Flow Distribution Inside An Air Conditioning Distributor Using FLUENT Simulation

by

RAGHDA MOHAMED MOSTAFA MOHAMED HEIKAL

Dissertation submitted to the Mechanical Engineering Programme
in Partial Fulfilment of the Requirements
for the Degree
Bachelor of Engineering (Hons.)
(Mechanical Engineering)

SEPTEMBER 2013

UniversitiTeknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak DarulRidzuan

Abbreviations and Nomenclature

TEV: thermal expansion valve

COP: Coefficient of performance

CFD: Computational Fluid Dynamics

LabVIEW : software used for data acquisition, instrument control, and industrial automation

Omega FLR1001: device used to measure the flow rate in air/water

Table of Contents

List of figures

List of tables

CERTIFICATION OF APPROVAL

Flow Distribution Inside An Air Conditioning Distributor Using FLUENT Simulation

by

RAGHDA MOHAMED MOSTAFA MOHAMED HEIKAL

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfillment of the requirement for the
Bachelor of Engineering (Hons.)
(Mechanical Engineering)

Approved by,

____

MS. CHIN YEE SING
Project Supervisor

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
MAY 2013

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

____

RAGHDA MOHAMED MOSTAFA MOHAMED HEIKAL

ABSTRACT

Flow maldistribution and specifically in distributors has been a major area of study in engineering fluid dynamics. This is due to the abundance of distributors in all engineering process and applications, such as chemical processes, solar collectors, microchannels, heat exchangers, cooling equipment and refrigerant distribution in multi-split type of air conditioner. The literature shows some of the work done on the flow distribution in parallel tubes, ejectors and manifolds, and its effect on the pressure drop as well as the energy losses in the heat exchanger. This study aims at investigating (numerically) the non-uniformity of the flow in various tubes inside the distributor, and trying to solve it using various techniques as well as effect of changing the geometry of the distributor. Using FLUENT 14 software to carry out the CFD study in a one-phase flow (liquid). The results obtained show that the distributor inlet should be long enough in order for the flow to be fully developed. The role of the dispersion cone in making the flow uniform is clearly seen. Also the difference in lengths of the outlet tubes, shows that the short tubes have faster flow and better distribution due to less pressure drop along their lengths. And by changing the outlet tubes diameters the flow distribution became almost uniform throughout all the tubes, which follows the continuity equation.

ACKNOWLEDGMENT

I would like to express my deepest, humblest gratitude towards Allah s.w.t for His blessing in my undertaking to complete this Final Year Project. Truly without His blessing, this would never have been accomplished.

I would like to express my special thanks of gratitude to Prof. Vijay, who gave me the golden opportunity to do this wonderful project on Fluid flow/Air Conditioning applications which also helped me in doing a lot of Research and I came to know about so many new things. Also by being my mentor and teacher for three years, helping me find direction and inspiration in the mechanical engineering field.

My greatest appreciation goes to my supervisor, Ms. Chin Yee Sing for her relentless support in helping me complete my final year project. Her supervision and moral support push me to go the length require for me to complete this project. Thank you, Ms. Chin, for your assistance from the beginning of this project until the end.

Not to mention, my utmost thankfulness to a Universiti Teknologi PETRONAS Master Student, Mr. Hilmee Bin Ibrahim, for sharing his experience and insightful knowledge to get me around tight corners in completing my project. Without his full assistance and guidance, this project would have never been able to be completed.

Also, a great recognition and appreciativeness to Universiti Teknologi PETRONAS especially Mechanical Engineering Department for giving me the opportunity to gain valuable experience in research, experimental and simulation works.

Last but not least, special thanks to my family and friends for their never ending support in completing the project. Likewise, thanks to everyone that was involve during the passage of this project. It has been a pleasure.

CHAPTER 1: INTRODUCTION

1.1. Background Information

1.1.1. History of Air-conditioning

The first air-condition was designed by Willis Carrier, a mechanical engineer from Buffalo, New York, in 1902. His design was basically a spray driven temperature and humidity controlled system. His idea of the "Apparatus for Treating Air" was that, chilled coils were used in cooling the air and lowering its humidity to an extent that reaches 55%. The apparatus was even able to control the level of humidity to a desired level, which introduced, now known as modern air conditioning. [1]

1.1.2. Principles of refrigeration and air-conditioning:

Air conditioning/refrigeration is basically based on two important thermodynamics rules,

- The first fact is that for a liquid to change to gas, the liquid needs to absorb heat (evaporation)
- The second is that for a gas to change into liquid, the gas needs to release energy (heat)

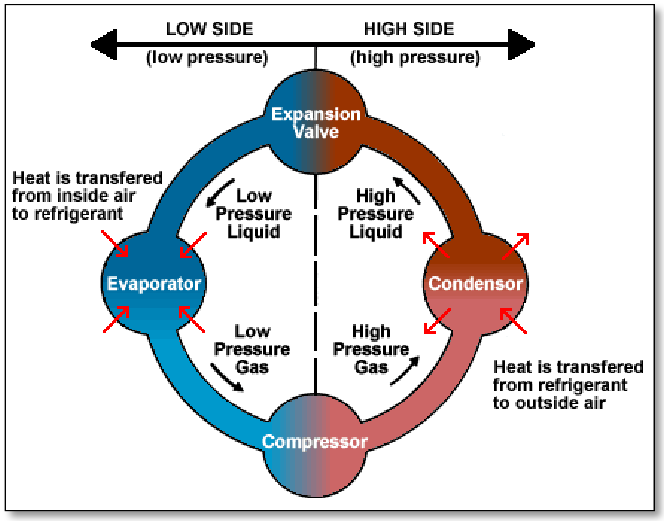

Figure 1: Air Conditioning/Refrigeration Cycle **[2]**

If an air conditioning system were to run efficiently and economically, the refrigerant needs to be used in a repeated cycle (preferably without losses). That is why, the same air conditioning closed cycle is repeated in all the air conditioning systems, where the same refrigerant is used to move the heat from one area (through evaporation) in order to cool it, then it needs to be compressed, then it expels the heat acquired previously (through condensation), then it needs to expand, in order to repeat the same the same cycle. [2]

1.1.3. Heat Exchangers

Heat exchangers are a vintage field of study that was developed long time ago. And since then a lot of research has been carried out in the area, to try to get the best efficiency out of the system, and minimize the energy losses throughout the cycle. A lot of studies have been done on heat exchangers and the maldistribution throughout the heat exchanger tubes.

1.2.Scope of study

1.2.1. Maldistribution in heat exchangers

One of the most crucial aspects of heat transfer is the uniformity of the flow distribution, since it is one of the main key controls of the system performance in heat transfer devices. In the case of heat exchangers, especially with parallel tubes, the performance can be vastly altered due to the flow maldistribution. When it comes to two-phase flow applications, such as, evaporators, maldistribution can lead to dryout, due to the maldistribution of the fluid velocity. [3]

1.2.2. Computational fluid dynamics (CFD)

Computational fluid dynamics (CFD) is a means to study flow and heat distribution through bodies, it replaces the partial differential equations with algebraic equations that approximate the partial differential equations. Solving these equations numerically, gives flow field values at the discrete points in space and/or time.

The CFD simulation solves these equations, at the discrete points for the applicable flow variables, which make up the grid or the mesh of the solution. Then, using interpolation, solutions at the non-grid point locations can be obtained. In other words, CFD is considered to be a numerical experiment [4].

- Discretization:

Discretization is the process that comes up with a set of algebraic equations to replace the partial differential equations. A lot of discretization techniques are available, these include: 1) the finite element (or finite volume) 2) the boundary element 3) the finite difference

For the first method of finite element (finite volume), the flow field is divided into smaller fluid elements. The equations of conservation are written for each element, and the resulting algebraic equations for the flow field are solved numerically.

In the boundary element method, only the boundary of the flow field (not the whole field as in the finite element method) is broken into discrete segments and appropriate singularities such as sources, sinks and doublets are distributed on these boundary elements.

In the finite difference method, the flow field is divided into grid points and the continuous functions (velocity, pressure, etc...) are approximated by discrete values of these functions calculated at the grid points. Then the difference between the function values at the grid points are divided by the grid spacing values is calculated, in order to find the approximated values of the derivatives of the continuous governing functions.

When the continuous governing equations are discretized, algebraic equations need to be used, but that might lead to some percentage of error, as a result to the approximations. This error is called truncation error. In order the truncation errors decrease to minimum, the grid needs to be refined.

- Grids:

When CFD is applied with the finite difference method gives the flow field at discrete points in the flow domain. Then those points are arranged in what is called the grid or the mesh. The solution and its accuracy for a given simulation can be seriously affected by the type of grid developed. The grid must be accurate in representing the geometry, since an error in this representation can affect the solution severely.

- Boundary Conditions:

Boundary conditions are one of the most essential components of the mathematical model. The boundary conditions enable the governing equations to differentiate between different flow fields and produce a unique solution to each of the different geometries of the flow [4]. They direct the motion of flow, specify fluxes (mass, momentum, and energy) into the computational domain. Fluid and solid regions are shown as cell zones. Material and source terms are assigned to those cell zones. Face zones are then use to represent boundaries and internal surfaces. Boundary data are assigned to face zones. There are a lot of boundary conditions types, first are the general ones, pressure inlet, pressure outlet. Second type, are used for the incompressible flow, velocity inlet, outflow. Third type, are the ones for the compressible flow, mass flow inlet, pressure far-field. [5]

1.3. Problem Statement

Maldistribution takes place in the air conditioning distributor, causing undesirable non-uniformity of the flow.

1.4. Objectives

1. To investigate (numerically) the non-uniformity of single phase flow in various tubes inside the distributor.
2. To try various techniques to reduce the non-uniformity or maldistribution across the distributors.
3. To study the effects of changing the distributor characteristics on the maldistribution across the distributors.

CHAPTER 2: LITERATURE REVIEW

2.1.The distributor

The distributor is a device connected to the outlet of the Thermal expansion valve (TEV). The outlet of the distributor is connected to tubes of different lengths, where each of the tubes is connected to one evaporator coil, in the evaporator circuit. The function of the distributor is to distribute the refrigerant flow equally, from the thermostatic expansion valve (TEV) into the tubes of each circuit, of a multi-circuit evaporator coil (as shown in Figure 2). The distributor available geometries is shown in Figure 3.

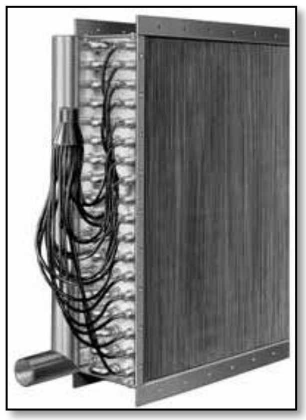

Figure 2: connection of the distributor tubes to evaporator circuits **[6]**

Figure 3: Available geometries of the distributor **[6]**

2.1.1. The distribution problem

The distribution problem happens when a fraction of the liquid refrigerant come through the thermostatic expansion valve (TEV) in two-phase (liquid and vapor) flow at the valve outlet, as (shown in Figure 1). This mixture is dominated by the liquid due to its weight, but the volume is occupied mostly by the vapor.

Another problem occurs, because of the difference in velocities between the liquid and vapor. This is known as slip, as the gravity has a greater impact on the liquid portion of the flow.

If a simple header is used to distribute the fluid flow into each of the evaporator circuits, this might lead to unequal distribution of the refrigerant in the evaporator circuits. The lower circuits will obviously receive higher percentage of the liquid, which might lead to hunting or floodback in the TEV. Consequently, the upper circuits may be starved, reducing the effective evaporator surface, as shown in Figure 4.

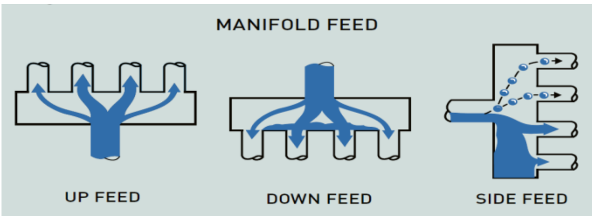

Figure 4: Different orientations for manifold feed/flow **[6]**

In order to get equal distribution, the liquid portion needs to be divided equally in each circuit. The solution to this is to first, mix both portions (liquid and vapor) of the refrigerant flow, second, maintain the mixture in a homogeneous two-phase form until equal portions of the flow are divided into each of the evaporator circuits.

2.1.2. Solving the distribution problem using a distributor

When the two-phase flow leaves the TEV, it then enters the distributor nozzle. The nozzle leads to an increase in the velocity of the two-phase flow, also better mixing of its liquid and vapor components. In addition to that, the nozzle is designed such that the flow should be focused onto the dispersion cone, which will equally divide the mixture into holes spaced equally around the cone. Then the refrigerant is conveyed, through the distributor tubes, to each evaporator circuit. The Pressure drop that takes place across the distributor geometry, leads to the high velocity, which is needed for effective flow distribution. High velocity outlet is the main point behind the distributor. While the pressure drop across the nozzle is what focuses the flow, to provide the mixing required, it also helps balance the flow into equal portions coming out of the holes. As a result, distributor tubing and nozzle sizing is critical to the optimum operation of the distributor. [6]

2.2. The maldistribution

Pacio et al. [7] used two models to predict the effect of maldistribution on performance in parallel channel evaporators. The first neglects the interaction between channels, whereas the second model incorporates an equal pressure drop constraint. After comparing all his models to a homogeneous reference case, remarkable reduction in performance was observed in all cases.

Ranganayakulu et al. [8] used finite element analysis to come up by a mathematical equation, in order to model different types of flow maldistributions in cross flow tube-fin heat exchangers. Lalot et al. [9] used a model of cross flow electrical heater, and concluded that the ratio of the maximum to the lowest velocity in the inlet of the counterflow heat exchanger is about 4.

According to Watanabe et al. [10] most of the heat exchangers, especially those used in refrigeration and air conditioning systems, the refrigerant is distributed using several tubes so as to have high cycle performance and keep the equipment at a small size.

N. Ablanque et al. [3] Conducted two numerical simulation models, where it first considered the phase split and the pressure drop, then considered the thermal and the fluid-dynamic behaviour of the two-phase flow and lastly the global momentum and continuity conservation governing equations. The model helped predict the thermal and fluid-dynamic behavior of two-phase flow system with branching tubes.

Chin et al. [11] studied how the flow maldistribution can lead to great degradation in the thermal performance of a heat exchanger, then based on Taylor series was able to derive a mathematical model in order to describe the contribution of each of the four statistical moments of distribution on the problem.

Wang [12] examined theoretical models and methodology of solutions in flow in manifolds. The main existing models, such as Bernoulli theory and momentum were unified to one theoretical framework. The procedure of design calculation is considered straightforward without much requirements i.e. iteration, successive approximations or computer programming.

Kandlikar et al. [13] developed a novel technique, where the pressure drop at the entrance region was measured in order to monitor the flow maldistribution in individual channels. The validation for the method was by using liquid water flow with four tubes in parallel, and then tested the air flow maldistribution with an experimental setup simulating the two-phase flow in parallel channels. Severe maldistribution was detected in the channels.

The experiment included intentionally induced flow non-uniformity in the tubes, by adding extension tubes at the exits of a few selected tubes. Then the pressure drop in individual tubes is recorded by software called LabVIEW. And the total water flow rate is monitored by device called Omega FLR1001, which is air/water flow meter (used for measuring the flow rate for water and air). Then by collecting the water from the tube outlets and weighing it, the flow rate in each of the tubes can be determined, over a known time-interval after the flow has stabilized.

Ruangtrakoon et al. [14] had an experiment on the effects of the nozzle geometry on the maldistribution. Their results showed that the geometry of the primary nozzle has strong effects to the ejector performance and therefore the system Coefficient of performance (COP).

Gandhia et al. [15] conducted a CFD study, where the flow and pressure distribution of pure steam was examined. Then the uniformity of flow rates among the parallel tubes is compared against the fluid pressure in the system under consideration. Also The effects of design geometries and parameters, such as the tube pitch, header diameter, tube diameter, number of tubes, inlet or outlet pipe diameter, was investigated as well. For validating the CFD models, experiments were conducted on a similar geometry as the CFD, but with a smaller scale, using air and water as a working fluid. As a conclusion, it was found that the tube diameter, number of tubes and their orientation with respect to inlet and/or outlet pipe are the most important design parameters affecting the flow and pressure distribution in the pipeline networks.

Experiment was set as shown in Figure 5:

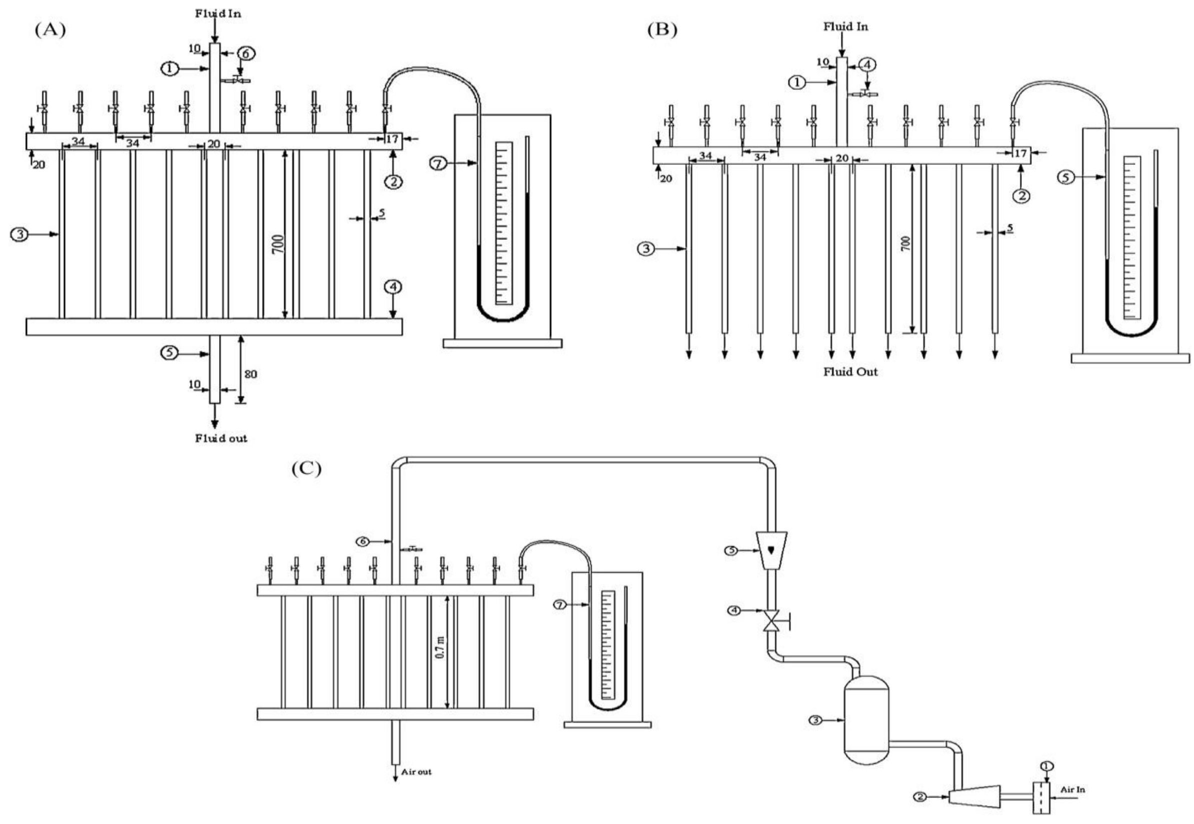

Figure 5: (A) Schematic diagram of the experimental header configuration *'E1'* (B) Schematic diagram of the experimental header configuration '*E*2' (C) Schematic diagram for experimental setup *'E3'* **[15]**

	E1	E2	E3
1	Inlet pipe	Inlet pipe	Air filter
2	Top header	Top header	Surge tank
3	tubes	Tubes	Reciprocating compressor
4	Bottom header	Pinch cock	Control valve
5	Outlet pipe	U-tube maometer	Rotameter
6	Pinch cock		Test section
7	U-tube mercury manometer		U-tube mercury manometer

The results (shown in Figure 6 and 7) show that the variation of geometry affects the velocity profile as proposed by the literature, which is in agreement with this proposed research.

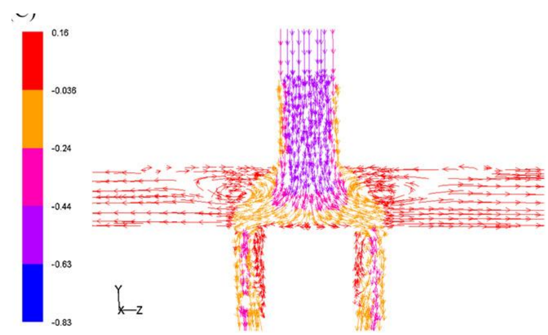

Figure 6: Effect of header diameter variation on the velocity magnitude (m/s) CFD simulations FLUENT **[15]**

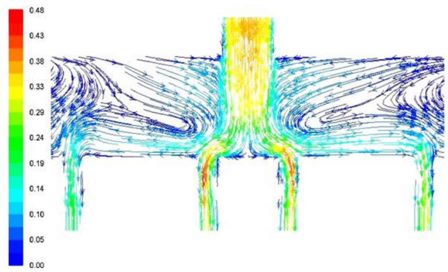

Figure 6: Effect of inlet pipe diameter variation on the velocity magnitude (m/s), CFD simulations FLUENT **[15]**

2.3. The grid independence study

Gandhi et al. [15] used a mixed grid combining both hexahedral and tetrahedral grids. To conduct the grid independence study, he used 4 different grids, 86,000, 150,000, 350,000 and 774,000. After comparing his results using all the 4 grids, he then reached the conclusion that that the grid size of 350,000 is adequate for his case. Coeiho [16] conducted a finite volume method, with an embedded-grid method, then the predictions were compared with data published in the literature. The result was that neither the convergence rate nor the stability of the method was affected by the presence of embedded grids. So as a conclusion, grid-embedding technique can lead to considerable savings in computing time and achieve the same accuracy as when conventional grids are used.

CHAPTER 3: METHODOLOGY

Study the concepts and causes of flow maldistribution

Generate the geometry of the distributor using CATIA, at different parameters; by changing inlet lengths, outlet lengths and diameters.

CFD is used in the commercial package ANSYS 14, and FLUENT 6.3, by using the continuity equation, conservation of mass and energy. Importing the geometry from CATIA to ANSYS 14 workbench

The flow velocity inside the distributor and its geometry is to be investigated.

Using different geometries of the distributor, to see their effect of varying the upstream and downstream resistances on the maldistribution

The simulation is considering only the 1phase flow

Set the boundary conditions ; The inlet velocity is 1 m/s, pressure is 2 bar absolute or 1 bar gauge and the fluid is liquid water at default/room temperature and solving model; Model: k-ε turbulence viscosity model.

Governing Equations: Bernoulli theory and momentum theory, as well as conservation of mass.

Grid is then generated and refined

The grid in imported into FLUENT then the simulation is run until convergence is achieved.

The result of velocity is obtained at the outlet tubes of the distributor

The results are analyzed and interpreted

Discussion, conclusion& recommendations

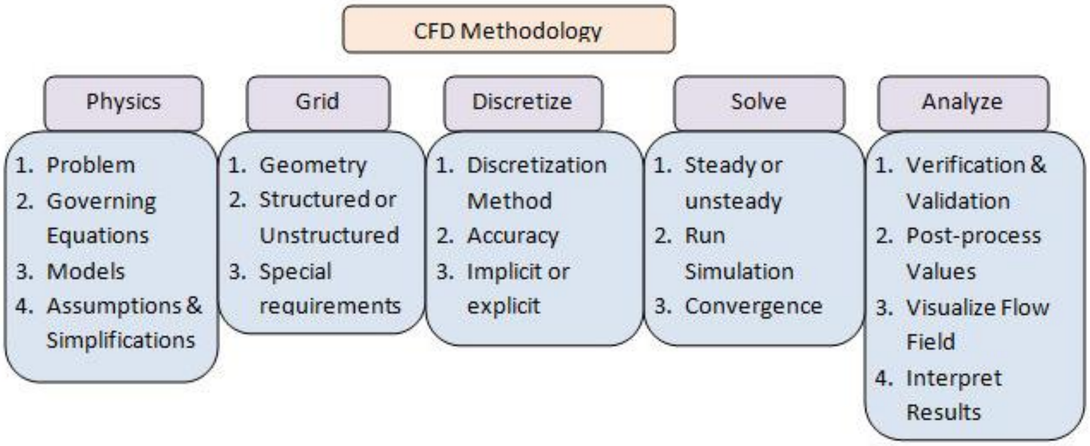

Figure 7: Methodology of CFD

3.1. Physics

1. Problem: Investigate the behavior of the fluid flow for 1 phase and 2 phase flow, leaving the distributor of heat exchanger and how it affects the maldistribution.
2. Governing Equations: Bernoulli theory and momentum theory, as well as conservation of mass.
3. Model: k-ε turbulence viscosity model to govern the turbulence flow phenomena
4. Assumptions:

a. The flow is to incompressible, constant density/specific weight, the fluid is liquid water.

b. Only straight tubes (unbent) are used for simplification.

c. The flow is adiabatic, so there is no need to consider the heat transfer aspect, only the flow is studied, no heat transfer. Thus energy equations are not applied in the simulation.

d. The flow is viscous, using Model k-ε turbulence viscosity model to govern the turbulence flow phenomena.

e. No slip conditions are applied

5. Boundary Conditions and Working fluid properties:

The inlet of the distributor is set as velocity inlet at 1 m/s, pressure at the inlet is 2 bar absolute or 1 bar gauge and the fluid is liquid water at default/room temperature

3.2. Grid

Grid Independence:

Although the solution reached might have converged, but there might be still a probability of error, since it is not known if it is dependent on the mesh resolution used or not. So after the first simulation convergence, the mesh is refined to larger number of cells (or elements) throughout the domain. The values obtained from the first simulation are compared against the values after refinement. If they are the same (within the allowable tolerance), then the first mesh used was accurate enough to capture the result. If they are not the same, then this means the solution is changing according to the mesh resolution, hence, not mesh independent. Thus, the mesh needs to be refined further. Using a smaller cell size, and larger cell (element) number, then after comparing, if the values obtained are similar to the previous mesh, then this means, the previous one is the most optimum for the calculation, if they do not match, then it means, the smallest mesh size available should be used to ensure accuracy and mesh independence.

So it is important to check, to make sure it does not cause any errors, also to find the optimum mesh size to be used in the simulation, so as to set it for next time for the same problem. This will lead to more confidence in the results obtained. [17]

3.3. Discretization

i. Discretization method: finite volume method to convert all governing equations to algebraic form so that those governing equations can be solved numerically.

ii. Explicit or implicit: Density-based implicit solver was selected to solve the governing equations.

3.4. Solve

i. Steady or unsteady: The model is steady, since there is no change with time.

ii. Convergence: the simulation is run until the solution reaches convergence, otherwise, the grid may need to be changed or the model and boundary layers checked.

3.5. Analyze

The simulated model is to be compared to an already established experiment or a similar simulation, or a benchmark, in order to validate the results and see if they follow the expected trend.

By using where the conservation of mass as well as the conservation of volume equations at the inlet and outlet are valid, the outlet flow rate and the inlet flow rate can be calculated, then they can be used as a validation benchmark to the results obtained using FLUENT 14.0.

CHAPTER 4: RESULTS & DISCUSSION

4.1. Geometry Generation in CATIA V5

The first geometry was generated in CATIA V5 then imported in ANSYS14 workbench to be used in FLUENT 14.0 as shown in the Figures below, using the dimensions shown in Table 1

Table 1: Parameters for the distributor

	Parameter	Value (mm)
1	Inlet diameter	120
2	Inlet length	100
3	Number of outlet tubes	9
4	Diameter of tube set 1	30
5	Diameter of tube set 2	27-30
6	Diameter of tube set 3	27-30
7	Length of tube set 1	200
8	Length of tube set 2	400
9	Length of tube set 3	600
10	Diameter of cone	120
11	Height of cone	130
12	Diameter of cylinder	240
13	Height of cylinder	80

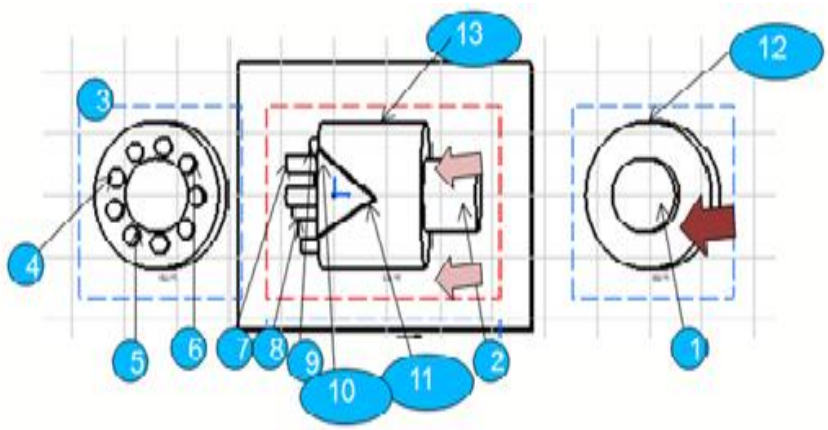

Figure 8: Schematic diagram and dimensions of the distributor

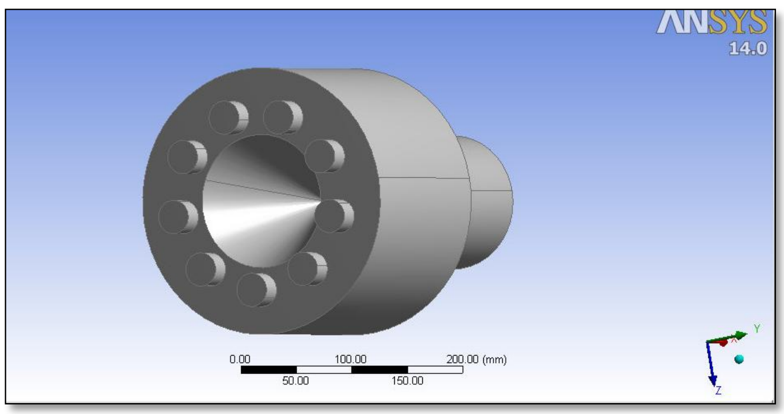

Figure 9: Geometry of the distributor as produced by ANSYS 14

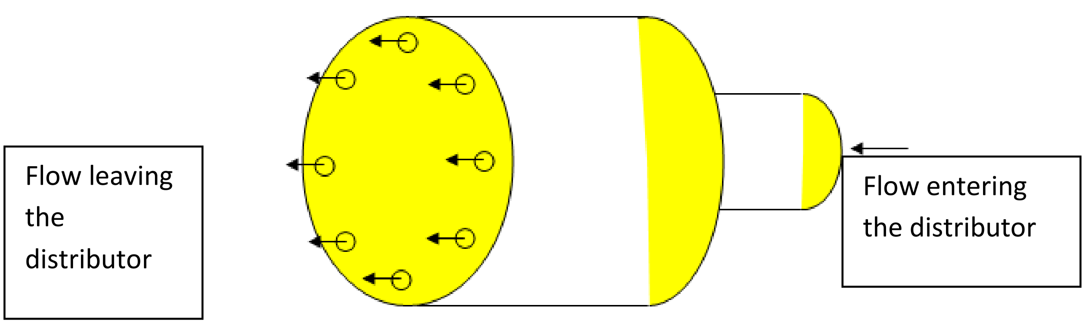

Figure 10: Direction of flow inside the distributor

4.2. Grid Independence Study

For the sake of the grid independence study, three different grids were generated, the first (coarse) with 1799 elements, the second (medium) with 21232 elements and the third (fine) with 118428 elements,

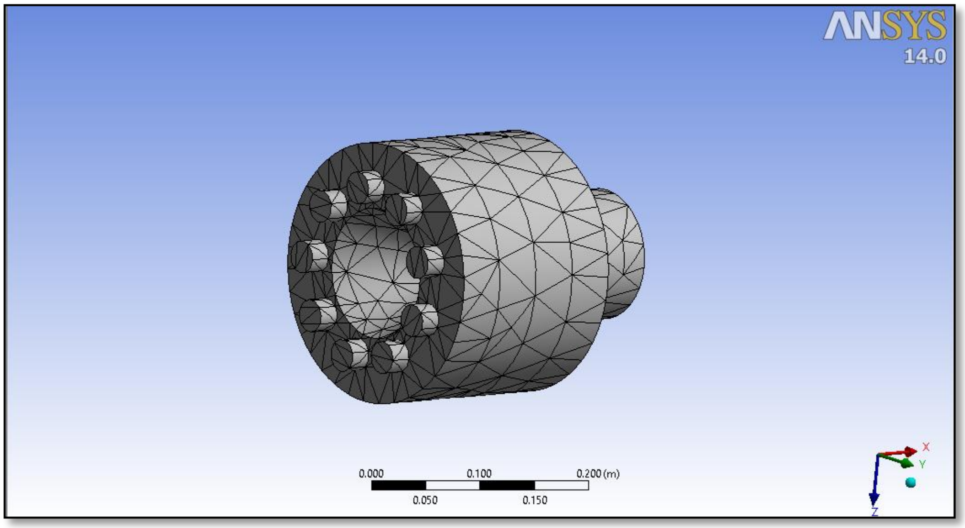

Figure 11: Mesh produced with 3324 Nodes and 1799 elements

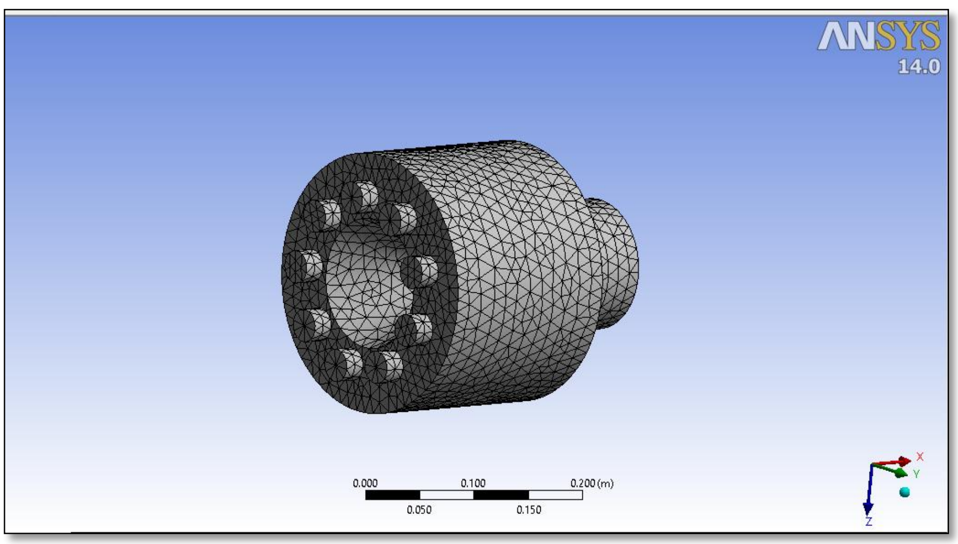

Figure 12: Mesh produced with 32382 nodes and 21232 elements

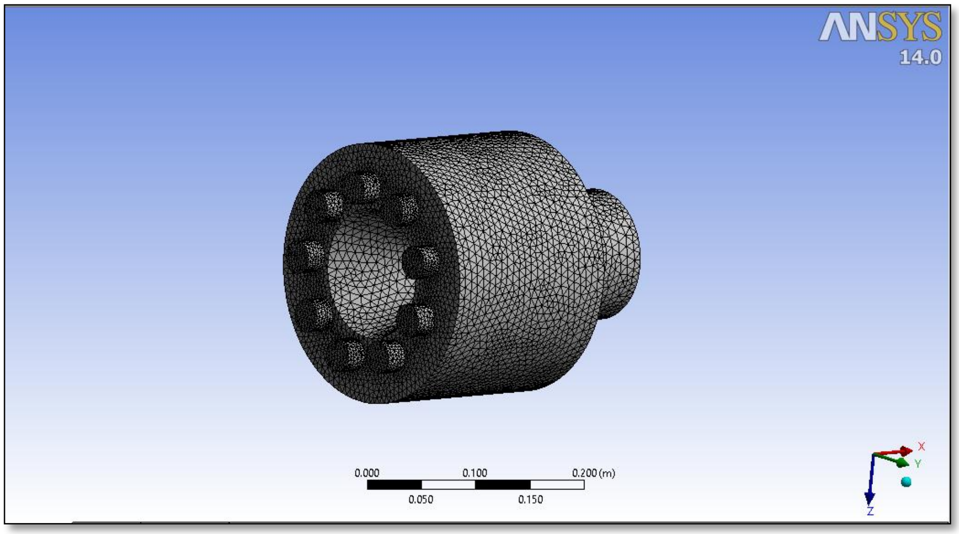

Figure 13: Mesh produced with 173845 nodes and 118428 elements

The three generated grids were imported to FLUENT 14.0 and the simulation was run, and after comparing the results, it was shown that the medium and the fine have almost the same results, so the medium grid can be used, in order to save computation time. See Figures 13 and 14 for FLUENT 14.0 results.

4.3. Simulation results from FLUENT

4.3.1. First set of results:

In order to study the effect of changing the geometry of the distributor nozzle,

I. By removing the cone, that is supposed to be present inside the nozzle, to study how the cone helps with the flow uniform distribution.

The simulation was carried out twice, on two steps,

A. Horizontally

The distributor nozzle is simulated horizontally, in order to study the effect of the flow horizontally, with respect to gravity, i.e. how the gravity can affect the horizontal flow.

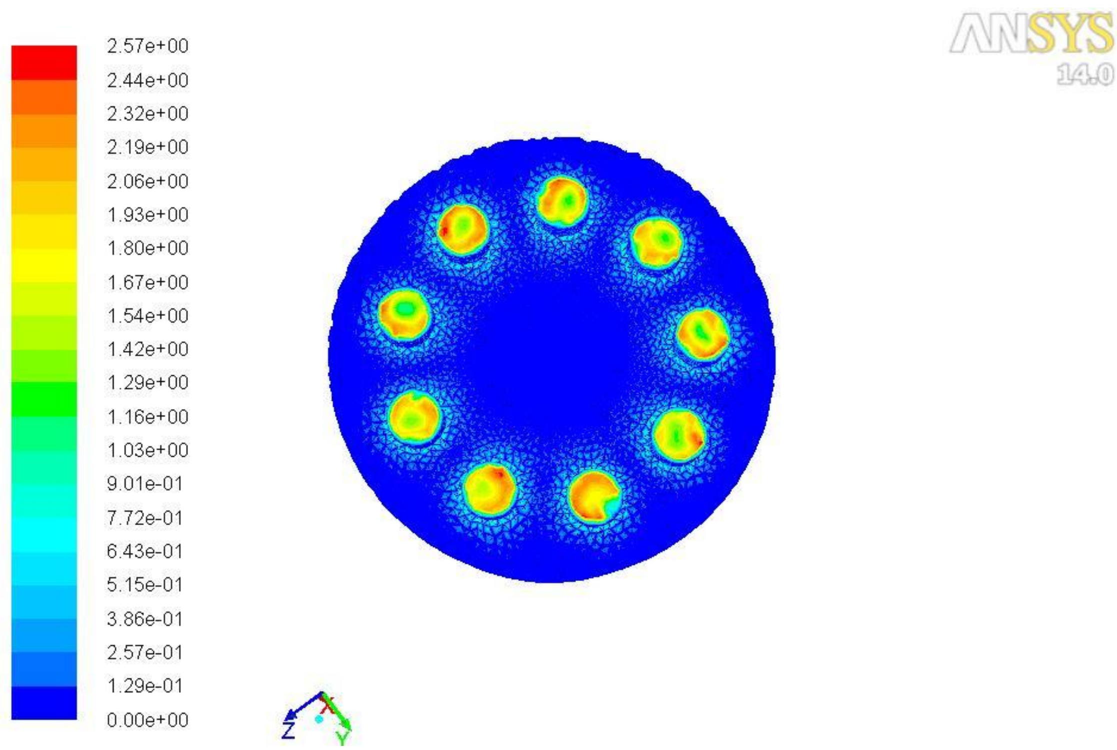

Figure 14: Velocity (m/s) contours for the horizontal orientation, of the distributor, with equal outlets lengths, short inlet and no cone

When the cone is removed, its effect can be seen, since by its removal the flow becomes in uniform in each of the tubes and it gives the dispersion effect which is needed in order to distribute the flow equally in each of the tubes.

B. Vertically

The distributor nozzle is then simulated vertically, in order to study the effect of the gravity on the flow, in the absence of the cone.

And also in order to compare the results obtained from the nozzle simulated horizontally, to the one simulated vertically, in order to study the significance of the orientation of the nozzle, and decide on which is a better orientation.

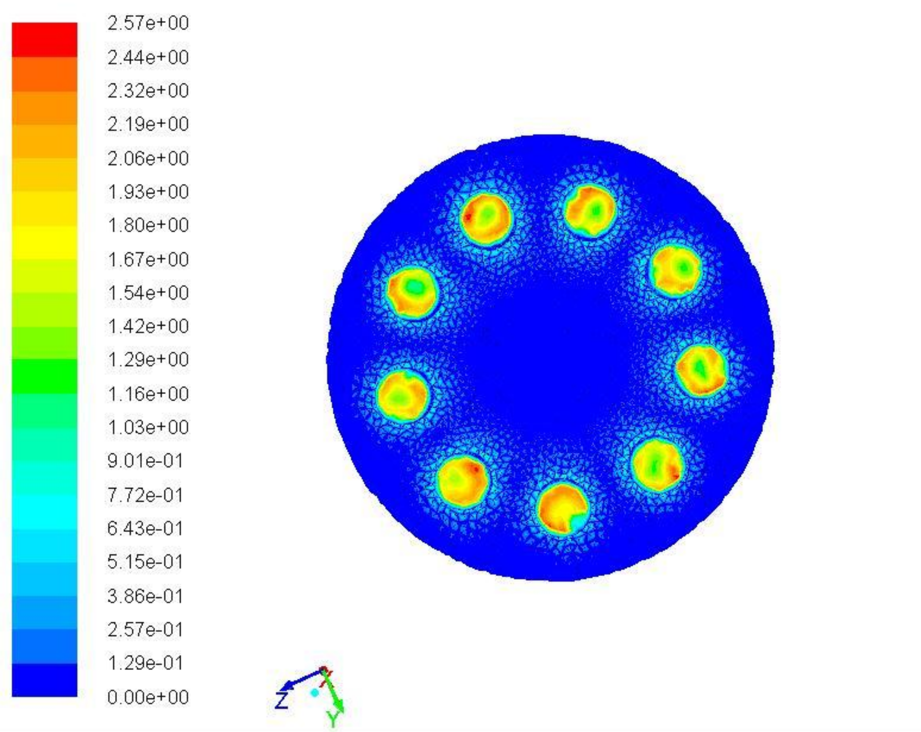

Figure 15: Velocity (m/s) contours for the vertical orientation, of the distributor, with equal outlets lengths, short inlet and no cone

When the distributor is simulated vertically and horizontally, there is not much difference seen, which means the effect of gravity in this case is negligible.

I. By using a short distributor inlet in the presence of the cone inside the nozzle, to study how the length of the inlet affects the flow uniform distribution.

The simulation was carried out twice, on two steps,

A. Horizontally

The distributor nozzle is simulated horizontally, in order to study the effect of the flow horizontally, with respect to gravity, i.e. how the gravity can affect the horizontal flow.

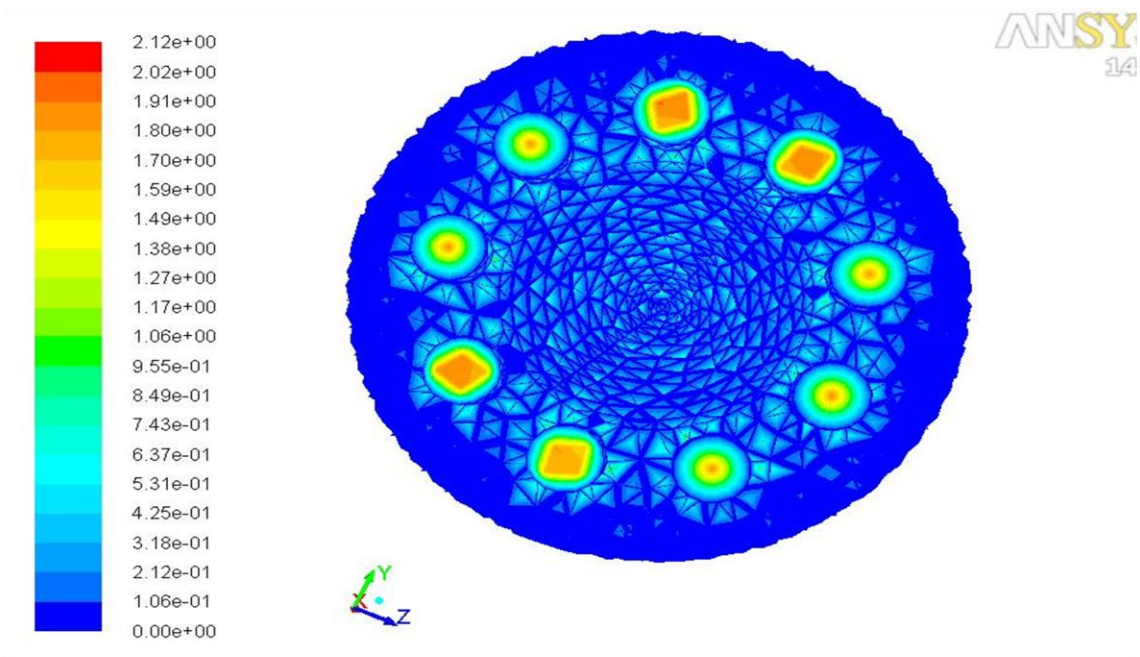

Figure 16: Velocity (m/s) contours for the horizontal orientation, of the distributor, with equal outlets lengths and short inlet

When the distributor is simulated with a short inlet, the distribution is remarkably non-uniform, which can be because the flow is not fully developed at the end of the short inlet.

B. Vertically

The distributor nozzle is then simulated vertically, in order to study the effect of the gravity on the flow, in the absence of the cone.

And also in order to compare the results obtained from the nozzle simulated horizontally, to the one simulated vertically, in order to study the significance of the orientation of the nozzle, and decide on which is a better orientation.

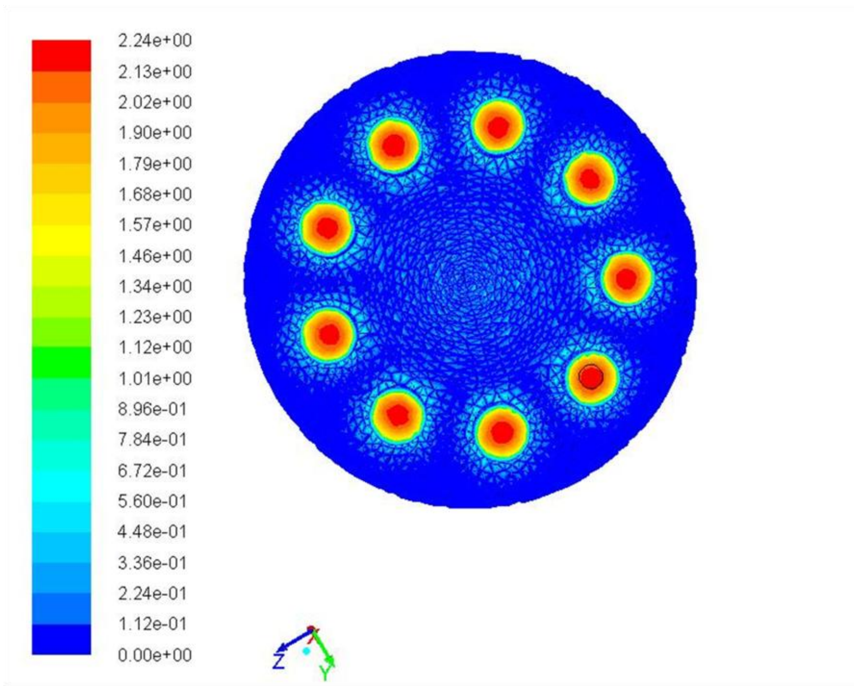

Figure 17: Velocity (m/s) contours for the vertical orientation, of the distributor, with equal outlets lengths and short inlet

When the distributor with the short inlet is simulated vertically downwards, the distribution is almost perfect, which shows that the gravity plays an important role in this case.

I. By using a long distributor inlet in the presence of the cone inside the nozzle, to study how changing the length of the inlet affects the flow uniform distribution.

The simulation was carried out twice, on two steps,

A. Horizontally

The distributor nozzle is simulated horizontally, in order to study the effect of the flow horizontally, with respect to gravity, i.e. how the gravity can affect the horizontal flow.

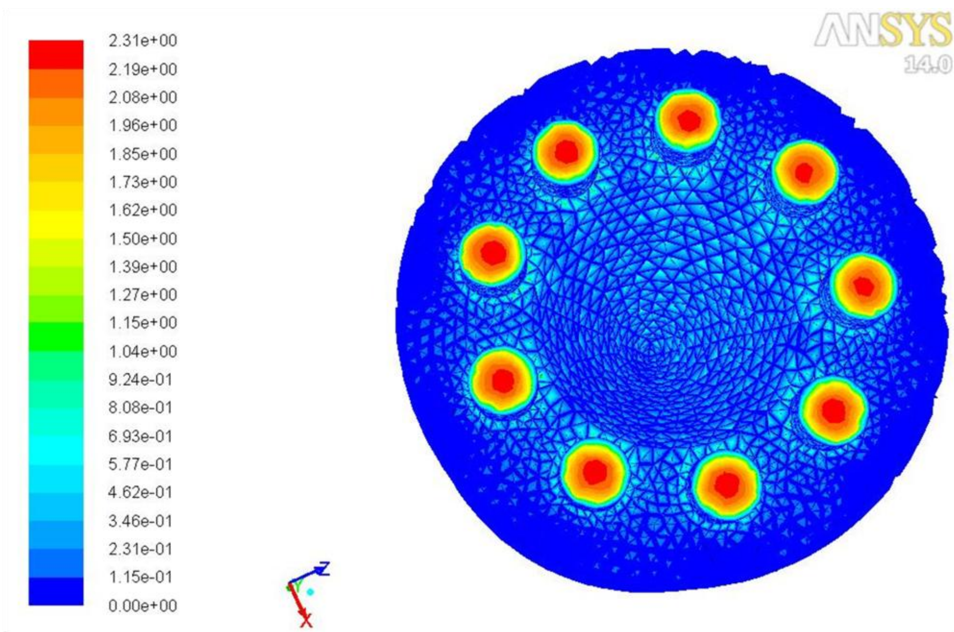

Figure 18: Velocity (m/s) contours for the horizontal orientation, of the distributor, with equal outlets lengths and long inlet

When the distributor is simulated with a long inlet, the distribution is almost perfect, which can be because the flow is fully developed already and has a uniform pattern.

B. Vertically

The distributor nozzle is then simulated vertically, in order to study the effect of the gravity on the flow, in the absence of the cone.

And also in order to compare the results obtained from the nozzle simulated horizontally, to the one simulated vertically, in order to study the significance of the orientation of the nozzle, and decide on which is a better orientation.

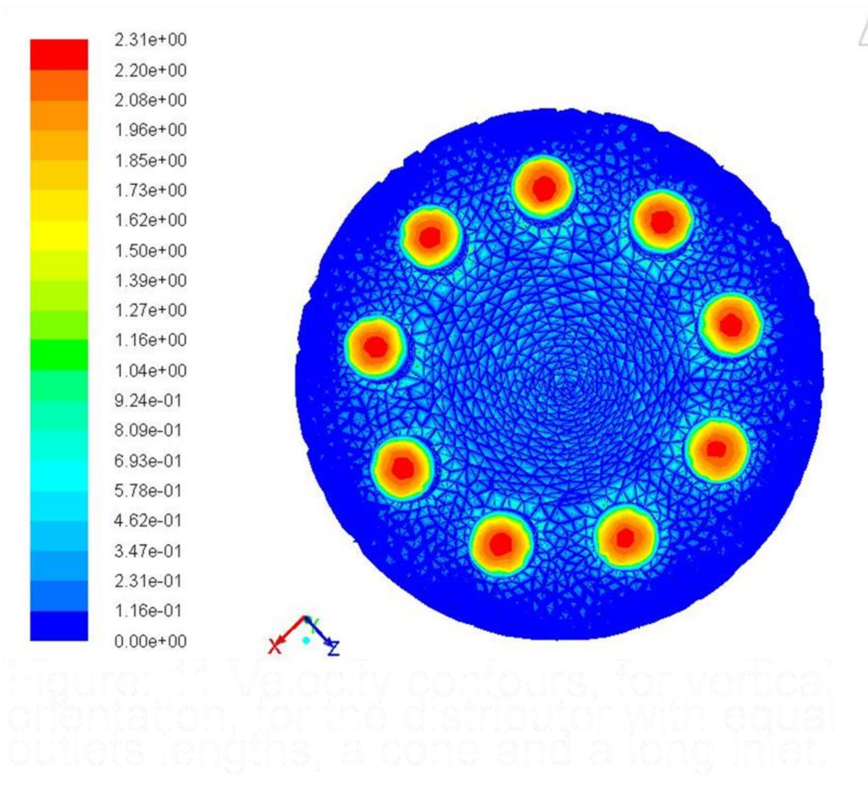

Figure 19: Velocity (m/s) contours for the vertical orientation, of the distributor, with equal outlets lengths and long inlet

When the long inlet distributor is simulated vertically downwards, the distribution is almost the same (almost perfect) which shows that the gravity does not play a big role in this case.

4.3.2. Second set of results

In order to study the effect of the outlet tube lengths on the flow distribution uniformity, By using different lengths of outlet tubes in 3 groups (200 mm, 400 mm & 600 mm) in 3 different orientations,

I. By setting the 200 mm group to be at the top, to study the effect of gravity as well as the outlet tube lengths, in terms of their orientation. On two major steps,

A. Horizontally

The distributor nozzle is simulated horizontally, in order to study the effect of the flow horizontally, with respect to gravity, i.e. how the gravity can affect the horizontal flow.

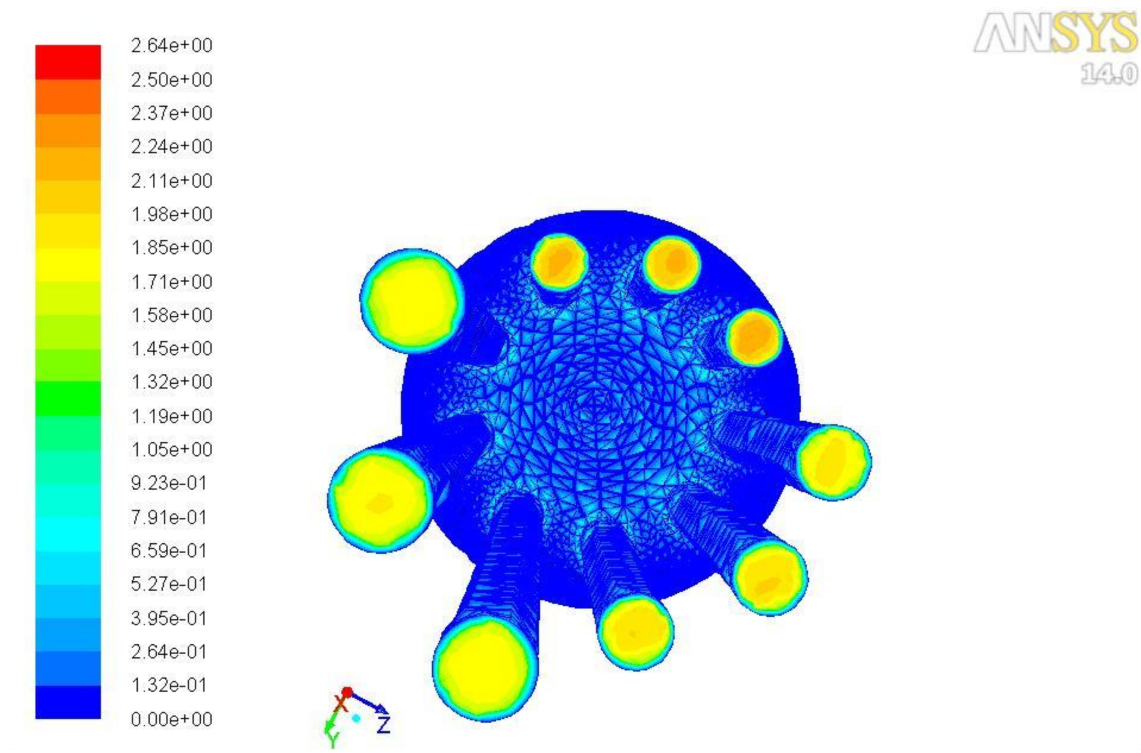

Figure 20: Velocity (m/s) contours for the horizontal orientation, of the distributor, with different outlets lengths, with the 200's tubes at the top

B. Vertically

The distributor nozzle is then simulated vertically, in order to study the effect of the gravity on the flow, in the absence of the cone.

And also in order to compare the results obtained from the nozzle simulated horizontally, to the one simulated vertically, in order to study the significance of the orientation of the nozzle, and decide on which is a better orientation.

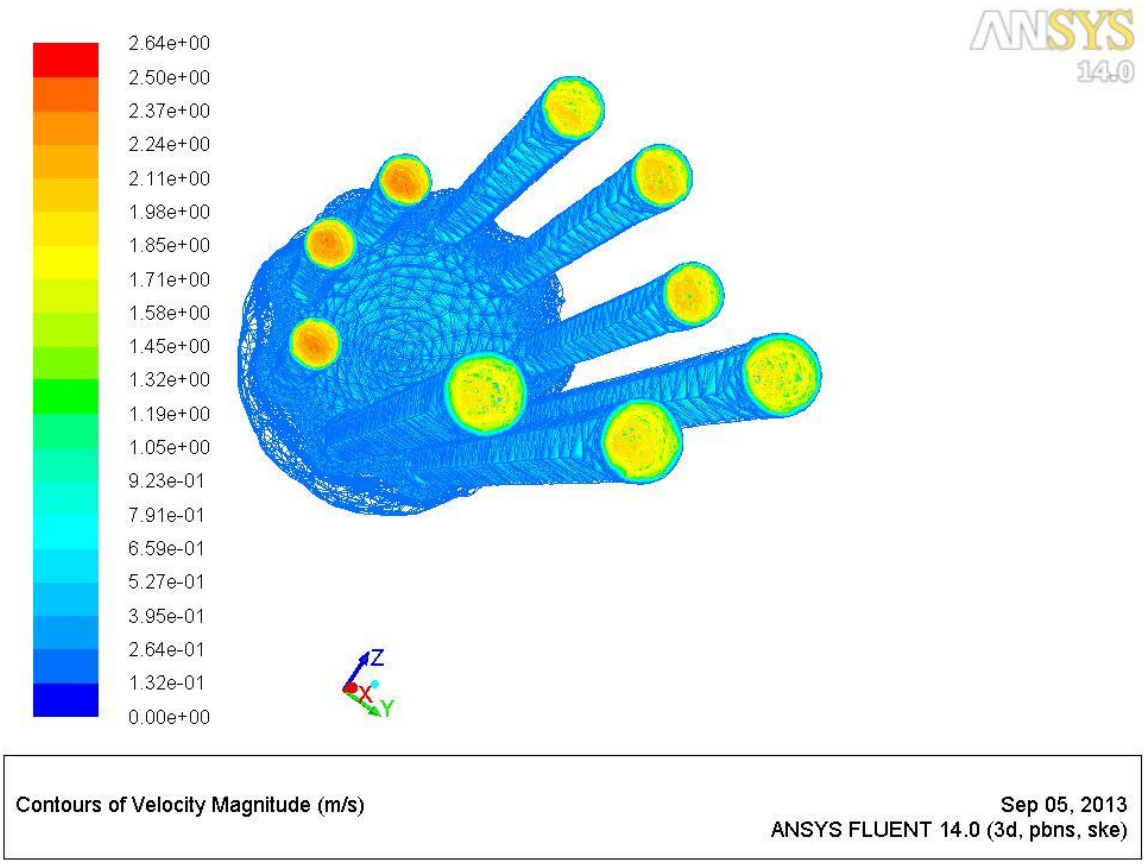

Figure 21: Velocity (m/s) contours for the vertical orientation, of the distributor, with different outlets lengths, with the 200's tubes at the top

I. By setting the 400 mm group to be at the top, to study the effect of gravity as well as the outlet tube lengths, in terms of their orientation. On two major steps,

A. Horizontally

The distributor nozzle is simulated horizontally, in order to study the effect of the flow horizontally, with respect to gravity, i.e. how the gravity can affect the horizontal flow.

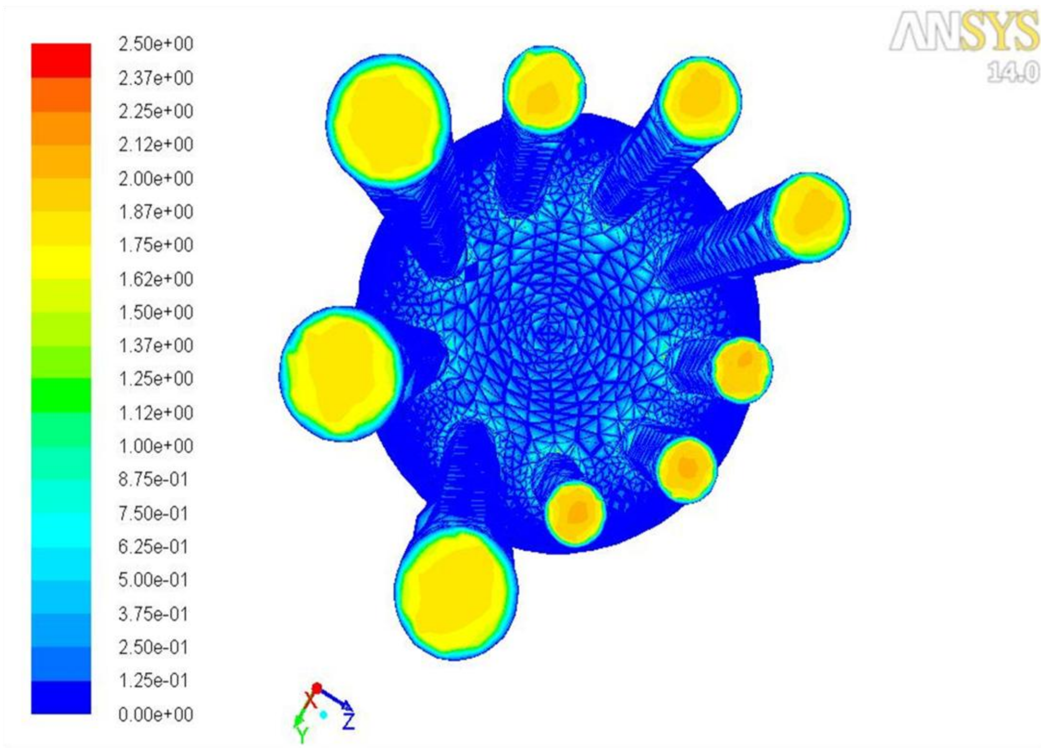

Figure 22: Velocity (m/s) contours for the horizontal orientation, of the distributor, with different outlets lengths, with the 400's tubes at the top

B. Vertically

The distributor nozzle is then simulated vertically, in order to study the effect of the gravity on the flow, in the absence of the cone.

And also in order to compare the results obtained from the nozzle simulated horizontally, to the one simulated vertically, in order to study the significance of the orientation of the nozzle, and decide on which is a better orientation.

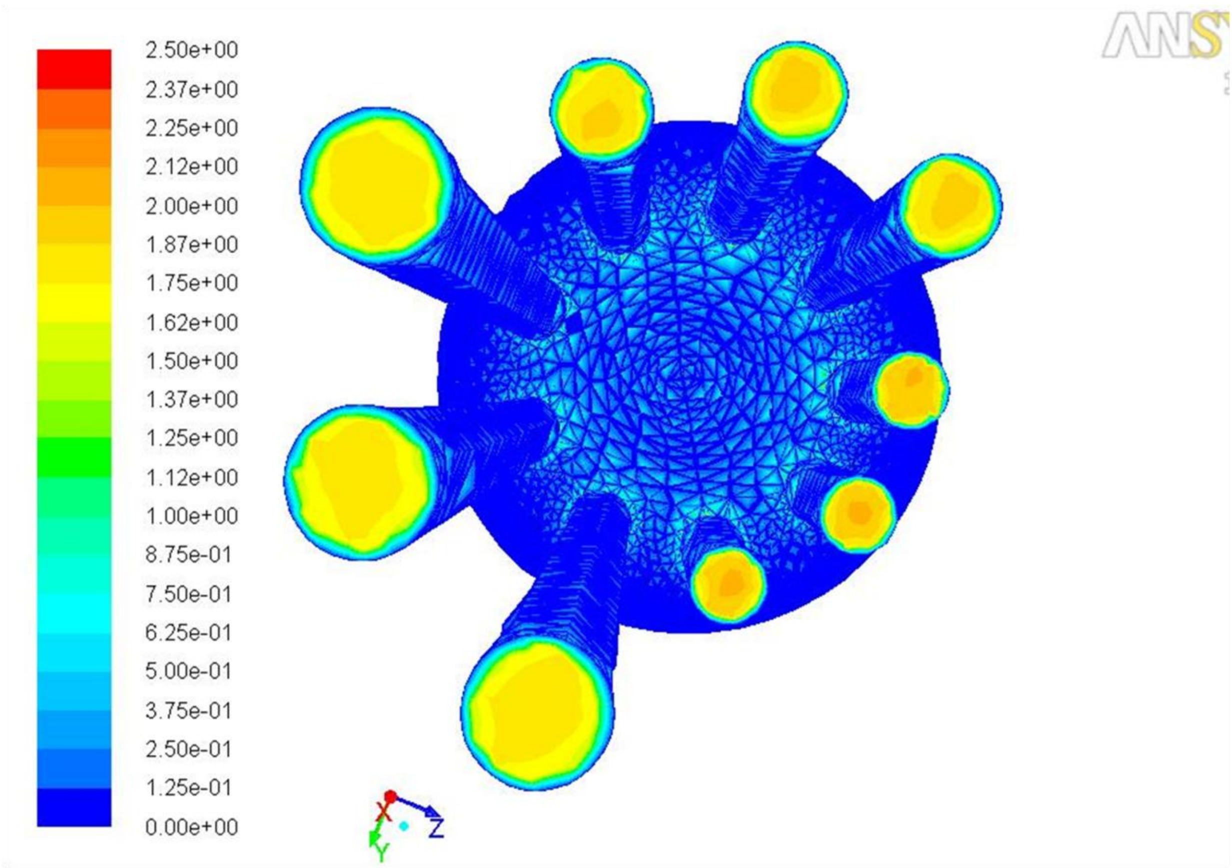

Figure 23: Velocity (m/s) contours for the vertical orientation, of the distributor, with different outlets lengths, with the 400's tubes at the top

I. By setting the 600 mm group to be at the top, to study the effect of gravity as well as the outlet tube lengths, in terms of their orientation. On two major steps,

A. Horizontally

The distributor nozzle is simulated horizontally, in order to study the effect of the flow horizontally, with respect to gravity, i.e. how the gravity can affect the horizontal flow.

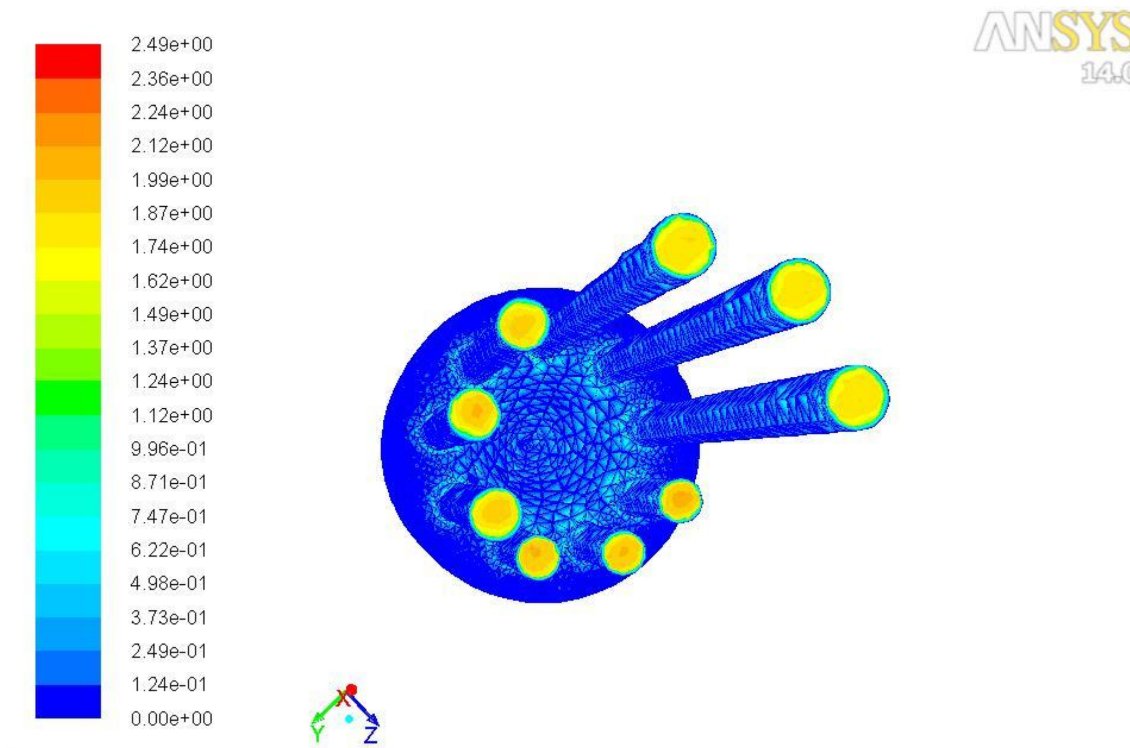

Figure 24: Velocity (m/s) contours for the horizontal orientation, of the distributor, with different outlets lengths, with the 600's tubes at the top

B. Vertically

The distributor nozzle is then simulated vertically, in order to study the effect of the gravity on the flow, in the absence of the cone.

And also in order to compare the results obtained from the nozzle simulated horizontally, to the one simulated vertically, in order to study the significance of the orientation of the nozzle, and decide on which is a better orientation.

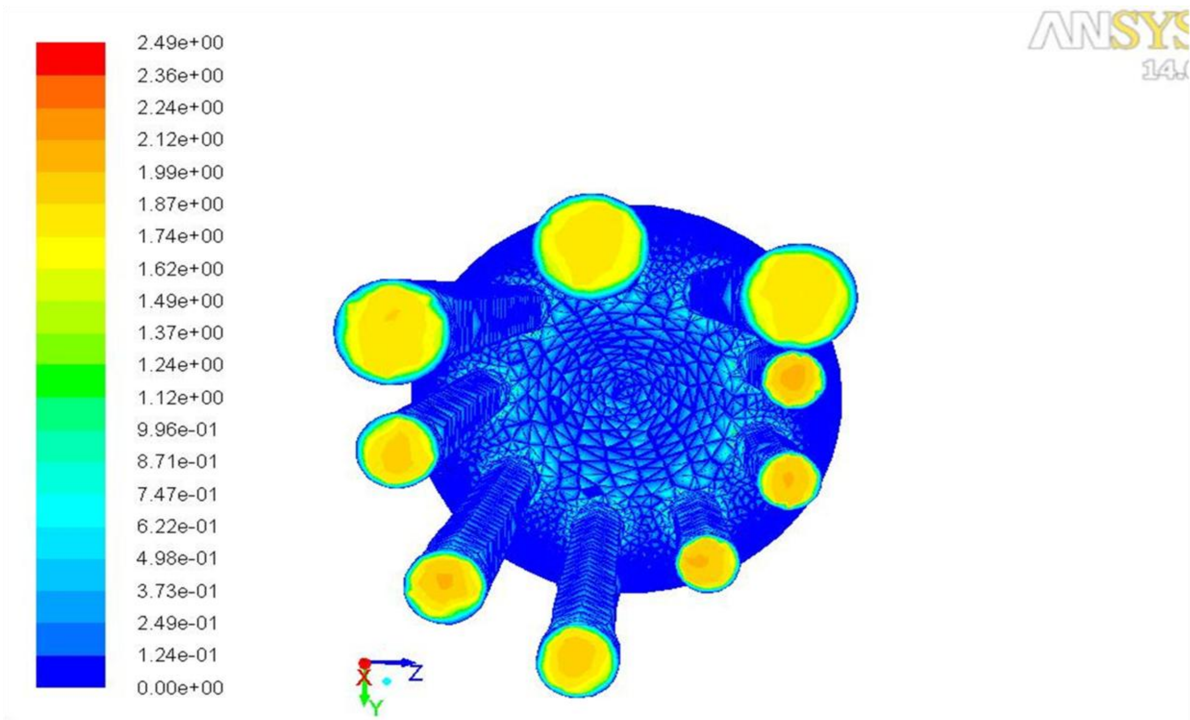

Figure 25: Velocity (m/s) contours for the vertical orientation, of the distributor, with different outlets lengths, with the 600's tubes at the top

When the distributor is simulated with different groups of length for outlet tubes (200mm, 400mm and 600mm), When investigating the effect of the orientation of each group (200mm at the top, 400 mm at the top and 600 mm at the top), there was not such significant difference between these orientations, which shows that the orientation does not play a big role in this case.

When investigating the effect of gravity, by simulating these different geometries horizontally and vertically, there was not such a big difference in the results, which shows that the gravity does not play such a big role in this case

When investigating the flow at each of the outlet tubes, it is found that the 200mm group had the best distribution, i.e. the fastest, while the 600mm group had the slowest. Which can be because of the effect of the pressure losses along the tube lengths, which is more remarkable in the 600mm group than the 200mm group.

4.3.3. Calculation of the outlet tubes varied/enhanced diameters

In order to improve the distribution at the outlets and make it more uniform,

By setting the 200mm group as reference of the best uniform distribution obtained and comparing both the 400mm and 600mm groups to it,

By calculating Volume flow rate at the outlet of the 200 mm group (reference),

Q_v= A*V

Where,

- Q_v is the volume flow rate at the outlet.
- A is the area of the outlet, where A=$\frac{\pi D^2}{4}$ and D is the diameter of the outlet.
- V is the velocity of the flow at the outlet

1. Then by using continuity equation and equating the volume flow rate at each of the other two groups of outlets (400mm tubes & 600 mm tubes), the optimum diameter can be found to use at the outlets of these two tube groups (400mm & 600mm).

Continuity equation,

$$Q_{v_1} = Q_{v_2}$$

$$A_1 V_1 = A_2 V_2$$

$$\frac{\pi D_1{}^2}{4} V_1 = \frac{\pi D_2{}^2}{4} V_2$$

Example of the calculation,

I. By setting $\frac{\pi D_1{}^2}{4} V_1 = \frac{\pi D_2{}^2}{4} V_2$

Where D_1 and V_1 are used for values obtained from the 200 mm group, while the V_2 is first used for the value obtained from the 400 mm group, then the 600 mm group

II. When substituting these values D_1, V_1 and V_2 into the above equation, the optimum outlet diameter could be found, in order to improve the distribution at the outlet tubes.

III. Velocity at the 200mm group, when at the top, in the horizontal orientation, V_1= 2.2408381m/s

IV. The diameter of the 200mm group, when at the top, in the horizontal orientation,
D_1= 0.03 m

V. Velocity at the 400mm group, when the 200 group at the top, in the horizontal orientation, V_2= 2.109024 m/s

VI. Therefore by using these values and substituting in the equation above, the new diameter is easily obtained, where the new diameter for the 400mm group in this case is D_2= 29.0473751 mm

4.3.4. Third set of results

After carrying out the calculations and finding the enhanced diameters for the outlet tubes, the simulations are repeated using the new geometries, in order to investigate the effect of changing the diameters on the uniform distribution improvement.

In order to study the effect of the outlet tube lengths on the flow distribution uniformity, By using different lengths of outlet tubes in 3 groups (200 mm, 400 mm & 600 mm) in 3 different orientations,

I. By setting the 200 mm group to be at the top, to study the effect of gravity as well as the outlet tube lengths, in terms of their orientation. On two major steps,

A. Horizontally

The distributor nozzle is simulated horizontally, in order to study the effect of the flow horizontally, with respect to gravity, i.e. how the gravity can affect the horizontal flow.

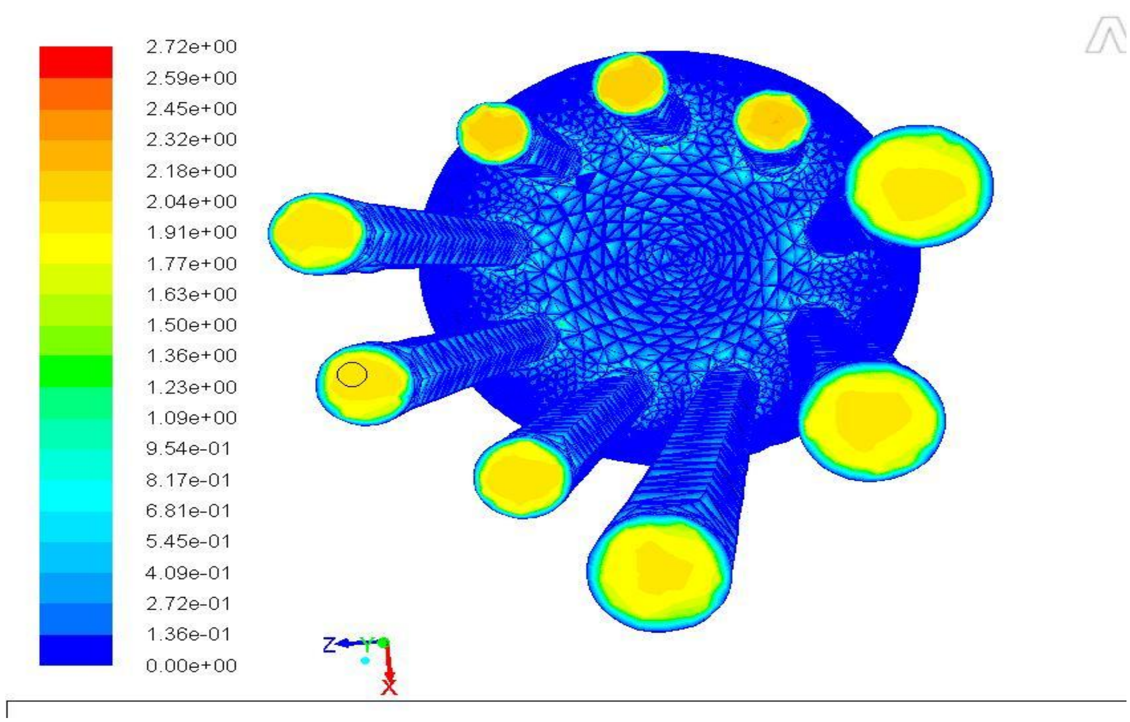

Figure 26: Velocity (m/s) contours for the horizontal orientation, of the distributor, with different outlets lengths, with the 200's tubes at the top, after changing the outlet tubes diameters

B. Vertically

The distributor nozzle is then simulated vertically, in order to study the effect of the gravity on the flow, in the absence of the cone.

And also in order to compare the results obtained from the nozzle simulated horizontally, to the one simulated vertically, in order to study the significance of the orientation of the nozzle, and decide on which is a better orientation.

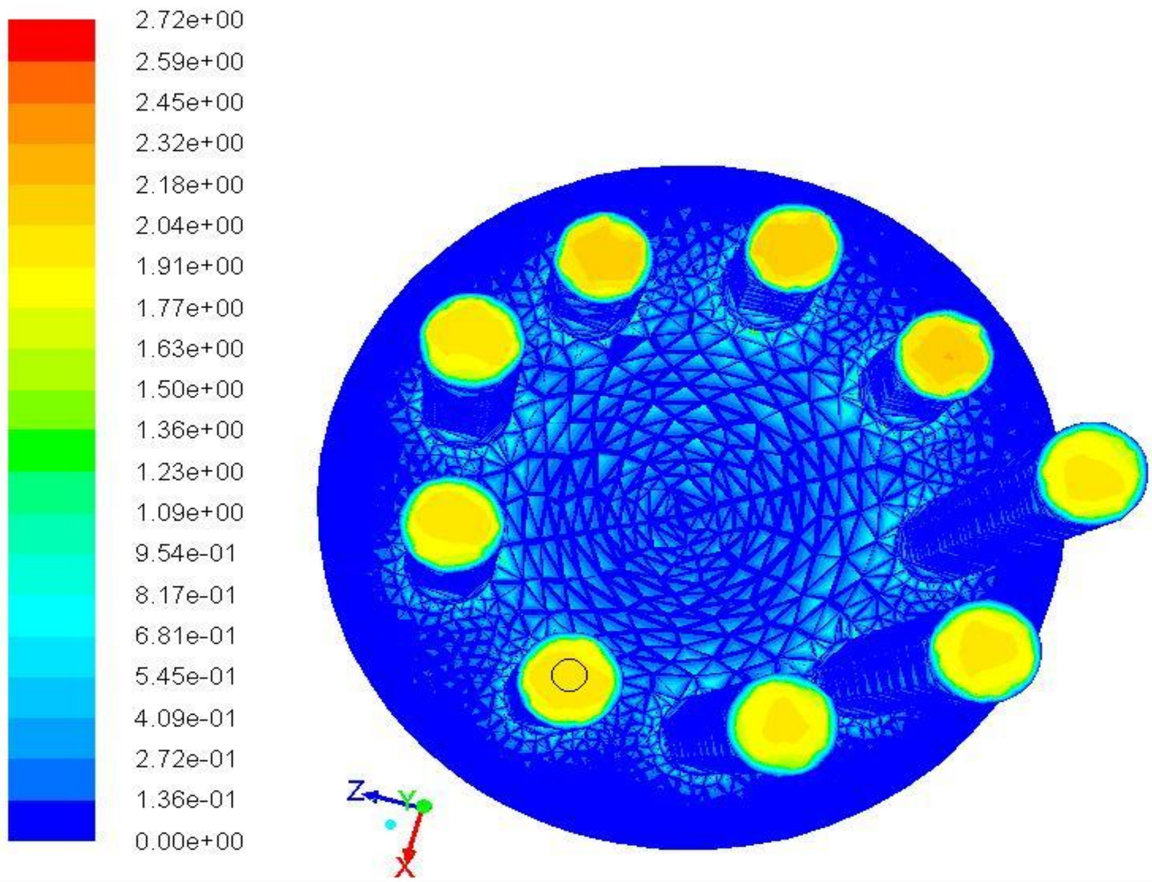

Figure 27: Velocity (m/s) contours for the vertical orientation, of the distributor, with different outlets lengths, with the 200's tubes at the top, after changing the outlet tubes diameters

II. By setting the 400 mm group to be at the top, to study the effect of gravity as well as the outlet tube lengths, in terms of their orientation. On two major steps,

A. Horizontally

The distributor nozzle is simulated horizontally, in order to study the effect of the flow horizontally, with respect to gravity, i.e. how the gravity can affect the horizontal flow.

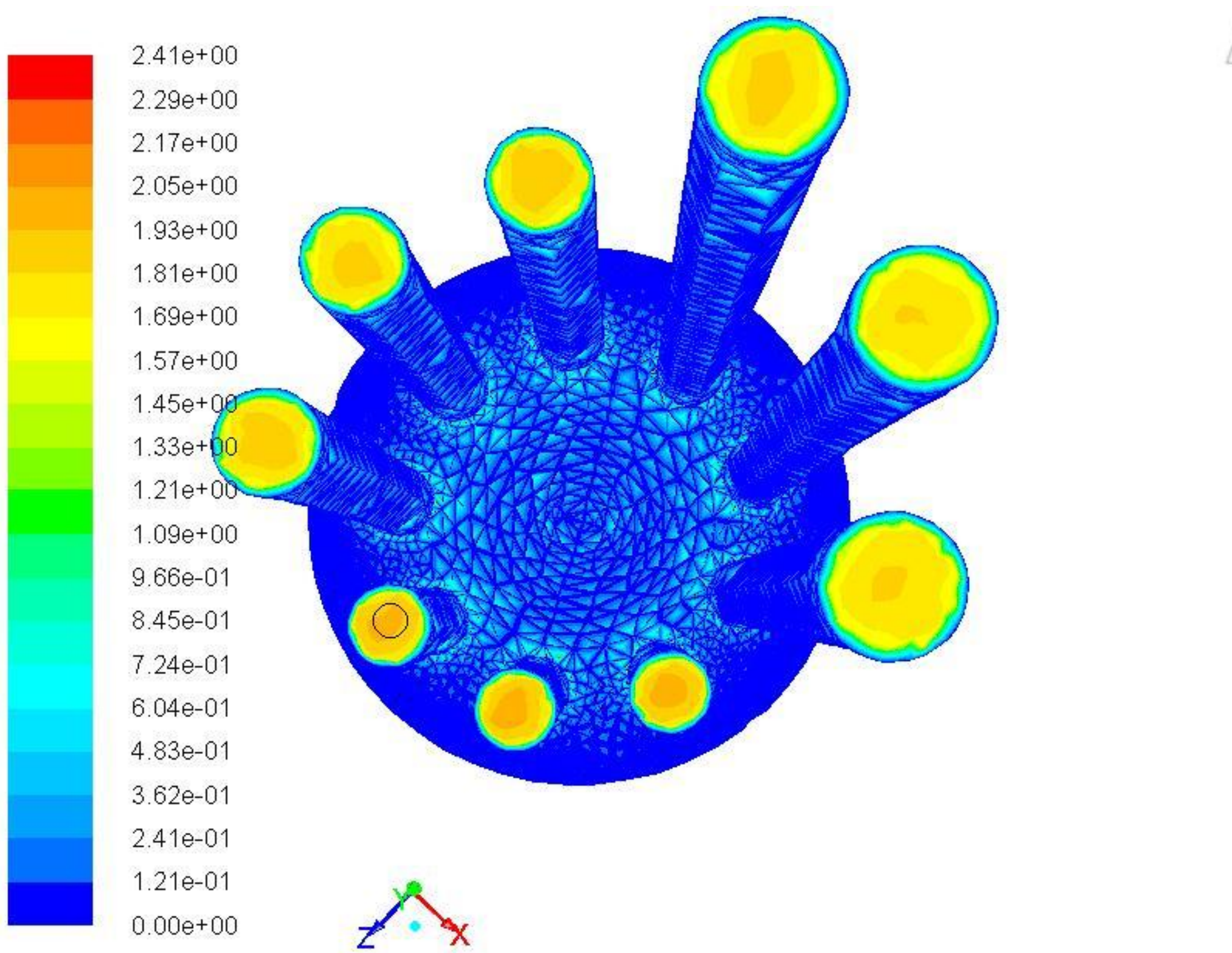

Figure 28: Velocity (m/s) contours for the horizontal orientation, of the distributor, with different outlets lengths, with the 400's tubes at the top, after changing the outlet tubes diameters

B. Vertically

The distributor nozzle is then simulated vertically, in order to study the effect of the gravity on the flow, in the absence of the cone.

And also in order to compare the results obtained from the nozzle simulated horizontally, to the one simulated vertically, in order to study the significance of the orientation of the nozzle, and decide on which is a better orientation.

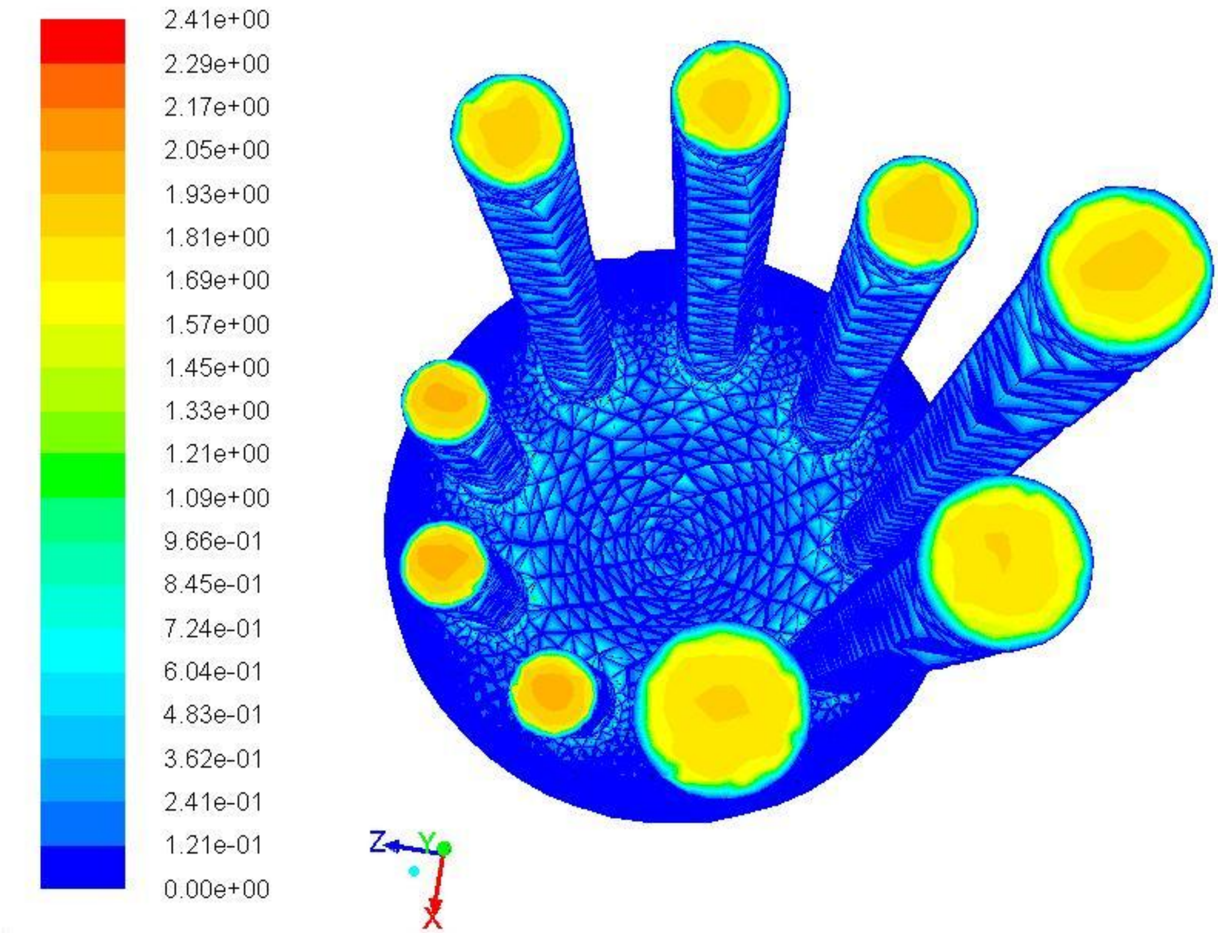

Figure 29: Velocity (m/s) contours for the vertical orientation, of the distributor, with different outlets lengths, with the 400's tubes at the top, after changing the outlet tubes diameters

III. By setting the 600 mm group to be at the top, to study the effect of gravity as well as the outlet tube lengths, in terms of their orientation. On two major steps,

A. Horizontally

The distributor nozzle is simulated horizontally, in order to study the effect of the flow horizontally, with respect to gravity, i.e. how the gravity can affect the horizontal flow.

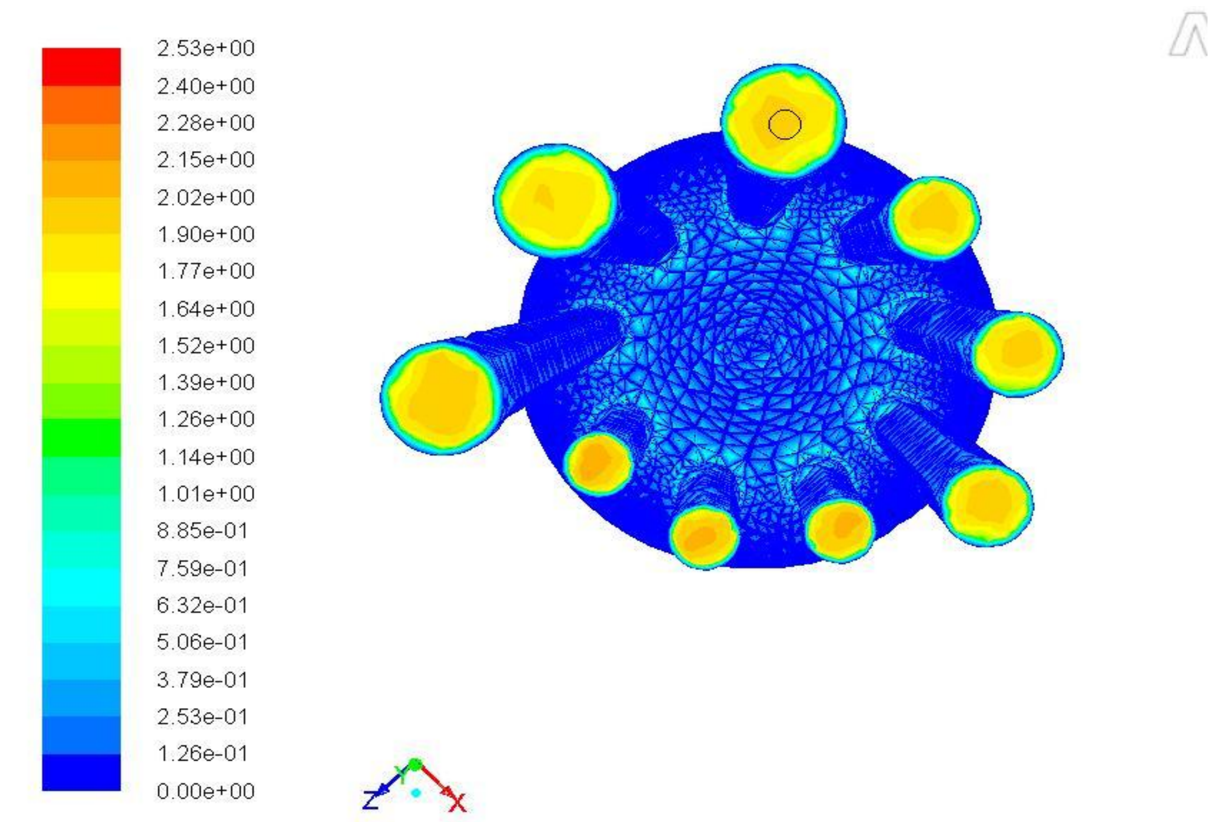

Figure 30: Velocity (m/s) contours for the horizontal orientation, of the distributor, with different outlets lengths, with the 600's tubes at the top, after changing the outlet tubes diameters

B. Vertically

The distributor nozzle is then simulated vertically, in order to study the effect of the gravity on the flow, in the absence of the cone.

And also in order to compare the results obtained from the nozzle simulated horizontally, to the one simulated vertically, in order to study the significance of the orientation of the nozzle, and decide on which is a better orientation.

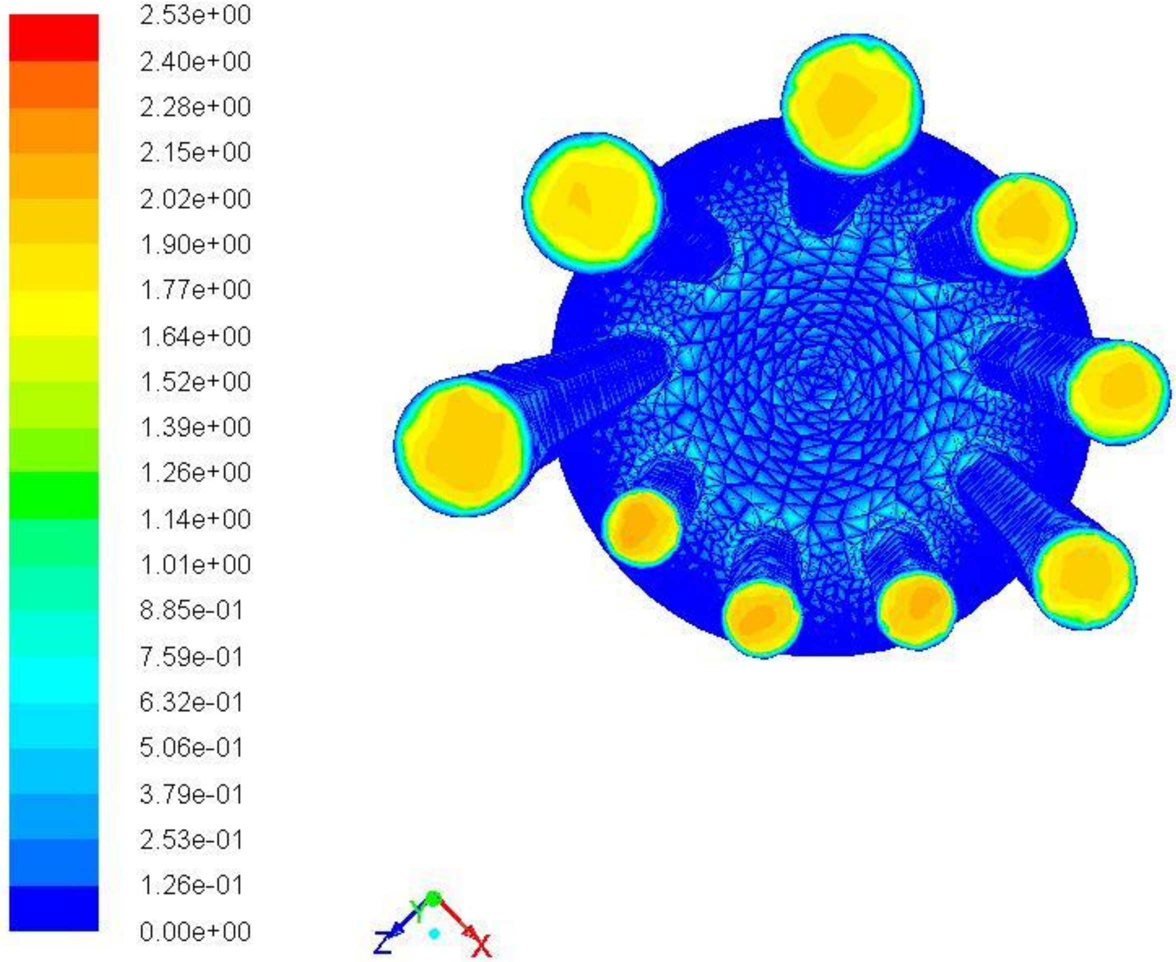

Figure 31: Velocity (m/s) contours for the vertical orientation, of the distributor, with different outlets lengths, with the 600's tubes at the top, after changing the outlet tubes diameters

When the outlet tubes diameters are adjusted, the velocities at the longer tubes (the 600 mm group) are noticed to get higher and more uniform i.e. closer to the reference values (at the 200 mm group), which shows that flow uniformity can be obtained by modifying the outlet tubes diameters.

CHAPTER 5: CONCLUSION & RECOMMENDATIONS

After carrying out the simulation and completing the study, it was concluded that the non-uniformity of the flow, in various tubes inside the distributor, was investigated, according to many aspects and factors, such as the dispersion cone present inside the distributor, the inlet length, and the output tube lengths. As well as, the orientations of the distributor and the outlet tubes.

Many techniques have been studied in order to reduce the non-uniformity of the flow. That was done by changing the lengths of the outlet tubes, as well as their orientations (above and below with respect to gravity) as well as the orientation of the distributor in total (horizontal and vertical). And it was found that the shorter tubes had faster flow and uniform distribution, unlike the longer tubes which had slower flow and non uniform distribution. Which can be justified to be due to the pressure drop losses along the tube length. Thus, the longer the tube, the higher the losses and the slower the flow will become. Then by changing the outlet tube diameters, the flow is found to have improved, which follows the continuity equation, making the flow faster and bringing it closer to uniformity when the diameter becomes smaller.

The distributor characteristics are studied, first by removing the dispersion cone, which shows the great role the cone plays in making the flow more uniform along all the outlet tubes. Then by changing the length of the inlet of the distributor, which shows that the inlet should be long enough in order for the flow to be fully developed.

It is recommended that this study be further pursued by studying the two phase flow in the distributor, with the primary phase being the liquid refrigerant and the secondary phase being the vapor refrigerant, as there are many other factors to consider, such as the density and the viscosity of each.

It is also recommended that the effect of the bent outlet tubes be studied, as the bents can cause more losses in energy, due to friction losses, which will majorly affect the uniform distribution in each of the tubes.

BIBLIOGRAPHY

[1] John Varrasi. (2011, Date of access: April 2013, June) ASME.com. [Online]. http://www.asme.org/kb/news---articles/articles/technology-and-society/global-cooling--the-history-of-air-conditioning

[2] (2006, Date of access: April 2013) Ag Power Web Enhanced Course Materials. [Online]. http://www.swtc.edu/ag_power/air_conditioning/lecture/basic_cycle.htm

[3] N. Ablanque, C. Oliet, J. Rigola, C.D. Pérez-Segarra, A. Oliva, , "Two-phase flow distribution in multiple parallel tubes," *International Journal of Thermal Sciences*, vol. 49, pp. 909-921, 2010.

[4] Donald F. Young, Theodore H. Okiishi, Wade W. Huebsch, Bruce R. Munson, *Fundamentals of Fluid Mechanics*, 6th ed.: Wiley.

[5] André Bakker. (2002, Date of access: April 2013) Bakker.org. [Online]. http://www.bakker.org

[6] Sporlan-Parker. (2011, February, Date of access: March 2013) Sporlan Online. [Online]. http://sporlanonline.com/literature-and-educational-materials/refrigerant-distributors/

[7] Julio C. Pacio, Carlos A. Dorao, "A study of the effect of flow maldistribution on heat transfer performance in evaporators," *Nuclear Engineering and Design*, vol. 240, pp. 3868–3877, September 2010.

[8] Ch. Ranganayakulu, K.N. Seetharamu, "The combined effects of wall longitudinal heat conduction, inlet fluid flow nonuniformity and temperature nonuniformity in crossflow plate-fin heat exchangers," *Int. J. Heat Mass Tran*, vol. 42, pp. 263–273, 1999.

[9] P. Florent, S.K. Lang, A.E. Bergles S. Lalot, "Flow maldistribution in heat exchangers," *Applied Thermal Engineering*, vol. 19, pp. 847–863, 1999.

[10] M. Katsuta, K. Nagata M.Watanabe, "General characteristics of two-phase flow distribution in a multipass tube, heat transfer," *Japanese Research*, vol. 24 (1), pp. 32-44, 1995.

[11] Wai Meng Chin, Vijay R. Raghavan , "On the adverse influence of higher statistical moments of flow maldistribution on the performance of a heat exchanger,"

International Journal of Thermal Sciences, vol. 50, pp. 581-591, 2011.

[12] Junye Wang, "Theory of flow distribution in manifolds, ," *Chemical Engineering Journal* , vol. 168 , pp. 1331–1345, 2011.

[13] S.G. Kandlikar, Z. Lu, W.E. Domigan, A.D. White, M.W. Benedict, "Measurement of flow maldistribution in parallel channels and its application to ex-situ and in-situ experiments in PEMFC water management studies," *International Journal of Heat and Mass Transfer*, vol. 52 , pp. 1741–1752, 2009.

[14] Natthawut Ruangtrakoon , Satha Aphornratana, Thanarath Sriveerakul Sirindhorn, "Experimental studies of a steam jet refrigeration cycle: Effect of the primary nozzle geometries to system performance," *Experimental Thermal and Fluid Science*, vol. 35, pp. 676–683, 2011.

[15] 1.Mayurkumar S. Gandhia, Arijit A. Gangulia, Jyeshtharaj B. Joshia, Pallippattu K. 2. Julio C. Pacio, Carlos A. Dorao, "CFD simulation for steam distribution in header and tube assemblies," *Chemical engineering research and design*, vol. 90 , pp. 487–506, 2012.

[16] P. J. Coeiho, "A local grid refinement technique based upon Richardson extrapolation," *Appl. Math. Modelling* , vol. 21, pp. 427-436, July 1997.

[17] LEAP Support Team. (2012, January, Date of Acess: April 2013) [Online]. http://www.computationalfluiddynamics.com.au/convergence-and-mesh-independent-study/

[18] Ablanque, N.,C. Oliet, J. Rigola, C.D. Pérez-Segarra, A. Oliva, "Two-phase flow distribution in multiple parallel tubes," *International Journal of Thermal Sciences* , vol. 49, pp. 909-921, 2010.

[19] (2010, Date of access: April 2013) Air Conditioners and Heaters. [Online]. http://air-conditioners-and-heaters.com/air_conditioning_history.html

[20] Lingai Luoa, Zhiwei Fan , Herv´e Le Gall, Xinggui Zhoub, Weikang Yuan, "Experimental study of constructal distributor for flow equidistribution," *Chemical Engineering and Processing*, vol. 47, pp. 229–236, 2008.

[21] K.N. Seetharamul Ch. Ranganayakulu, "The combined effects of longitudinal heat conduction, flow nonuniformity and temperature nonuniformity in crossflow plate-fin heat exchangers," *Int. Commun. Heat Mass Tran.*, vol. 26, pp. 669–678, 1999.

[22] P. Yuan, "Effect of inlet flow maldistribution on the thermal performance ofa three-fluid crossflow heat exchanger," *Int. J. Heat Mass Tran.*, vol. 46, 2003.

[23] (2010, Date of access: April 2013) Ra-Jac. [Online]. http://air-conditioners-and-heaters.com/air_conditioning_history.html

[24] John Varrasi. (2011, Date of access: April 2013, July) ASME.com. [Online]. http://www.asme.org/kb/news---articles/articles/technology-and-society/global-cooling--the-history-of-air-conditioning

[25] Amanda Green. (2012, July, Date of access: April 2013) Popular Mechanics. [Online]. http://www.popularmechanics.com/home/improvement/electrical-plumbing/a-brief-history-of-air-conditioning-10720229

TABLE OF APPENDICES

APPENDICES

Appendix A

A.1. Gantt Chart and milestones for study of FYP 1:

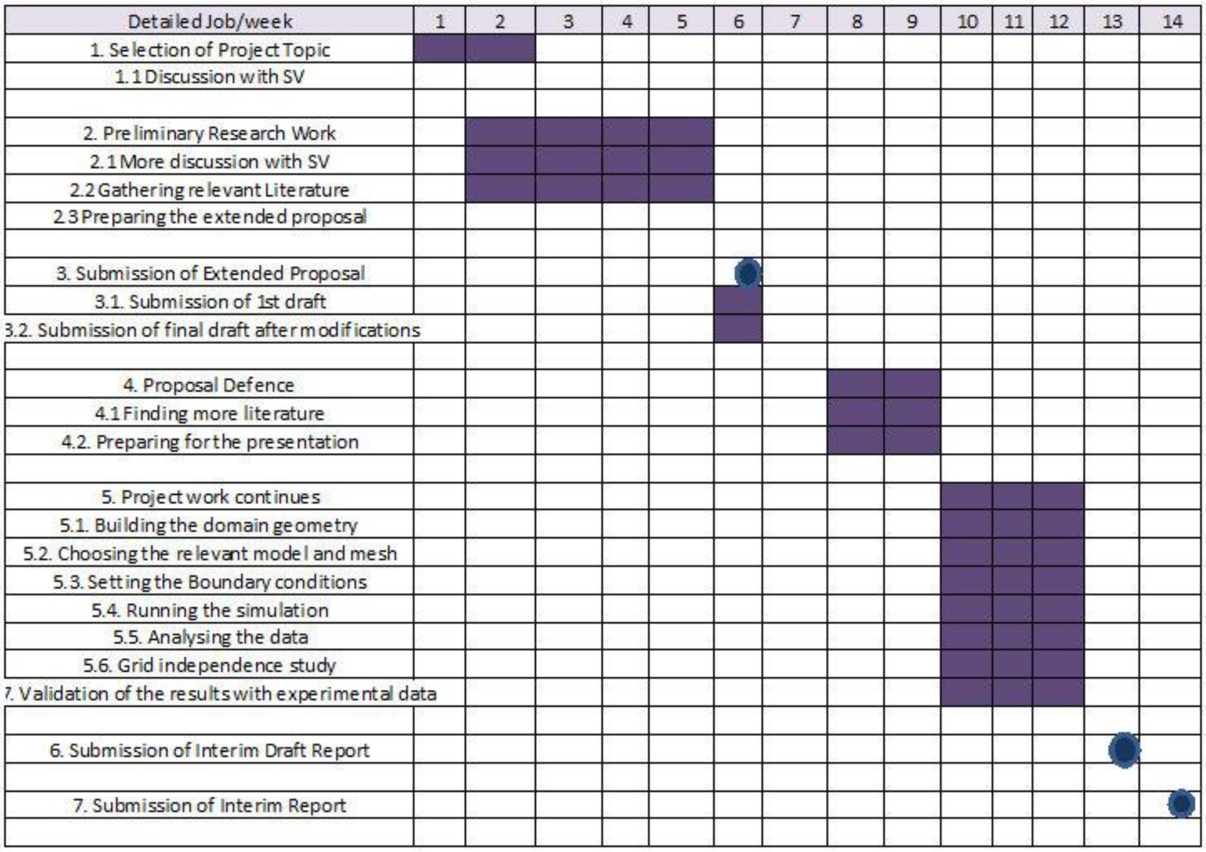

Figure 32: Gantt Chart and milestones for FYP1

A.2. Gantt Chart and milestones for study of FYP 2:

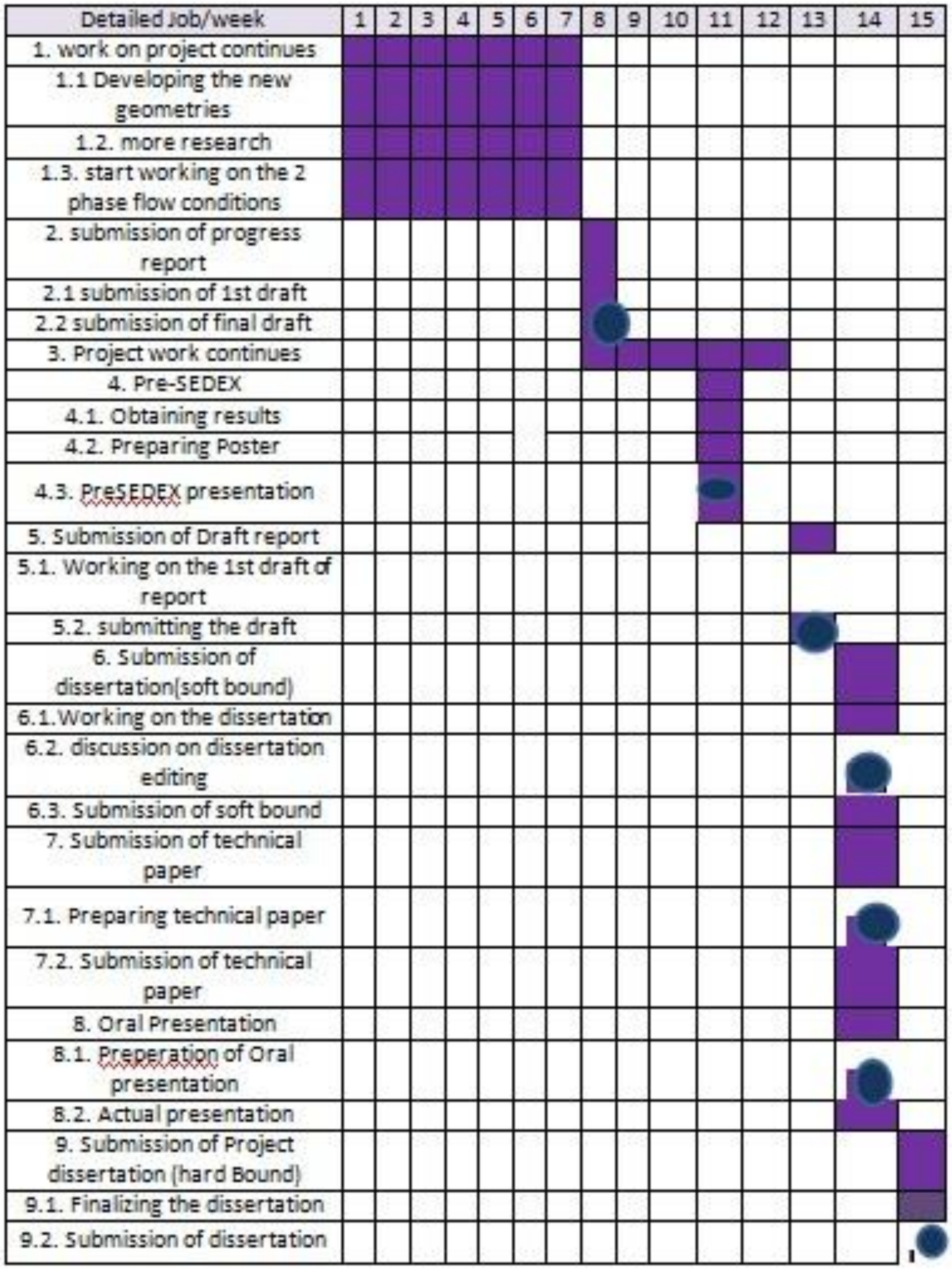

Figure 33: Gantt Chart and milestones for FYP2

Appendix B

Calculations of 3rd set of results

B.1. Calculation of Original values obtained for the velocity (m/s) from the simulation results, in two orientations; horizontal and vertical.

Minimum velocity value (m/s)	Horizontal orientation			Vertical orientation		
	200mm group results	400mm group results	600mm group results	200mm group results	400mm group results	600mm group results
200mm group at the top	2.109	1.977	1.845	2.240	1.977	1.845
400mm group at the top	1.851	1.975	1.851	1.999	1.874	1.749
600mm group at the top	1.991	1.991	1.866	1.991	1.991	1.866
Maximum velocity value (m/s)	**Horizontal orientation**			**Vertical orientation**		
	200mm group results	400mm group results	600mm group results	200mm group results	400mm group results	600mm group results
200mm group at the top	2.240	2.109	1.977	2.372	2.109	1.977
400mm group at the top	1.975	2.098	1.975	2.124	1.999	1.874
600mm group at the top	2.115	2.115	1.991	2.115	2.115	1.991

B.2. Calculation of the volume flow rate (Q) at the outlet of the tubes for the values of velocities obtained from the simulation, in two orientation; horizontal and vertical

Minimum flow rate value (m^3/s)	Horizontal orientation			Vertical orientation		
200mm group at the top	200mm group results	400mm group results	600mm group results	200mm group results	400mm group results	600mm group results
	1490.781	1397.607	1304.433	1583.955	1397.607	1304.433
400mm group at the top	1308.904	1396.164	1308.904	1413.388	1325.051	1236.714
600mm group at the top	1407.615	1407.615	1319.640	1407.615	1407.615	1319.640
Maximum flow rate value (m^3/s)	Horizontal orientation			Vertical orientation		
200mm group at the top	200mm group results	400mm group results	600mm group results	200mm group results	400mm group results	600mm group results
	1583.955	1490.781	1397.607	1677.128	1490.781	1397.607
400mm group at the top	1396.164	1483.425	1396.164	1501.724	1413.388	1325.051
600mm group at the top	1495.592	1495.592	1407.615	1495.592	1495.592	1407.615

B.3. New area obtained after obtaining the new diameters

Minimum Area value (m^2)	Horizontal orientation		Vertical orientation	
200mm group at the top	400mm group results	600mm group results	400mm group results	600mm group results
	662.679	618.501	623.698	589.048
400mm group at the top	753.982173	706.858347	662.679714	623.6985
600mm group at the top	706.858	662.679	706.858	665.278
Maximum Area value (m^2)	Horizontal orientation		Vertical orientation	
200mm group at the top	400mm group results	600mm group results	400mm group results	600mm group results
	665.278	623.698	628.318	589.048
400mm group at the top	751.037	706.858	665.278	623.698
600mm group at the top	706.858	665.278	706.858	665.278

B.4. New diameter obtained from the area

Minimum diameter value (m)	Horizontal orientation			Vertical orientation		
200mm group at the top	200mm group results	400mm group results	600mm group results	200mm group results	400mm group results	600mm group results
	30	29.047	28.062	30	28.180	27.386
400mm group at the top	30	30.983	30	30	29.047	28.180
600mm group at the top	30	30	29.047	30	30	29.104
Maximum diameter value (m)	**Horizontal orientation**			**Vertical orientation**		
200mm group at the top	200mm group results	400mm group results	600mm group results	200mm group results	400mm group results	600mm group results
	30	29.104	28.180	30	28.284	27.386
400mm group at the top	30	30.923	30	30	29.104	28.180
600mm group at the top	30	30	29.104	30	30	29.104