EXPERIMENTAL AND NUMERICAL INVESTIGATION ON THE ENHANCEMENT OF NATURAL UP DRAFTING IN A CHIMNEY

BY

KAM CHONG PING

16474

Dissertation submitted in partial fulfillment of

the requirements for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

JANUARY 2016

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

I

CERTIFICATION OF APPROVAL

EXPERIMENTAL AND NUMERICAL INVESTIGATION ON THE ENHANCEMENT OF NATURAL UP DRAFTING IN A CHIMNEY

By

KAM CHONG PING

16474

A project dissertation submitted to the

Mechanical Engineering Department

University Technology PETRONAS

In partial fulfillment of the requirement for the

BACHELOR OF ENGINEERING (Hons)

(MECHANICAL ENGINEERING)

Approved by,

_____

(Dr Jundika Candra Kurnia)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

January 2016

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.

_____

KAM CHONG PING

Abstract

The global initiative to go green and the potential benefits in terms of operational cost, energy requirement and carbon emissions have raised more interest than ever to develop passive ventilation systems. Significant amount of research work have been done on chimneys since the 1990s, most studies focuses on the effects of geometry and the inclination angle of the ventilation performance of the solar chimney. However, published studies showed that natural passive ventilation has not been fully understood. The objectives of this research are to identify the effective design parameters on natural up drafting using top cup of a chimney and to investigate the influence of the design parameters on the up drafting. The parameters investigated are the various design angles of the chimney cup from 180˚ to 90˚ and the position of the chimney cup varying from 3cm to 7.5cm from the tip of the chimney. Numerical investigation were carried out by using ANSYS 15.0 Fluent simulation while experimental investigation involved setting up the experiment setup, conduct test and obtain measurements of actual velocities and temperature readings. The results obtained by both investigation procedures were compared to validate the findings and identify the chimney cup design angle design and position which produces the optimum natural up drafting velocities. The results indicate the chimney cup design angle of 120˚ produced the highest flow velocity, while the optimum gap length to be set is 4.5cm. The combination of these two design parameters produced an experimental flow velocity of 2.87m/s and a simulated flow velocity of 3.79m/s.

ACKNOWLEDGEMENT

I would like to take this opportunity to express my utmost thankfulness to Dr. Jundika Candra Kurnia and Prof. Hussain Al-Kayiem, my UTP supervisor and co-supervisor for willingly spending their time to give me guidance and share their valuable knowledge with me throughout my Final Year Project I & II. They have given me great motivation and encouragement for me to achieve the best results possible as well as given me constructive criticism to further improve my work. They were patient and given me the independence to work freely throughout these past few months.

Furthermore, I would like to thank my family and friends who have given me their love and support during my most trying times. Without them, this FYP would have been very difficult to achieve on my own.

Lastly, I would like to express my appreciation to Universiti Teknologi PETRONAS (UTP) for giving me this opportunity to complete my Final Year Project. This project has helped in my development of my technical skills as well as my individual development in becoming a future engineer.

Table of Contents

LIST OF FIGURES

List of Tables

Chapter 1: Introduction

1.1 Background

Chimneys are an effective means of providing ventilation. It takes advantage of the partial negative pressure created when wind goes through the chimney, or the turbine upon the chimney rotates [1]. Industrial-made chimneys are made of stainless steel material which are coated with galvanized steel or wrapped with stainless steel sheeting as insulation to prevent the high levels of heat loss. Several parameters influence the draft produced by the chimney structure such as the height of the chimney, diameter, temperature difference between the fuel gas and the ambient air, as well as the pressure difference between the internal housing and the external ambient air.

Chimney designs should take into consideration the nature of the buoyancy airflow, the wind direction on the building sites, the wind force [2]. Downward draft is the movement of wind which flows down the chimney instead of going up [3]. This mainly occurs when ambient wind is blowing pass an obstacle such as the roof lining or tree and towards the top of a chimney. This produces a buildup of pressure which forces the flue gases down the chimney, creating flow resistance which affects the ability of the chimney of removing fuel gases and toxic smoke from the building effectively. Flow resistance decreases the induced draft [3,4].

Consequences of downward draft is taken very seriously as it produces a buildup of toxic gases which leads to carbon monoxide poisoning in the chimney housing. "It is known that the pollution level decreases exponentially with the airflow rate. Hence, optimizing the supply of air is essential to ensure adequate indoor air quality while maintaining the ventilation rates within a certain range" [5]. It also affects the efficiency of the combustor by increasing amounts of carbon dioxide and monoxide in the system while reducing the oxygen needed for combustion. The buildup of coal soot deposits and wood tar may also cause a chimney fire. Heat from the fire heats up the deposits, releasing combustible volatiles which cause sparks to ignite.

Chimney fires are a health hazards as they risk burning the entire building down if not controlled.

To avoid downward draft while increasing upward draft is by installing a chimney top device such as a chimney cup. Chimney cups have the goal of dissipating exhaust air into the atmosphere while preventing down draughts, rain, insects, etc. into the building [7]. The formation of a band between the chimney cup and the skirt prevents the chimney top from direct exposure to ambient wind conditions. This simple design channels the wind through the given band opening, producing suction at the top of the chimney as a result of wind speed and wind angle [7]. "This suction effect is dependent upon three primary factors prevailing wind conditions, type of cowl (discharge coefficient) and also the flow rate." [8].

1.2   Problem Statement

Downdraft in chimney is a main problem faced by users as it does not only affect the overall efficiency of the chimney but is unsafe as well. Besides, industrial manufacturing operations mainly require expelling of unwanted byproducts through an exhaust system which involves a chimney as an outlet. Companies invest heavily on purchasing and maintaining exhaust systems like the exhaust fans as they use thin airfoil blades which are fragile and complex in design. With these characteristics, an expert is required to be hired for maintenance and cleaning which is of high cost. Furthermore, without proper maintenance, the exhaust system can turn ineffective and result in high risk of carbon monoxide poisoning. Carbon monoxide is an odourless, tasteless and colourless gas [9,10]. Symptoms include nausea, drowsiness, fainting and in some cases, death [10]. It is difficult to detect and highly hazardous. Taking this into consideration, it is timely that efforts are directed at designing a passive chimney top enhancement which relies on natural up drafting to expel exhaust gases which is low in cost and easy to maintain.

1.3   Objectives and Scope of Study

1.3.1 Objectives

The objectives of this project are:

- To identify the effective design parameters on natural up drafting using top cup of a chimney
- To investigate the influence of the design parameters on the up drafting.

1.3.2 Scope of Study

The scope of study of this project is as given below:

- Procedure of investigating numerically through modeling and simulation.

- Procedure of investigating experimentally design and fabrication.

Chapter 2: Literature Review

2.1 Chimney Top Enhancements

Known as rain, weather, or termination cups, metal chimney cups are located on top of metal chimney flue liners [11]. During a low production periods, the plant's fuel consumption is reduced, lowering flue gas volumes of dispersion, hence reducing gas outlet velocity. Low outlet velocity results in rapid cooling of the flue gas, wash-down, and scattering of contaminants in the housing of the chimney. Each of the conditions is a risk towards the top of the chimney, increasing corrosion and weakening of the structure [12]. Contaminant build up in the liner is also caused by low exit velocities. Resulting in frequent maintenance and cleaning. A chimney cup is designed to be placed on the end of a chimney to increase the amount of draft and assist in the proper exhaust of combustion gases. The chimney cup increases flue gas exit velocity when properly installed and designed at an optimum gap width and design angle. The effective height of the stack and the rate of dispersion are increased. "Nevertheless, to achieve designs that assure a correct gas evacuation, there should also be considered the parameters associated with the constructive and environmental characteristics of the region" [13].

According to Sakonidou in his research of solar chimney tilt, his work uses a composite engineering model to estimate the optimum angle to tilt a chimney accordingly to location latitude to gain uniform solar irradiance for the solar chimney and enhance the air flow rate within the chimney. In his research, he tested various angles from 30° to 90°, and found that a tilt angle of 60° provided the maximum air flow rate while a tilt angle of 30° provided the maximum uniform solar irradiance. He tested various lengths of the chimney as well, from 1m to 12m in length. The height of chimney 3 meters and above showed significant increase in velocities. However, he did not anticipate the instability of the chimney as flow changed from laminar state to turbulent state. For lengths below 3 meters, significant changes to velocities were not able to be obtained as the measuring device, anemometer, was

frequently providing fluctuating readings. From this research paper, we can observe a relationship between the effective height of the chimney with air velocity as well as the relationship between tilt angles with air flow rate. Unfortunately, results of chimney height of 3 meters and below could not be provided and further research could be conducted for this region [14].
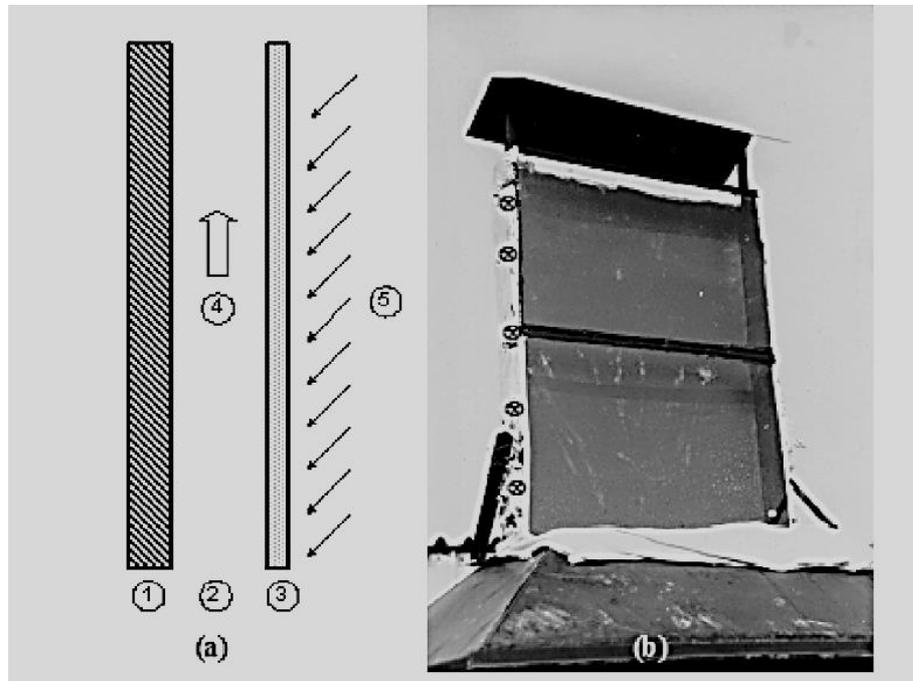
Figure 2.1: Schematic diagram of solar chimney (a) actual experimental model (b) [14]

Alfriyie's research regarding Simulation and optimisation of the ventilation in a chimney-dependent solar crop dryer studied the effects of the parameters of the tilt angle of the roof on the velocity of the air flow as well as the effects of inlet-outlet area ratios. This research was highly beneficial as the chimney roof used in our study consist of a tilted angle of 45° which suggested in his study to produce the maximum exit velocity flow. Whereas, the inlet-outlet ratio used for our study is 1:2 but in the study, maximum velocity can be achieved with a ratio of 1:4. In this study, the author concluded that site of experiments near the equator is best suited using high drying chamber with a short chimney height. This is similar to our experimental set up to fully maximize the outlet velocity [15].

In the study by Khanal and Lei of solar chimney, one of the parameters studied was the inclination of the chimney roof in defining ventilation rate and air flow. In the

study, various angles of tilt was listed to match the latitude of the experimentation site. For location like Malaysia, the latitude is estimated of 3˚ N and the optimum tilt angle is 55˚in enhancing the velocity air flow according to Table 1 attached. This research further supports the shape and design of the roof of the chimney used for our current study. Regrettably, the literature of the study done by Khanal and Lei face contradiction claims to other literature in their study, making their findings less viable as to indicate further extension into their research should be carried out [16].

Table 1: Variation of Optimum Inclination of Solar Chimney with Latitude [16].

Latitude (˚)	Optimum inclination (˚)
0	55
5	50
10	50
15	50
20	45
25	45
30	45
35	50
40	50
45	55
50	55
55	60
60	60
65	60

Yong and Nyuk conducted an experiment in a 3 story Zero Energy building located in Singapore concerning natural ventilation using a solar chimney. The interesting findings obtained from this experimental study is that by using a chimney width of 1m, and a length of 15.9m, they are able to obtain exit velocities up to 1.9m/s with temperatures achieving 47˚C while the solar chimney surface temperature reach up to 60˚C on a hot day by selecting the best inlet location within the building. This is extremely productive highlighting that no external heat source was required other than solar [17].
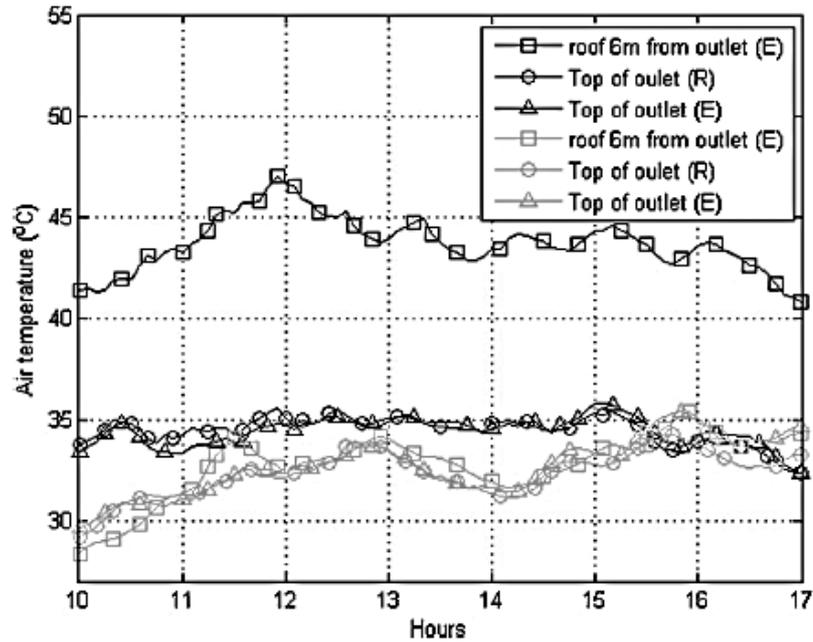
Figure 2.2: Air temperatures within solar chimney system along rooftop (black: hot day,grey: cool day) [17]
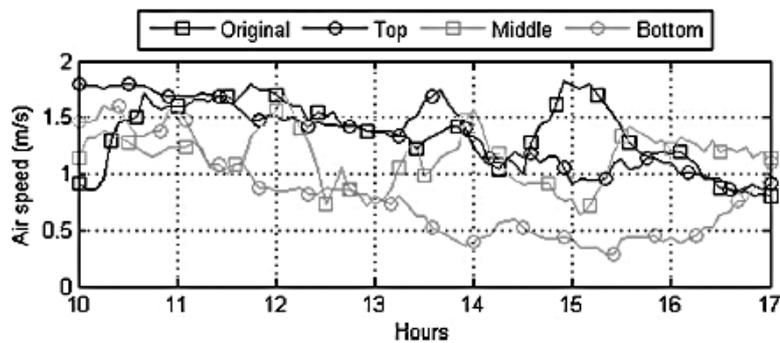
Figure 2.3: Air speeds within solar chimney system for various inlet positions. [17]

Arcea's research of an experimental study of natural ventilation on solar chimney proofed a valuable piece of literature to our research as it studies the relationship between the air flow rate of the chimney and the influence of a pressure difference caused by thermal gradients of the inlet and outlet. The research as well studies the influence of external wind on the air flow rate of the chimney. Although, from this research, the cross sectional area of the solar chimney was fairly huge being $0.3m^2$. This results in a low average exit velocity of 0.05m/s. From the findings of the research, it is found that the maximum air flow rate is obtained during the period of

high temperature and high ambient wind velocities achieved between 15:00 hours and 18:00 hours of the day [18]. These findings are in line with the current research we are carrying out regarding using the ambient wind velocities and temperature gradient to aid the air flow rate with the use of a chimney cup.
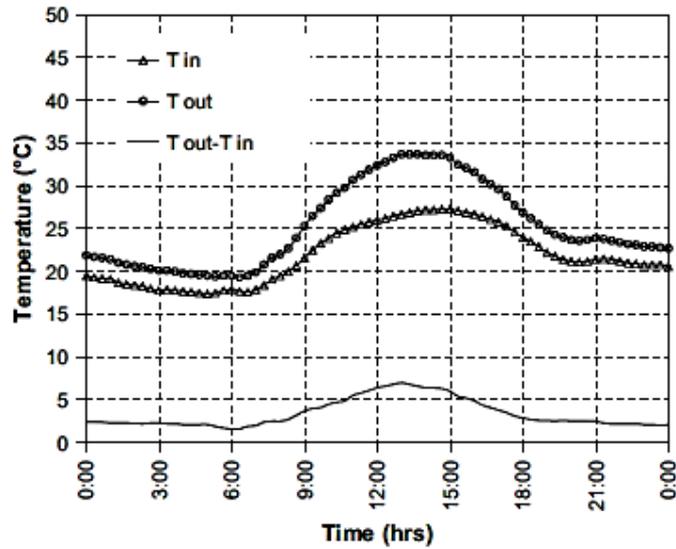
Figure 2.4: Inlet and outlet air temperatures, and their difference (September 15th 2007) [18]
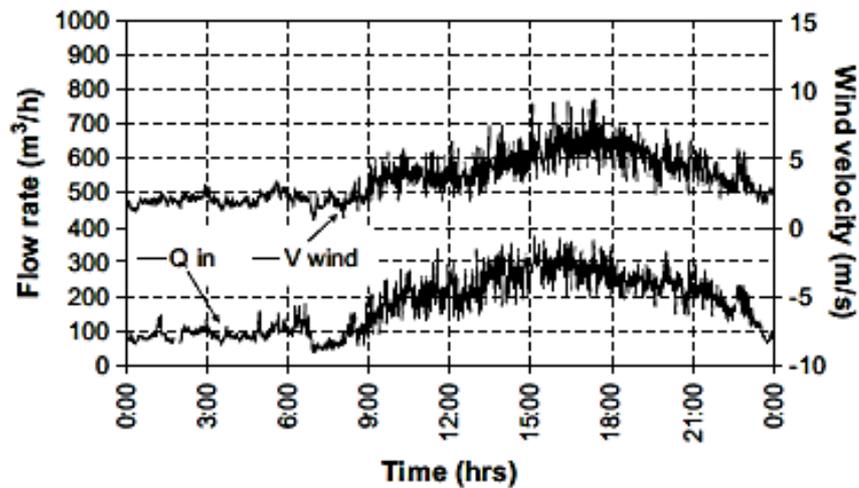
Figure 2.5: Air flow rate (Q in) in m3/h and (V wind) wind velocity at 10 m high (September 15th 2007) [18].

8

Based on the research done by Zamora and Kaiser, which did a numerical study on mixed buoyancy-wind driving induced flow in a solar chimney, they managed to discover an interesting relationship between external wind velocity and buoyancy effect. It is found in that research that when external wind velocities are low, the buoyancy forces are analogous for wind driving effects. Alternatively, when external wind velocities are high, the wind suction effects become the dominant factor in enhancing air flow rate. Due to the computational set up, the wind direction plays a significant role as a negative value in wind velocity (opposite wind direction), it creates inflow of air into the chimney. [19] This would not be the case for our study as the chimney cup is made to alter external wind flow from any direction to aid the airflow rate of the chimney.
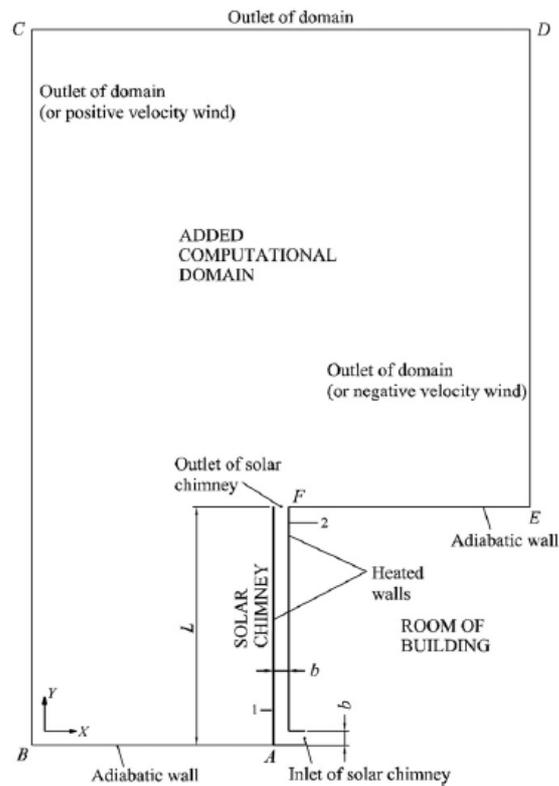
Figure 2.6: Scheme of the complete computational set up [19].
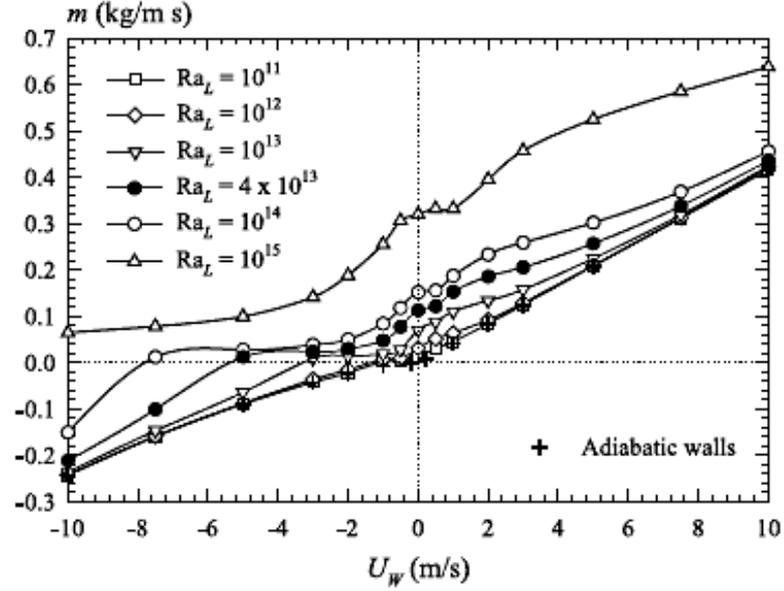
Figure 2.7: Mass flow rate versus external wind velocity, for Ra values of different lengths, with uniform heat flux heating condition [19].

2.2 Working Principle

2.2.1 Stack Ventilation

"Buoyancy or stack effect is a result of the air temperature difference between the micro and macro climate. The subsequent variation of air density and pressure gradient of the indoor and outdoor air masses causes the warm air (less dense) to rise up and escape through the wind tower's exhaust. Consequently, new air is drawn into replace the air that escaped." [20]

$$dP_s \quad = \quad - r_{ins} \cdot g \cdot T_{ins} (h_2 - h_1) \left(\frac{1}{Tout} - \frac{1}{Tins}\right) \qquad (2.1)$$

where;

dP_s is the stack effect pressure (Pa), and r_{in} is the air density inside stack (kg/m^3), and g is the acceleration due to gravity (9.81 m/s^2), and h_1 is height of inlet of stack above datum (m), and h_2 is the height of outlet of stack above datum (m), while T_{out} is the temperature of air outside stack (^{o}K) and T_{ins} is the temperature of air inside stack (^{o}K).

10

2.2.2 Bernoulli's Principle

Bernoulli's principle is based on wind speed differences to displace fluid (air). As the velocity of the fluid increases, the pressure of the fluid is lowered. This is core working principle of the chimney cup as it is driven by differential internal and ambient pressures elaborates that these are significantly influenced by the opening geometry and compressibility of the fluid flowing through that opening [21]. The broad range of applications of the Bernoulli's principle in the engineering field spans from aerospace wing design to designing pipes for fluid movement.

$$\frac{p}{\rho g} + \frac{u^2}{2g} + z = H = \text{constant} \tag{2.2}$$

$$P + \frac{1}{2}\rho u^2 + \rho g z = \text{constant} \tag{2.3}$$

where;

pressure head is $\frac{p}{\rho g}$ and P, while the velocity head is $\frac{u^2}{2g}$ and $\frac{1}{2}$, and potential head is z and $\rho g z$ and the total head is H.

With the following assumptions:

- Flow is steady;

- Density is constant (which also means the fluid is incompressible);

- Friction losses are negligible.

- The equation relates the states at two points along a single streamline, (not conditions on two different streamlines).

2.3 Related Work

The project research is focused on comparing the different types of chimney top enhancements available in the market, and thus Table 2 shows the analysis on the advantages and disadvantages of various enhancements for comparison purpose.

Table 2: Comparison of Different Types of Chimney Top Enhancement.

No.	Type of Enhancement	Energy Dependency	Working Principle	Advantages	Disadvantages
1	Chimney Cup	Passive	Increase exit velocity creating pressure difference	Low cost, easy to produce	Improper design can compromise efficiency
2	Chimney Fan	Active	Use a fan to create pressure difference producing updraft	Prevents downdraft	Requires pump and electrical input.
3	Damper	Passive	Reduce heat loss, stack ventilation	Prevents heat leakage	High maintenance due to moving parts
4	Chimney Extension	Passive	Increases high difference to increase updraft.	Increase updraft, prevents downdraft	Increased wind resistance produces unwanted vibrations

2.4 Summary

From the literature review, it can be seen that various types of chimney top enhancements were compared of various attributes. However, there are current research done on chimney cups conclude that the main function of the chimney cup is to reduce the damage caused by water penetration such as rain and snow [5]. We would like to test the theory that chimney cups function more than just a barrier but acts as a flow enhancement as well for the chimney. With that aim in mind, it appears that the chimney cup provides the most potential to be further studied in this thesis such as the design parameters and the effect of those design parameters on the enhancement of the flow. The design parameters to be studied are the geometrical shape which is the design angle and the gap length between the chimney cup from the outlet of the chimney using an experimental and numerical approach.

Chapter 3: Methodology

Reviewing different working principles on creating natural updraft, the chimney cup is most suited as it relies on both the Bernoulli's principle and stack ventilation effect to optimize its functionality. By conducting experimental and numerical evaluation on the cup design angle design and gap length between the cup and the outlet, the aim is to determine the design parameters of the chimney cup which produces the maximum updraft flow.

Before any investigations can be carried out, we first need to understand the project by carrying out extensive study of previous related literature. This would us a comprehensive overview and understanding of the related works produced by other researchers and enable us to define our problem statement and objectives. Subsequently, we are required to identify and understand the working principles of our project. This will highlight the parameters that would affect our results and ultimately be the focus of our investigations. Only then, we are able to carry out our investigations via experiments and computer simulations.

The measurements and data collected from the experimental evaluation will be compiled and results will be produced based on readings obtained from the measuring instruments to determine the average exit velocities. A computational model will be created and simulated to produce an ideal interpretation of the experimental model. The results obtained from the experimental model and the simulated model will be compared to determine the percentage of error to improve the credibility and reliability of the findings. The flow of project methodology is shown in Figure 3.0.1.
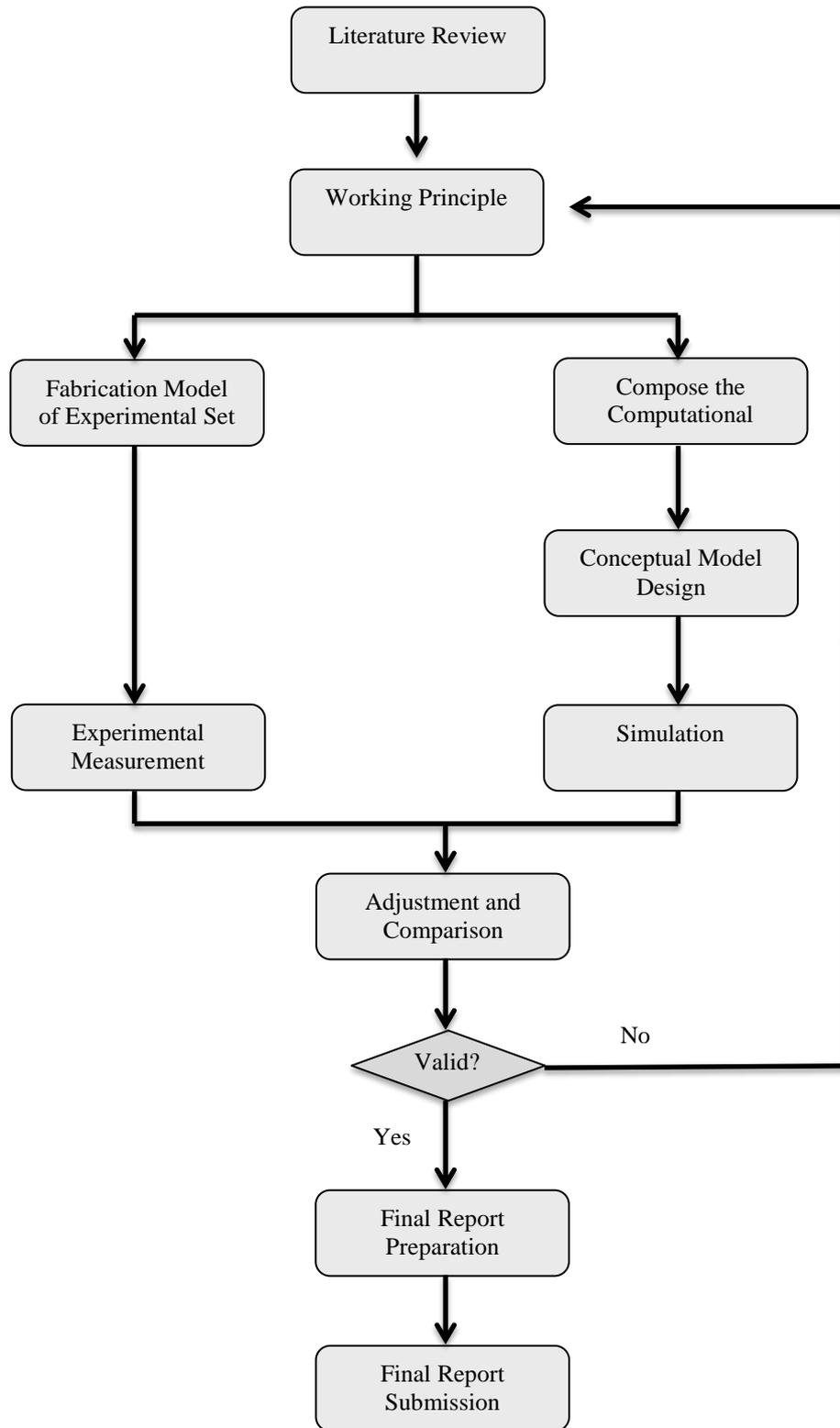
Figure 3.1: Project Methodology

3.1 Experimental Investigation

3.1.1 Fabrication and Modification

A total of 4 chimney cups with various design angles consisting of 180˚, 150˚, 120˚ and 90˚ were manufactured with a support diameter of 4". In the second half of FYP duration, modifications to the chimney cup support was made to reduce its diameter from 4" to 2" to accommodate the 2" pipe being used for the experimental investigation of this research.

a) 180˚   b)150˚

c) 120˚   d)90˚

Figure 3.2: Actual chimney cups to be used for experimental investigation.

Besides the chimney cups, a heater support stand was fabricated as well. The two heater coils purchased had a 0.28m x 0.28 dimension. These dimension were uncommon and the requirements for them to be located at least 10cm above the trolley rack and 5cm apart from one another has made it difficult to find a support stand for these coils. Hence, a custom made support stand is required to be fabricated. Using steel bars, these bars are welded to form a steady cross base with a 0.28m x 0.28m dimensions that extends upwards with the first set of holders located at 10cm and the second set of holders are located at 15cm from the base. The support stand is meant to be placed on the trolley rack. The purpose for the minimum height of the support stand to be of 10cm is to ensure the heat from the heater coils do not burn or damage the wooden trolley rack.

3.1.2 Experimental Set Up

For the experimental investigation, the set up consisted of several parts and components which is listed Table 3 below :

Table 3. List of Components and Quantities Used in the Experimental Set Up.

Part/Component	Quantity
Trolley Rack	1
Heater Support Stand	1
1 kW Air Heater Coil	2
Chimney Base	1
4" to 2" Pipe Reducer	1
2m length x 2" diameter PVC Pipe	1

Figure 3.3 below represents the actual experimental set up, which was assembled in laboratory of University Technology Petronas, located in block 18-00-05. The location for the experiment was set to be held in an indoor laboratory with high ceiling to eliminate the chance of outdoor wind velocities which would affect the natural updraft and provide inconsistent results.
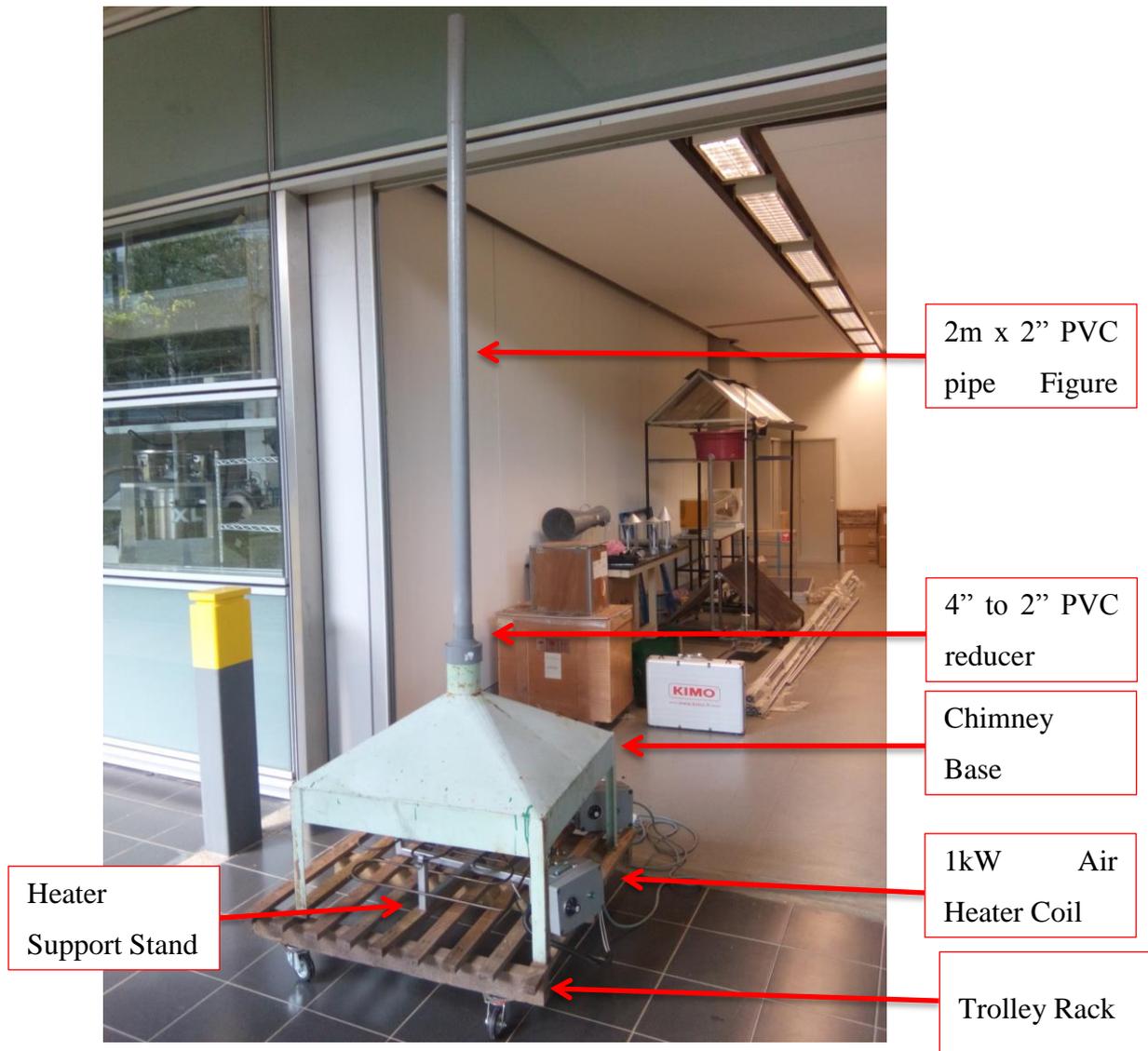
Figure 3.3: Experimental Set Up of Natural Up drafting Experiment Model.

3.1.3 Instrumentation

For this experiment, we will only be using one measuring instrument which is the Kimo VT 200.This measuring instrument consist of many different probes to measure various parameters, but we will only be using the hot wire anenometer as the measuring device to measure the velocity and temperature of the up drafting air.

Figure 3.4: Hot wire anenometer.

3.1.4 Experiment Procedure

- Set up the experiment as shown in Figure 3.1.2.1.
- Switch on the heaters and turn the temperature knob to 80˚C for both heaters.
- Wait for 10 minutes to allow the temperature to stabilize and the air flow to be fluent and consistent.
- Switch on the hot wire anenometer and place the probe at the chimney base, with the hot wire perpendicular to the flow of the air.
- Note the temperature and velocity readings.
- Place the probe at the tip of the chimney, at the end of the 2 meter pvc pipe, with the hot wire perpendicular to the flow of air.
- Note the temperature and velocity readings.
- Take 3 readings, and calculate the average temperature and velocity readings.

- Repeat the entire process with the addition of installing a chimney cup of 90˚, 120˚, 150˚ and 180˚ for positions 3cm, 4.5cm, 6cm and 7.5cm from the tip of the chimney.

- After the initial 10 minutes, readings can be taken at 5 minutes intervals for each change of chimney cup or position.

3.2 Numerical Investigation

3.2.1 Computational Modelling

A computational model of the actual experimental model is designed in both 3D and 2D using Autodesk Inventor Professional 2016 and ANSYS Workbench 15.0. The dimensions of the experimental setup is measured and recorded to ensure similarity between the computational model and experimental model. For this research, only the 2D model designed using the ANSYS Workbench 15.0 is used for the numerical simulation. The 3D model can be imported into ANSYS for future works using 3D simulation.

Chimney Cup

2m x 2" PVC pipe   Figure

4" to 2" PVC reducer

Chimney Base
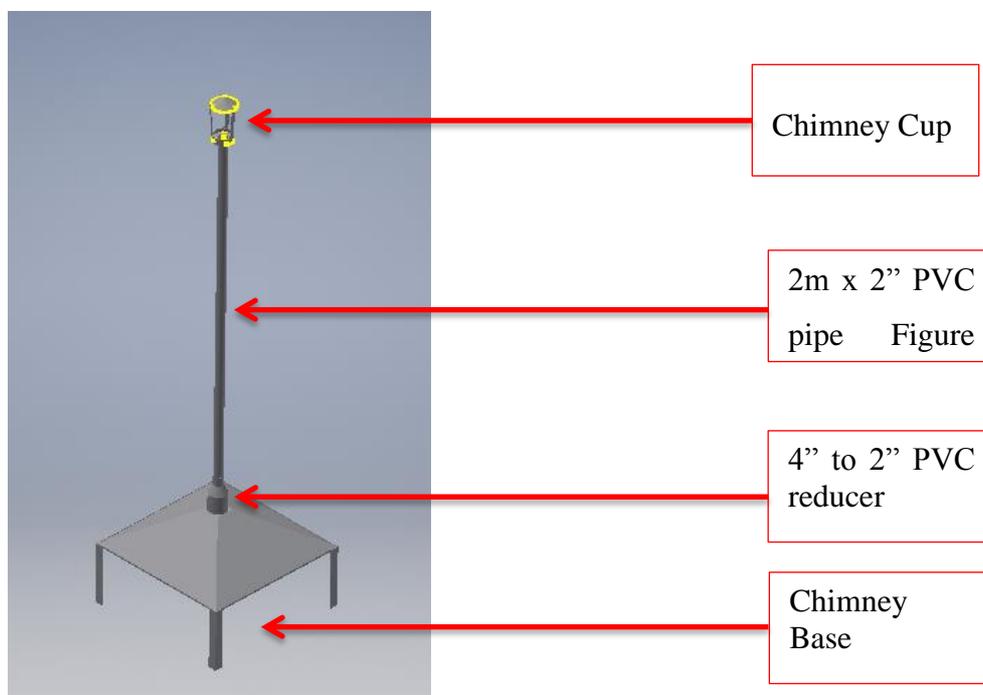
Figure 3.5: 3D Computational model of the actual experiment model designed using Autodesk Inventor Professional 2016 with an addition of a chimney cup attached.

20
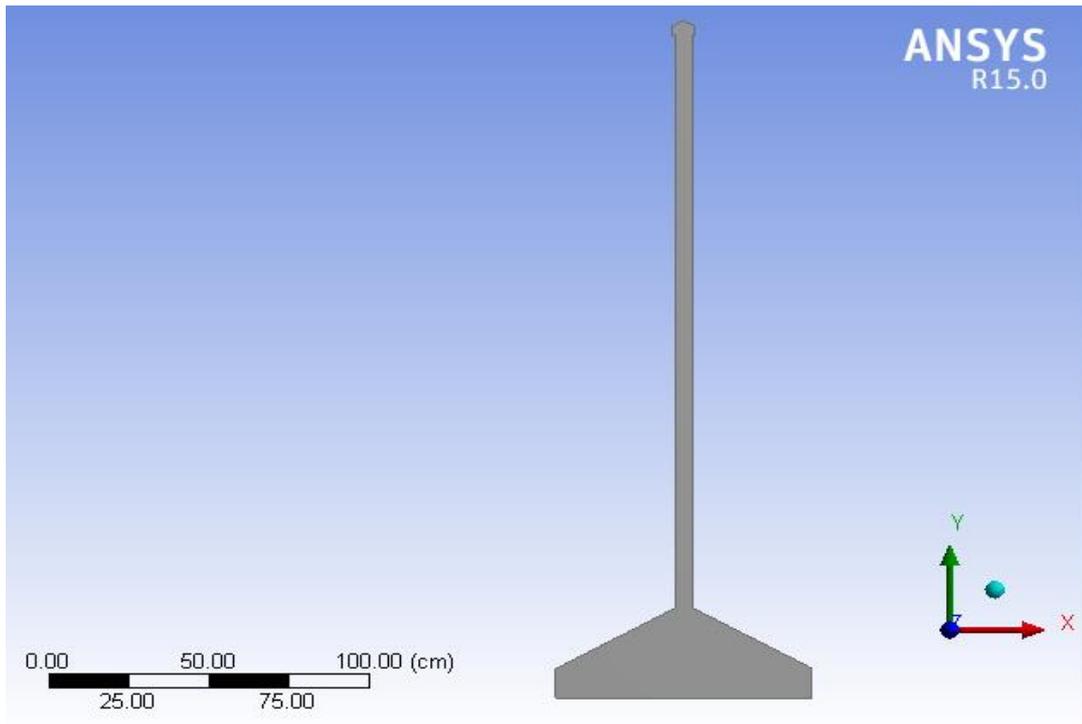
Figure 3.6: 2D Computational model of the actual experiment model designed using ANSYS Workbench 15.0 with an addition of a chimney cup attached.

The mesh setting of the 2D computational model is done with the relevance center and span angle size both set to 'Fine'. The rest of the settings are left as default resulting in a total of 593 nodes and 471 elements.
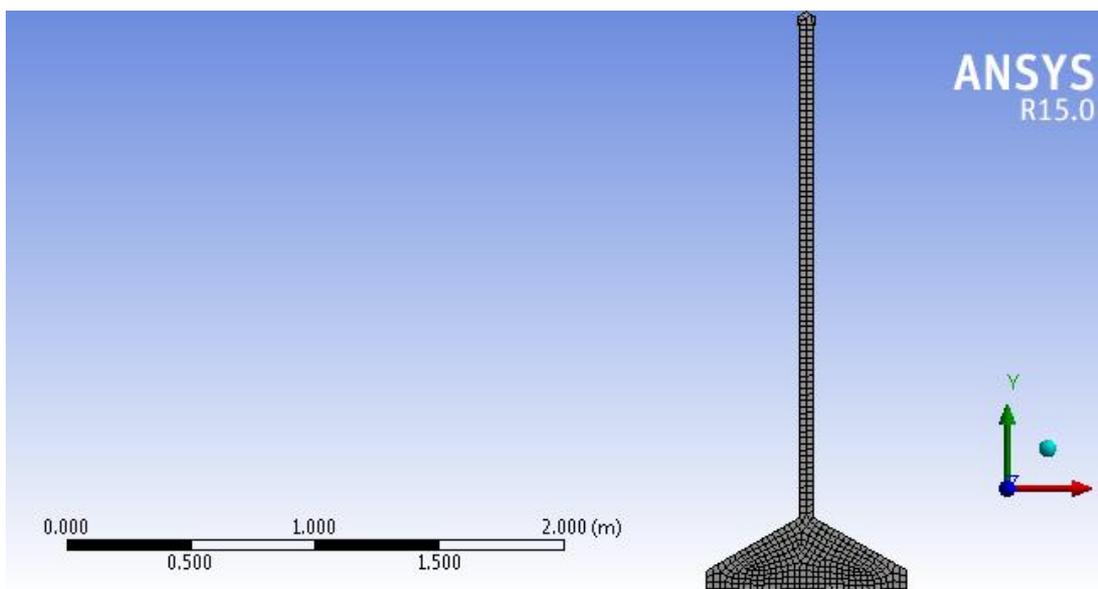
Figure 3.7: 2D mesh model for simulation using ANSYS Fluent.

3.2.2 Simulation Procedure

- Replicate the experimental model by creating a 2D computational model using Geometry on ANSYS Workbench with all required dimensions.

- Select and name the inlet and outlet of the geometry.

- Mesh the geometry using 'Fine' relevance centre and span angle size. The rest of the settings should be set as their default settings.

- In the setup, the Energy Equation and K-Epsilon Equation should be employed.

- The velocity at the inlet should be set to 0.22m/s to replicate the experimental inlet flow velocity.

- The model is initiatized using hybrid initilization.

- The simulation is then run until the results converge.

- Both the velocity vectors and pressure contours should be captured as the results for further analysis.

- Repeat all steps for each model with design angles of 180˚, 150˚, 120˚ and 90˚ with gap length of 3cm, 4.5cm, 6cm and 7.5cm. There should be an overall of 17 models including the model without a chimney cup.
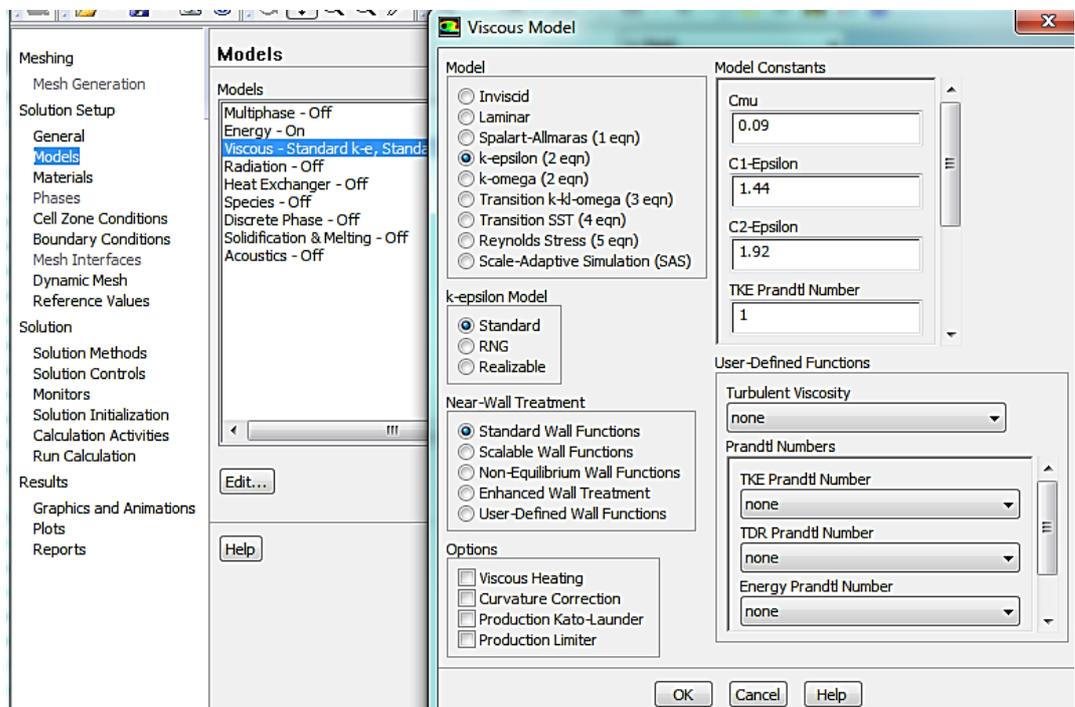
Figure 3.8: Simulation settings

3.3 GANTT CHART AND KEY MILESTONE

Table 4: Project Timeline Within 29 Weeks For Fyp I And Fyp II

No.	Details/ Week	FYP 1													
		1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Literature Review	█	█	█	█	█	█	█	█	█	█	█	█	█	█
2	Working Principle Identification				█	█									
3	Conceptual Model Design										█	█		▲	
4	Fabrication Model of the Experimental Set Up												█	▲	
5	Compose the Computational Model														█

No.	Details/ Week	FYP II														
		1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	Experimental Measurements	█	█	█			█	█	▲							
2	Simulation				█	█				█	▲					
3	Adjustment & Comparison					█	█				█	▲				
4	Final Report Preparations											█	█			
5	Submission													█		

▲ Key milestone

█ Process

3.3.1 Project Milestones

Table 5 : Project Milestones Within 29 Weeks For Fyp I And Fyp II

Key Milestones Achieved	Start Date	End Date
Creating the conceptual model design	Week 10	Week 13
Fabricating the experimental set up	Week 12	Week 13
Carrying out experimental measurements	Week 15	Week 22
Carrying out simulation of computational model	Week 18	Week 24
Comparing of experimental and numerical results and make appropriate adjustments	Week 19	Week 25

Chapter 4: Results and Discussion

4.1 Results

4.1.1 Experimental Results

After carrying out the experiment, the data collected using the anemometer instrument is recorded down and later calculated to obtain the average reading which would be taken as the final reading. The flow velocity data collected are registered into a table and using the information gathered from the table, a graph was plotted using Microsoft Excel. By plotting the graph, it illustrates the relationships between the experimental flow velocities in response to the design parameters which were the gap length and the chimney cup design angle.

Table 6. Table of Experimental Flow Velocity Corresponding with Various Cup Gap Lengths and Various Chimney Cup Design Angles.

	Experimental Flow Velocity (m/s)			
Cup Gap Length (cm)	3.0	4.5	6.0	7.5
Chimney Cup Angle Design (°)				
180	2.57	2.66	2.55	2.51
150	2.59	2.80	2.69	2.60
120	2.72	2.87	2.78	2.69
90	2.69	2.83	2.75	2.70
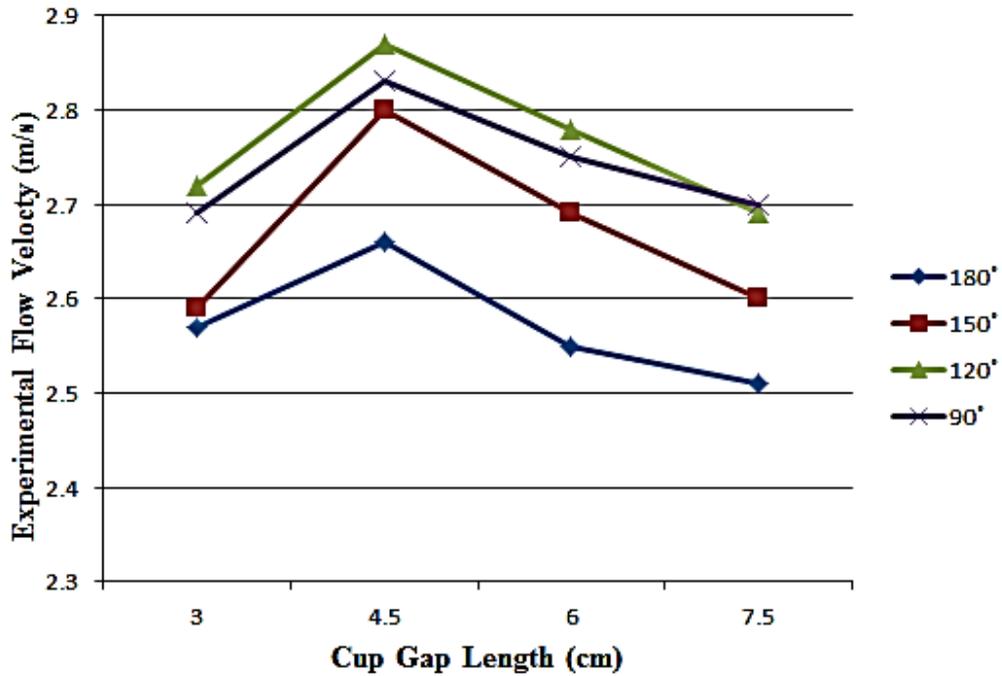
Figure 4.1: Graph of experimental flow velocity corresponding with various cup gap lengths and various chimney cup design angles.

4.1.2 Numerical Results

After the experiment, crucial data such as the inlet velocity and dimensions of the experimental setup were recorded to produce a replicate 2D model using ANSYS Workbench. The model is then used to simulate the air flow within the chimney and the data of the velocity vectors and pressure contours are then collected and recorded into a table. Using Microsoft Excel, a graph is plotted with the tabulated data in the table. By plotting the graph, it illustrates the relationships between the numerical flow velocities in response to the design parameters which were the gap length and the chimney cup design angle.

Table 7: Table of Numerical Flow Velocity Corresponding with Various Cup Gap Lengths and Various Chimney Cup Design Angles.

	Simulated Flow Velocity (m/s)			
Cup Gap Length (cm)	3.0	4.5	6.0	7.5
Chimney Cup Angle Design (°)				
180	3.41	3.50	3.33	3.15
150	3.46	3.62	3.51	3.32
120	3.58	3.79	3.61	3.53
90	3.53	3.72	3.61	3.50
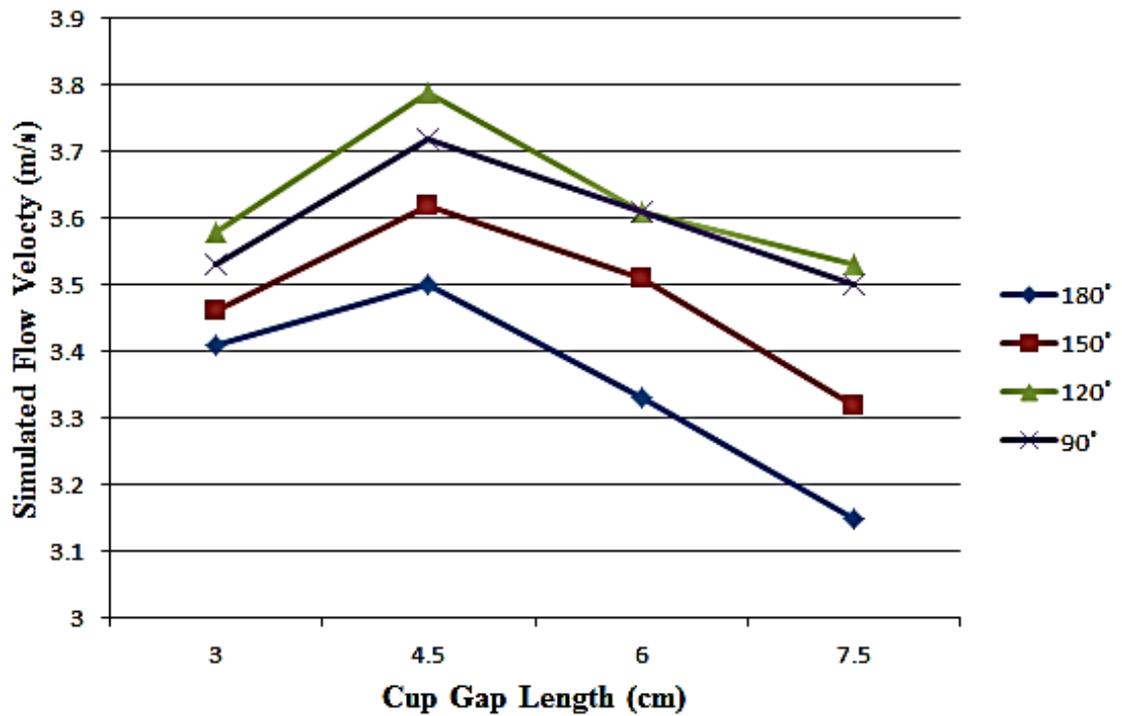
Figure 4.2: Graph of numerical flow velocity corresponding with various cup gap lengths and various chimney cup design angles.

4.2 Discussion

4.2.1 Results Analysis

From the results of the experimental and numerical investigations, we can observe a very similar trend in both graph patterns and the peak of both graphs is at 120˚ chimney cup design angle with a 4.5cm cup gap length. For the achievement of such high flow velocity, we would need to study the velocity vectors and pressure contours from the simulation for further understanding.
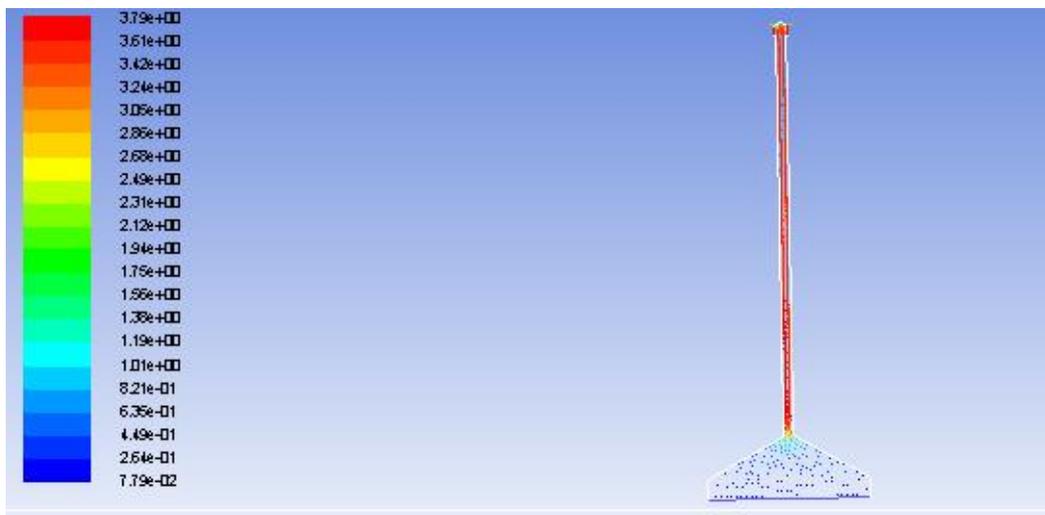
Figure 4.3: Velocity vector of the chimney cup design angle at 120˚ with gap length of 4.5cm.
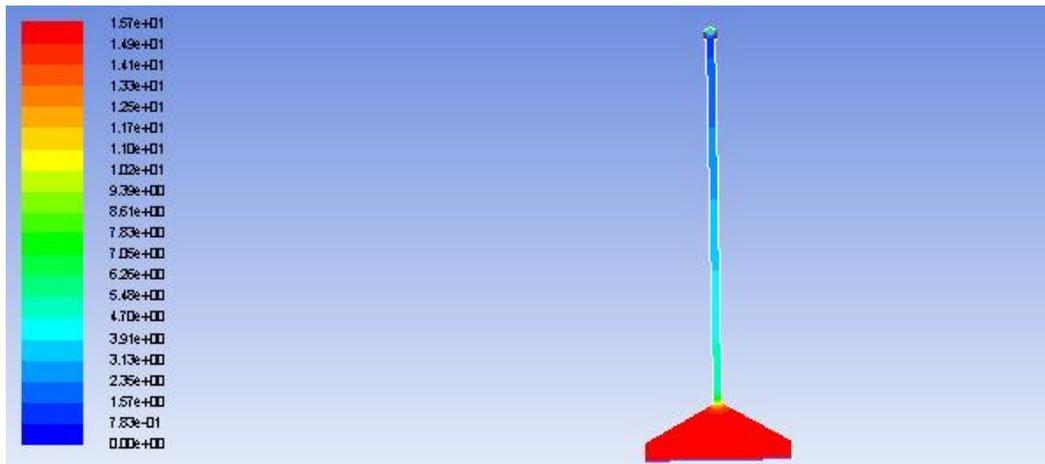
Figure 4.4: Pressure contour of the chimney cup design angle at 120˚ with gap length of 4.5cm.
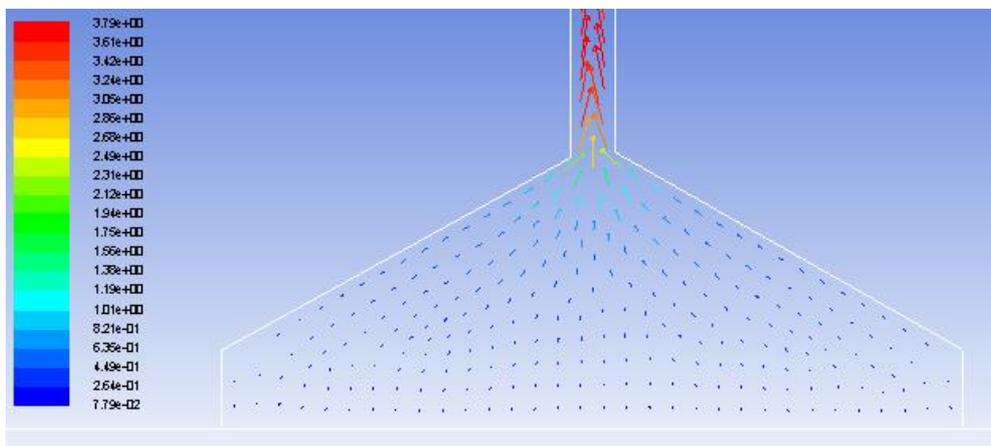
Figure 4.5: Velocity vector converging at the entrance of the pipe.

From studying the velocity vectors, we can observe that the velocity of the flow gradually increases as they are channeled from an inlet with a large cross sectional area into a small pipe opening. Upon entering the pipe, the flow continues to develop progressively until it has reached its maximum flow velocity at the outlet. This analysis can be further supported by the pressure contour which shows the pressure of the air at the inlet initially at high pressure and it gradually loses its pressure as it passes through the chimney. This is due to the stack effect and the Bernoulli's principle. According to Bernoulli's principle, as the velocity is inversely proportional to pressure, hence as velocity increases, pressure decreases, and as the velocity of the flow develops and increases, it pressure will gradually decrease inversely with it. The

stack effect explains the development of the flow velocity as the air is heated by the heating coils, the density decreases, causing the air to rise through the chimney. The pressure difference between the inlet and outlet causes a suction effect which then increases its flow velocity.

In discussion of the cup gap lengths, it can be seen that 4.5cm is the optimum gap length for all chimney cups used, following with both 3.0cm and 6.0cm having similar results and lastly 7.5cm with the poorest results. One of the reasons that may explain the decrease in flow velocities with increasing gap length beyond 4.5cm is the pressure losses to the environment. By increasing the gap length, the outlet is more exposed and this causes the pressure difference between the inlet and outlet to decrease, hence, reducing the suction effect which enhances the velocity of the flow. As for the 3.0cm gap length, it can be due to the outlet area was unable to accommodate the mass flow rate of the air channeled through the chimney causing a buildup of pressure within the chimney, hence, the flow was inhibited, hindering the full development.

Figure 4.6: Pressure contour of the chimney cup design angle at 120˚ with gap length of 7.5 cm.

Figure 4.7. Pressure contour of the chimney cup design angle at 120˚ with gap length of 3.0cm.

As for the discussion on chimney cup design angles, the readings produced by the 150˚, 120˚ and 90˚ cup does not differ much in comparison, this is because due to additional area created by the these 3 cup designs does not impact as much as compared to the overall scale of the experiment. Although the highest velocity output is by the 120˚ even when compared with the larger area 90˚ chimney cup, this could be caused by design of the 90˚ which traps the exhaust air, which then causes a buildup of pressure opposing the flow. Large variation can be seen by the results of the 180˚ design chimney cup, this could be due to the flat design that acts as a barrier to the exhaust air, resulting in poor overall performance of the chimney.

4.2.2 Numerical and Experimental Result Comparison and Percentage of Error

The numerical and experimental results will be compared to as a measure to verify the reliability of the results documented. The percentage of error will be calculated by employing the follow formula.

$$\% \, \text{Error} = \frac{|\text{Experimental} - \text{Theoretical}|}{\text{Theoretical}} \, \text{x} \, 100\%$$

For this calculation, the experimental results would be represented as the experimental value and the simulated results will be represented as the theoretical value.

<u>180˚ Chimney Cup</u>

Table 8: Table of Percentage of Error for 180˚ Chimney Cup with Various Gap Lengths.

Gap Length (cm)	Experimental Flow Velocity (m/s)	Numerical Flow Velocity (m/s)	Percentage of Error (%)
3cm	2.57	3.41	24.6
4.5cm	2.66	3.50	24.0
6cm	2.55	3.33	23.4
7.5cm	2.51	3.15	20.3

<u>150˚ Chimney Cup</u>

Table 9: Table of Percentage of Error for 150˚ Chimney Cup with Various Gap Lengths.

Gap Length (cm)	Experimental Flow Velocity (m/s)	Numerical Flow Velocity (m/s)	Percentage of Error (%)
3cm	2.59	3.46	25.1
4.5cm	2.80	3.62	22.7
6cm	2.69	3.51	23.3
7.5cm	2.60	3.32	21.7

120˚ Chimney Cup

Table 10: Table of Percentage of Error for 120˚ Chimney Cup with Various Gap Lengths.

Gap Length (cm)	Experimental Flow Velocity (m/s)	Numerical Flow Velocity (m/s)	Percentage of Error (%)
3cm	2.72	3.58	24.0
4.5cm	2.87	3.79	24.3
6cm	2.78	3.61	23.0
7.5cm	2.69	3.53	23.8

90˚Chimney Cup

Table 11: Table of Percentage of Error For 90˚ Chimney Cup with Various Gap Lengths.

Gap Length (cm)	Experimental Flow Velocity (m/s)	Numerical Flow Velocity (m/s)	Percentage of Error (%)
3cm	2.69	3.53	23.8
4.5cm	2.83	3.72	23.9
6cm	2.75	3.61	23.8
7.5cm	2.70	3.50	22.9

Based on Tables 8 to 11, the range of errors is from 20.3% to 25.1%, for all four chimney cup design geometry. This is due to the higher simulated values obtained as compared to the experimental values. Several factors could contribute to the variation in results as the simulated value is considered to be an ideal model that does not take into account of the instantaneous surrounding conditions, the flow resistance of different materials and geometry involved.

Chapter 5 : Conclusion and Recommendation

5.1 Conclusion

The first objective of the study was achieved by identifying the effective design parameters of the chimney cup which were the cup gap length and the chimney cup geometry. Furthermore, we have successfully achieved the second objective which was to investigate of the influence of those parameters through numerical and experimental investigations. The experimental investigations were carried out by manufacturing and setting up the experimental setup, followed by conducting the experiment and collecting data for in-depth analysis. A numerical investigation was carried out following the collection of experimental data, as the initial conditions were input into a replicated computational 2D model to simulate the actual flow within the chimney. The results obtained experimentally were consistent with the simulated results, both results reflecting the theories involved such as the stack effect in a chimney and the Bernoulli's principle. From the findings, it can be concluded that the optimum gap length required to produce the highest flow velocity is 4.5cm, whereas for cup design angle, 120° is the ideal design as it enhances the flow significantly.

5.2 Recommendation

Despite the conclusive statement, there were errors present throughout the research that would affect the accuracy of the results, hence, resulting in the percentage of error. Although the percentage of error is moderately acceptable, further identifications can be made to rectify and improve future work.

Firstly, one of the possible errors made in the experiment is the experimental setup. It would be recommended if a holder for the sensor should be installed as measuring the data with the anemometer by hand causes inconsistencies in the data collected.

Secondly, the measuring instrument should have a more real-time response and reading. The Kimo VT 200 interface encounters a lag time to make large variation in readings, for example, from the still air of the lab to the outlet of the chimney, the interface requires a slow inclination through time, this may cause a false data being collected. It is recommended a more modern and recent measuring device be used that delivers real-time data.

Third, during the simulation, two assumptions were made which caused the variation in results. The first assumption was the inlet velocity be at a constant 0.22m/s throughout the entire cross section of the inlet. This is not accurate as the heater coils were slightly smaller than the chimney base cross sectional area, hence, causing the inlet velocity to not be constant across the inlet. The second assumption made was the roughness of the material remained constant throughout the entire model. This is also not accurate as this assumption does not account for the different roughness of the steel and PVC material used, as well as resistance induced by bends and turns.

Lastly, the simulated model was done in a 2D model whereas the experimental model is carried out as a 3D model in the real world, hence, the 2D model is not able to capture the full geometry of the entire model. Therefore, the simulated model is slightly too ideal to replicate the experimental model, resulting in the variation in outcomes. It is recommended in future that the simulation done in future be of a 3D model with assigned material to produce more realistic findings.

References

1. Engineers, B. M. C. (1999). Wind Towers–Detail in Building Academy Editions, New York: John Wiley & Sons Ltd.

2. Lai, C.-m. (2003). "Experiments on the ventilation efficiency of turbine ventilators used for building and factory ventilation." Energy and Buildings 35(9): 927-932.

3. Ansmann, A., Fruntke, J., & Engelmann, R. (2010). Updraft and downdraft characterization with Doppler lidar: cloud-free versus cumuli-topped mixed layer. Atmospheric Chemistry and Physics, 10(16), 7845-7858.

4. Pemberton-Pigott, C., Robinson, J., Cook, C., Molapo, V., & Annegarn, H. (2010). Emissions reductions from domestic coal burning: demonstration of an innovative bottom lit down draft stove.

5. Farshadmanesh, P., et al. (2014). "Impact of Chimney-top Appurtenances on Flue Gas Flow."

6. Santamouris, M. and F. Allard (1998). Natural ventilation in buildings: a design handbook, Earthscan.

7. Khan, N., et al. (2008). "A review on wind driven ventilation techniques." Energy and Buildings 40(8): 1586-1604.

8. Santamouris, M. and P. Wouters (2006). Building ventilation: the state of the art, Routledge.

9. Guzman, J. A. (2012). Carbon monoxide poisoning. Critical care clinics, 28(4), 537-548.

10. Hampson, N. B., Piantadosi, C. A., Thom, S. R., & Weaver, L. K. (2012). Practice recommendations in the diagnosis, management, and prevention of carbon monoxide poisoning. American journal of respiratory and critical care medicine, 186(11), 1095-1101.

11. Koski-Harja, A. M. (1995). Draft Enhancing and Downdraft Inhibiting Characteristics of Chimney Caps.

12. Shreir, L. L. (Ed.). (2013). Corrosion: corrosion control. Newnes.

13. Farías, O., et al. (2008). "Theoretical and experimental study of the natural draft in chimneys of buildings for domestic gas appliances." Energy and Buildings 40(5): 756-762.

14. Sakonidou, E., et al. (2008). "Modeling of the optimum tilt of a solar chimney for maximum air flow." Solar Energy 82(1): 80-94.

15. Afriyie, J. K., et al. (2011). "Simulation and optimisation of the ventilation in a chimney-dependent solar crop dryer." Solar Energy 85(7): 1560-1573

16. Khanal, R. and C. Lei (2011). "Solar chimney—A passive strategy for natural ventilation." Energy and Buildings 43(8): 1811-1819.

17. Tan, A. Y. K. and N. H. Wong (2012). "Natural ventilation performance of classroom with solar chimney system." Energy and Buildings 53: 19-27.

18. Arce, J., et al. (2009). "Experimental study for natural ventilation on a solar chimney." Renewable Energy 34(12): 2928-2934.

19. Zamora, B. and A. Kaiser (2010). "Numerical study on mixed buoyancy-wind driving induced flow in a solar chimney for building ventilation." Renewable Energy 35(9): 2080-2088.

20. Hughes, B. R., et al. (2012). "The development of commercial wind towers for natural ventilation: A review." Applied Energy 92: 606-627.

21. M.P. Straw, C.J. Baker, A.P. Robertson, Experimental measurements and computation of the wind induced ventilation of a cubic structure, Journal of Wind Engineering and Industrial Dynamics 88 (2000) 213–230.

APPENDICES
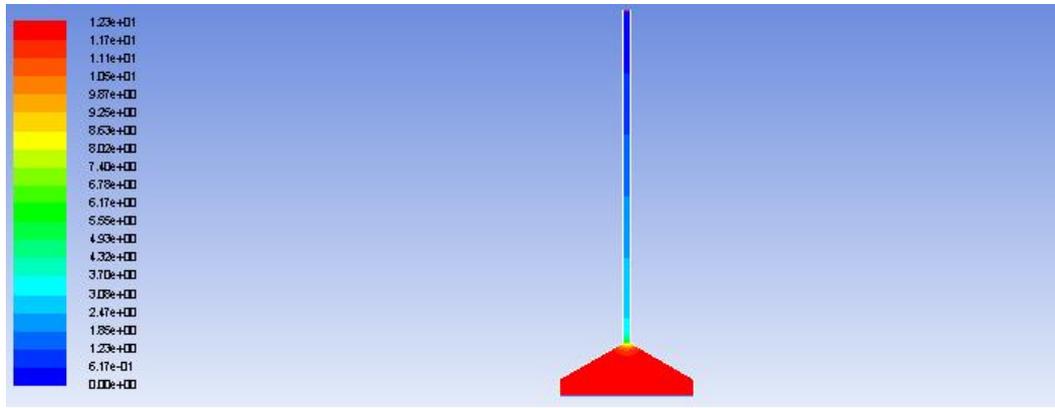
Figure A.1: Pressure contour for without cup
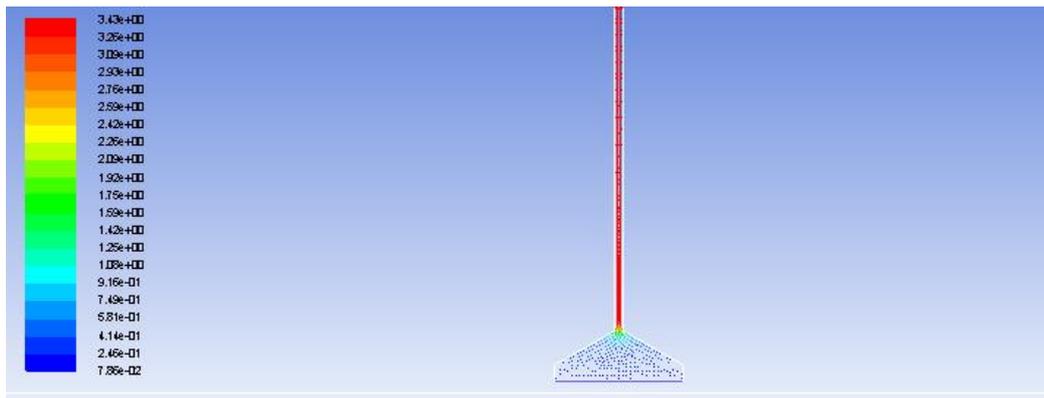
Figure A.2: Velocity vector without cup
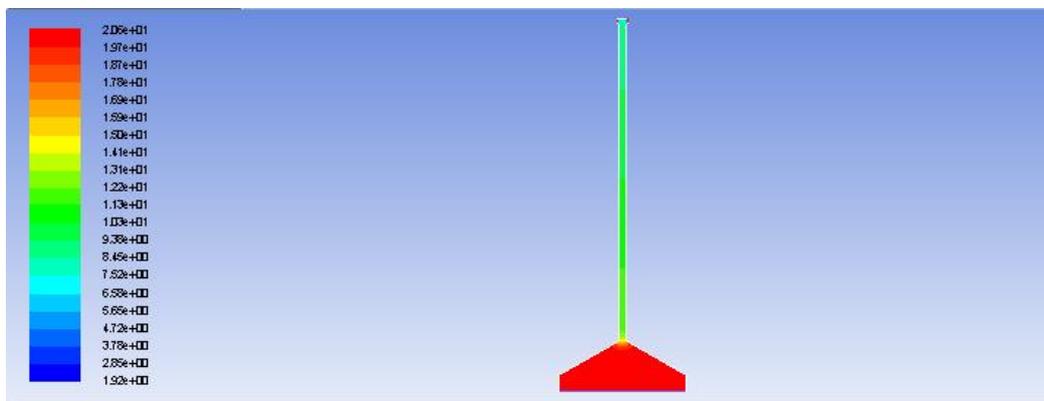
Figure A.3: Pressure contour for 180˚ cup at 3cm
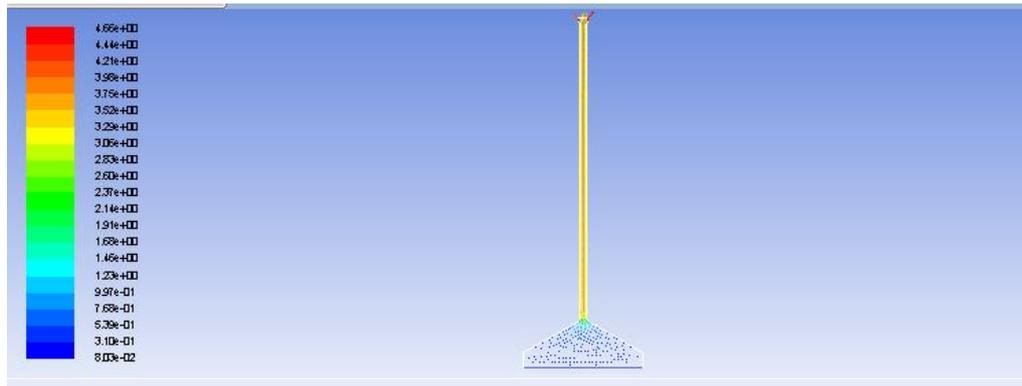
Figure A.4: Velocity vector for 180˚ cup at 3cm
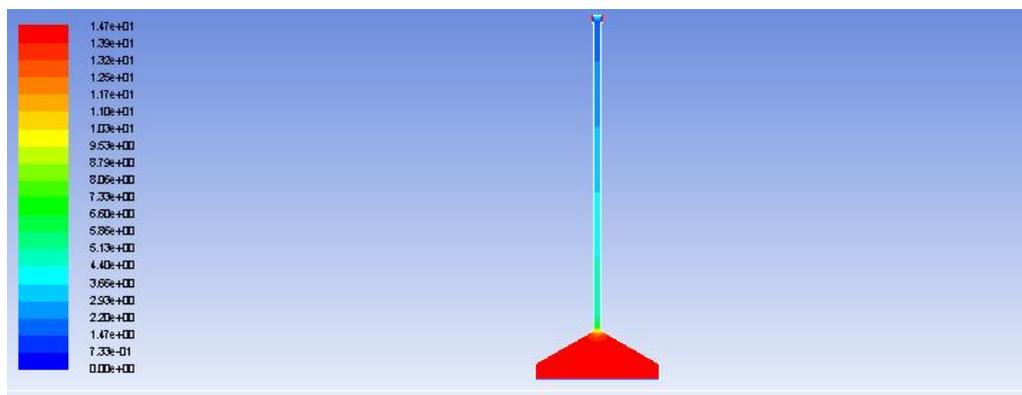
Figure A.5: Pressure contour for 180˚ cup at 4.5cm
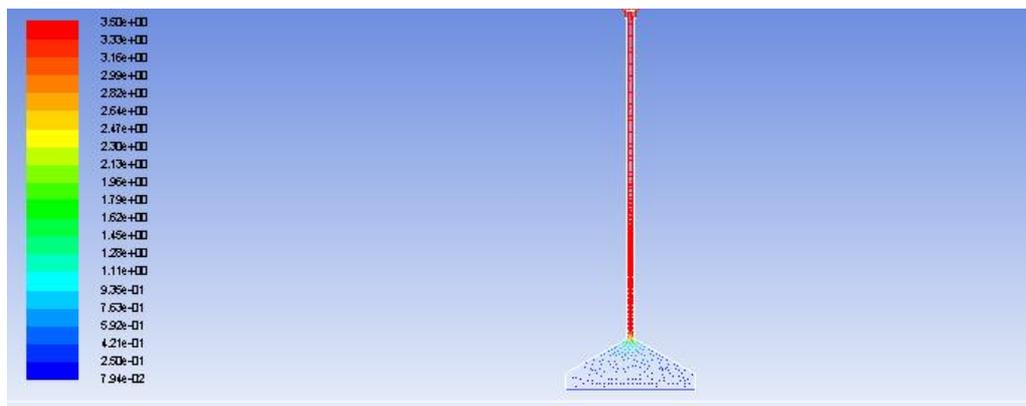
Figure A.6: Velocity vector for 180˚ cup at 4.5cm
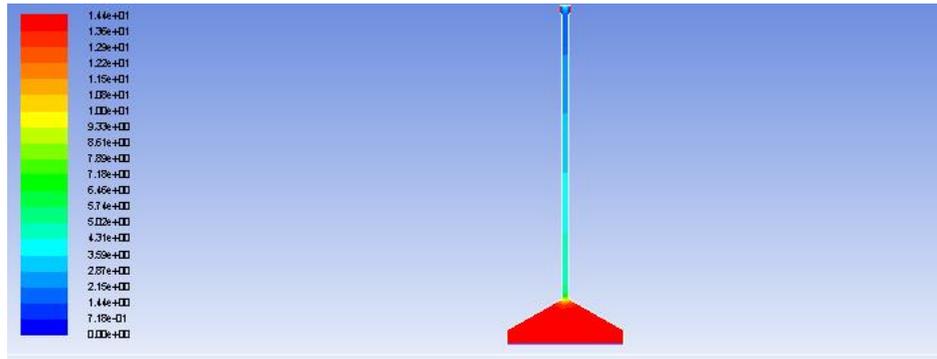
Figure A.7: Pressure contour for 180˚ cup at 6cm
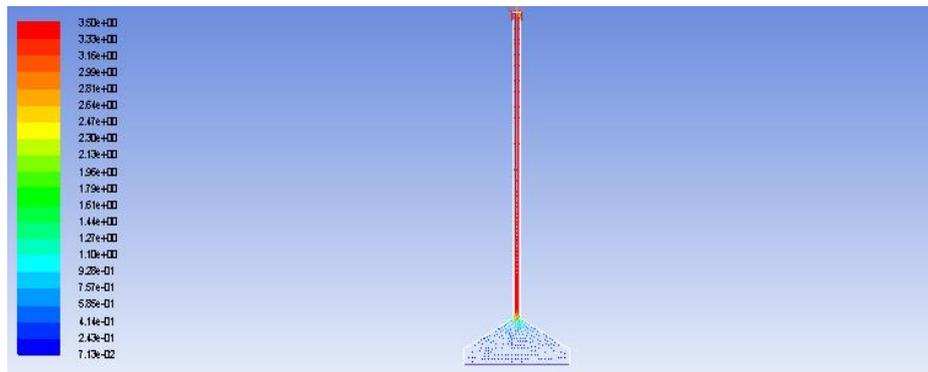
Figure A.8: Velocity vector for 180˚ cup at 6cm
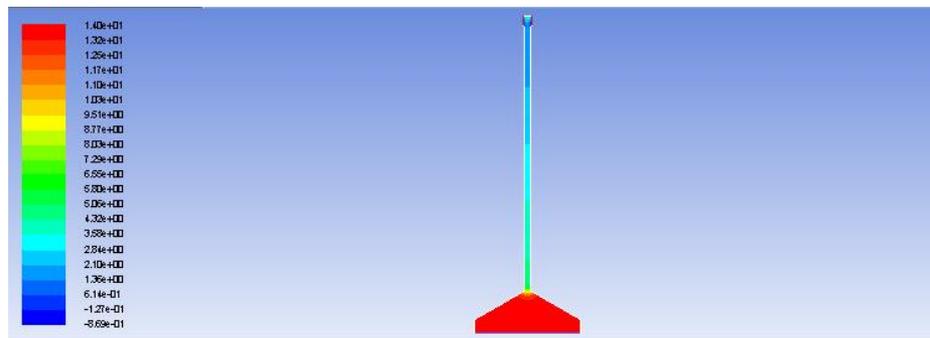
Figure A.9: Pressure contour for 180˚ cup at 7.5cm
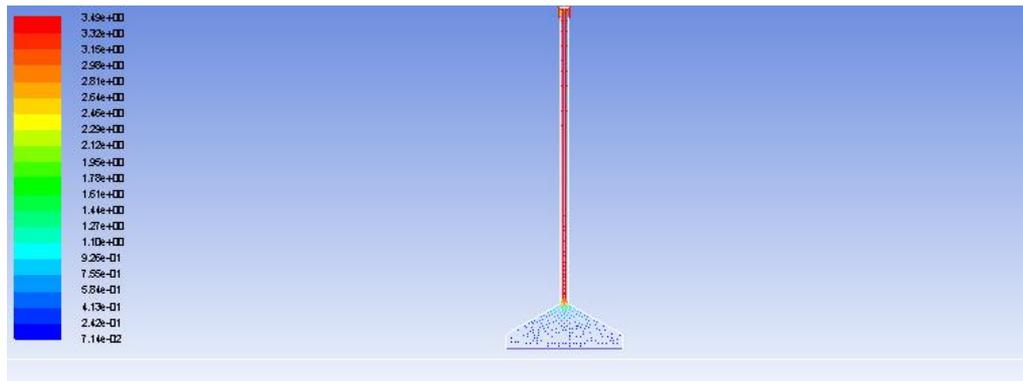
Figure A.10: Velocity vector for 180˚ cup at 7.5cm
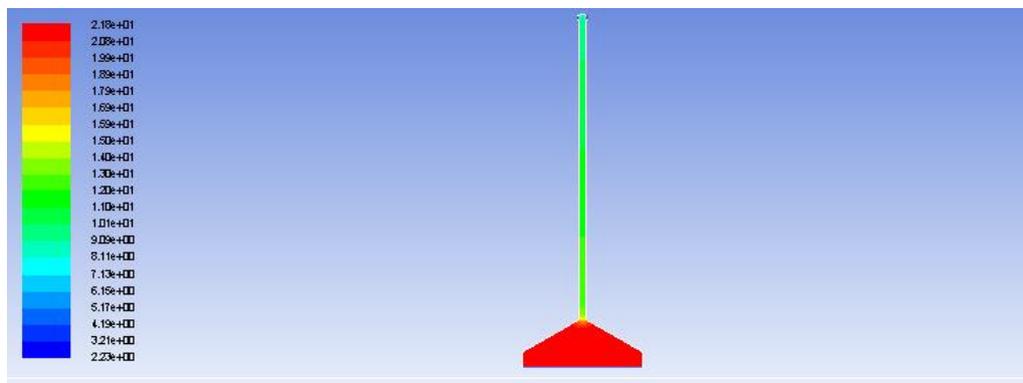
Figure A.11: Pressure contour for 150˚ cup at 3cm
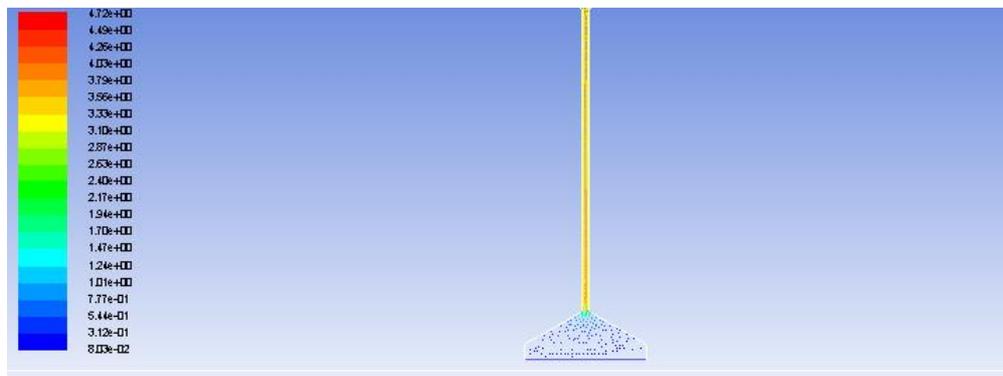
Figure A.12: Velocity vector for 150˚ cup at 3cm
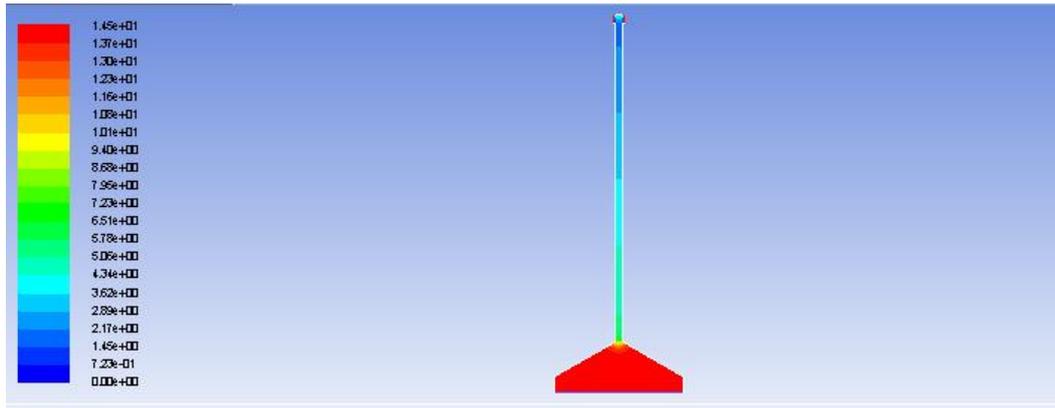
Figure A.13: Pressure contour for 150˚ cup at 4.5cm
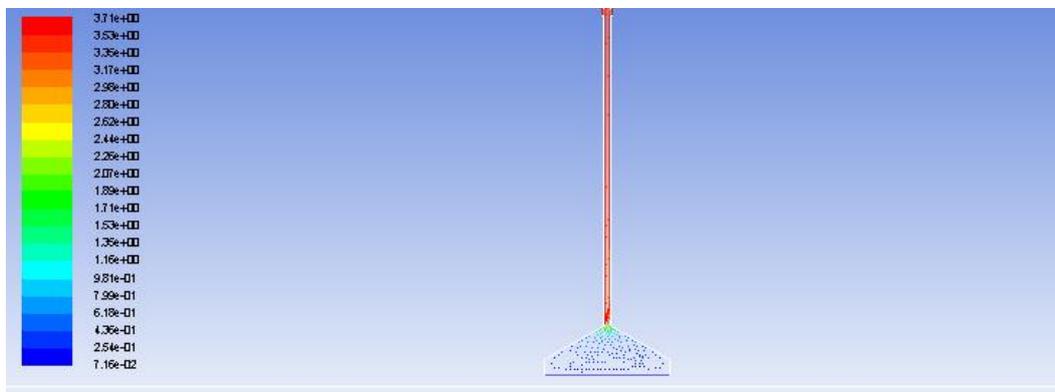
Figure A.14: Velocity vector for 150˚ cup at 4.5cm
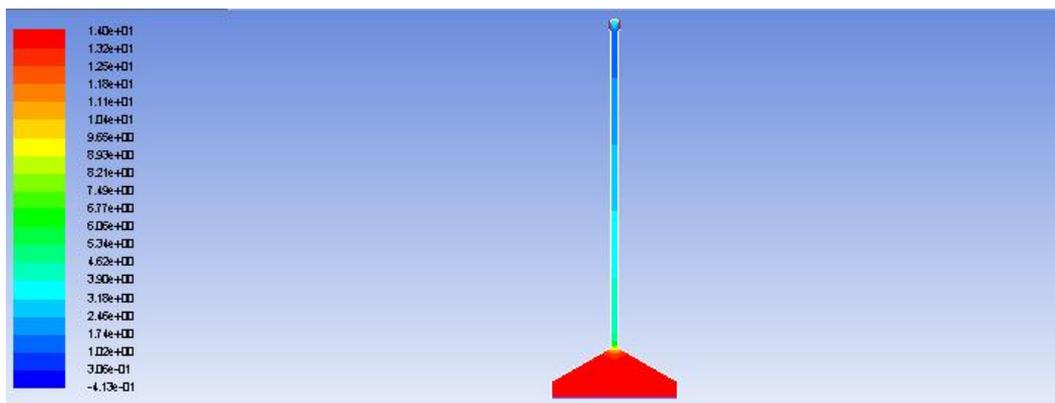
Figure A.15: Pressure Contour for 150˚ cup at 6cm
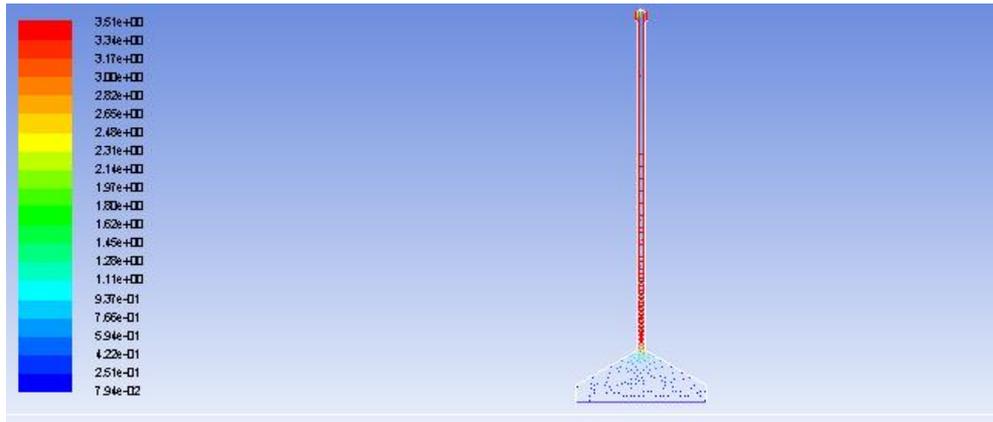
Figure A.16: Velocity Vector for 150˚ cup at 6cm
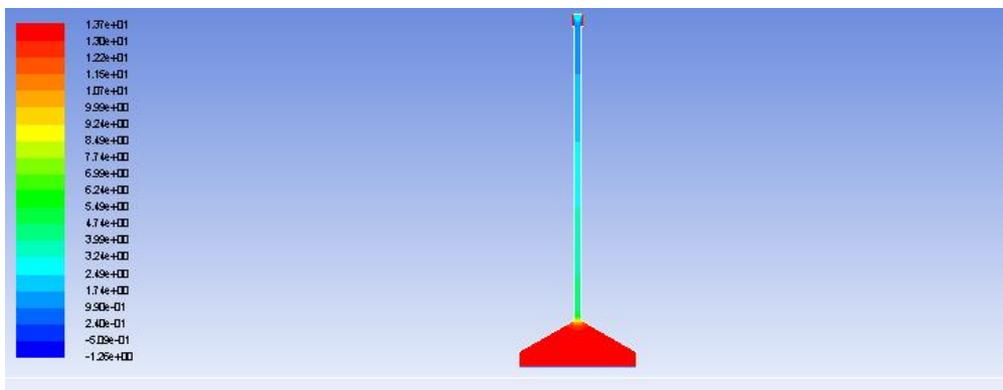
Figure A.17: Pressure contour for 150˚ cup at 7.5cm
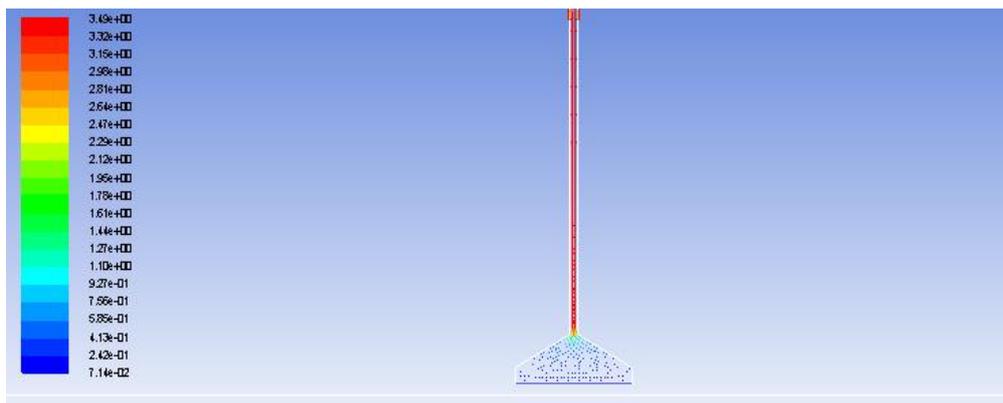
Figure A.18: Velocity vector for 150˚ cup at 7.5cm
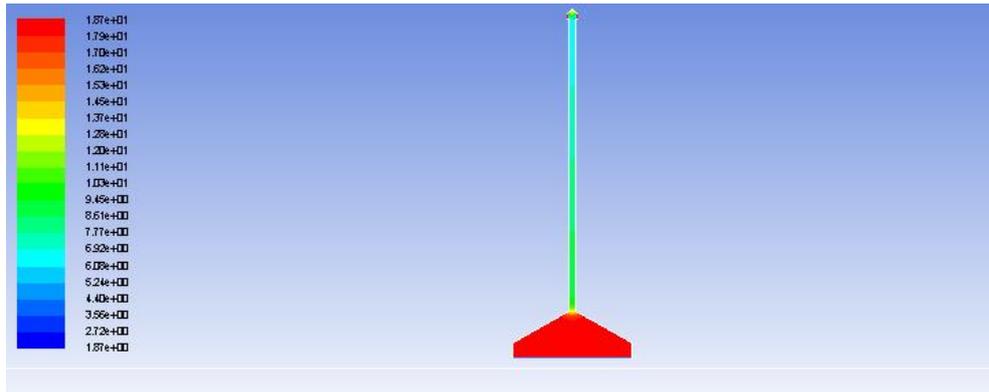
Figure A.19: Pressure contour for 90˚ cup at 3cm
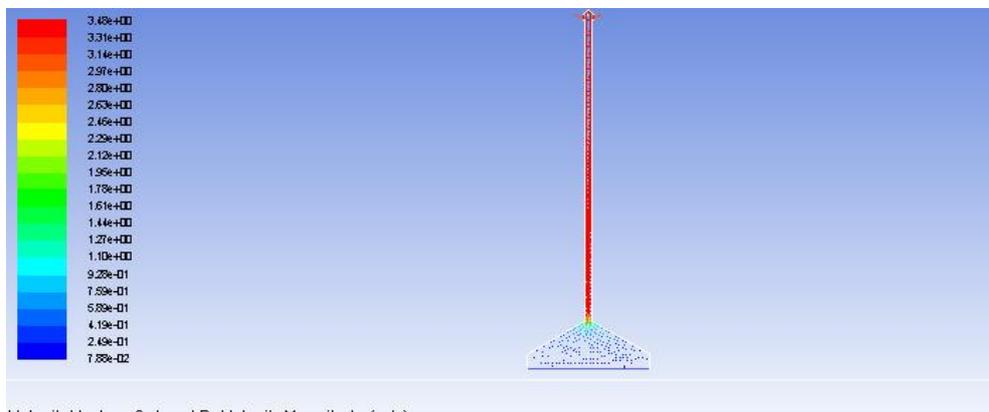
Figure A.20: Velocity vector for 90˚ cup at 3cm
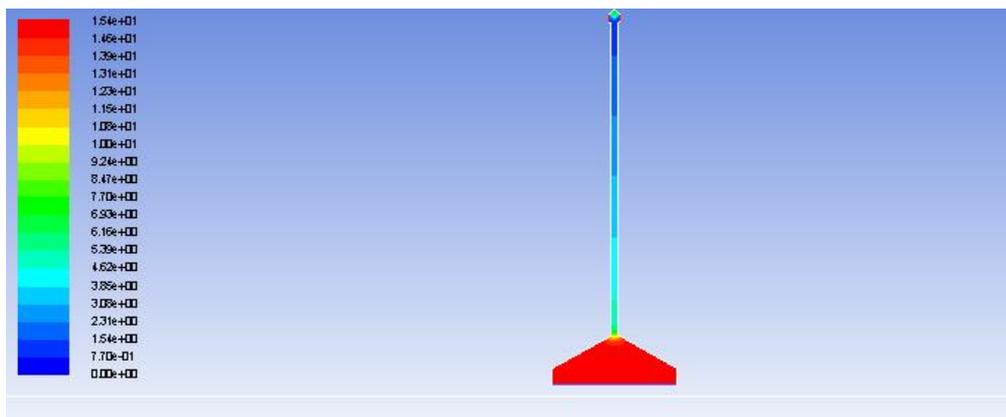
Figure A.21: Pressure contour for 90˚ cup at 4.5cm
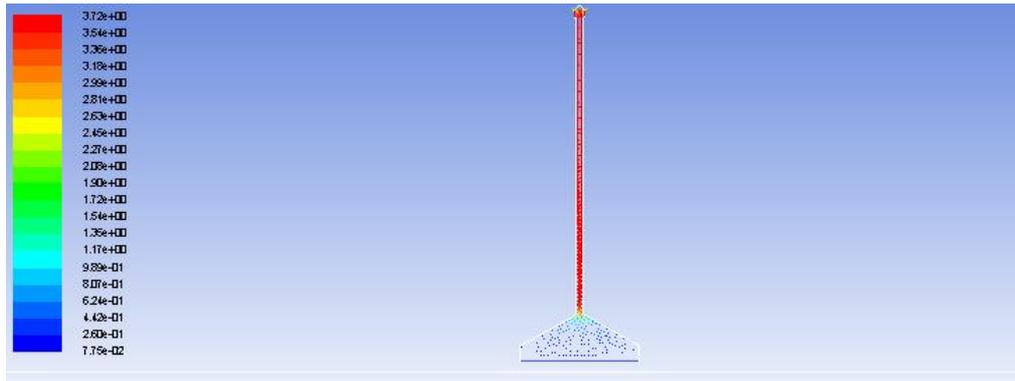
Figure A.22: Velocity vector for 90˚ cup at 4.5cm
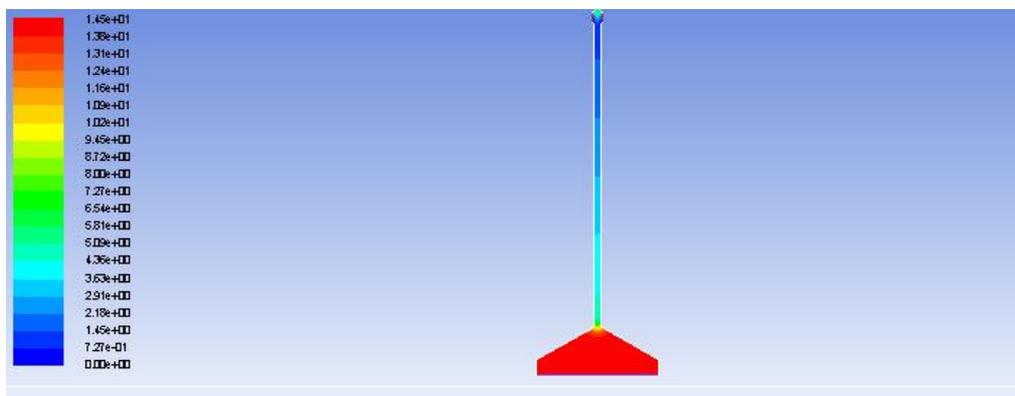
Figure A.23: Pressure contour for 90˚ cup at 6cm
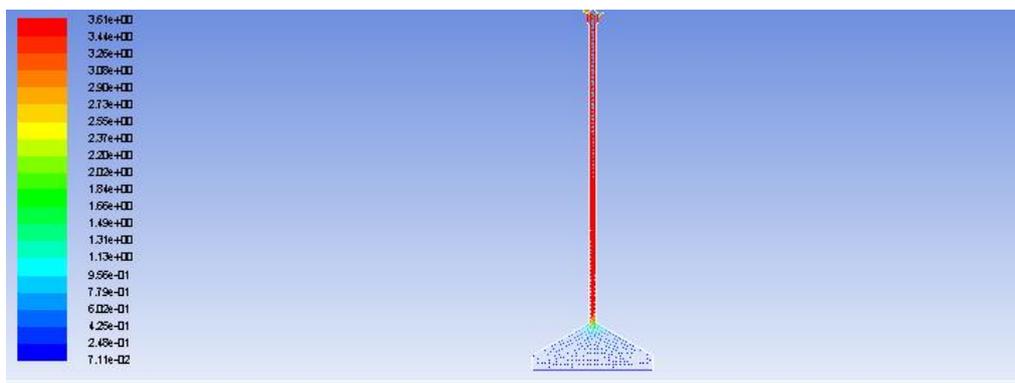
Figure A.24: Velocity vector for 90˚ cup at 6cm
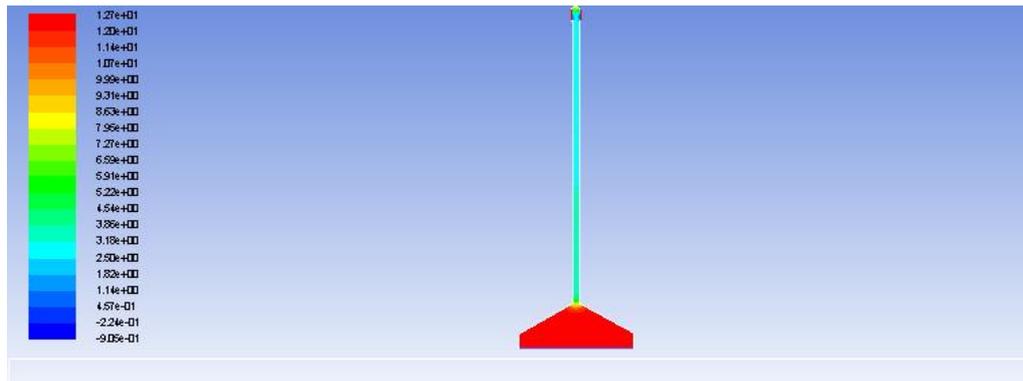
Figure A.25: Pressure contour for 90˚ cup at 7.5cm
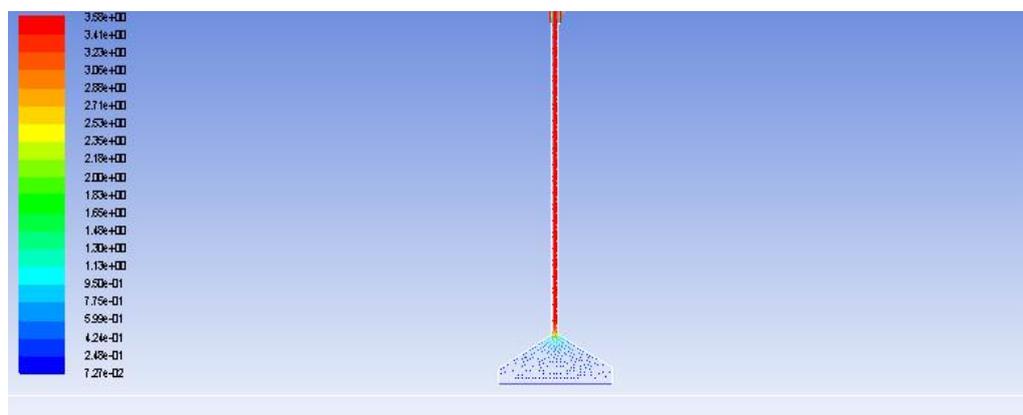
Figure A.26: Velocity vector for 90˚ cup at 7.5cm