Tutor/s

Dra. Alexandra Bonet Ruiz
Departament d'enginyeria química

Dr. Joan Llorens Llacuna
Departament d'enginyeria química

Màster en Enginyeria Química

Treball Final de Màster

Multistack immision modelling using CFD

Modelització de la immissió produïda per múltiples xemeneies simulat mitjançant CFD

Antoni Tauler Marimón

February 2016

UNIVERSITAT DE BARCELONA

Cada dia sabemos más y entendemos menos.
Albert Einstein

Vull agrair aquest treball als meus tutors, la doctora Alexandra Bonet i el doctor Juan Llorens, i també al doctor Jordi Bonet per haver-me guiat en aquest projecte tan complicat per mi. El món de la simulació era completament desconegut per mi i la seva ajuda i paciència ha estat inestimable.

REPORT

Index

1. Summary

The use of industrial chimney rise up overall in 20th century, during industrial revolution. Until now, the estimation of values of pollutants emitted by industry were made through atmospheric dispersion equations. These models are very effective, because it is not necessary a lot of information to obtain a pretty good estimation.

Nowadays, due to technological advances, the computer became a powerful calculus engine. Because of that, it has been possible to develop programs to perform more accurate simulations, like Computer Fluid Dynamic (CFD) software. CFD is predicting what will happen, quantitatively, when fluids flow is affected by different variables, like heat, chemical reaction, mechanical movement, mass transfer…

This project is focused on the study of the immission in a multiple stack system. The CFD selected to perform this work is Ansys. Through this, different variables, such as airstream and height of chimney will be changed to study the effect on maximum immission.

2. Objectives

The main objective of this project is modelling the smoke flux using a CFD software to study the immission. Into this one, there are other goals:

- Make optimum mesh to perform an accuracy simulation of the system.
- Study several cases to provide an insight into the behaviour of the immision. Two variables are going to be analysed:
    - Influence of different airstreams velocity on maximum immission
    - Influence of different stack height on maximum immission.

3. Introduction

Nowadays, the pollution of industrialization is one of the biggest problems of humanity. Although, pollution had been known to exist for a very long time (at least since people started using fire thousands of years ago), it had seen the growth of truly global proportions only since the onset of the industrial revolution during the 19th century.

Due to this, people got some commodities and facilities in their daily life. The industrial revolution brought with it technological progress, such as discovery of oil and its virtually universal use throughout different industries. Technological progress facilitated by the capitalist business practices had probably become one of the main causes of serious deterioration of natural resources.

At the same time, the natural sciences was developed, and of course this act led to the better understanding of negative effects produced by pollution on the environment [1].

Figure 1. Real chimneys

The first measure applied in the history to reduce the effect of anthropogenic emissions is by dilutions of the contaminants. Therefore, chimneys can be considered the first measure taken to reduce the contaminants impact. The noxious effect of a contaminant depends on its concentration which are nowadays regulated (threshold limit value) and its effect does not depends on the emissions but the immissions received. Hence, the determination of the immisions around a chimney are of paramount importance to determine its effects on the health and environment. Usually, experimental data about this subject is scarce as it is not desirable for an enterprise to spend efforts to make public available their negative effects on the surroundings.

The monitoring of chimneys emissions is performed nowadays and also there are some stations analysing the quality of the air. Immissions are usually calculated using models such as the Gauss, however more accurate simulation models are available today, e.g. Ansys. The increase of computing power has promoted the apparition of software which can do better simulations solving the microscopic balance. The goal of the present work is to study the chimney immisions using this kind of software [2].

3.2. Methods for immission calculus

3.2.1. Traditional methods

The theoretical model is called Stack Plumes. In the most of diffusion models is applied this theory, which is basically a mass balance. A mathematical model of atmospheric dispersion must be attempt to simulate the gross behaviour of plumes emitted from ground-level or stack-height sources. For localized point sources such a stack, the general appearance of the plume might be represented by the schematic shown in Figure 2 . Although the plume originates at a stack height h it rises an additional height Δh, owing the buoyancy of the hot gases and the momentum of the gases leaving the stack vertically with a velocity vs. Consequently, the plume appears as if it originated as a point source at an equivalent or effective stack height H= Δh+h. The virtual point source may also lie somewhat upwind of the centre line of the stack position, although in most cases the point is assumed to be directly above the stack [3].

The resulting general equation, used frequently as the basis for modeling emissions from continuous point sources of emissions, is:

$$c = \frac{Q}{2\,\pi\,u\,\sigma_y\,\sigma_z}\exp\left[\frac{-(y-y_0)^2}{2\sigma_y^2}\right]\exp\left[\frac{-(z-z_0)^2}{2\sigma_z^2}\right] \qquad (1)$$

Here, Q is the emission strength of the source (mass/time) and u is the average wind speed taken through the plume. Equation 1 describes the change in concentration as the plume travels in the downwind direction x and gradually disperses in the vertical (z) and perpendicular to the direction of travel (y direction). For the completely generalized equation, it is assumed that there is no interference or limitation to dispersion in any direction. Equation 1 is referred to as the general Gaussian dispersion equation [4].

The advantages of the Gaussian plume models are short computation time, a number of scenario scan be quickly run to assist planning, minimal meteorological data required, and predicts maximum hourly concentrations well when time and space variations are not critical. However, some limitations and their implication on model result interpretation have to be taken into account, as does not provide any estimate of variance from predicted values, models cannot track changing meteorological conditions such as rain or fog, cannot treat spatial inhomogeneity like wind shear or terrain specific features and validations show models do not predict hourly observations at a specific time and location beyond the immediate vicinity of the release [5].

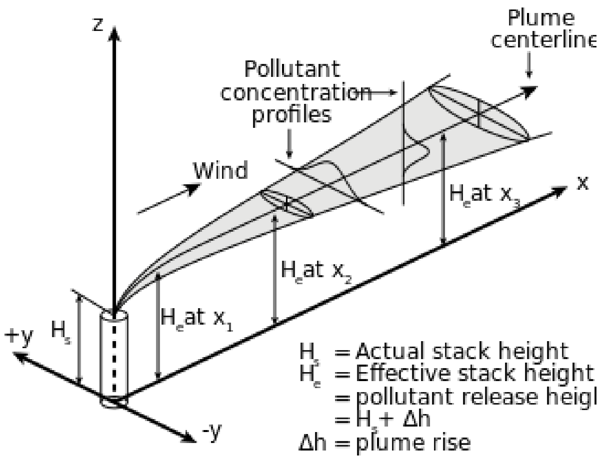


Figure 2. Gaussian plume model

3.2.2. CFD methods

The use of computational fluid dynamics to predict internal and external flows has risen dramatically in the past twenty years. The widespread availability of engineering workstations together with efficient solution algorithms and sophisticated pre- and post-processing facilities enable the use of commercial CFD codes by graduate engineers for research, development and design tasks in industry. The codes that are now on the market may be extremely powerful, but their operation still requires a high level of skill and understanding from the operator to obtain meaningful results in complex situations.

Computational Fluidl Dynamics or CFD is the analysis of system involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulations. The technique is very powerful and spans a wide range of industrial and non-industrial application areas. Some examples are: aerodynamics or aircrafts, combustion in engines and gas turbines, chemical process engineering (mixing and separation, polymer moulding…) etc.

The main reason why CFD has lagged behind is the tremendous complexity of the underlying behaviour, which precludes a description of fluid flows that is at the same time economical and sufficiently complete. The availability of affordable high performance computing hardware and the introduction of user-friendly interfaces have led to recent upsurge of interest and CFD is poised to make an entry into the wider industrial community. Moreover, there are several unique advantages of CFD over experiment-based approaches to fluid systems design [6]:

- Substantial reduction of lead times and costs of new designs
- Ability to study systems where controlled experiments are difficult or impossible to perform
- Ability to study systems under hazardous conditions at and beyond their normal performance limits
- Practically unlimited level of detail of results

Figure 3. 3D draw designed with Ansys

In the present project, Ansys is the CFD program used, so it will be amply explained in the next chapter.

3. Ansys® fundamentals

ANSYS, Inc. is an engineering simulation software (computer-aided engineering, or CAE). It is a general purpose software, used to simulate interactions of all disciplines of physics: structural, vibration, fluid dynamics, heat transfer and electromagnetic. So ANSYS, which enables to simulate tests or working conditions, virtual environment before manufacturing prototypes of products. Furthermore, determining and improving weak points, computing life time and predict probable problems are possible by 2D and 3D simulations in virtual environment. In addition, this software with its modular structure gives an opportunity for taking only needed features.

ANSYS can import CAD data and also enables to build geometry in the software in question. Similarly in the same preprocessor, finite element model (a.k.a. mesh) to solve the underlying governing equations and the associated problem-specific boundary conditions. After defining the values of every material, boundary condition… and carrying out analyses, results can be viewed numerically and graphically.

Considering that ANSYS is a powerful software tool which can carry out advanced engineering analyses quickly, safely and practically by its variety of contact algorithms, time based loading features and nonlinear material models.

Companies in a wide variety of industries use ANSYS software. The tools put a virtual product through a rigorous testing procedure, such as crashing a car into a brick wall, or running for several years on a tarmac road, before it becomes a physical object.

When this CFD project began, it was known what each of the steps requires and be able to plan accordingly:

- Geometry
- Mesh
- Setup
- Solution
- Results

4.1. Geometry modelling

Once the system has been decided, it has to draw and design into the software. In order to do this, there is a tool dedicated only for this function, CAD ("Computer aided design").

Computer-aided design (CAD) is the use of computer systems to aid in the creation, modification, analysis, or optimization of a design. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD is an important industrial art extensively used in many applications, including automotive, shipbuilding, and aerospace industries, industrial and architectural design, prosthetics, and many more [7].

There are much CAD programs on the market, which it is possible to mix with Ansys, but Ansys also has its own CAD function. In this work, only Ansys software will be used due to a greater compatibility with the following steps.

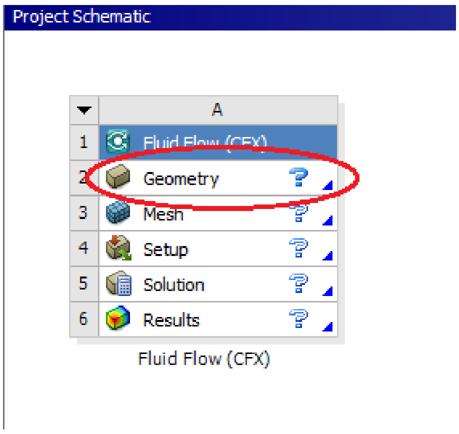


Figure 4. Basic CFX menu (Geometry)

The Ansys designmodeler application is designed to be used as a geometry editor of existing CAD models. The Ansys design modeller application is a parametric feature-based solid modeller designed so that you can intuitively and quickly begin drawing 2D sketches, modelling 3D parts, or uploading 3D CAD models for engineering analysis preprocessing. In CAD systems, features are collections of geometric shapes with which can be add or cut material from a model. DesignModeler application also use features to slice a model into separate bodies for improved mesh generation or to imprint faces for patch loading.

The principal tools to create the most basics structures with DesignModeler are follows:

- Primitives: DesignModeler allows to create models quickly by defining primitive shapes that do not require sketches. All the primitive features require several point and/or direction inputs. These inputs may be defined by either specifically typing in the coordinates or components, or by selecting geometry on the screen. Also, each primitive contains a base plane which identifies the coordinate system in which the primitive is defined. The primitives in DesignModeler are sphere, box, parallelepiped, cylinder, cone, prism, pyramid, torus and bend.

- Sketching: If the design to carry out must be created entirely from scratch, the program offers a section called sketching from which can draw any kind of 2D shape.

- Extrude: Become a 2D structure to a 3D adding the 3rd dimension chosen by user.

Typically, the generation of 3D feature consists of two steps; generate the feature bodies, and merge the feature bodies with the model via Boolean operations. It can be applied five different Boolean operations to the 3D Features:

- Add material: Creates material and merges it with the active modies in the model
- Cut material. Removes material from the active bodies in the model.
- Slice material: Slices bodies into multiple pieces. Active bodies in the Slice operation will be automatically frozen. This option is available when at least one body is present in the model.
- Imprint Faces: Similar to Slice, Imprint faces imprints curves onto the faces of active bodies in the model.

- Add Frozen: Creates material, but adds it to the model as frozen bodies, without merging them with other bodies in the model. This allow, for example, to import a model as a set of frozen bodies without the need to manually aply the freeze feature afterwards.

Finally, when the layout was finished, the tool Named selection allows the creation of named selections that can be transferred to the Ansys Mechanical application [8].

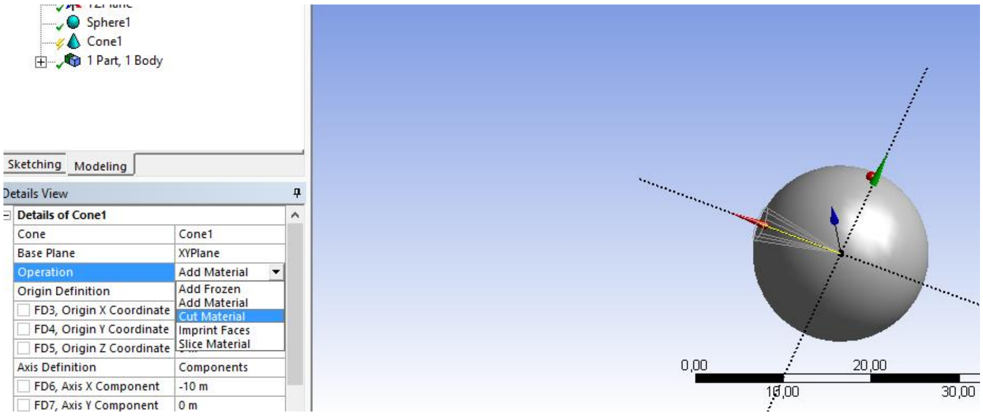


Figure 5. Design Modeler

4.2. Meshing

A mesh is a discretization of a geometric domain into small simple shapes, such as triangles or quadrilaterals in two dimensions and tetrahedral or hexahedra in three. Meshes find use in many application areas. In geography and cartography, meshes give compact representations of terrain data. In computer graphics, most objects are ultimately reduced to meshes before rendering. Finally, meshes are almost essential in the numerical solution of partial differential equations arising in physical simulation.

A structured mesh is one in which all interior vertices are topologically alike. In graph-theoretic terms, a structured mesh is an induced subgraph of an infinite periodic graph such as a grid. An unstructured mesh is one in which vertices may have arbitrarily varying local neighbourhoods. A block-structured or hybrid mesh is formed by a number of small structured meshes combined in an overall unstructured pattern.

In general, structured meshes offer simplicity and easy data access, while unstructured meshes offer more convenient mesh adaptivity (refinement/derefinement based on an initial solution) and a better fit to complicated domains. High-quality hybrid meshes enjoy the advantages of both approaches.
The division between structured and unstructured meshes usually extends to the shape of the elements: two-dimensional structured meshes typically use quadrilaterals, while unstructured meshes use triangles. In three dimensions the analogous element shapes are hexahedra, meaning topological cubes, and tetrahedra. There is, however, no essential reason for structured and unstructured meshes to use different element shapes [9].

In the Ansys mesh, there are different methods that they make this step the strongest. The methods we can find are:

- Tetrahedrons.
    - Patch Conforming (TGrid based) & Path Independent (ICEM CFD based).
- Sweep.
    - Generates prims or hexahedral.
- MultiZone.
    - Mainly hexahedral elements.
- Hex Dominant.
- Cut Cell Mesh.
    - Generates Cartesian Cut Cell mesh.
- Automatic.
    - Combines Tetrahedral Patch Conforming & Sweep Mesh based on complexity of the geometry.
- Interoperability between different meshing methods.

Depend of the geometry, useful or physics requirements we will choose the method that we consider the best, because the "Fluid dynamics" simulations require very high-quality meshes in both element shape and smoothness of sizes changes [10].

In the Outline Ansys Meshing contains three default sections:

- Geometry.
    - Bodies.
- Coordinate Systems
    - Default global & user defined systems.

- Mesh.
    - Meshing operations (control & methods) displayed in the order in which they are inserted.

Once different types of mesh in Ansys are known, one of them must be chosen for the meshing process depending on the form of the geometry.

Finally, the feature of steps to create an accurate mesh in the software are follow:

1. Specify Global Mesh Controls (physics, sizing, inflation, pinch, etc). Allow to specify the mesh based on the physics to be solved.

- Defaults: Set Physics and Solver preferences.
- Sizing: Specify sizing function (curvature, proximity, fixed), mesh sizes, growth rate, etc.
- Inflation: Prims layer growth.
- Assembly Meshing: Assembly meshing method (None/Cut Cell/Tetrahedrons).
- Patch Conforming Options: Tri Surface Mesher.
- Advanced: Advanced mesh parameters.
- Defeaturing: Ignore small features in geometry for improving mesh quality.
- Statistics: View meshes count and mesh quality.

In this step, the features of the global mesh will be added.

2. Insert Local Mesh Controls (sizing, refine, pinch inflation, etc)

- Sizing.
- Contact Sizing.
- Refinement.
- Mapped Face Meshing.
- Match Control.
- Pinch.
- Inflation.

If there are locations in the geometry which require special mesh, this method allows to select a specify type of mesh only for these zones.

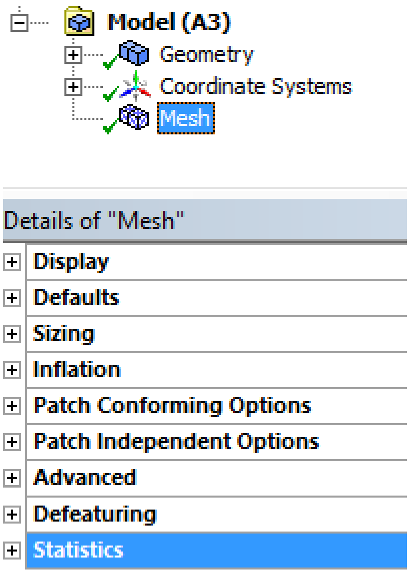


Figure 6. Options menu of mesh design

3. Preview & Generate Mesh (preview surface mesh, inflation)

Entire volume mesh can be generated on all bodies or on individual bodies, as surface or inflation mesh only:

- Allows surface or inflation mesh quality to be checked before volume meshing.

- Not available when using Patch Independent Tetrahedron, MultiZone or Cut Cell methods.

Another important thing is to create named selections:

- Can be created in Meshing by selecting entity(s).
- Entities within a Named Selection must be of the same topology (edge/surface/volume).
- Easy to reselect groups that will be referenced often.
- Listed under Name Selections object Outline.

The Named Selections can be applied to the entities of the same size, type or location, by using selection options.

4. Check Mesh Quality (mesh metrics, charts)

- Displays global Node/Element count and quality:
  - For per-body statistics select body in Tree.

- Quality defined by Metrics:
  - Element Quality
  - Aspect Ratio
  - Jacobian Ration
  - Warping Factor
  - Parallel Deviation
  - Maximum Corner Angle
  - Skewness
  - Orthogonal Quality
- Show min, max average and standard deviation.
- Different physics and different solvers have different criteria for mesh quality.

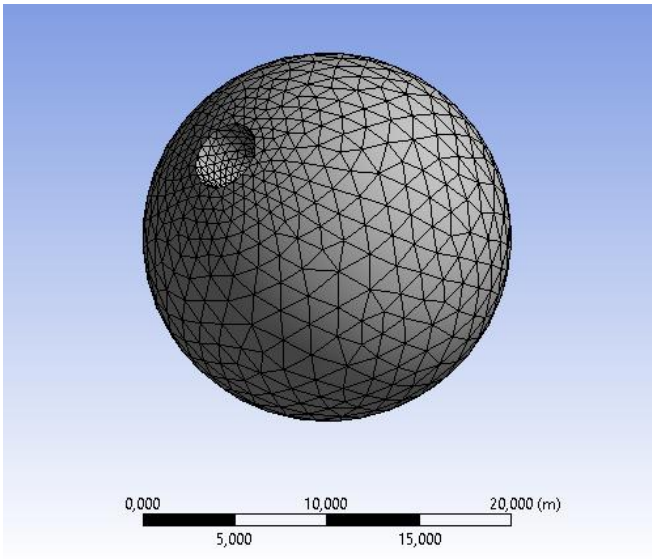


Figure 7. Object with mesh created with Ansys

4.3. Setup

When the layout and mesh of the system have been finished, system conditions have to be defined. In this step of the program are defined the materials, inlets, outlets, temperatures, flow rate… i.e. whole features which defined the case of the project.

Furthermore, in this part the boundary conditions have to be established. In order to allow at calculation engine could solve differential equations, it's imperative to choose correct features of the limits.

Until this point, Ansys used practically the same way to carry out Geometry and meshing in its different tools. From now, it is necessary pick out a determinate module of Ansys in function of the system. In the present project, the module Ansys CFX has selected, because this is designed for work with fluids, heat transfers and chemical reactions.

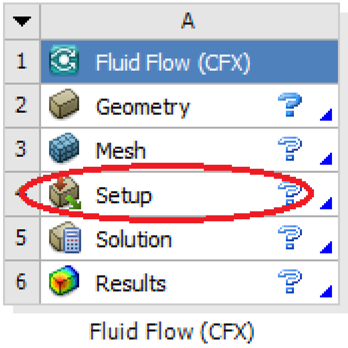


Figure 8. Basic CFX menu (Setup)

4.3.1. Ansys CFX

First of all, it is necessary to define at the program de state of the system (steady or transient) in 'Analysis type' [11].

Then, the materials which will be used at this case (water, air, exchanger fluid…) will be added in Domain section. CFX uses the concept of domains to define the type, properties, and region of the fluid, porous, or solid. Domains are regions of space in which the equations of fluid flow or heat transfer are solved.
Once done, subdomains are added for each boundary limit of the system. For each one, the features have to be defined:

- Inlet (Fluid flows into domain)
- Outlet (Fluid flows out of the domain)
- Opening (Fluid can simultaneously flow both in and out of the domain. This is not available for domain with more than one fluid present).
- Wall (impenetrable boundary to fluid flow)
- Symmetry Plane (Plane of both geometric and flow symmetry)

The two next steps it is possible to add or create new materials which are not in data base and specify the chemical reactions there are in the system. In the present project there are not any reactions or new materials (only air is used), then this section won't be explained in detail.

Finally, in Solver part the features of solving equations are defined. The most important of these ones are follow:

· Solutions unities (kg, m3, atm…)

· Number of iterations (minimum and maximum)

· Variables to study (select the variables to show on results)

The next figure show the CFX setup console.

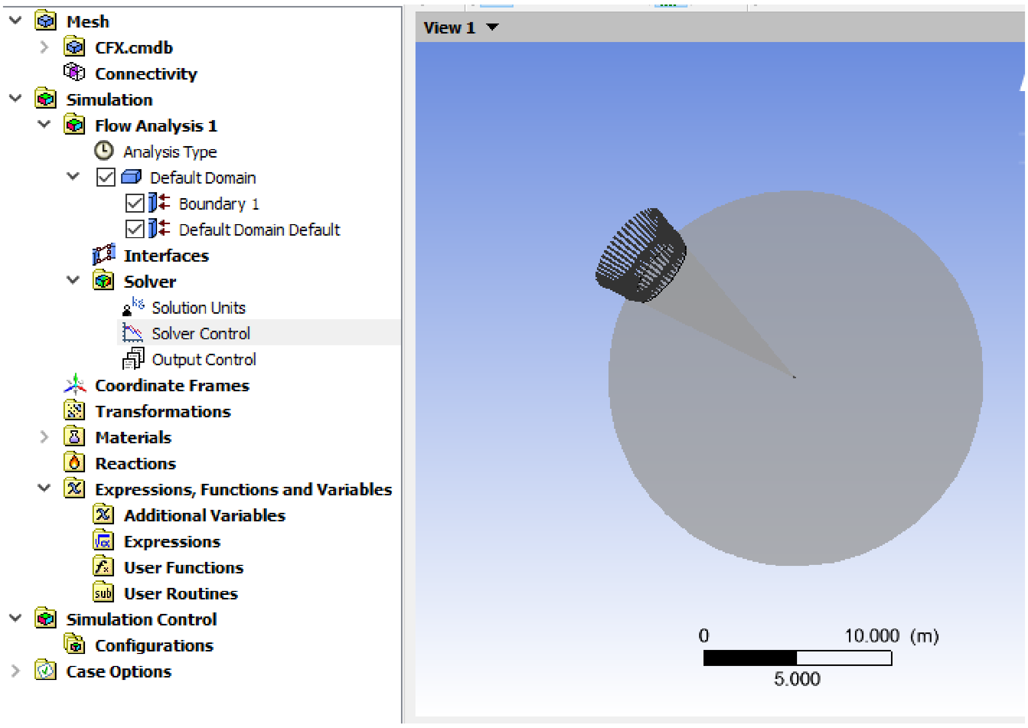


Figure 9. Setup menu into CFX program

4.4. Solver

CFX-Solver is a graphical user interface that allows to set attributes for CFD calculation, control the CFX-Solver interactively, and view information about the emerging solution. Before starting results calculation, it is need to define the simple precision or double precision. Simple precision is acceptable in little systems with simple mesh. On the other hand, double precision is available to permit more accurate numerical mathematical operations. Double precision accurancy might be needed if the computational domain involves a huge variation in grid dimension, aspect ratio, pressure range, etc. [12].

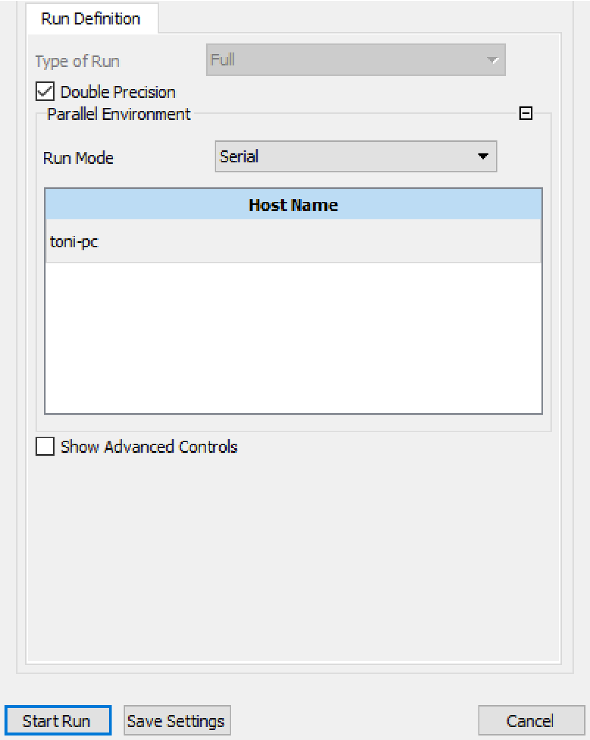


Figure 10. CFX-Solver console

4.5. Results (CFX-Post)

CFX-Post is a flexible, state-of-the-art port processor. It is designed to allow easy visualization and quantitative analysis of the results of CFD simulations.

This section of the program support a variety of graphical and geometric objects used to create post processing plots, to visualize the mesh, and to define locations for quantitative calculations.

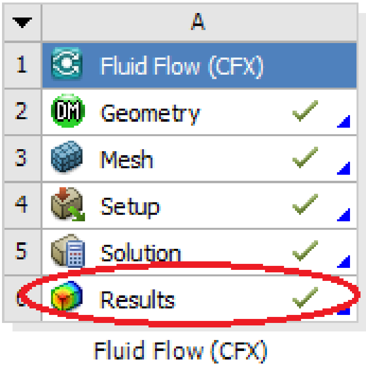


Figure 11. Basic CFX menu (Results)

The Insert menu application in CFX-Post is used to create new objects (such as locators, tablets, charts, etc.), variables, and expressions.

A locator is a place or object that another object uses to plot or calculate values. Some of locators using in this project are followings [13]:

- Point: Used to locate the position of variable minimum or maximum or as an object with which other objects can interact.
- Line: Can exist between two points anywhere inside or outside the domain. Used to analyze the value of the variables trough the line.
- Plane: Two dimensional area that exists only within the boundaries of the computational domain.
- Volume: Is a collection of mesh elements that can be used as a locator for graphic objects or calculations.
- Isosurface: Is a surface upon which a particular variable has constant value, called the level. In CFX-Post, isosurfaces can be defined using any variable.
- Contour plot: Is a series of lines linking points with equal values of a given variable.
- Streamline: Is a path that a particle of zero mass would take through the fluid domain. Streamlines start at each node on a given location.

- Chart: Are graphs that use lines and/or symbols to display data. Charts can be created used on their own or in reports. It is possible to export this data to spreadsheet.

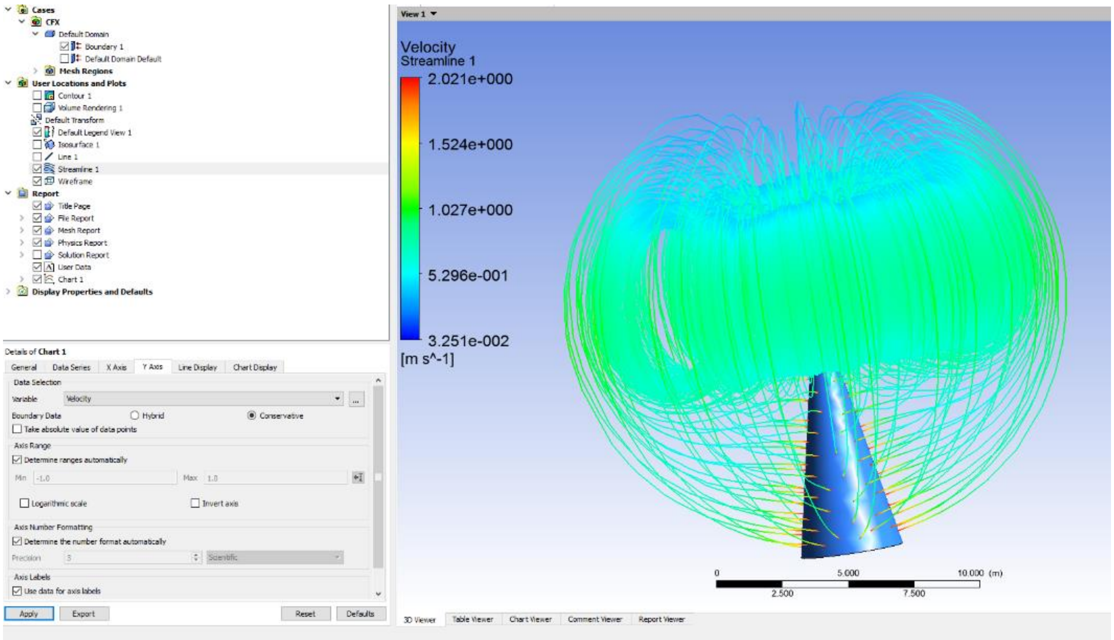


Figure 12. Results screen example

5. Project characterization

Once the Ansys operation has been explained, the next point will be to expose the features of the present study.

5.1. Layout

The design specifications of the system is two chimneys in an open area.

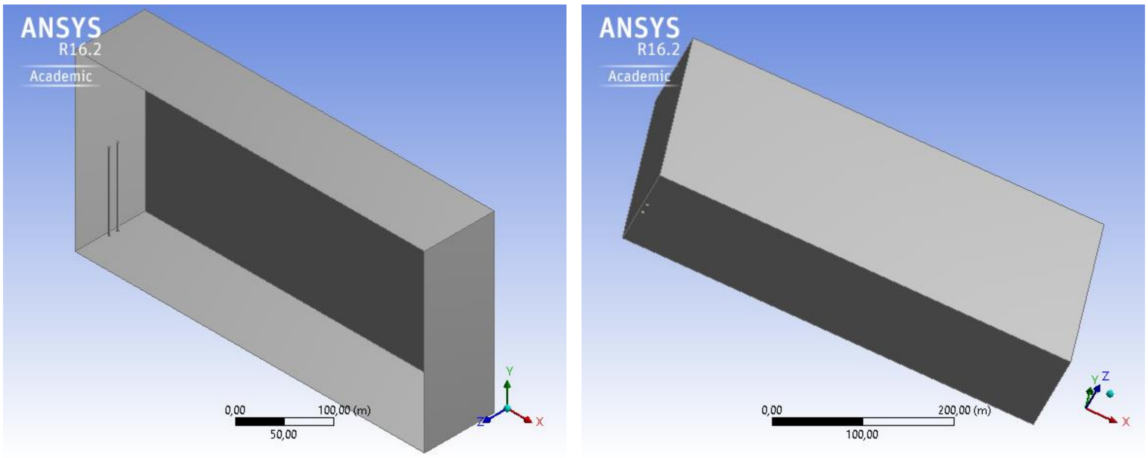


Figure 13. 3D designed by DesignModeler

First of all, a prisms has been created through primitives shapes. This zone represents an open place, where the ZX plane is the base at the prisms and the ground. The others faces are atmosphere.

Then, at the base of the prisms (ZX plane), it create two equals cylinders also with primitives shapes. In this case, cut material option is used because Ansys take into volume of the figure to carry out the mesh and consequently for calculate the equations.

The dimensions of the systems are showed in table 1.

Table 1. Dimensions of the system

Geometry dimensions	
Chimney height (h)	70, 110 or 150 (m). It depends of the case
Chimney radius (r)	1.5 (m)
Y axis	200 or 250 (m)
X axis	500 (m)
Z axis	100 (m)

Where h is the height of the stack, and r the radius. Y is the height of the prism, X the length and Z the width.

5.2. Mesh

Because of the chimney throw out the smoke at high point in the prism, the systems needs a homogeneous mesh for obtain an accuracy result of immission.

It has been necessary to perform an exhaustive work to build this mesh. The distribution of nodes will determinate the accuracy of the simulation. This turns into the mesh an important element of the simulation, consequently the efforts dedicated to perform the mesh has been higher than the other parts. As a result, a mesh with the maximum quantity of nodes allowed has been obtained, which is showed in figure 14.

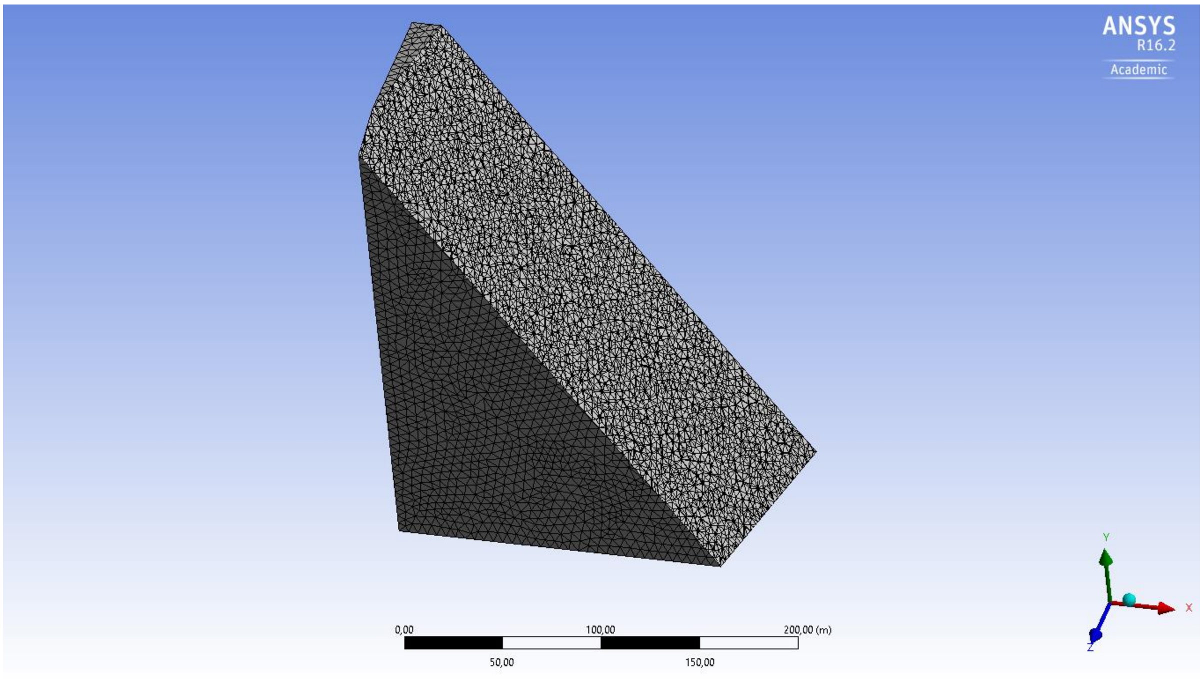


Figure 14. Mesh designed for the simulation

The features of the muilt mesh is illustrated on tables 2, 3 and 4.

Table 2. Body Sizing mesh

Body Sizing	
Supressed	No
Element Size	5 (m)
Behaviour	Soft

Table 3. Mesh Sizing

Mesh Sizing	
Relevance Center	Coarse
Element Size	Default
Smoothing	Medium
Transition	Fast
Nodes	1212158

Table 4. Inflation features

Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
Collision Avoidance	Stair Stepping
Gap Factor	0.5
Maximum Height over Base	1
Growth Rate Type	Geometric
Maximum Angle	140 º
Use post Smoothing	Yes

5.3. Setup

In this part all data characterization are added. These are inlets, outlets and boundaries of the system:

Air

Subdomain inlet. Specified velocity 3, 7 or 10 m/s depends on the case.

Temperature = 298 K.

Ground boundary

Subdomain specified as Wall.

Smoke

Subdomain inlet. Specified velocity in 15 m/s, temperature 600 K and concentration 1 kg/m3.

Atmosphere

Subdomain specified as Opening, and temperature 298 K (the same as air).

The number of iterations is defined in 100 and convergence criteria in 10-4.

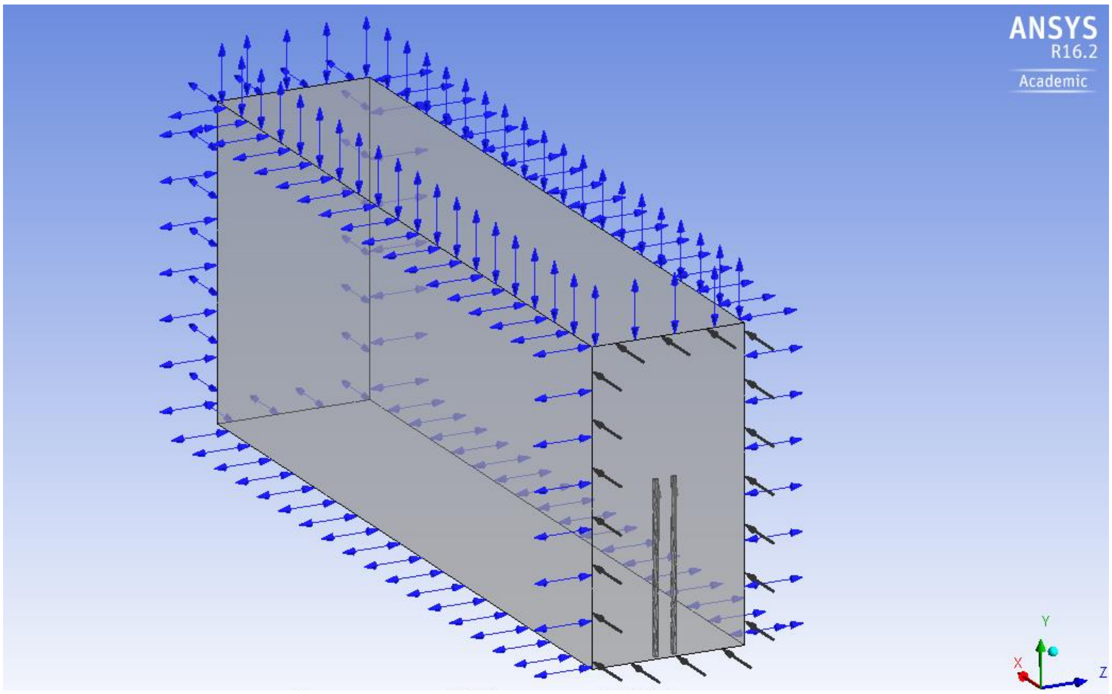


Figure 15. Setup illustration

5.4. CFX-Post

In results, only the line command is used to obtain the values of concentration to the ground. However, others commands, like plane, isosurface and volume, are visually helpful.

Line: Point 1 (X=0, Y=1, Z=50) and point 2 (500, 1, 50)
Plane: ZX plane (Y=1m)
Isosurface: It is just necessary to introduce a value of whatever variable to illustrate this option.

The figures of results will be presented in its corresponding chapter.

6. Results

The results obtained are presented in function of two variables: the height of the chimneys and the velocity inlet of air.

The first of following figures (A) illustrates the scenario. The colormap of the scenario scheme varies together with the smoke concentration on the horizontal plane situated at 1 meter. Therefore, the legend shows the corresponding value of the concentration depending on the colour.

On the other hand, second part of figures (B) illustrates in a more visible manner the variation of the concentration versus the X coordinate.

6.1. Chimney of 70 meters height

The analysis of different wind velocities with 70 meters chimney are shown in the following figures.

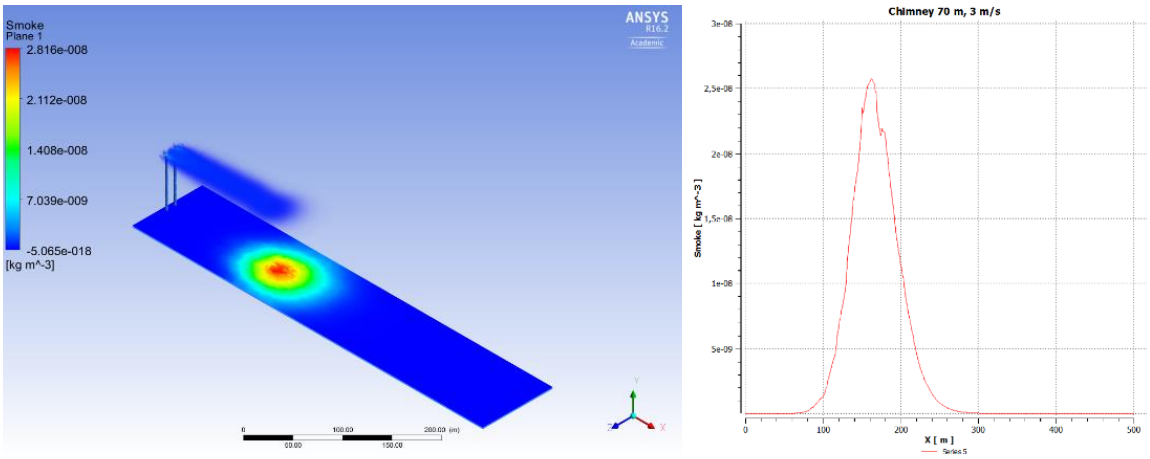


Figure 16. (A) Picutre of Ansys CFX-Post. (B) Graphic of smoke concentration versus X distance

Results of simulation with air velocity = 3 m/s and height chimney = 70 m

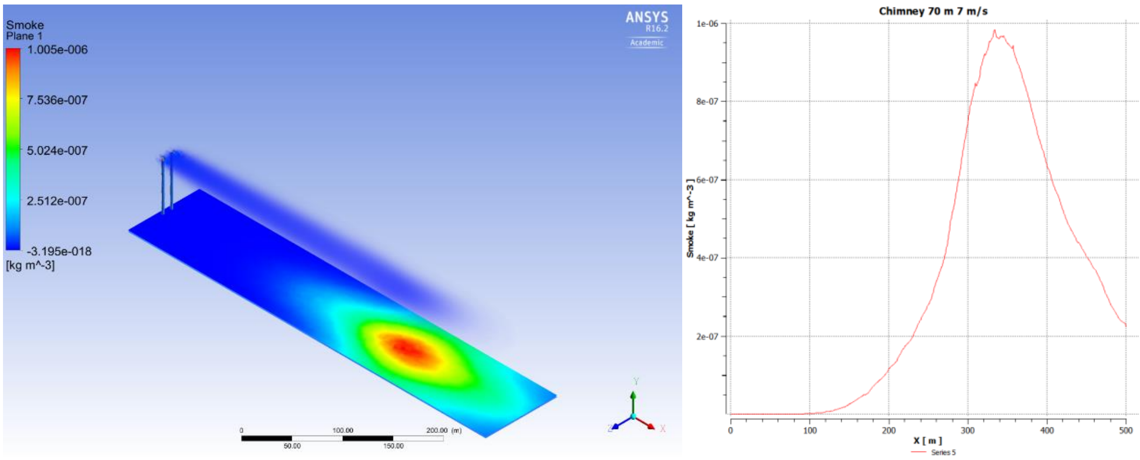


Figure 17.(A) Picutre of Ansys CFX-Post. (B) Graphic of smoke concentration versus X distance

Results of simulation with air velocity = 7 m/s and height chimney = 70 m

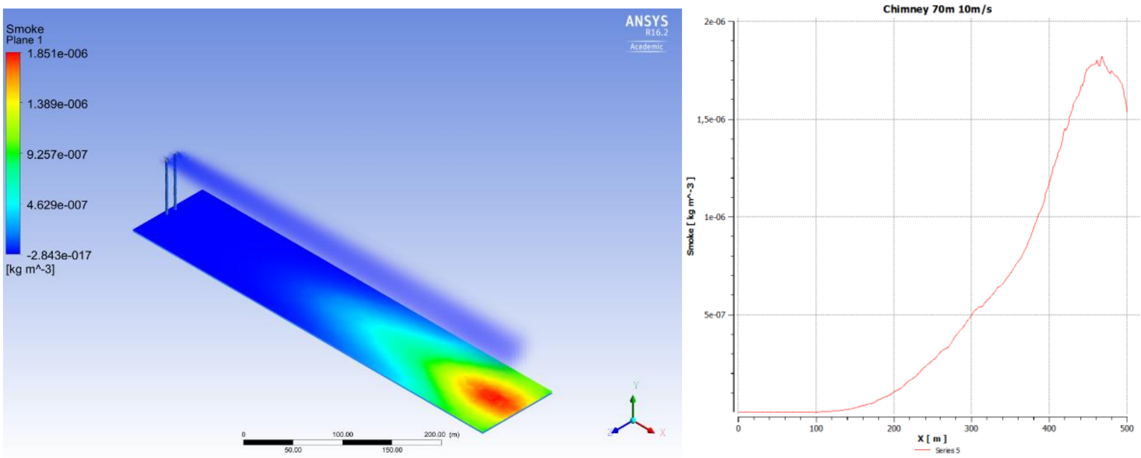


Figure 18. (A) Picutre of Ansys CFX-Post. (B) Graphic of smoke concentration versus X distance

Results of simulation with air velocity = 10 m/s and height chimney = 70 m

The figures show that the variation of the air velocity has a clear influence on the smoke concentration which arrives to the ground. At more velocity, higher is the maximum immision attainable. Table 5 provides the values which reflect the influence of the air velocity, for a chimney of 70 m high.

Table 5. Maximum immission values at 70 meters chimney

70 meter	
Air velocity inlet (m/s)	**Maximum immission (kg/m^3)**
3	2.57 x10^{-8}
7	9.83 x10^{-7}
10	1.82 x10^{-6}

Also, on studied plane, the zones with the same colour can be identified, corresponding to areas of the same value of immission concentration. These zones have a circular shape for smaller air velocity values, but this shape changes to oval as the air velocity is increased.

6.2. Chimney of 110 meter height

The analysis of different wind velocities with 110 meters chimney are shown in the following Figures.

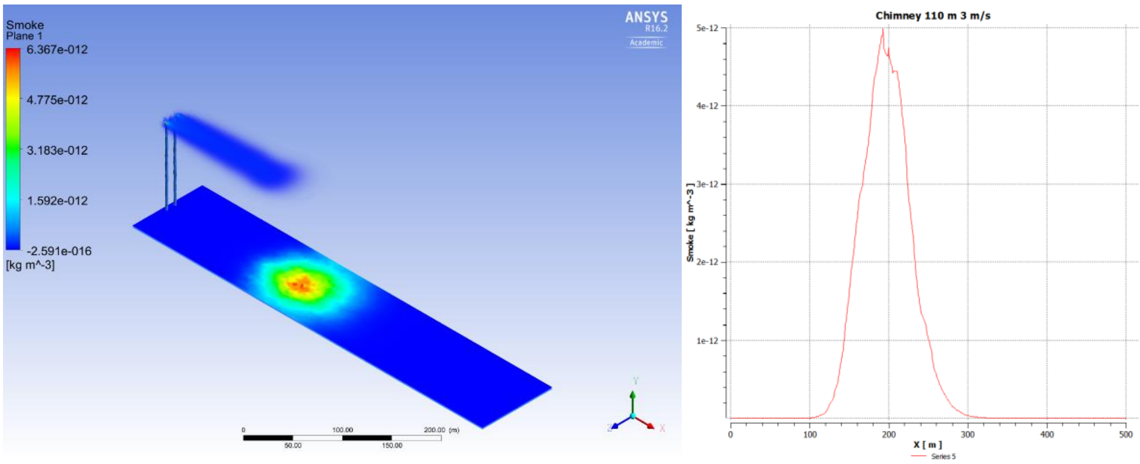


Figure 19.(A) Picutre of Ansys CFX-Post. (B) Graphic of smoke concentration versus X distance

Results of simulation with air velocity = 3 m/s and height chimney = 110 m

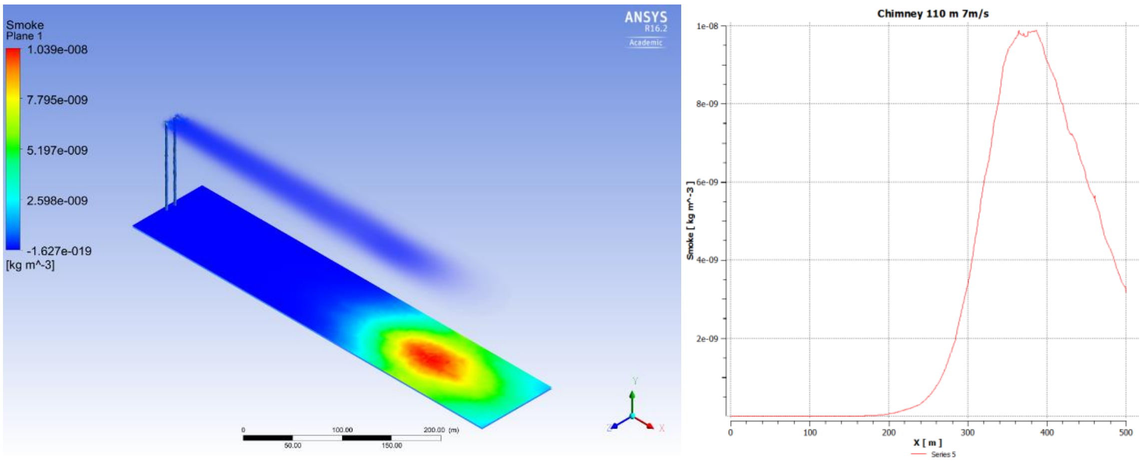


Figure 20.(A) Picutre of Ansys CFX-Post. (B) Graphic of smoke concentration versus X distance

Results of simulation with air velocity = 7 m/s and height chimney = 110 m

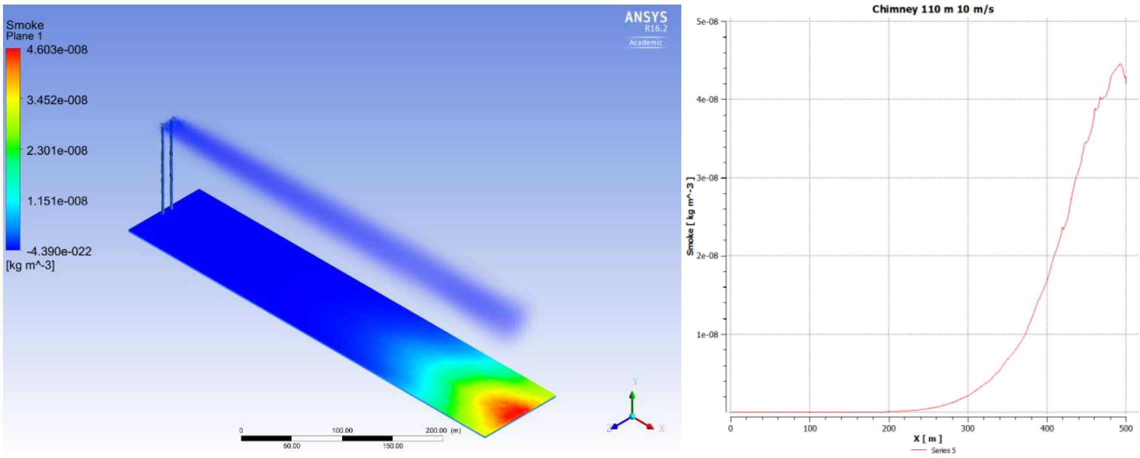


Figure 21.(A) Picutre of Ansys CFX-Post. (B) Graphic of smoke concentration versus X distance

Results of simulation with air velocity = 10 m/s and height chimney = 110 m

In this case, the height of chimneys is increased up to 110 meters. The effect of air velocity keep the same pattern as previous case, because the only difference between them is the height of stacks.

Table 6 provides the values which reflect the influence of the air velocity on the maximum immission obtained, for a chimney of 110 m high.

Table 6. Maximum immission values at 110 meters chimney

110 meter	
Air velocity inlet (m/s)	**Maximum immission (kg/m^3)**
3	5.00 x10^{-11}
7	9.88 x10^{-9}
10	4.46 x10^{-8}

6.3. Chimney of 150 meter height

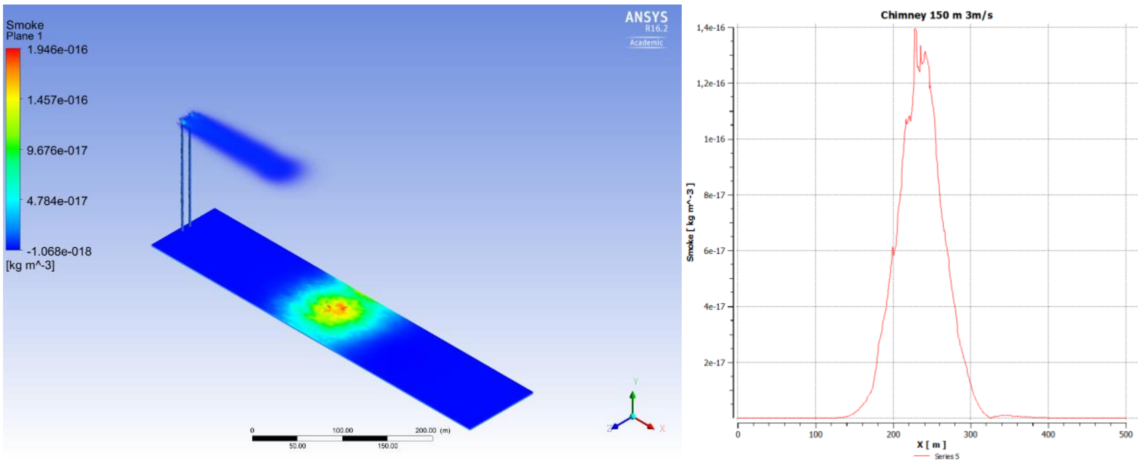


Figure 22.(A) Picutre of Ansys CFX-Post. (B) Graphic of smoke concentration versus X distance

Results of simulation with air velocity = 3 m/s and height chimney = 150 m

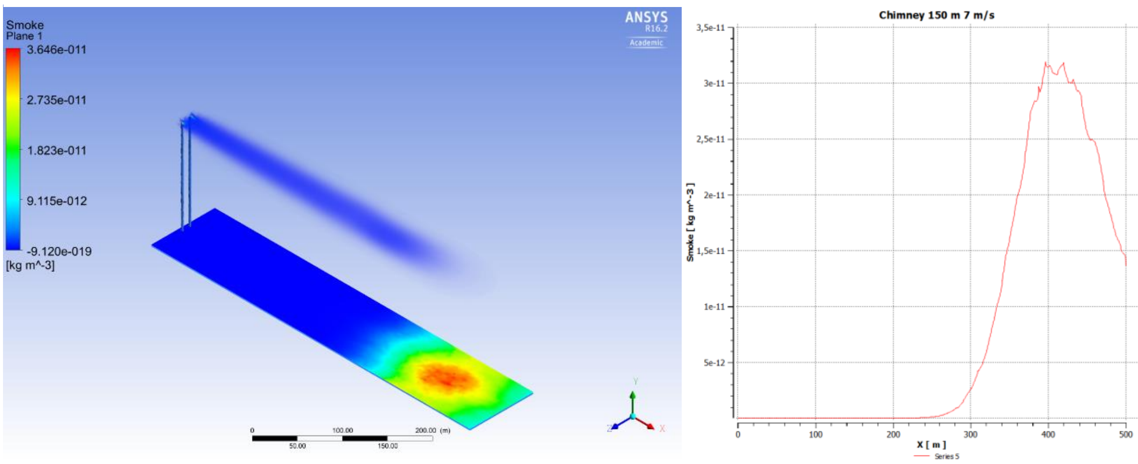


Figure 23.(A) Picutre of Ansys CFX-Post. (B) Graphic of smoke concentration versus X distance

Results of simulation with air velocity = 7 m/s and height chimney = 150 m

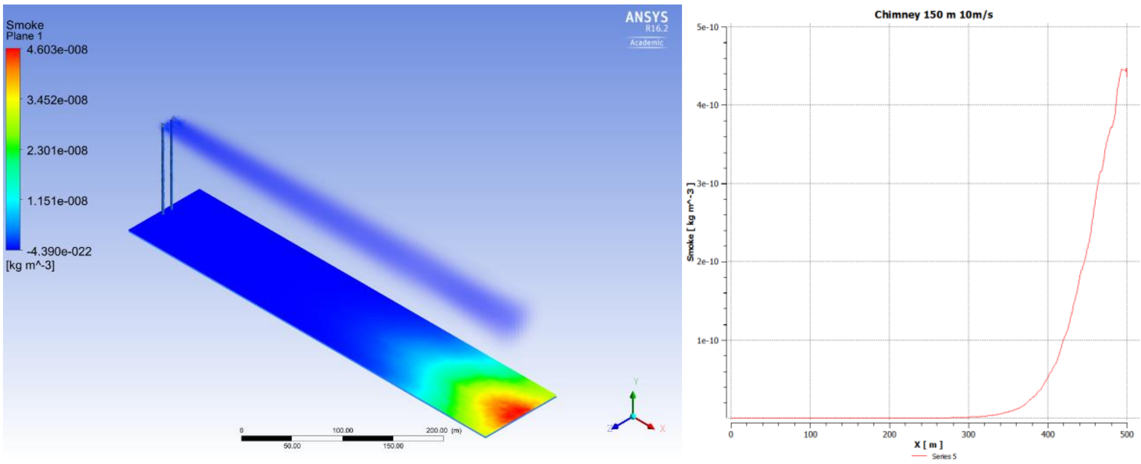


Figure 24. (A) Picutre of Ansys CFX-Post. (B) Graphic of smoke concentration versus X distance

Results of simulation with air velocity = 10 m/s and height chimney = 150 m

Finally, the last height to study is 150 meters. It is confirmed that the tendency observed in the previous two case studies is also respected in the present one.

Table 3 provides the values which reflect the influence of the air velocity on the maximum immission obtained, for a chimney of 150 m high.

Table 7. Maximum immission values at 150 meters chimney

150 meter	
Air velocity inlet (m/s)	**Maximum immission (kg/m^3)**
3	1.4 x10^{-16}
7	3.19 x10^{-11}
10	4.47 x10^{-10}

With this results obtained, as it was expected, the height of the stacks affect the values of the maximum immission attained in each scenario. Analysing the data provided in Tables 1 to 3, the smaller height of chimneys determine higher values of immission, as the immission depends of the height of the emission source. As higher was the emission point, there is more quantity of air between the emission source and ground, so that the smoke dilution depends on this distance.

The hot exhaust gases has rising inertia due its initial higher temperature at the chimney emissions point but later the smoke is entrained by wind that circulated parallel to the ground and dispersed reaching the ground level. An increases of the wind velocity produces the maximum peak immission at a larger distance as expected. However, the time required by the smoke to reach the maximum peak immission decreases and therefore has less time to disperse reaching higher immision concentration as shown in Figures where the air velocity is 10 m/s. Furthermore, the wind produces that the horizontal estabilization of the plume is at a lower high what contributes to higher immision values.

In the last case (150 meters chimney and 10 m/s air velocity) the maximum immission point is situated at limit of the figure. This is because this point directly depends on the height of the stack and the air velocity. But the software license used in this project doesn't allow scenarios longer than 500 m, so it is impossible to analyse velocities higher than this, at least for these altitudes.

The immission zone increases its area and maximum peak concentration accordingly to the wind velocity when it increases from 3 to 10 m/s. This fact is observed for all the chimney hights. As mentioned previously, due to the limitations inherent to the academic licence, it is not possible to check what happens at higher wind speeds, e.g. 20 m/s, because the maximum peak becomes ouside the range of 500 m.

7. Conclusions

Finally, the conclusions of this work are explained:

- Ansys is a powerful calculation engine. The software offers the possibility to select all the parameters and tools needed to personalize the resolution of the system: mathematical methods, geometry of the system, type of mesh, variables (temperature, heat flux, flow rates, materials, inlets, outlets…), number of iterations, units etc.
-
- The height of the chimney has a very important role in the maximum immission evaluation. As more height, the maximum immission attainable is smaller.
- The velocity of the air increases the maximum immission. The smoke doesn't rise up due to the direction and speed of air, then the concentration of smoke at ground level increases.

7.1 Future work

This study lays the groundwork for future projects. People who want to work with smoke emission and immision and its effects on environment and society, have a starting point.

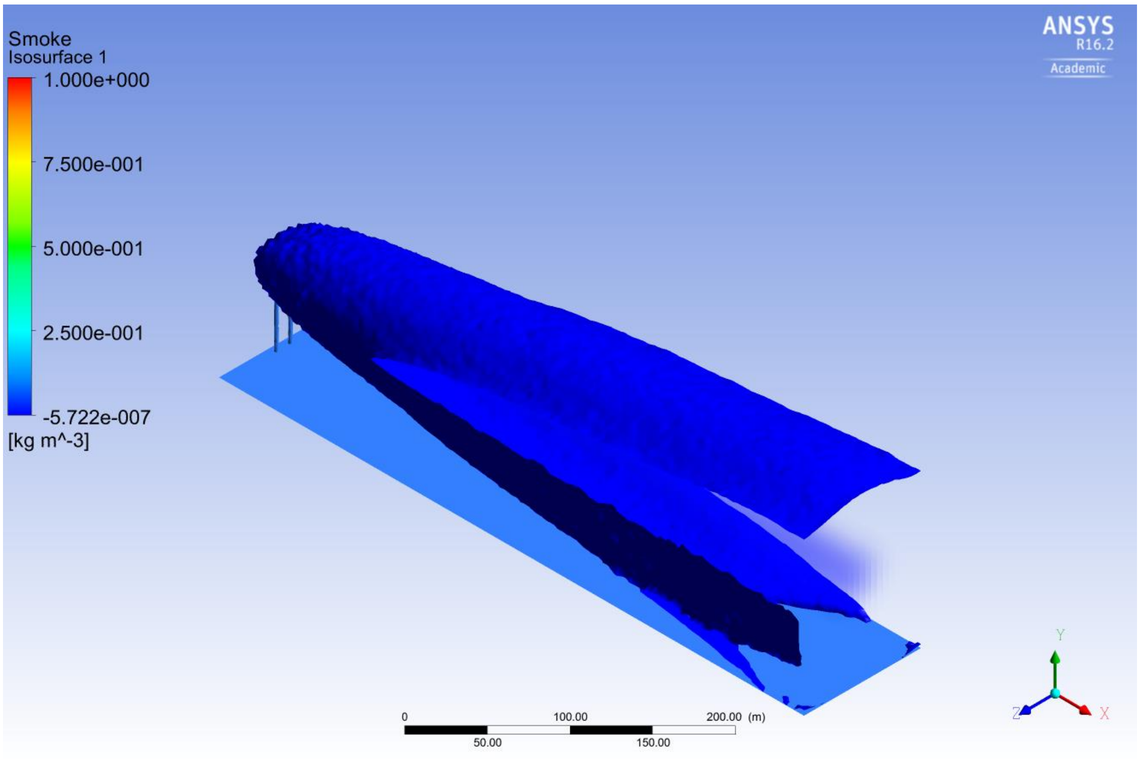
Some examples could be:

- Studies of specific pollutants types: Ansys allows to work with specific pollutants.
- It could be interesting to apply this type of programs (CFD) in emergencies response planning. For example, a program which could predict the direction and concentration of smoke in a fire depending on the weather conditions would help to improve the emergency plan.
- Studies of immissions in different types of geography and weather. These studies could involve scenarios such as beaches, mountains, cold and hot zones etc.

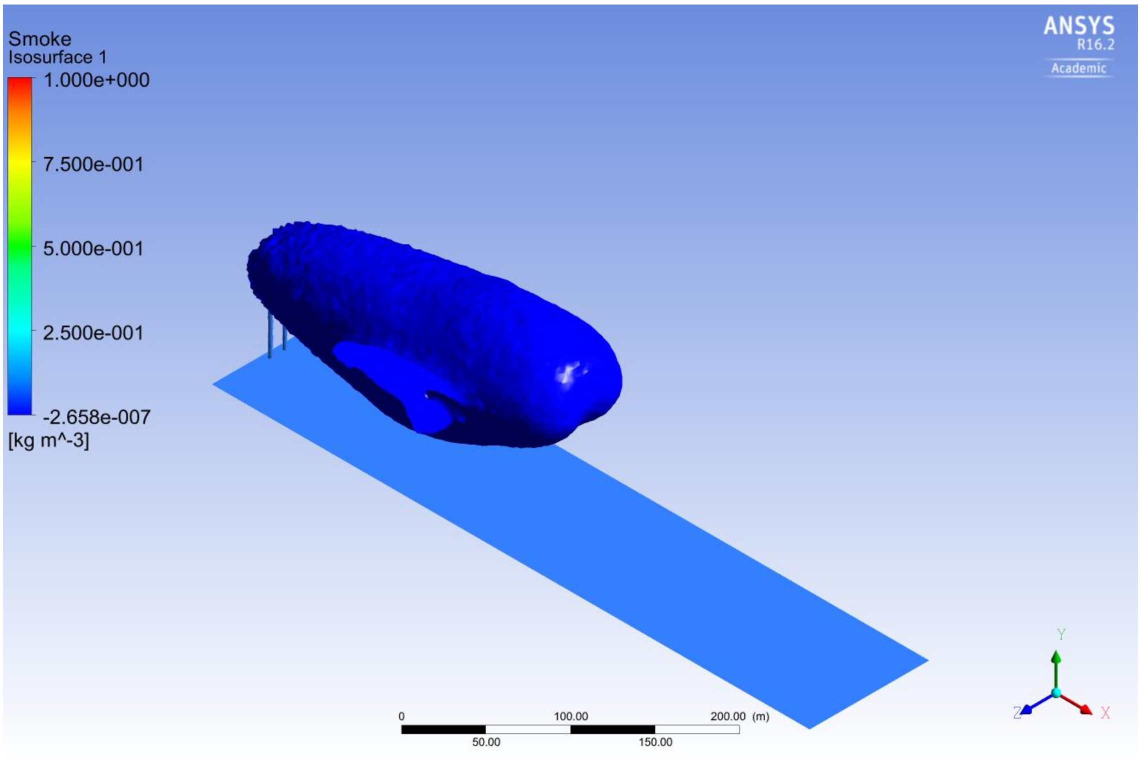
8. References

1. Parker, Albert. *Industrial Air Pollution Handbook*. McGraw-Hill, second edition, 1978.
2. Wayne R. Davis. *Air Pollution Engineering Manual*. Wiley, 2nd edition, 2000.
3. Turner, D. B. (1973*).* *Workbook of atmospheric dispersion estimates*. US Government Printing Office, 5th edition, 1972.
4. Seinfeld, John H., Spyros N. Pandis. *Atmospheric chemistry and physics: from air pollution to climate change*. Wiley-Interscience, second edition, 2006.
5. Meroney, Robert N. *Dispersion in Non-Flat Obstructed Terrain and Advanced Modeling Techniques*. Department of Civil engineering, Colorado State University, 1992.
6. Versteeg, H. K., Malalasekera, W. *An introduction to Computational Fluid Dynamics. The finite volume method*. Longman Scientific & Technical, 1995.
7. Madsen, David A. *Engineering Drawing & Design*. Delmar, 2012.
8. *148.204.81.206/Ansys/150/DesignModeler%20Users%20Guide.pdf* (*DesignModeler User's Guide*). Ansys, Inc. Release 15.0, 2013
9. www.cs.berkeley.edu/~jrs/meshpapers/BernPlassmann.pdf (Bern, Marshall; Plassmann, Paul. *Mesh Generation*). Visited 2016
10. 148.204.81.206/Ansys/150/ANSYS%20Meshing%20Users%20Guide.pdf (*ANSYS Meshing user's Guide*). Ansys Inc, 2013
11. orange.engr.ucdavis.edu/Documentation12.1/121/CFX/xpre.pdf. (*Ansys CFX-Pre User's Guide*). Ansys, Inc 2009
12. orange.engr.ucdavis.edu/Documentation12.1/121/CFX/xthry.pdf (*Ansys CFX-Solver Theory Guide*). Ansys, Inc. 2009
13. orange.engr.ucdavis.edu/Documentation12.1/121/CFX/xpost.pdf (*Ansys CFD-Post User's Guide).* Ansys, Inc. 2009d
14. Mateos Perez, Nuria. *Treball final de Grau. Immission determination from stacks analysed by Computer Fluid Dynamics*. Universitat de Barcelona, 2015

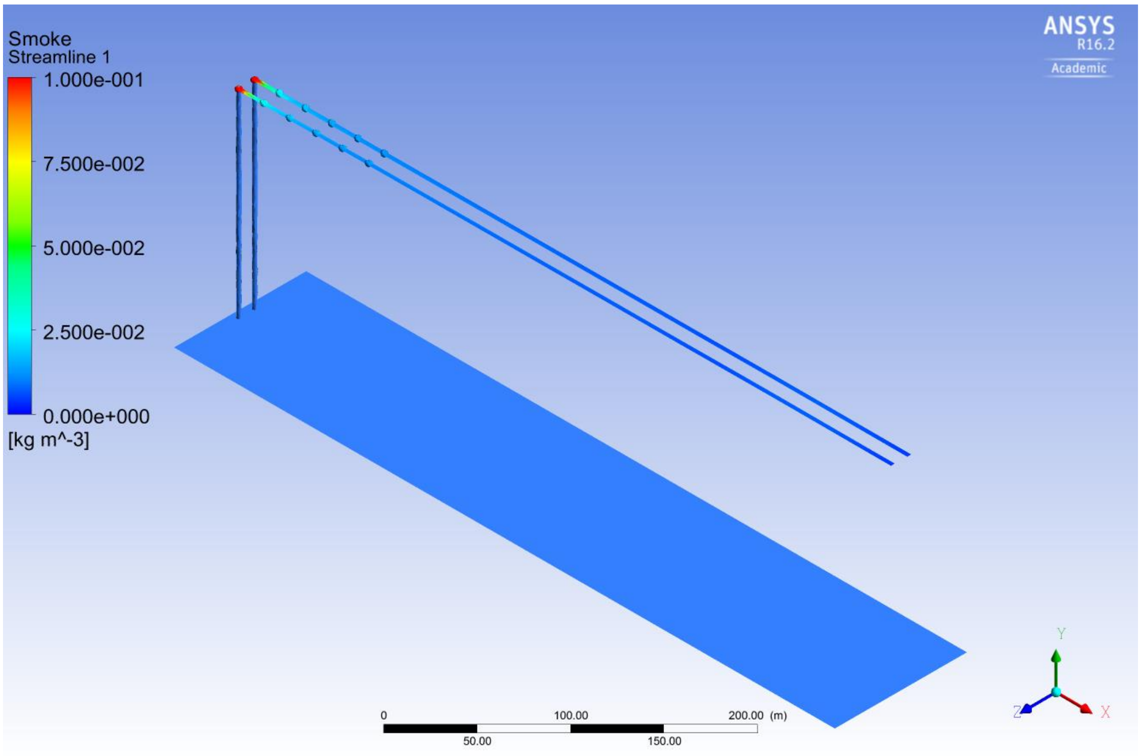
APPENDICES



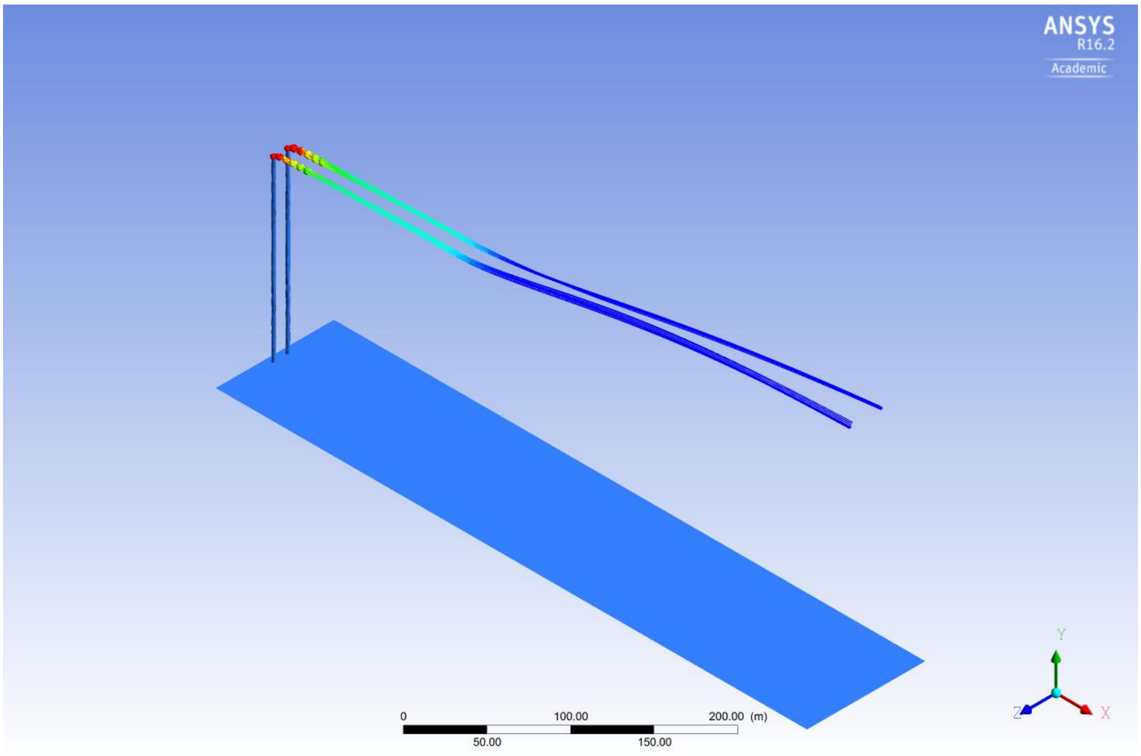
Appendix 1. Isosurface, chimney height = 70 m, air velocity = 10 m/s,, smoke concentration = 1e-6 kg/m3



Appendix 2. Isosurface, chimney height = 70 m, air velocity = 3 m/s, smoke concentration = 1e-6 kg/m3



Appendix 3. Streamline (variable: Smoke). Chimney height = 150 m, air velocity = 10 m/s



Appendix 4. Streamline (variable: Smoke). Chimney height = 150 m, air velocity = 3 m/s