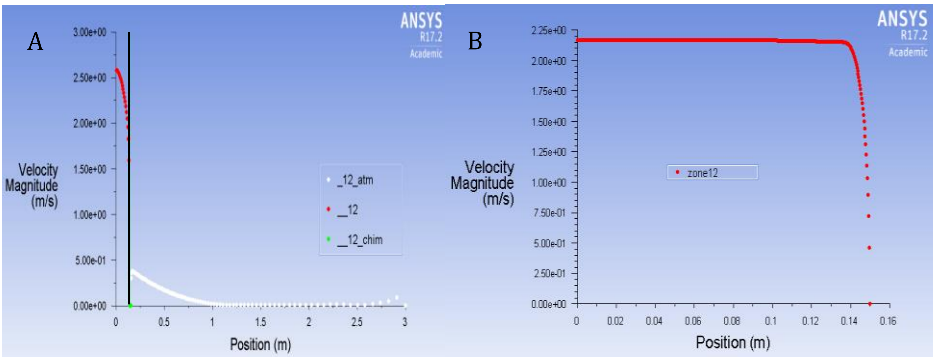

Figure 32. A) Model with atmosphere, 12#, and 12# atm radial velocity profiles inside and outside the chimney. The vertical black line references the chimney wall. B) Model without atmosphere, radial velocity profile of 12#. (These are not comparative graphics)

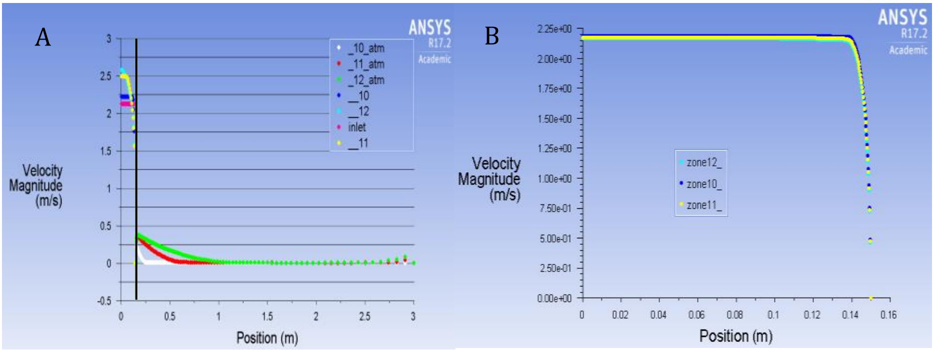

Figure 33. A) Model with atmosphere, inlet, 10#, 11#, 12#,10# atm, 11# atm and 12# atm radial velocity profiles inside and outside the chimney. The vertical black line references the chimney wall. B) Model without atmosphere, 10#, 11#, 12# radial velocity profiles. (These graphics are not comparative between them)

The surface velocity integrals of each height are shown in *table 5* and the surface Reynolds integrals of each height will be shown in *table 6*, to show the flux model.

Table 5: Velocity surface integrals in each height. In this case, there is no comparison between the experimental data (Zhou et al. 2007) and the simulation data, because velocity experimental data is unknown

Height Tag	(1), Atmosphere model. Velocity (m/s)	(2), Non-atmosphere model. Velocity (m/s)	Similarity % between (1) and (2)
Outlet	2.111	2.094	99.19
12# atm	0.030	---	---
11# atm	0.014	---	---
10# atm	0.003	---	---
12#	2.118	2.095	99.71
11#	2.124	2.107	99.71
10#	2.128	2.122	99.99
Inlet	2.130	2.130	100

Table 6: Reynolds surface integrals for both models at each height. There is no comparison between the experimental data (Zhou et al. 2007) and the simulation data, because Reynolds experimental data is unknown

Height Tag	(1), Atmosphere model. Reynolds	(2), Non-atmosphere model. Reynolds	Similarity % between (1) and (2)
Outlet	18.211	17.957	98.61
12# atm	1.132	---	---
11# atm	3.127	---	---
10# atm	2.352	---	---
12#	18.876	17.971	95.21
11#	44.648	18.165	40.68
10#	94.459	18.348	19.42
Inlet	84.798	20.316	23.95

In this case, some additional pictures are built to show the behavior of velocity contours and vectors near the walls in both models. To show this effect, some pictures with a high zoom and vertical format are used. Also, a picture for the air turbulence was built for both models.

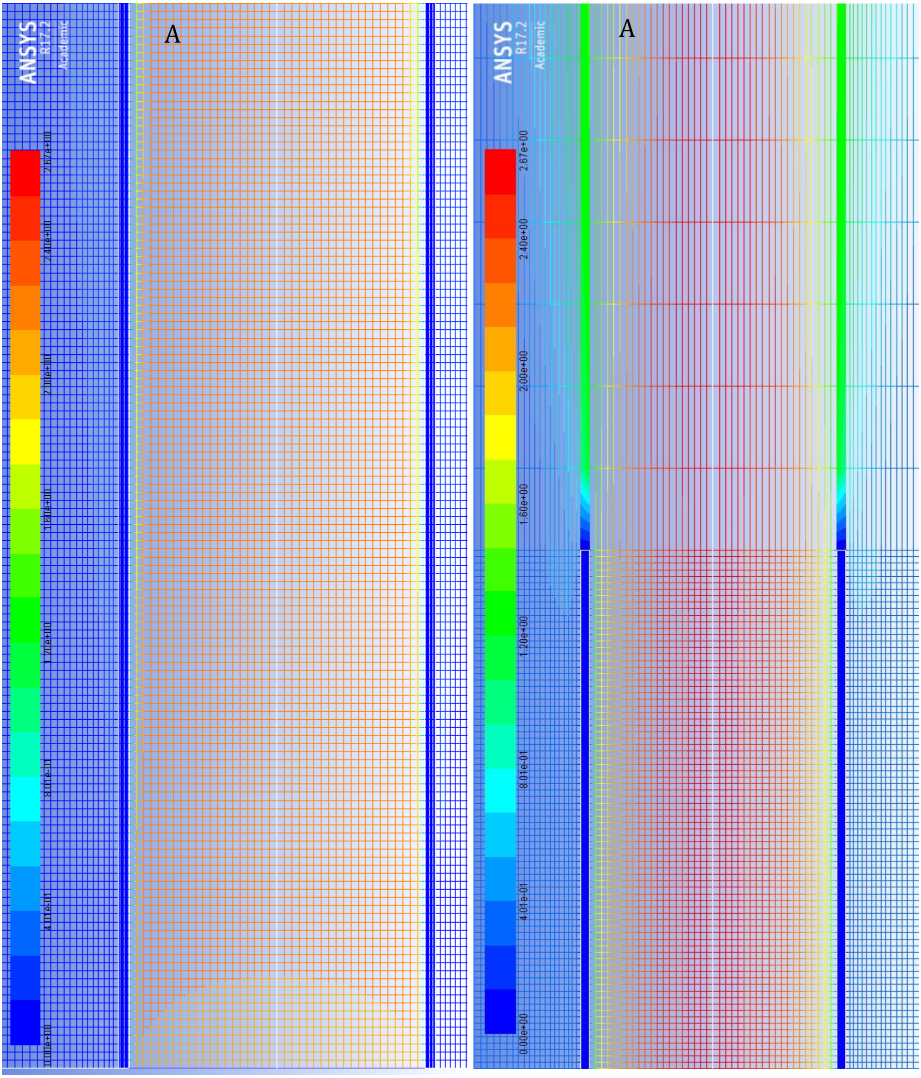

Figure 34. A) Model with atmosphere, velocity contour, zoom applied. Left picture, fluid entrance into the system. Right picture, continuation of the left picture until the outlet. Axis-symmetry applied

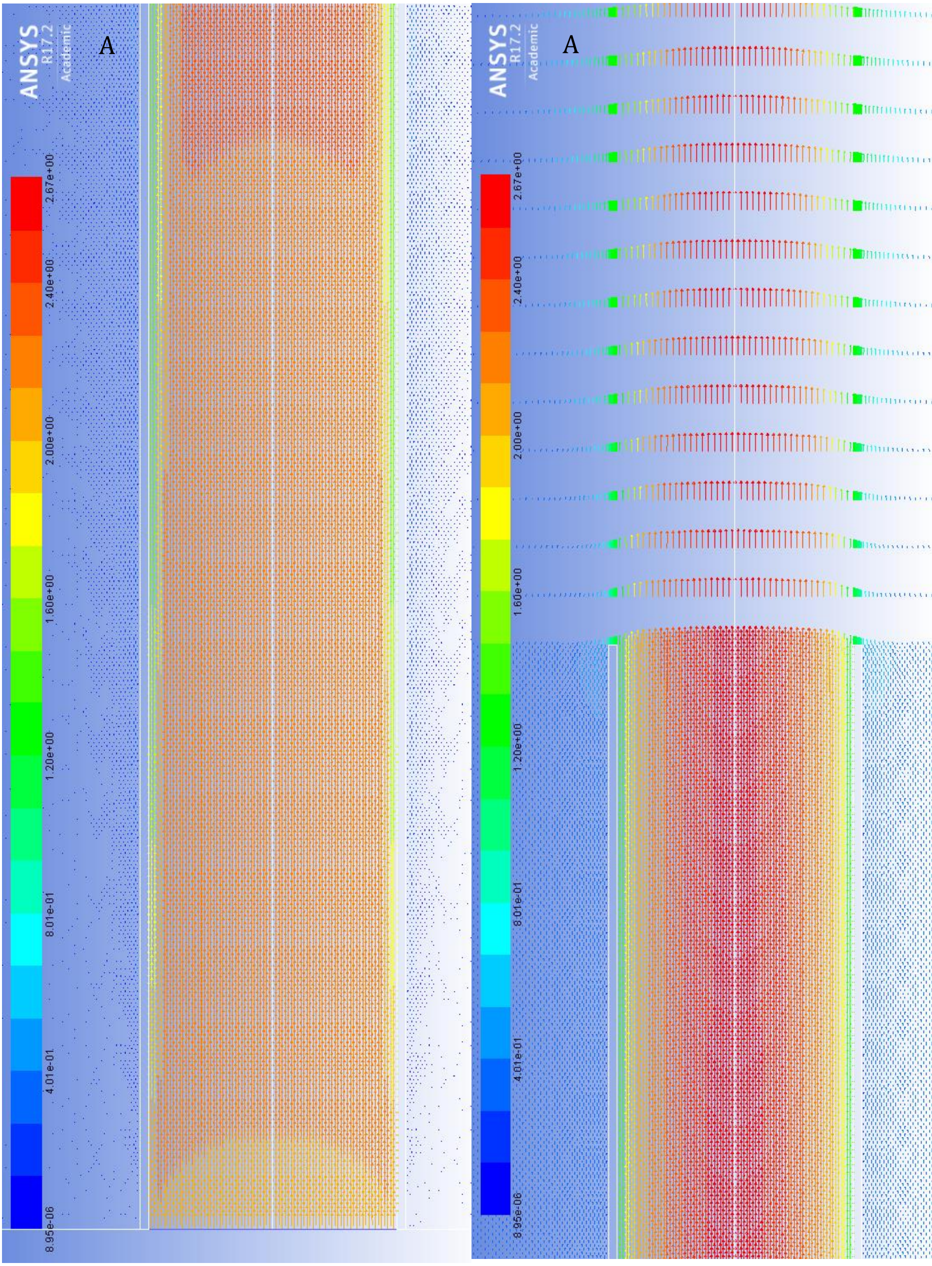

Figure 35. A) Model with atmosphere, velocity vectors, zoom applied. Left picture, fluid entrance into the system. Right picture, continuation of the left picture until the outlet. Axis-symmetry applied

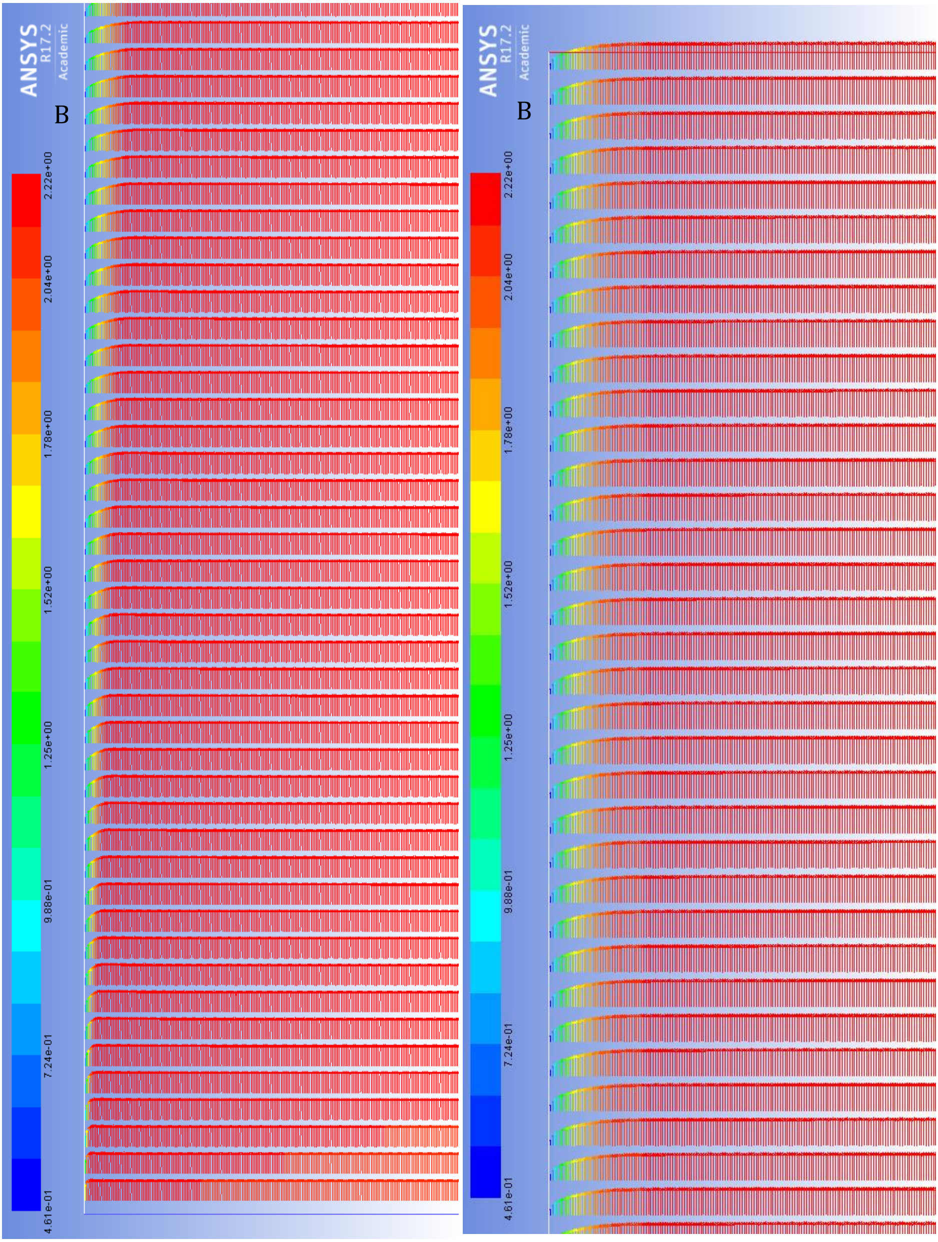

Figure 36. B) Model without atmosphere, velocity vectors, zoom applied. Left picture, fluid entrance into the system. Right picture, continuation of the left picture until the outlet. No axis-symmetric applied, only radial direction

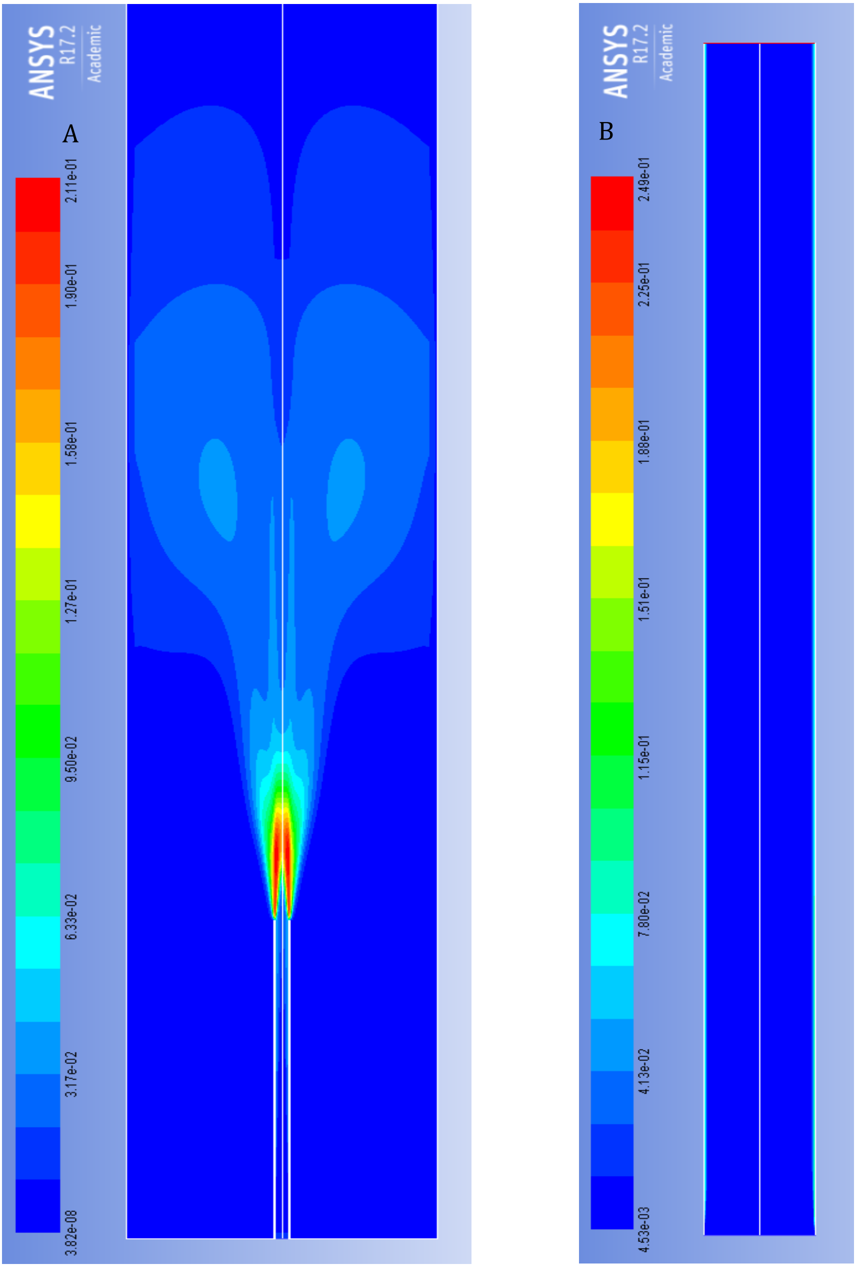

Figure 37. A) Model with atmosphere, turbulence distribution inside and outside the chimney, zoom applied (the most important part of the system). B) Model without atmosphere, turbulence distribution inside the chimney, (all the system). Axis-symmetry applied

The velocity is variable in the whole simulation for both models, because of the density variations. Where the density is closely to be constant like in the middle part of the chimney, (see *Figures 29, 34, 35 and 36)*, the velocity keeps changing. This effect is caused because of the "section reduction". This "section reduction" is caused by the high-density gas near the closest parts of the chimney wall. The heat flux liberated to the atmosphere cause also a decreasing longitudinal temperature distribution near the walls of the pipe. This makes the density become higher there. Therefore, the velocity in these parts is smaller than in the middle ones, creating the above mentioned "static gas wall" near the walls.

In the atmosphere model, if the atmosphere is analyzed in terms of velocity, a poor but real increment of velocity will appear due to the density changes. This updraft advection wins the gravity acceleration. In *Figure 35* the velocity vectors show with their proportional magnitude to the real direction that the flow is going always to the top. This means that there is not reversed flow in this simulation, which means that the gravity effect against a gas is not as important as it is against a liquid. Therefore, in this simulation a lower decrease value in the gas density pushes it up.

The radial velocity profiles for both models can be seen in T*able 5*. The velocity in 10# (_10) is higher that the velocity in 11# (_11) and 12# (_12), due to the friction that the fluid suffers against the chimney walls.

Also, checking the velocity in the atmosphere model, in *table 5,* the velocity in 12# atm (_12_atm) is higher than in 11# atm (_11_atm) and this one is higher than the velocity in 10# atm (_10_atm). This happens because the temperature in the atmosphere grows up when the chimney inner temperature goes down due to the heat flux that is liberated to the atmosphere. Due to this, the density of the outdoor atmosphere gas decreases and, therefore, increases the up-flow velocity due to the density difference.

Looking at *Figures 30, 31, 32 and 33,* in the atmosphere model, a constant velocity zone can be seen. This is caused because the heat flux is not too high to change the temperature of the fluid that is far away from the chimney wall and, therefore, the density which would change the velocity, is not changing too much. This happens in all the heights where there is a wall domain. Where the chimney pipe finishes, the above mentioned "tongue", exchanges heath easier with pure convection heath transfer, decreasing the gas density and increasing its velocity. This can be seen in *Figures 29, 34 and 35*.

In *Figure 37* also a turbulence can be seen outside the pipe in the atmosphere model, but a laminar flow can be seen in the interior of the chimney in both models. That is caused because where there are no walls, the gas that comes from the chimney goes out of it expanding itself and trying to fill the whole infinite volume. In order to check this picture analysis, the Reynolds numbers have been calculated and are shown in the *table 6.* Analyzing that *table 6*, a laminar flow Reynolds values can be shown inside and outside the pipe at low heights in the radial direction.

7. CONCLUSIONS

1- After studying the state of the art of solar chimney simulations it can be stated that this work presents the first simulation of solar chimney with surrounding atmosphere. The atmosphere model of this project is determined as representative, given the similarity between the obtained results and the experimental data (Zhou, et al. 2007). Emphasizing the temperature results obtained in the simulation, in the different predetermined heights, it is concluded that the similarity between these and the experimental data is 99.7%.

2- After studying the state of the art of solar chimney simulations in ANSYS® Fluent, and knowing that it is the model used by other authors. The non-atmosphere model of this project is determined as representative, given the similarity between the results obtained and the experimental data (Zhou, et al. 2007). Emphasizing the temperature results obtained in the simulation, in the different predetermined heights, it is concluded that the similarity between these and the experimental data is 99.5%.

3- Considering that the goal of the study is to compare the results with the existing real data and not to design a device, both models are equally valid, but the atmosphere model is more representative.

4- Validated the temperature results for both models, it is possible to say that the obtained results of density, velocity and turbulence are expected to match with the experimental data (Zhou, et al. 2007).

5- Thanks to the atmosphere model, it has been possible to confirm changes of speed in the air surrounding the chimney given the variation in its density. By overcoming the action of gravity, all the surrounding air generates upward currents.

8. RECOMMENDATIONS

The following recommendations are based on trial and error tests. Also of knowledge obtained throughout the simulation process. And, obtained from other sources of information related to ANSYS Fluent, such as the Workbench operator's manual and the ANSYS® Fluent users guide.

- The use of "URF" relaxation factors is recommended whenever possible (see page 35).

- It is recommended to always carry out 2D solar chimney studies whenever possible, using the symmetry of the systems, to avoid unnecessary node calculation. Especially, in cases where other systems are adapted to the chimney or if it is a case with atmosphere, because of the substantial number of nodes generated.

- It is recommended the square meshing, to facilitate the computation of the results.

- To design a solar chimney it is recommended to do it with a model that includes an atmosphere. With an atmosphere surrounding the pipe, the outdoor possible impacts can be known.

- If it is desired to design a solar chimney in conjunction with a structure, be it a building or an installation, it is recommended to use the model with atmosphere if many nodes are available. In case that enough nodes are not available it is recommended to make the model without atmosphere, since the approximation will be correct enough.

- If it is desired to evaluate the interior behavior of the chimney, against external variations, whether changes of humidity, air velocity, ambient temperature, solar radiation, changes in the composition of the outside air, chimney construction materials, etc. It is recommended to use a model with atmosphere.

- If it is desired to evaluate the interior behavior of the chimney, against changes in temperature, pressure, speed, composition, etc. It is recommended to use any model. The approximations that will give both will be very similar.

9. FUTURE WORK

The project can be extended by different means, among others:

1- Addition:

   a. Of a structure under the chimney, being it a collector, a building, a factory, or a house, attaching to them an entrance of air.
   b. Once a system such as case (a) has been added, an air humidification system at the entrance can be added to it to reduce the temperature of the area and thus to create a more adequate ventilation system. Also, a good idea would be adding a cooling system in the air intake without having to change the composition of it.
   c. Add a turbine to the chimney top to take advantage of the kinetic energy of the fluid and create electrical energy.

2- Behavioral evaluation of the chimney:

   a. Against changes in solar radiation, whether changes in the angle or intensity of incident radiation.
   b. If there are modifications in the building materials of the chimney.
   c. If the fluid were flue gases, instead of natural air.
   d. Changing the indoor flow composition.
   e. Changing the outdoor fluid composition.
   f. Changing the outdoor temperature and humidity.

These ideas are proposed to carry out a much more exhaustive and concrete study of this phenomenon, being able to contribute in the creation of a new concept of ventilation for living homes or industries, applicable anywhere in the world and being completely free, sustainable and natural.

10. REFERENCES AND NOTES

Bernardo Buonomo et al. *Thermal and fluid dynamic analysis of solar chimney building systems.* Via Roma 29, 81031 Aversa (CE), Italy, ResearchGate, 2014.

Hu Siyang, Dennis Y.C.Leung, Jhon C.Y.Chan *Impact of the geometry of divergent chimneys on the power output of a solar chimney power plant.* Pokfulan Road, Hong Kong, China / Shilong South Road, Foshan, Guangdong 528200, China, ELSEVIER, 2017.

Islamuddin Azeemuddin, HussainH Al-Kayiem and Syed I Gilani *Simulation of solar chimney power plant with an external heat source.* Department of Mechanical Engineering, Universiti of Teknologi Petronas, Malaysia, IOPSCIENCE, 2013.

Krisst R.J.K *Energy transfer system. Energy 63, 8-11.* Alt. Sour., s.n., 1983.

Kulunk *A prototype solar convection chimney operated under Izmit conditions.* Miami Beach, Florida, Proceddings of 7th Miami International Conference on Alternative Energy Sources

Md. Mujibur Rahman Ville Vuorinen, Ramesh K. Agarwal, Md Mizanur Rahman *One-Equation Turbulence Model Based on k/epsilon.* s.l., Conference SciTech, 2017.

N. Pasumarthi S.A.Sherif *Experimental and theoretical performance of a demonstration solar chimney model.* s.l., International Journal of Energy Research, 1998.

Sudprasert Sudaporn, Chatchawin Chinsorranant, Phadungsak Rattanadecho *Numerical study of vertical solar chimneys with moist air in a hot and humid climate.* Klontuang, Patumthani, Thailand, ELSEVIER, 2016.

Zhou Xinping et al. *Experimental study of temperature field in a solar chimney power setup.* 1037 Luoyu Road. Wuhan, Hubei 430074, China, ELSEVIER, 2007.

***ANSYS Fluent User's Guide*, ANSYS®**s.l., ANSYS, inc. Southpointe 275 Technology Drive Canonsburg, PA 15317.

11. NOMENCLATURE & SYMBOLS

Nomenclature

A	Area
CFD	Computational Fluid Dynamics
$C_{1\in}$ and $C_{2\in}$	Epsilon coefficients 1 and 2
h_j	Sensible enthalpy of species j
I	Unit tensor
J_j	Diffusion flux of species j
K	Turbulent kinetic energy
k	Karman constant
keff and $\hat{z}eff$	Effective conductivity and stress tensor
P	Absolute pressure
P_k	Turbulent production term
R	Universal gas constant divided by molecular weight
S_h	Heat of chemical reaction
S_{ij}	Strain-rate tensor
S_m	Mass flow
T	Temperature
T= k/∈	Turbulent time scale
T_c	Critical temperature
T_r	Reduced temperature
U	Velocity
u_i	Cartesian mean velocity
$\bar{u}_i\bar{u}_j$	Reynolds stresses
u_T	Friction velocity
V	Specific volume
v_T and v	Eddy and molecular kinematic viscosities respectively
x_j	Cartesian horizontal coordinate normal to plate

Greek symbols

δ_{ij}	Kronecker delta function
∈	Dissipation rate
μ	Molecular viscosity
ρ	Density
σ_K and σ_ϵ	Turbulent Prandtl number
τ	Stress tensor

Tags - Reprint

10# = _10 = zone10_	0.7 (m) chimney height (inside pipe)
11# = _11 = zone11_	3.7 (m) chimney height (inside pipe)
12# = _12 = zone12_	7.7 (m) chimney height (inside pipe)
10# atm = _10_atm	0.7 (m) chimney height (atmosphere)
11# atm = _11_atm	3.7 (m) chimney height (atmosphere)
12# atm = _12_atm	7.7 (m) chimney height (atmosphere)
10# chim = _10_chim	0.7 (m) chimney height (pipe wall)
11# chim = _11_chim	3.7 (m) chimney height (pipe wall)
12# chim = _12_chim	7.7 (m) chimney height (pipe wall)