INVESTIGATION ON THE EFFECT OF THE CHIMNEY CUP ON THE NATURAL UPWARD DRAFTING

by

CHONG YEW POR

14514

MECHANICAL ENGINEERING

Project Dissertation is submitted in partial fulfillment

of the requirement for the Bachelor of Engineering (Hons)

(Mechanical Engineering)

SEPTEMBER 2013

Universiti Teknologi PETRONAS

Bandar Seri Iskandar

31750 Tronoh

Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Investigation On The Effect Of The Chimney Cup On The Natural Upward Drafting

by

CHONG YEW POR

14514

Project Dissertation is submitted to

Mechanical Engineering Department

Universiti Teknologi PETRONAS

as partial fulfillment of requirements for the

BACHELOR OF ENGINEERING (Hons)

(MECHANICAL ENGINEERING)

Approved By,

____

Dr. Hussain H. Al-Kayiem

Universiti Teknologi PETRONAS

Bandar Seri Iskandar

31750 Tronoh

Perak Darul Ridzuan

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 my acknowledgement, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

CHONG YEW POR

14514

Mechanical Engineering Department

Universiti Teknologi PETRONAS

Bandar Seri Iskandar

31750 Tronoh

Perak Darul Ridzuan

ACKNOWLEGDEMENT

First and foremost, my deepest gratitude to God, for He had given me strength and guidance to overcome the challenges and problems in completing the 9 months Final Year Project.

My sincerest appreciation to Dr. Hussain H. Al-Kayiem, my supervisor for giving me full mentorship, guidance, and support to perform my duties and also helping me develop as a professional and individual throughout my project period. His crucial contribution has kept my research work on the right track.

Secondly, I would like to acknowledge Mr. Tadahmun, for his continuous support and assistance over the experiment related matter, including the setup of the experiment and calibrating the measuring tools, all during Ramadhan month.

I would also like to express my gratitude to Ms. Sreejaya for her endless support and assistance over the simulation using ANSYS-Fluent. None of the simulation is possible without her selfless guidance and advise.

Not to forget are my friends and family for their moral support and encouragement along the project. For without them, I would not be able gain the courage to face the countless challenges in completing the project.

Last but not least is a thank you to Universiti Teknologi PETRONAS (UTP) for providing a conductive learning environment for the students of UTP.

ABSTRACT

Increasing interest is obvious nowadays on natural ventilation system which has been left out since 1970s due to the development of the mechanical ventilation systems. The potential benefits in terms of operational cost, energy requirement and carbon dioxide emission have raised more interest than ever in the passive ventilation systems. A significant amount of research work has been done on the solar chimney since the 1990s. Most of the studies focused on the effects of geometry and inclination angle on the ventilation performance of a solar chimney. Computational Fluid Dynamics (CFD) technique, which has been adopted as investigation tool in the present research, has also attracted considerable amount of interest due to the ability to compute numerical modeling of solar chimney. However, published studies showed that the natural passive ventilation has not been fully understood. The understanding is that the drafting force of the chimney is mainly due to the wind velocity at the top of the chimney and the thermal gradient. The geometry of the chimney cup could be another factor in creating a greater upward drafting force; hence, four different cup configurations have been investigated.

In addition to the CFD analysis, an experiment set up has been designed, fabricated and used to investigate different geometries of chimney cup in order to find out the optimum design conditions. The CFD analysis using ANSYS- FLUENT version 14 software has been validated by comparing with the experimental measurements. Some cases have shown large discrepancy and it is contributed to the measuring instrument installation. In many cases, reasonable agreement was achieved. Increased outlet surface of the chimney cup showed a decrease in the outlet pressure and thus resulted in greater pressure difference, and eventually a greater upward drafting force or higher air flow velocity.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLE

1.0 INTRODUCTION

1.1 Background of study

Natural ventilation has been around for centuries and is considered as an important sustainable building design strategy. However, little information has been found about this passive system, which possibly due to the invention and development of the mechanical ventilation system. Mechanical ventilation requires undesirable energy consumption such as electricity. When operational and climatic are suitable, natural ventilation has the ability to provide 10% cooling energy savings and 15% of annual energy consumption on fan power savings [1]. High carbon monoxide emission to the environment is proved to be yet another downside for the mechanical ventilation system. Global warming and the depletion of oil crisis become the global headlines and therefore interest on natural ventilation has been growing in recent years. The spotlight has been set on solar chimney, which combines features of the solar roof collector, Trombe wall and wind tower to remove heat effectively. The ventilation effect of a chimney is induced either by thermal buoyancy or by wind. To further increase the stack effect of the chimney, a larger pressure difference between the inside of the chimney and the top of it is desired. A chimney cup is installed in order to enlarge the outlet are at the top of the chimney which in turn reduce the pressure. The pressure reduction will produce higher pressure difference and hence a higher driving force of the flow will be created.

1.2 Problem statement

The chimney cup is a critical piece of equipment in creating a larger drafting force in a thermal chimney in order to fulfill the desired cooling effect. The cup geometry such as cup angle, diameter ratio and the height of the cap above the plane of exit could be affecting the chimney performance. There is no such study on the effect of the cup to improve the up drafting process of gases in the chimney to date.

1.3 Objectives

This project is aimed to:

- Develop a fundamental understanding of the effects of chimney cup design parameters on the performance of the thermal up draft chimney.
- Investigate various design parameters on the up drafting process.

1.4 Scope of study

There are many published literature related to both experimental study and mathematical model with regards to the thermal chimney. Relatively little or none has appeared on published literature related to the experiment study on chimney cup effects. This project is concerned with the pressure difference the chimney cup is able to create by alternating the cup's angle and diameter ratio and the height of each design. Hopefully a higher airflow rate could be achieved by using a different set of cup geometry. The author will not delve into varying chimney depth or different wind velocity profile due to time constraint. Moreover, there are a lot of similar studies on the solar chimney have been published.

2.0 LITERATURE REVIEW

The development of mathematical models for Trombe wall and roof collector has inspired the solar chimney ventilation. Yong and Nyuk [2] conducted experiment on a Zero Energy Building using a solar chimney system to enhance the air ventilation within the interior spaces. Results showed that by implementing the solar chimney system, the interior air speed of the experimental region is able to reach a maximum of 0.49m/s and the interior air temperature to heat up slower and cool down faster by 1–2 h as compared to the reference region. The last finding of the experiment also showed that the position of the solar chimney inlet has significant effect on the interior air speed. Air speed achieved 0.60m/s by lowering the inlet to occupancy's height of 1.20m.

Rahimi and Bayat [3] have carried out experiment to investigate the effect of thermal boundary condition of the pipe have on the air flow within a vertical pipe induced by buoyancy effect. Results showed that the pressure losses at the inlet and outlet of the pipe along with that of the main pipe are compensated by the buoyancy pressure. The three main parameters to influence air flow rate are pipe length, surroundings temperature and the mean temperature of the flow inside the pipe.

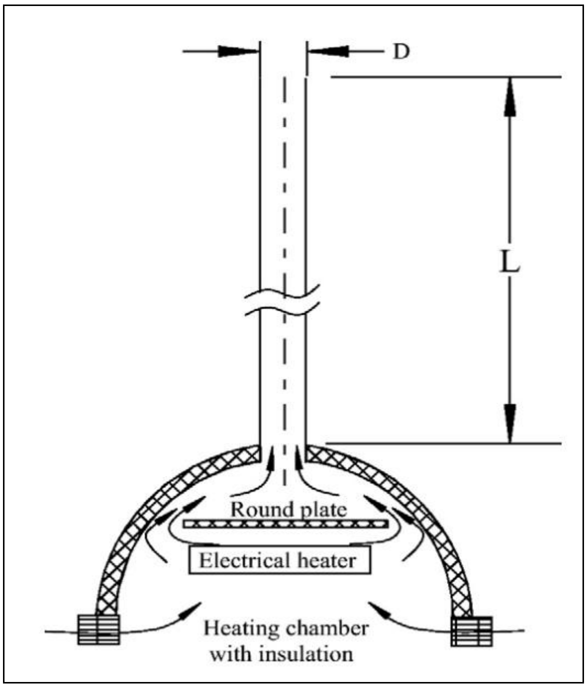


Figure 1: Rahimi and Bayat [3] test set up.

Zamora and Kaiser [4] studied the mixed-buoyancy flow induced by the atmospheric wind blows over the upper part of the chimney. Chimney is assumed to be free from any protective device at its upper part. They concluded that for positive wind velocity, the wind driving forces is dominant ranging from 2-3 m/s. While for a negative wind velocity, the mass flow rate becomes negative through the chimney, which means the air comes into the upper part of the chimney.

Lee and Strand [5] have constructed a solar chimney to study the effect of the chimney height, solar absorptance of the absorber wall, solar transmittance of the glass cover and the air gap width under various conditions. Results showed that chimneys have more potential cooling than heating and significant cooling energy saving can be achieved by implementing the chimneys properly. The climate of the location also affects the performance of a thermal chimney. They conclude that the Chimney height, solar absorptance and solar transmittance turned out to have more influence on the natural ventilation improvement than the air gap width.

Arcea et al. [6] set up a full scale experimental chimney model and implemented in real meteorological conditions. Observations showed that the air flow rate through the solar chimney is influenced by a pressure difference between input and output, caused mainly by thermal gradients and wind velocity.

Mehla et al. [7] built a solar updraft tower consists of an air collector 1.4m in diameter and 80cm tall chimney to investigate the variation of velocity with essential geometric parameter of the system. They found that with increasing and decreasing chimney diameter, the velocity decreases and increases respectively. Besides that, a larger tower radius also causes a lower temperature flow in the cover. Results also show that the material with higher transmittances to solar radiation increases the ground temperatures and result in higher flow temperature.

Farias et al. [8] elucidated the wind effect over the chimney cap by constructing a test facility based on the British Standard BS 715. Their results showed that when gas flow decrease, the values of the draft also decrease. The effect of the wind on the chimney cup is quantified abd obtained by adjustment of the experimental results with a polynomial approach using software. The pressure loss produced by the chimney cap basically depends on the gas flow rate, the wind speed, the chimney area and the gas density.

Sakonidou et al. [9] developed a mathematical model to determine the tilt that maximizes natural air flow inside a solar chimney using daily solar irradiance data on a horizontal plane at a site. The model predicts the temperature and velocity of the air inside the chimney as well as the temperatures of the glazing and the black painted absorber. Chimney tilt varies in a rather narrow range between 65 ° and 76 ° to yield the maximum air flow rate while it varies between 12 ° and 44 ° for maximum irradiation.

Mullet [10] indicates that overall efficiency of the chimney is directly related to the height of the chimney. His analysis shows that 1% of efficiency is increased for a height of 1000m. Afriyie et al. [11] combined an appropriately inclined roof of drying chamber for ventilation improvement in a chimey-dependant solar crop dryer (CDSCD). According to the results of parametric studies performed with the developed FORTRAN simulation code, maximum airflow can be achieved when the inlet-exit area ratio is around 4:1.

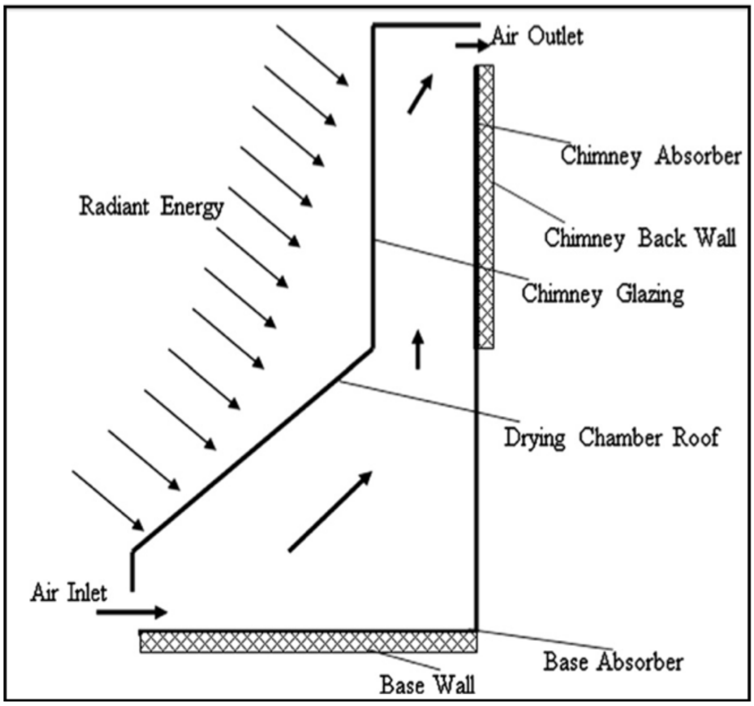


Figure 2: Afriyie et al. [11] - functional architecture of a chimney-dependent direct mode solar crop dryer.

There are many studies on solar chimney where most of them are searching for optimum design solutions to enhance the natural ventilation by varying the design parameters of the chimney. Among the parameters reported, the chimney aspect ratio (stack height/air gap width), ventilation height (height between inlet and outlet apertures), aperture areas, thermal characteristics of the absorber material and chimney tilt angle are found to have the most pre-dominant effect on ventilation performance. Based on the findings by Khanal and Lei [12], open ended vertical channel geometry, vertical chimney attached to the building and inclined chimney have been the major design configurations used for most of the solar chimney studies.

Dimoudi [13] states various advantages and disadvantages of solar chimneys based on a thorough investigation via both CFD simulation and experimental work. He finds that although solar chimneys are suitable for any type of building, they may not sufficiently cover the cooling needs during the whole cooling period and thus alternative techniques and backup systems may be required.

Afonso and Oliviera [14] identified two important parameters to satisfy the required airflow rate for solar chimneys, which are the chimney section and chimney height. The average flow rate is directly proportional with the chimney section. An increase in chimney width is more effective than an increase in height for a given solar collection area. Chen et al. [15] pointed that airflow rates increased continuously with chimney length/gap ratio up to 2.5 via their smoke visualization studies. However, they could not determine the optimum air gap for maximum flow rate.

3.0 RESEARCH METHODOLOGY

The author has divided the whole project into 5 phases, which is shown in the flow chart below:

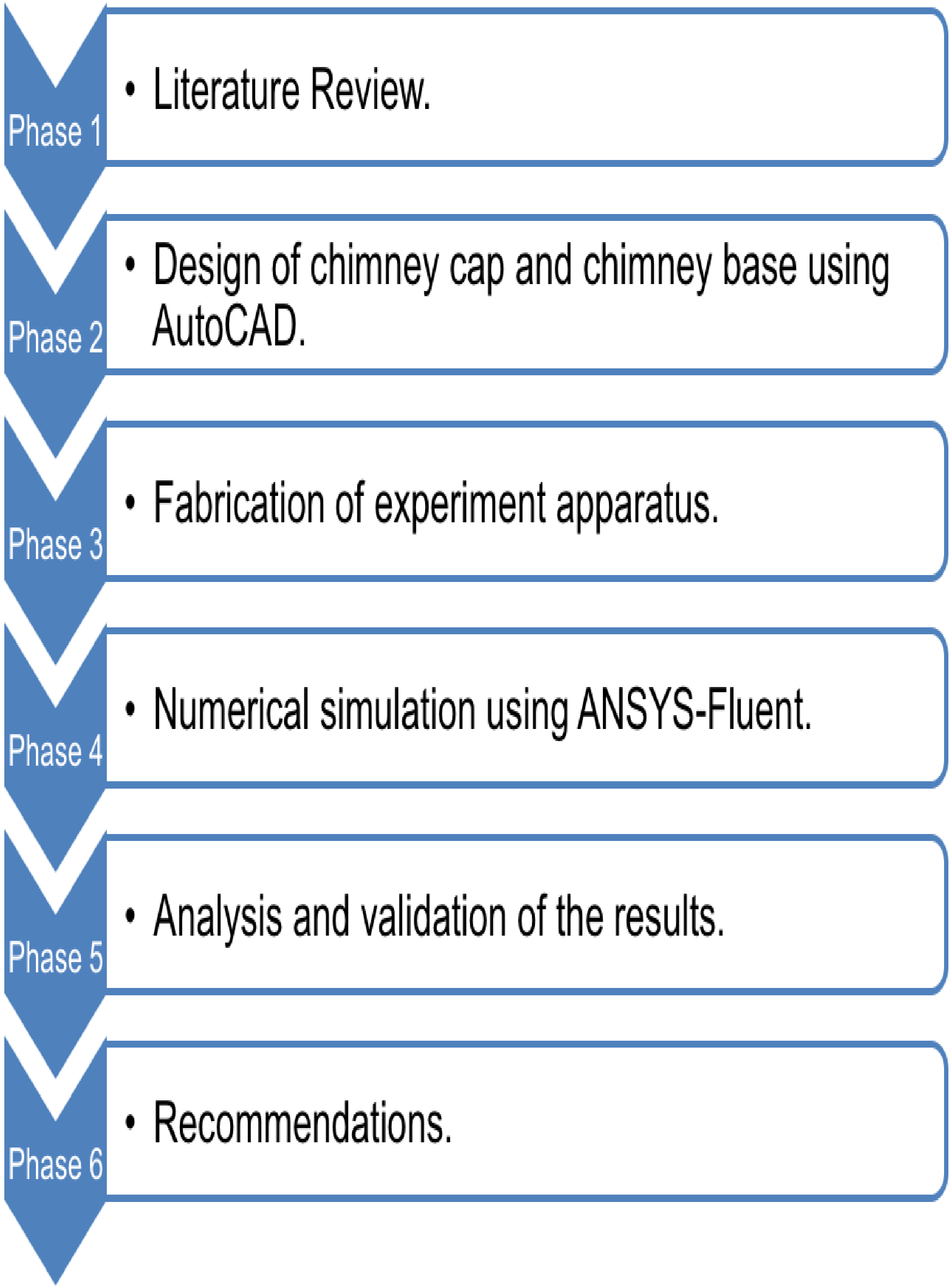


Figure 3: Research Flow Chart.

During the period of FYP 1, researches have been done by gathering all relevant journals or published papers to gain a better understanding of natural ventilation. Literature review has been produced in order to assist the author in establishing a feasible scope to be accomplished in the given timeframe. After further discussion with the supervisor, experimental parameters have been decided. The parameters are the chimney cup geometry and the chimney cup gap length with the chimney outlet. The project will be conducted with experiment and be validated with numerical simulation using ANSYS-Fluent.

The author started off with the drawings of the experiment set up using AutoCAD 2009 software. A total of 4 cup angles have been drawn: 180°, 150°, 120° and 90°. A chimney base, which will act as a cover for the heat source underneath the chimney is drawn out as well.

The completed drawings are handed to fabricator for fabrication. Simulations work have been initialized by the author at the same time to get familiarized with ANSYS-Fluent software.

During FYP 2, the author has scheduled to carry out the experiment and to complete the simulations as well. Data analysis and validation of results will be conducted, followed by documentations.

Project Gantt chart, key milestone and activities flow for both FYP 1 and FYP 2 are developed to boil down various timelines and easily comprehend where the author is in a progression (Refer to Figure 28 & 29).

The followings are some sample of the drawings:

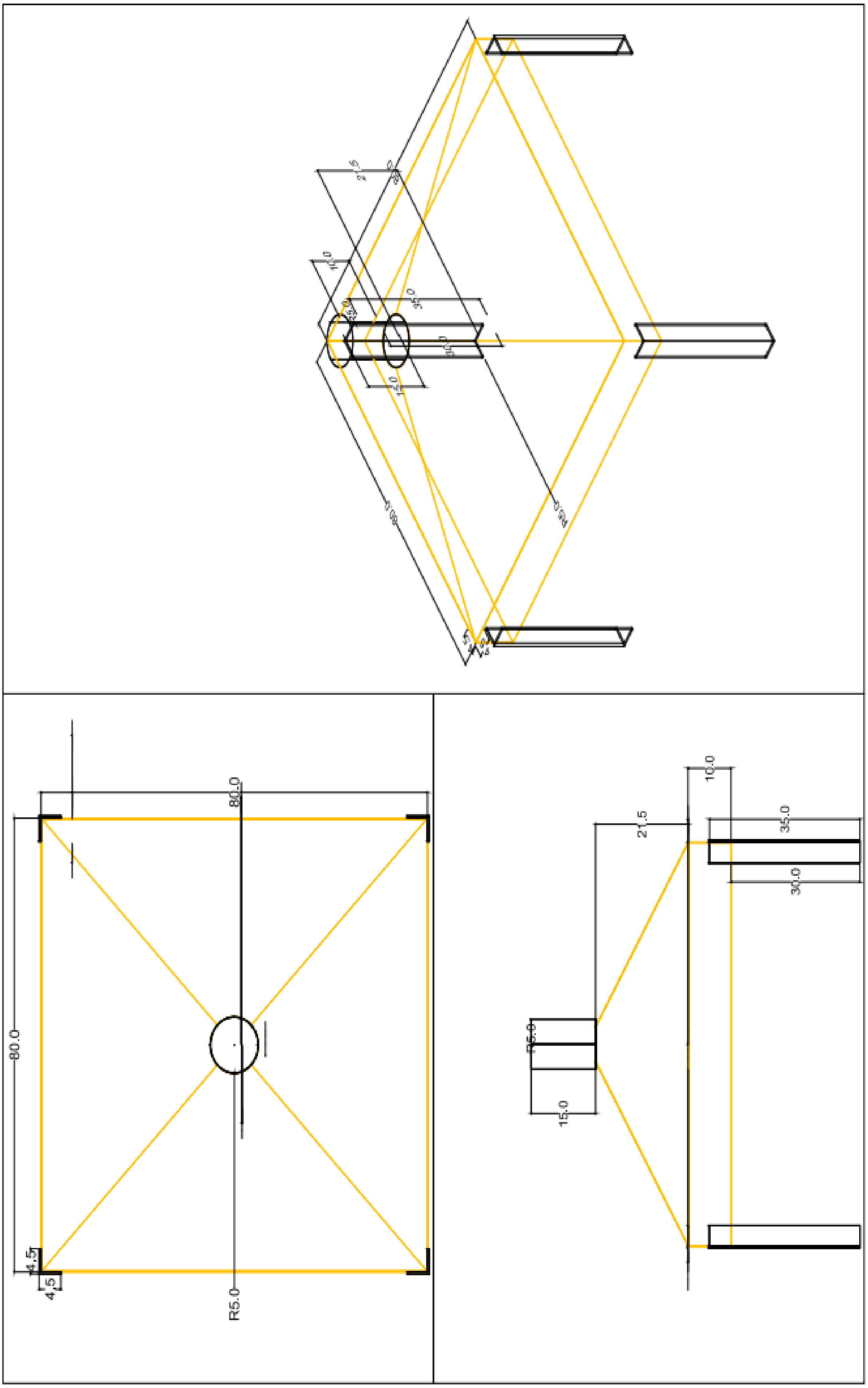


Figure 4: 3-Port View of Chimney base in AutoCAD.

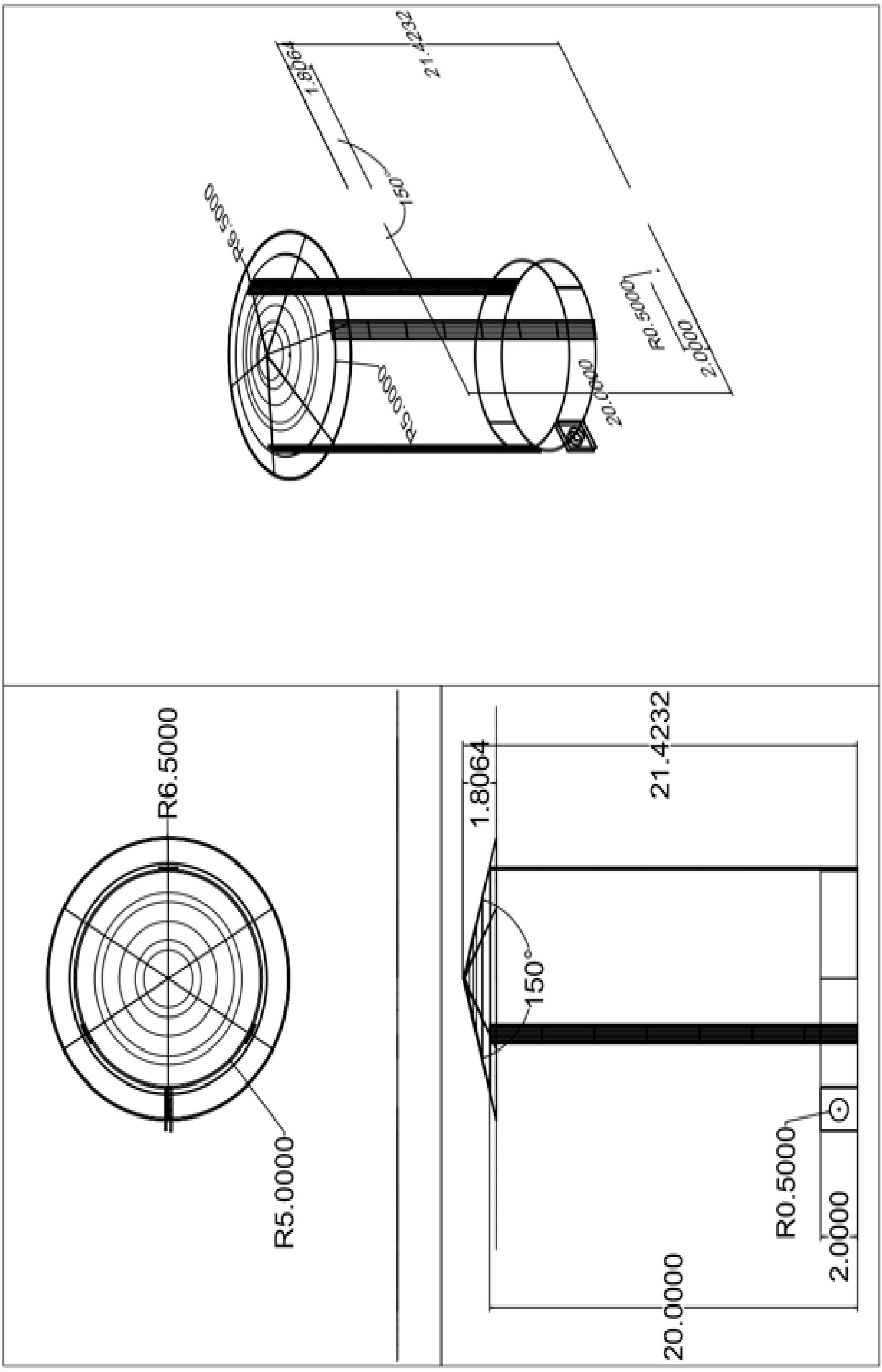


Figure 5: 3-Port View of Chimney cup 150 °in AutoCAD.

3.1 Experiment Equipment Setup and Hardware

The real experiment setup is shown in Figure 6. It consists of a chimney base, 4 different angle chimney caps, a coil-typed heater, a 0.93m 12 inch Outer Diameter PVC pipe, GRAPHTECH GL820 Data Logger (Figure 8), KIMO VT 200 Anemometer (Figure 9) and thermocouples. The PVC pipe is slotted onto the chimney base, and the chimney cup is slotted on top of the PVC pipe. The gap between the cup and the chimney exit varies from 3 cm to 7.5 cm. The heater was placed in the center and beneath the chimney base.

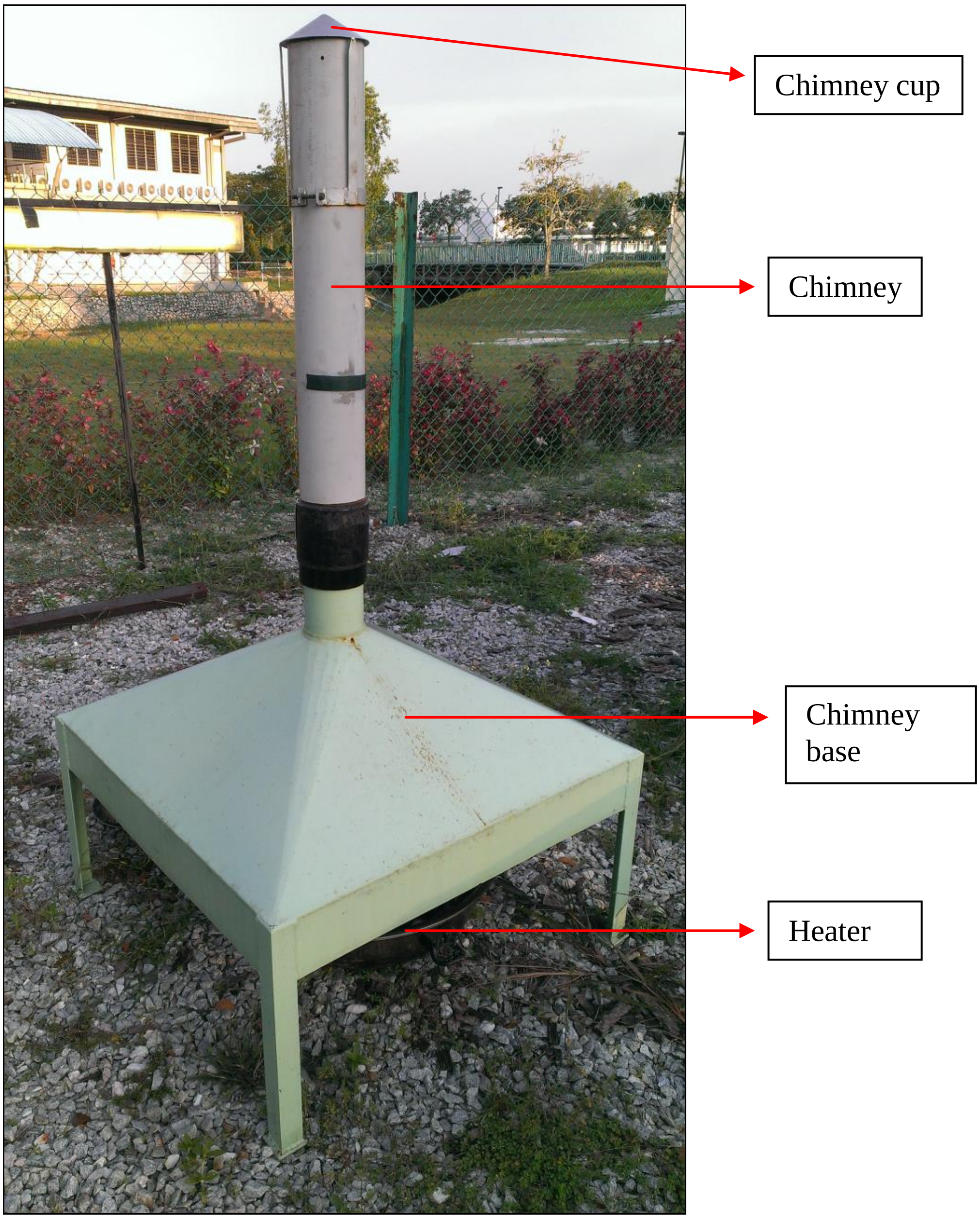
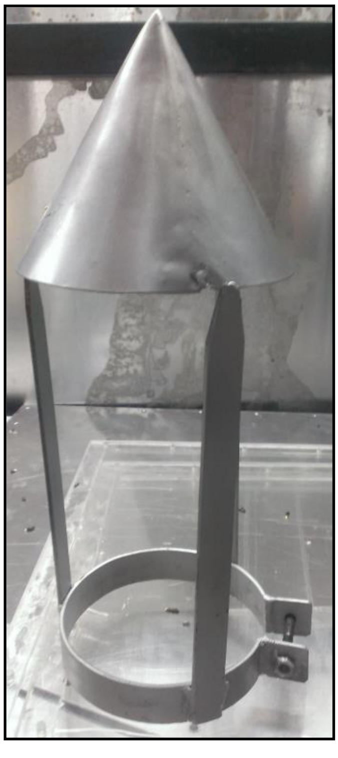


Figure 6: Actual experiment set up.

(a)

(b)

(c)



(d)

Figure 7: Actual chimney cups.
(a) 180°, (b) 150°, (c) 120° and (d) 90°.

3.2 Instrumentation

This experiment investigates the fluid flow parameters at the chimney inlet and outlet. The airflow velocity and the temperature of air in the chimney are the parameters to be investigated. Thermocouples have been attached at the inlet and the out of the chimney, and are connected to the GRAPHTEC GL 820 Data Logger (Figure 8). The GL 820 is a standalone data logger with built-in 20 analog channels. The maximum data-sampling rate is 10 ms, while the sampling rate used in this experiment is 10 s.

Figure 9 shows a KIMO VT 200 Anemometer used to measure the air velocity at the chimney outlet. The accuracy of the VT 200 different depends on the vane probe used. In this experiment, the 70 mm diamater is used and the accuracy achieved while measuring 0.1 m/s to 3 m/s is ± 3% of reading, while for a flow velocity ranged of 3.1 m/s to 35 m/s, the accuracy is ±1% of reading.

Figure 8: GRAPHTEC GL820 Data Logger.

Figure 9: KIMO VT 200 Anemometer.

3.3 Experimental Procedures

1. The heater is turned on and allowed to be heated for 20 minutes.

2. The data logger is turned on and is set to record the Inlet Temperature (Temp In) and Outlet Temperature (Temp Out) every 10 seconds.

3. The 180°chimney cap is placed and locked on the top of chimney.

4. The gap length is set to 3cm.

5. The cap is left to settle for 10 mins.

6. Anemometer is used to record the air velocity at the chimney outlet.

7. The air velocity and temperature are recorded.

8. Step 4 to step 7 are repeated with gap length of 4.5cm, 6cm and 7.5cm.

9. The cup is replaced with 150°cup.

10. Step 4 to step 8 are repeated.

11. The cup is replaced with 120°cup.

12. The gap length is set to 1.5cm.

13. Step 5 to 7 are repeated with gap length of 3cm, 4.5cm, 6cm and 7.5cm.

14. The cup is replaced with 90°cup.

15. Step 5 to 7, and step 13 are repeated.

3.4 Numerical Simulation with ANSYS-Fluent

The dimensions of the experiment set up is recorded and drawn in ANSYS-Fluent Workbench via Geometry. The drawing includes the surrounding air so that airflow direction into the chimney could be observed. In Setup, Energy and K-Epsilon Equations are employed. The ambient air velocity of left inlet is set to be 0.05 m/s while the inlet on the right is set to be -0.05 m/s. The heat source is set to have a constant temperature of 343 K.

The proposed model (Figure 16) is a 2D model adopting a technique called 'far-field', where the simulation includes a 1.2 m surrounding extension from the edge of the chimney model. The purpose of including the surrounding into the simulation is to observe the air movement into the chimney.

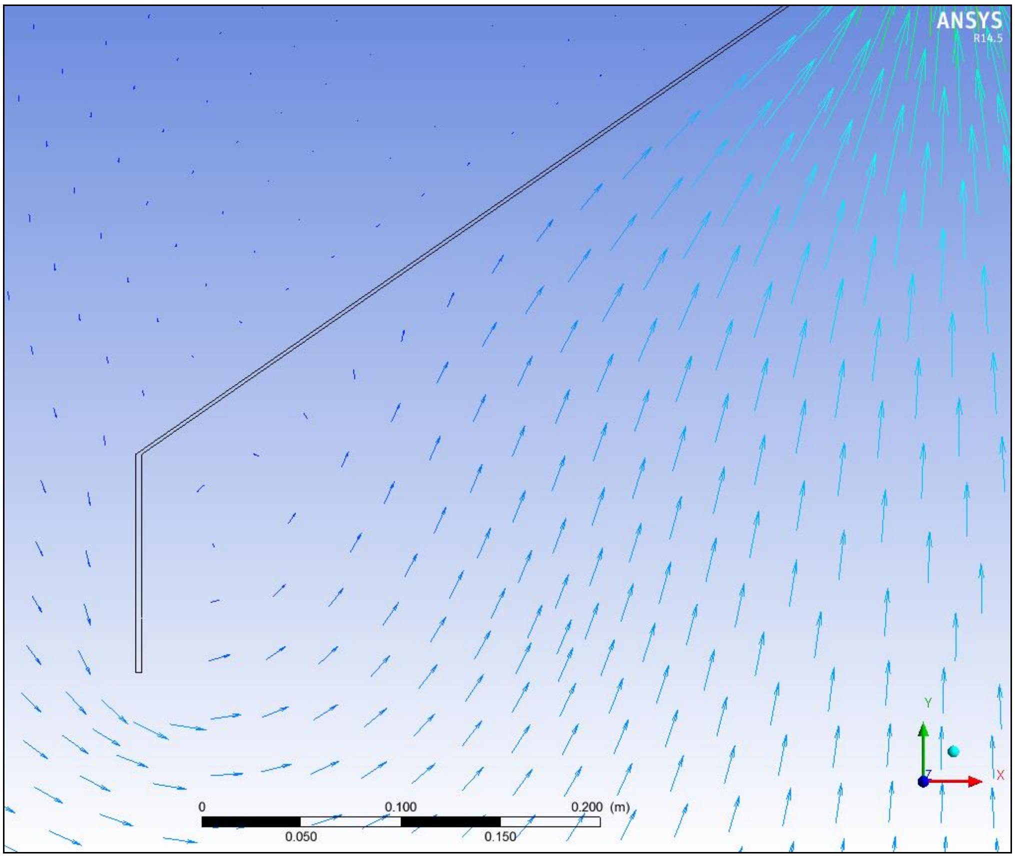


Figure 10: Air movement into the chimney.

Below are the velocity contours of flat plate, 150°, 120°, 90° cup and without cap, showing the air flow movement at the entrance of the chimney.

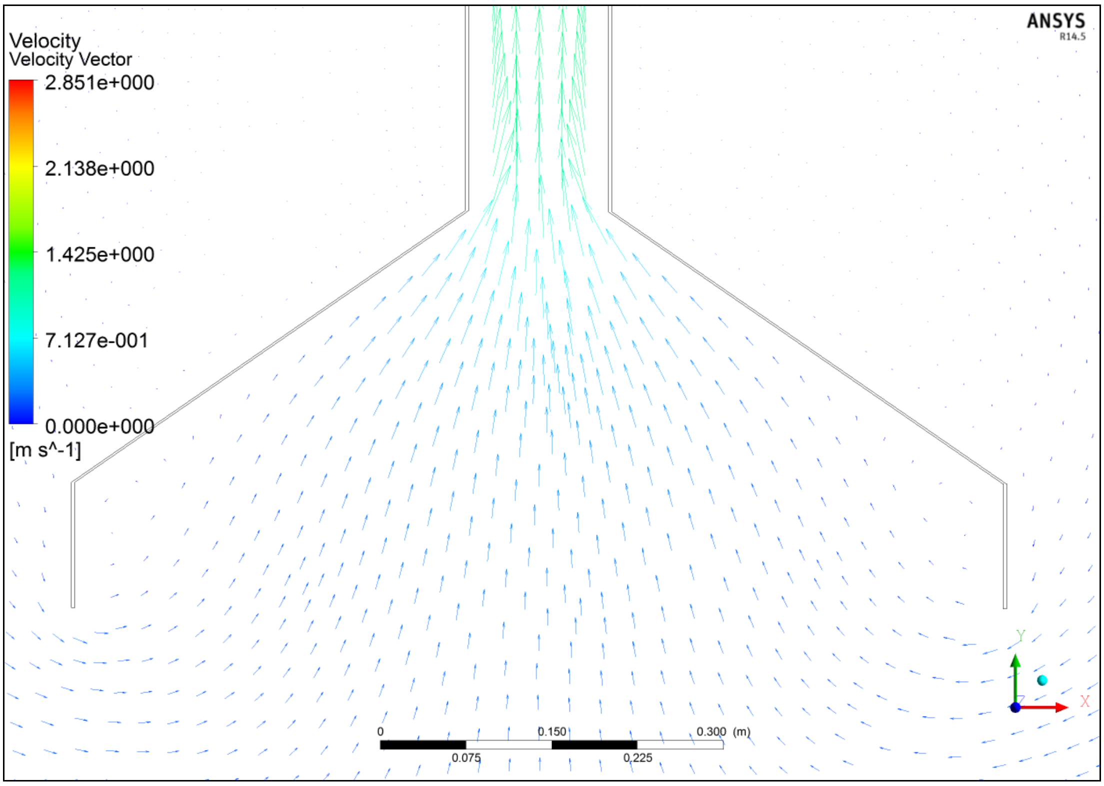


Figure 11: Airflow movement at the chimney entrance. (Flat Plate)

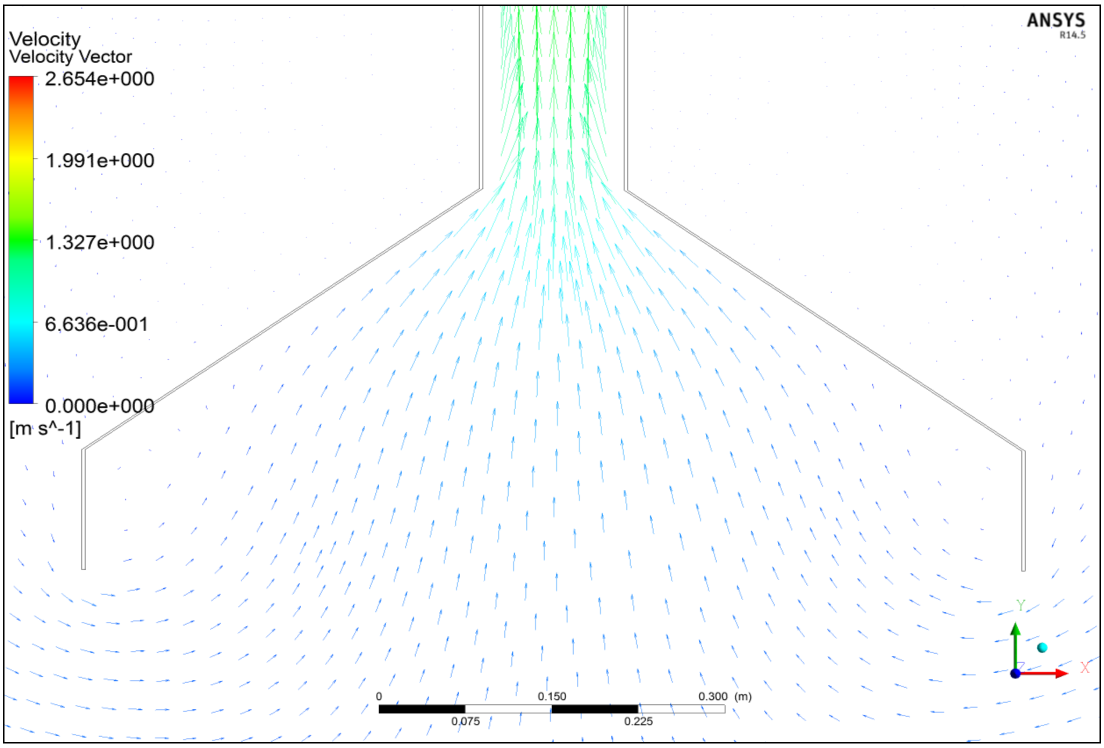


Figure 12: Airflow movement at the chimney entrance. (150°)

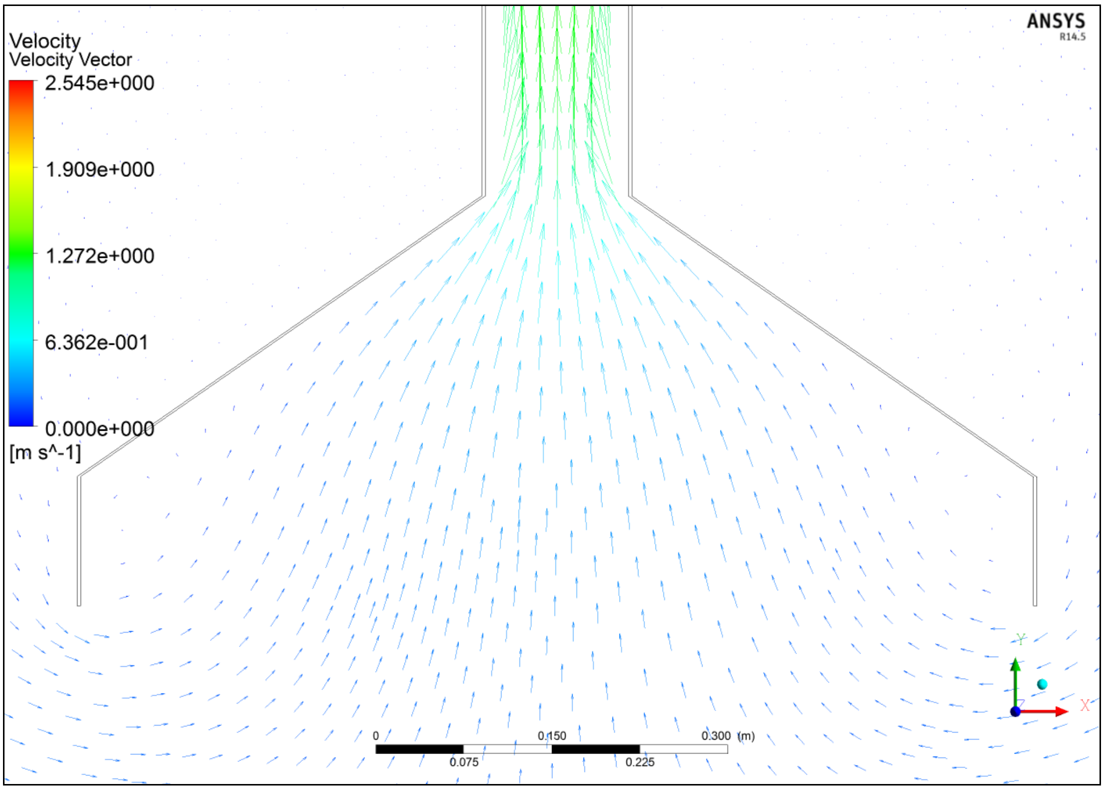


Figure 13: Airflow movement at the chimney entrance. (120 °)

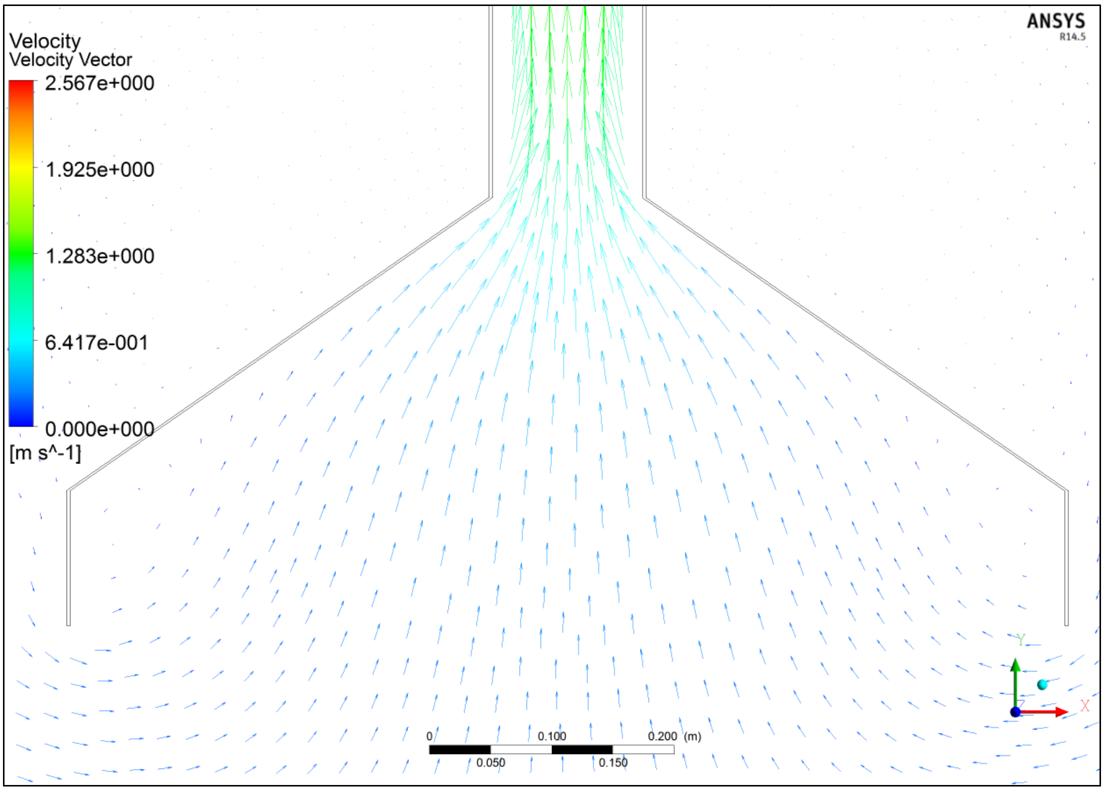


Figure 14: Airflow movement at the chimney entrance. (90 °)

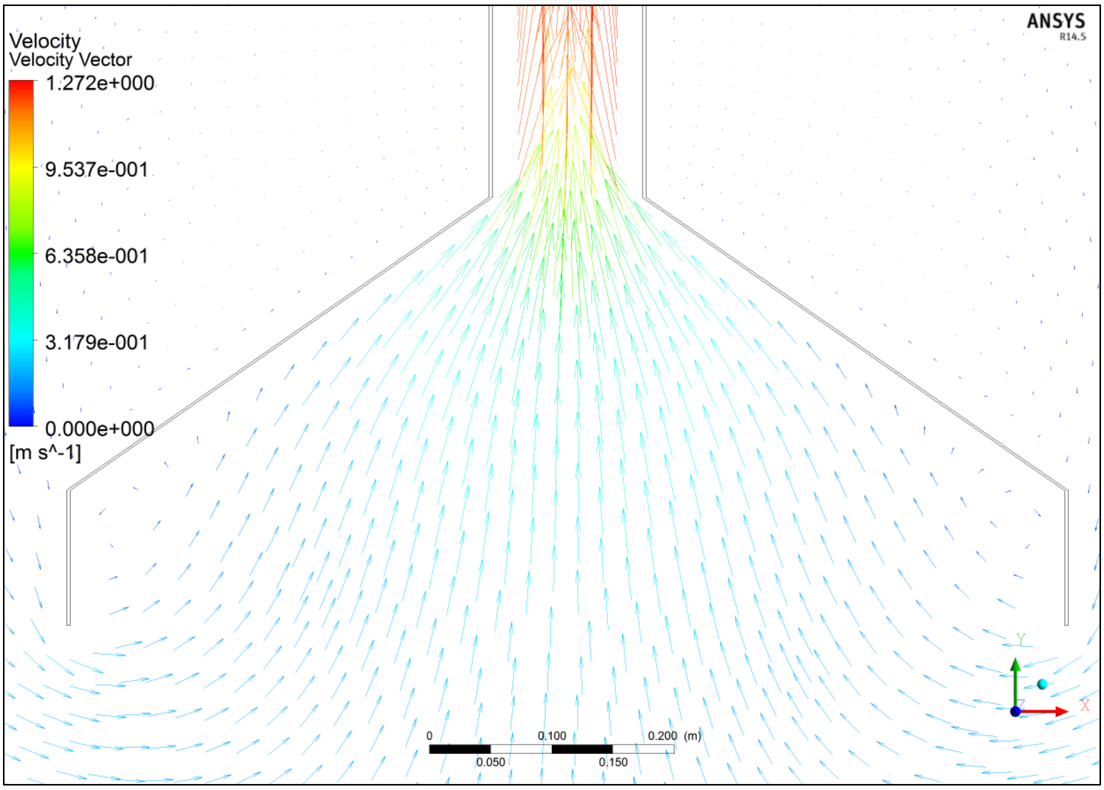


Figure 15: Airflow movement at the chimney entrance. (Without chimney cap)

In the meshing setup, the relevance center and the span angle size are both set to ‘Fine’. The rest of the settings are left as default. There are a total of 7714 nodes and 7409 elements after meshing.

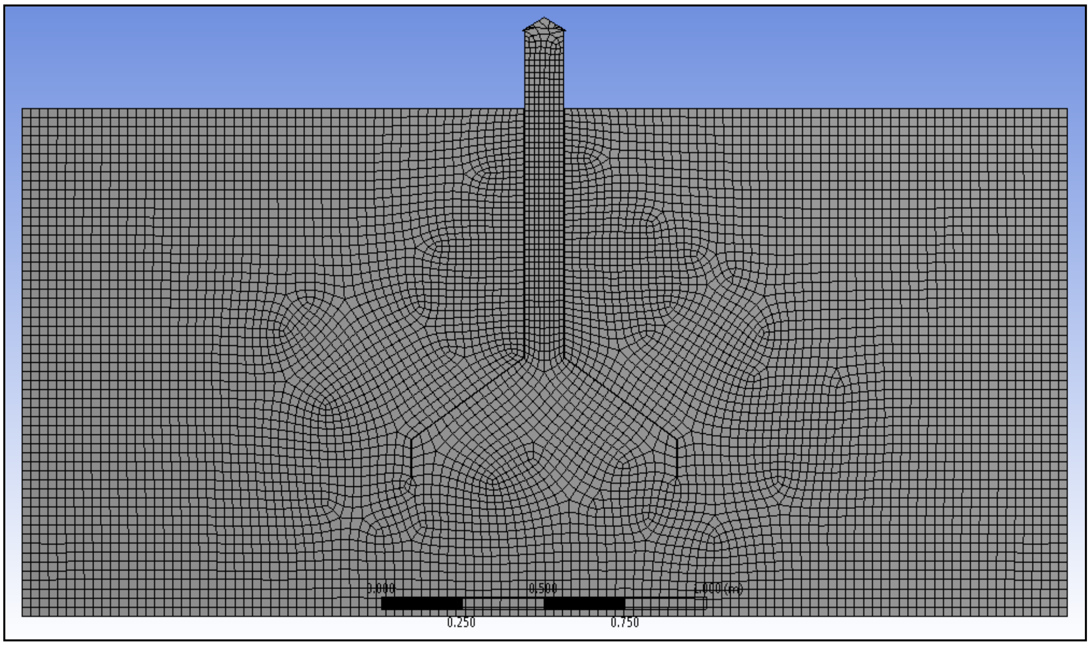


Figure 16: Proposed model with far-field mode in ANSYS-Fluent.

4.0 RESULTS & DISCUSSIONS

4.1 Induced Experiment Flow Velocity and Simulated Flow Velocity

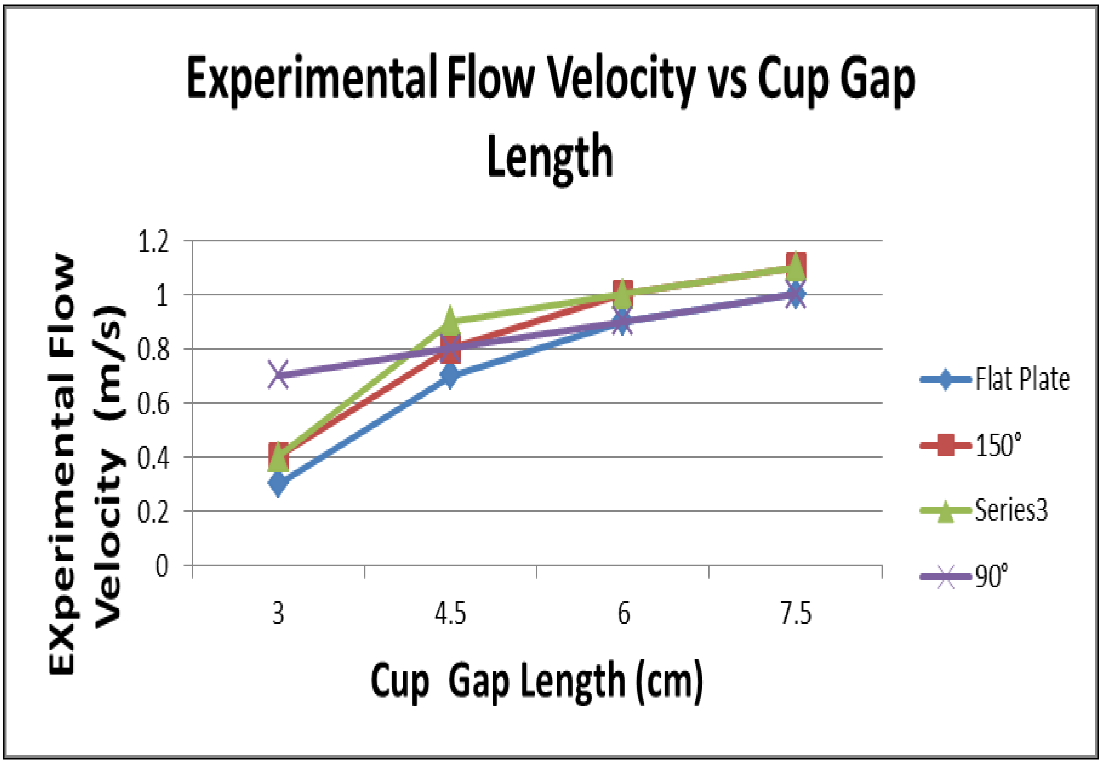


Figure 17: Experiment Flow Velocity vs Cup Gap Length.

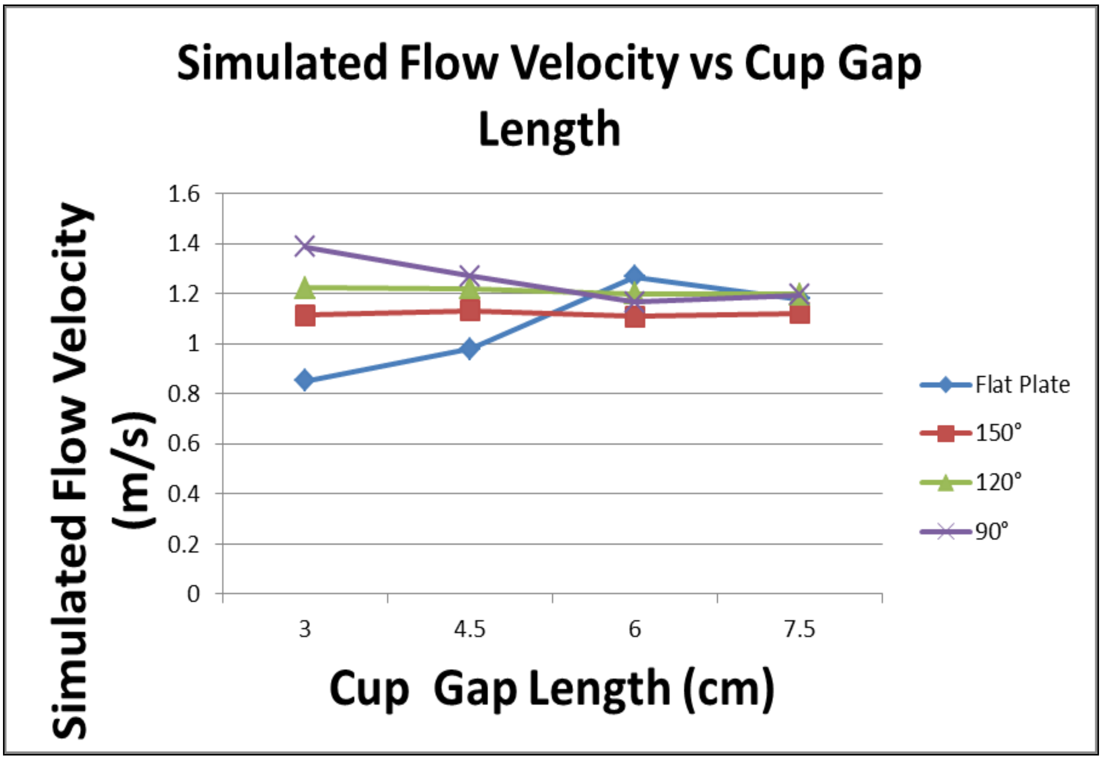


Figure 18: Simulated Flow Velocity vs Cup Gap Length.

Figure 17 shows that generally the flow velocity increases with the increased of gap length, regardless of the geometry of the chimney cup. This phenomenon might be due to the increased gap provides a larger vacant space for the air to flow out of the chimney. It is worth noting that among the 4 angle cups, 150 ° and 120 ° cup have outperformed the other two cups in inducing a higher flow velocity, peaking at 1.15 m/s with a cup gap length of 7.5cm. Even with the same gap length of 7.5cm, the 90 ° cup only produced air velocity of 1 m/s. This can be explained by pointing out the fact that the air flowing towards the inside of the chimney cup is experiencing a resistance flow created by the flow reflected off the top of the chimney. The area of the inside of the 90 ° chimney cup is larger comparing to the other cup, thus the pressure at the top of the chimney is smaller. According to the observations of Arcea et al. [6], the air flow rate through a solar chimney is influenced by a pressure difference between the input and output. Based on the observations, 90 ° is supposed to be producing a higher velocity among the four cups. This results shows that the geometry of the cup is not independent in influencing the chimney air outflow velocity.

Figure 18 shows that the simulated flow velocity follows the general trend of the experimental flow velocity, where the velocity increases with the increase of cup gap length. Air velocity started to stay flat after 3 cm gap length except for flat plate. 90 ° cup has been producing the highest velocity of all, producing 1.4 m/s at 3 cm gap length, but the velocity decreased when the gap increased.

4.2 Percentage of Error for Experimental Air Velocity and Simulated Air Velocity.

Based on Table 1, the percentage of error for cup gap length of 3 cm and 4.5 cm are large, ranging from 35.67% ~ 205%, for all four cups geometry. This might be caused by the error in the process of data recording. The KIMO VT 200 Anemometer used to measure the air velocity in the experiment is large compared to the size of the chimney. During the measurement, the anemometer occupied most spaces between the chimney cup and the outlet when the gap length is small, leaving a smaller space for the air to flow out of the chimney freely. Hence, a slower air velocity is recorded.

	Gap Length (cm)	Experimental Outlet Air Velocity (m/s)	Simulation Outlet Air Velocity (m/s)	% of Error
Flat Plate	3	0.3	0.852	-184
	4.5	0.7	0.9793	-39.9
	6	0.9	1.267	-40.78
	7.5	1	1.18	-18

(a)

	Gap Length (cm)	Experimental Outlet Air Velocity (m/s)	Simulation Outlet Air Velocity (m/s)	% of Error
150 °	3	0.4	1.115	-178.75
	4.5	0.8	1.132	-41.5
	6	1	1.109	-10.9
	7.5	1.1	1.122	-2

(b)

	Gap Length (cm)	Experimental Outlet Air Velocity (m/s)	Simulation Outlet Air Velocity (m/s)	% of Error
120 °	3	0.4	1.222	-205.5
	4.5	0.9	1.221	-35.67
	6	1	1.199	-19.9
	7.5	1.1	1.196	-8.73

(c)

90 °	Gap Length (cm)	Experimental Outlet Air Velocity (m/s)	Simulation Outlet Air Velocity (m/s)	% of Error
	3	0.7	1.386	-98
	4.5	0.8	1.271	-58.88
	6	0.9	1.165	-29.44
	7.5	1	1.194	-19.4

(d)

Table 1: Percentage of Error Between Experimental vs Simulated Air Velocity for a) Flat Plate,

4.3 Simulated Pressure Difference

Based on the observation on Figure 20, 90° cup have been producing the highest pressure difference, followed by the flat plate, 120° cup and 150° cup. The 90° cup which having a higher area is able to reduce the pressure at the top of the cup, hence increasing the pressure difference between the outlet and inlet. It is found that when the gap length increased, the pressure difference decreased. The effect of the chimney cup in reducing the outlet pressure is decreasing when the distance of the outlet and the cup is widened.

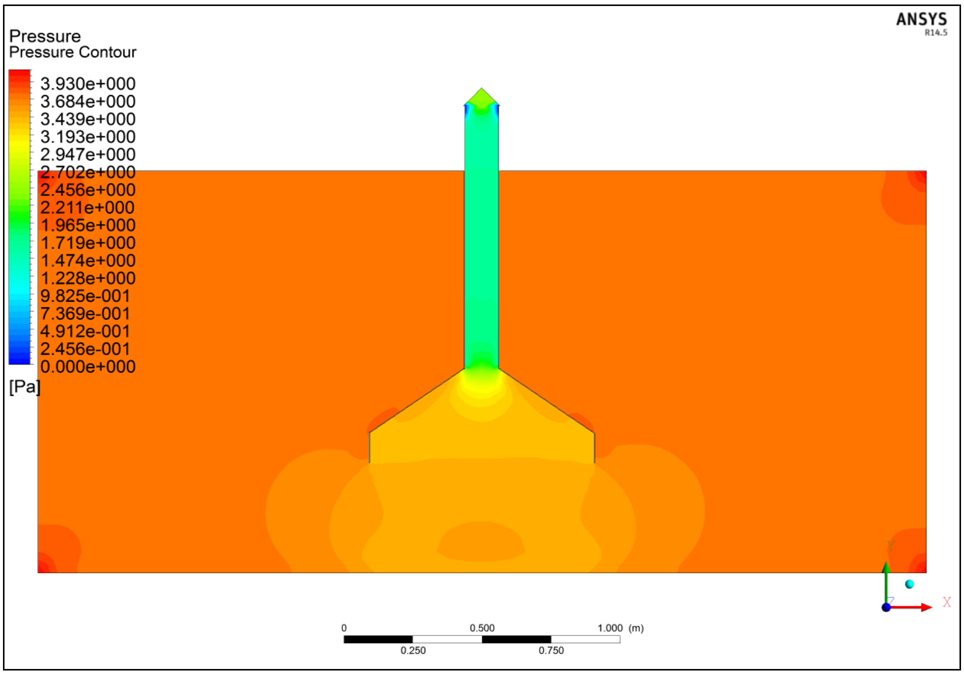


Figure 19: Pressure Contour of 90 cup with 4.5 cm gap length.

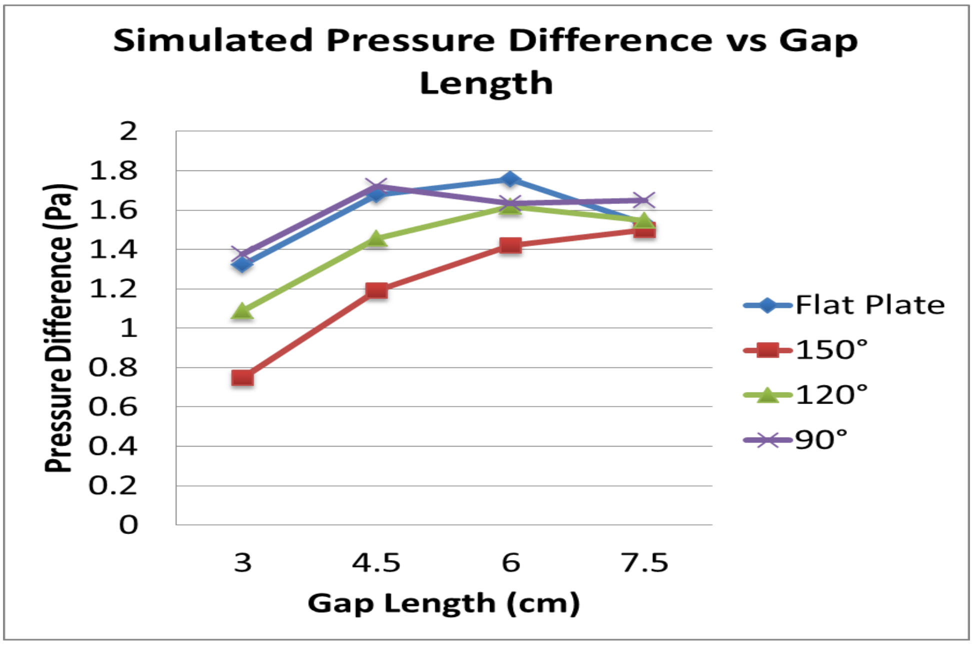


Figure 20: Simulated Pressure Difference vs Cup Gap Length.

4.4 Air Velocity Along the Chimney Outlet Curve Length

In ANSYS-Fluent, velocity profile along the chimney outlet of the airflow is plotted at the position where the experiment air velocity is recorded.

Air velocity along the wall is zero as observed from the Figure 16. This is due to the existence of ‘no-slip’ condition at the wall. Friction at wall resists any flow motions and the layer of fluid is moving in respect to the adjacent layer. When the air flow is in contact with the chimney wall, there is no relative motion and thus the flow is having zero velocity. The air velocity is increasing towards the middle of the chimney.

As shown in Figure 23, without chimney cap, the flow would peaked at the middle of the chimney. This is because all fluids possess viscosity; there is friction between layers of moving air. The part of air in the middle of the chimney, which is away from the chimney wall is away from the boundary layer formed at the wall, thus produced the highest velocity. In the case which the chimney cup is installed, the velocity towards the middle is decreasing as shown in Figure 8. This is because the air flow is facing a resistance flow which is created by the air flow reflection off the chimney cup. The resistance neutralized some of the outflow air velocity and creating a lower velocity towards the middle.

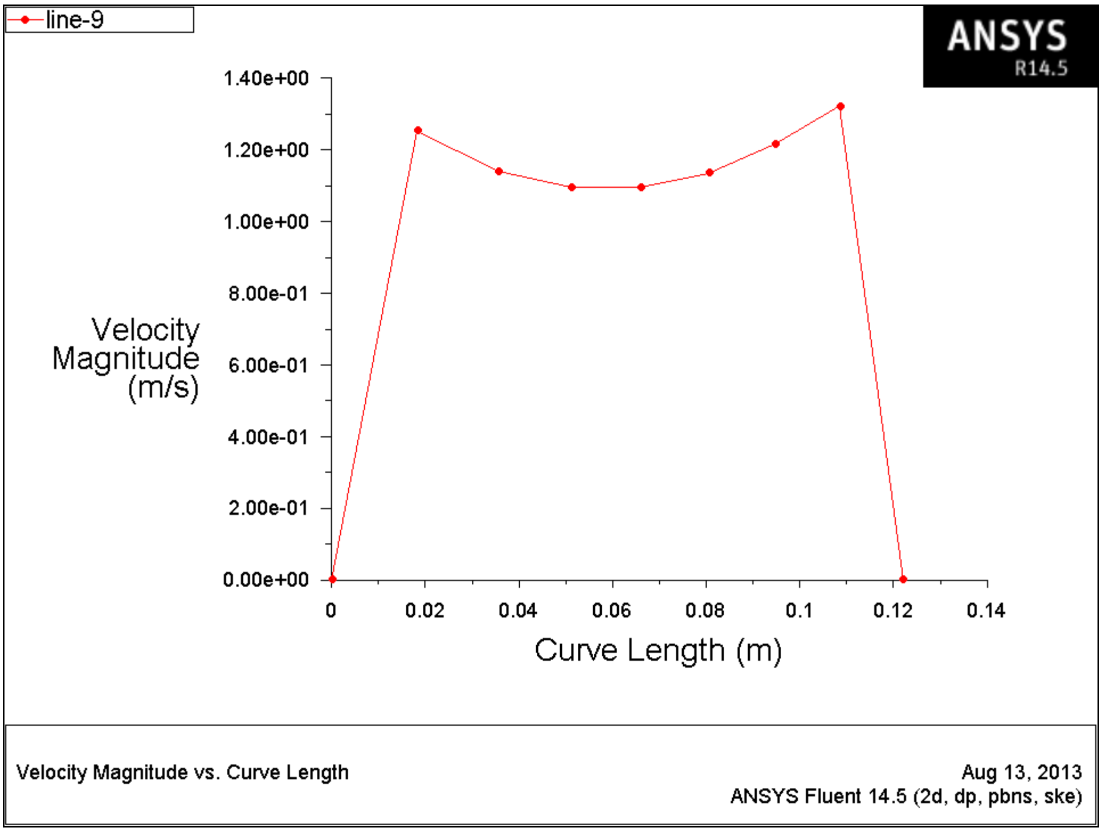


Figure 21 : Velocity profile plotted at the chimney outlet. (120° cup with 6 cm gap length.)

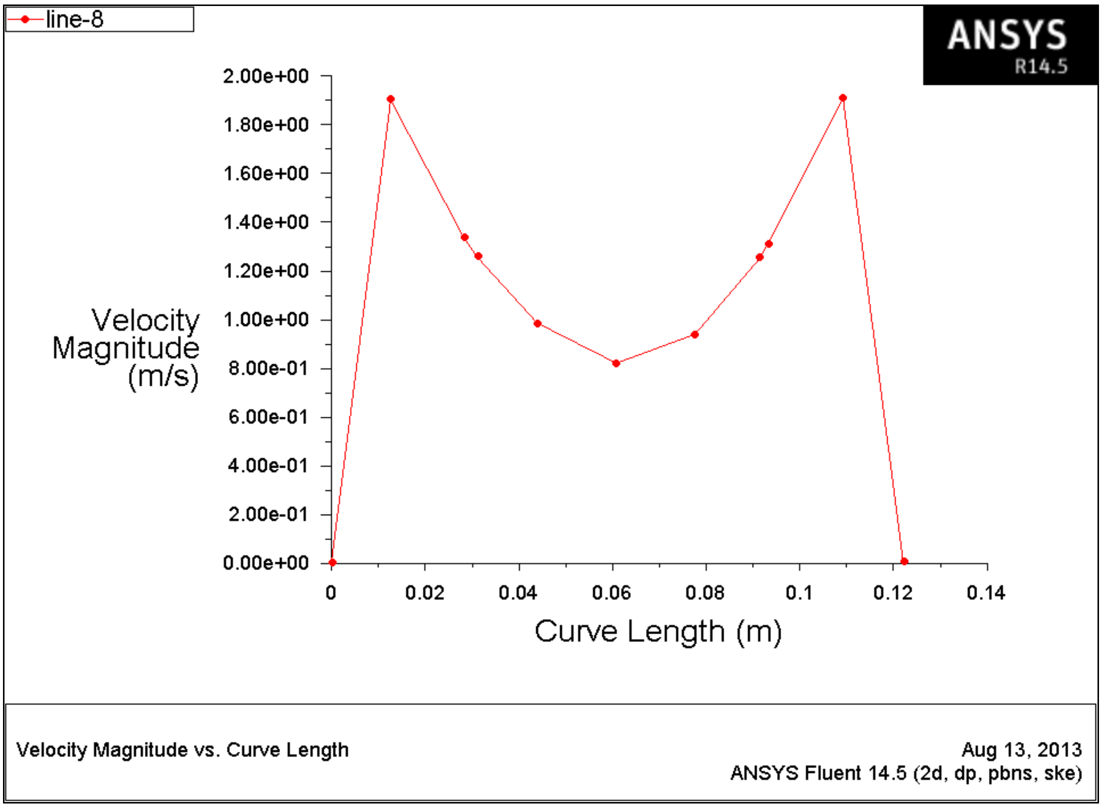


Figure 22: Velocity profile plotted at the chimney outlet. (90° cup with 1.5 cm gap length.)

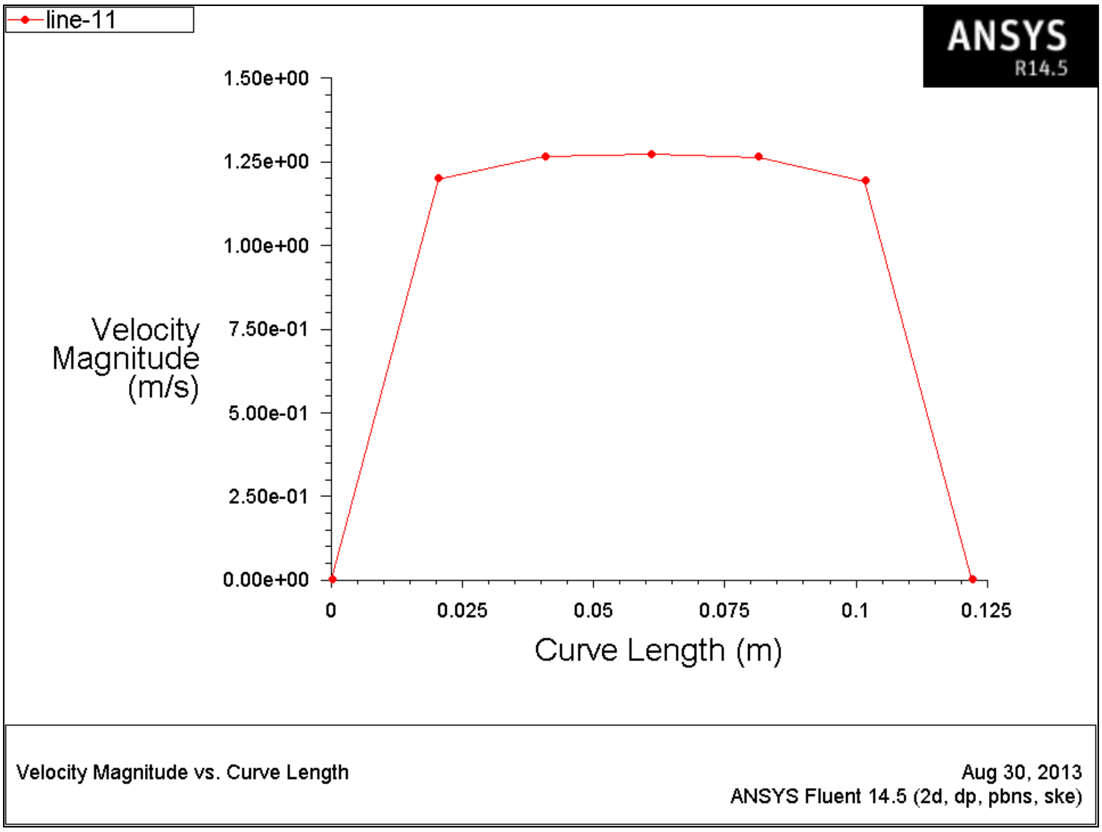


Figure 23: Velocity profile plotted at the chimney outlet. (Without Cap)

5.0 CONCLUSION & RECOMMENDATIONS

The effect of the chimney cup design parameters, mainly on the cup geometry and the gap length of the chimney cup are studied thoroughly in this project. 90° cup with 3 cm gap has produced the highest air velocity and the highest pressure difference, among the 4 cups. This finding concludes that the cup geometry do affect the performance of a thermal updraft chimney. The other findings show that the even cup gap length is crucial in inducing a faster air flow through the chimney, they should not exceed 3 cm, as the distance will hinder the cup's ability to reduce pressure. Based on the findings, there are still potential to be exploited with the chimney cup geometry, while the gap length should be limited to a certain degree. Hence, after combing both of the experiments and simulations results, the 90° cup with 3 cm gap should be recommended as the ideal chimney cup geometry to achieve best performance of a thermal updraft chimney.

Despite the conclusive statement, there are errors which might affect the accuracy of the results. The percentage of error between the experimental and simulated airflow is unacceptable. The anemometer should be replaced by a smaller device, or the experiment set up should be replaced with a larger scale set up in order to obtain a higher accuracy results. During data collecting, the surrounding wind velocity could have been affecting the temperature of the chimney, as well as the airflow velocity.

Besides, assumptions made in the simulation, such as the surroundings wind profile and the temperature of the chimney could also compromise the accuracy of the outcome.

The author would like to recommend that a larger scale experiments to be conducted, so that more gap lengths could be covered in the research. More cup angles or other parameters could be a focus for future work as well.

6.0 REFERENCES

[1] S.J. Emmerich, W.S. Dols, J.W. Axley, “Natural Ventilation Review and Plan for Design and Analysis Tools Report”, National Institute of Standards and Technology, Technology Administration, U.S. Department of Commerce, 2001.

[2] A. K. T. Yong., H. W. Nyuk, “Natural ventilation performance of classroom with solar chimney system”, Department of Building, School of Design and Environment, National University of Singapore, 4 Architecture Drive, Singapore 117566, Singapore, 2012.

[3] M. Rahimi, M. M. Bayat, “An experimental study of naturally driven heated air flow in a vertical pipe”, 2010.

[4] B. Zamora, A.S. Kaiser, “Numerical study on mixed buoyancy-wind driving induced flow in a solar chimney for building ventilation”, 2010

[5] K. H. Lee, R. K. Strand, “Enhancement of natural ventilation in buildings using a thermal chimney”, 2008.

[6] J. Arcea, M.J. Jimenez, J.D. Guzman, M.R. Heras, G. Alvarezc, J. Xaman, “Experimental study for natural ventilation on a solar chimney”, 2009.

[7] N. Mehla, R. Makade, N. S. Thakur, “Experimental analysis of a velocity field using variable chimney diameter for solar updraft tower”, 2011.

[8] O. Farias, F. Jara, R. Betancourt, “Theoretical and experimental study of the natural draft in chimneys of buildings for domestic gas appliances”, Mechanical Engineering Department, University of Concepcio´n, Casilla 160-C, Concepcio´n, Chile, Mechanical Engineering Department, University of La Fontera, Temuco, Chile, 2007.

[9] E.P. Sakonidou, T.D. Karapantsios, A.I. Balouktsis, D. Chassapis, “Modeling of the optimum tilt of a solar chimney for maximum air flow”, Department of

Mechanical Engineering, Technological Educational Institution of Serres, End of Magnesia's Str, GR-62100 Serres, Greece, 2007.

[10] L. B. Mullett, "The solar chimney—overall efficiency, design and performance", International Journal of Ambient Energy, Department of Engineering, University of Reading, Whiteknights, Reading, UK, 2011, 8:1, 35-40.

[11] J.K. Afriyie, H. Rajakaruna, M.A.A. Nazha, F.K. Forson, "Simulation and optimisation of the ventilation in a chimney-dependent solar crop dryer", Department of Engineering, De Montfort University, Queens Building, The Gateway, Leicester LE1 9BH, UK, 2011.

[12] R. Khanal, C. W. Lei, "Solar chimney—A passive strategy for natural ventilation", School of Civil Engineering, The University of Sydney, Sydney, NSW 2006, Australia, 2010.

[13] A. Dimoudi, "Solar Chimneys in Buildings—The State of the Art", Advances in Building Energy Research, 2009, 3:1, 21-44.

[14] C. Afonso, and A. Oliveira, 'Solar chimneys: Simulation and experiment', Energy and Buildings, 2000, vol 32, no 1, pp71–79.

[15] Z. D. Chen, P. Bandopadhayay, J. Halldorsson, C. Byrjalsen, P. Heiselberg, and Y. Li, 'An experimental investigation of a solar chimney model with uniform wall heat flux', Building and Environment, 2000, vol 38, no 7, pp89.

7.0 APPENDIX

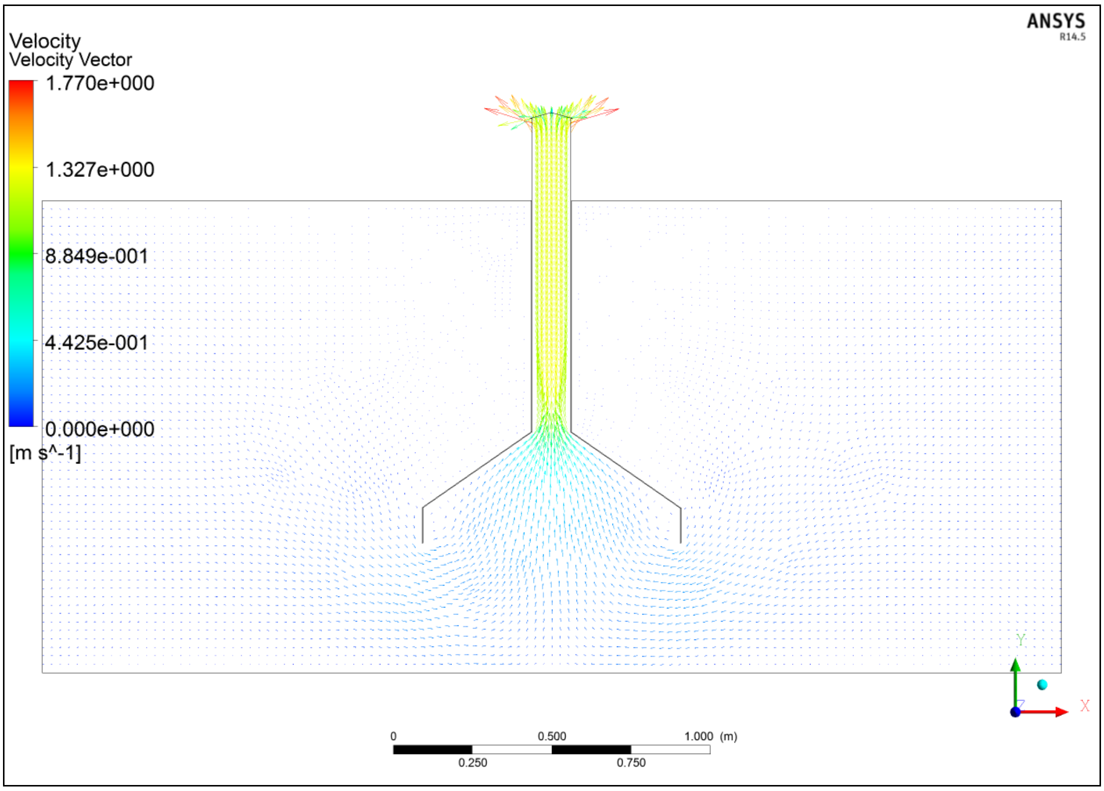


Figure 24: Velocity Vector for 150° cup, 6 cm gap length.

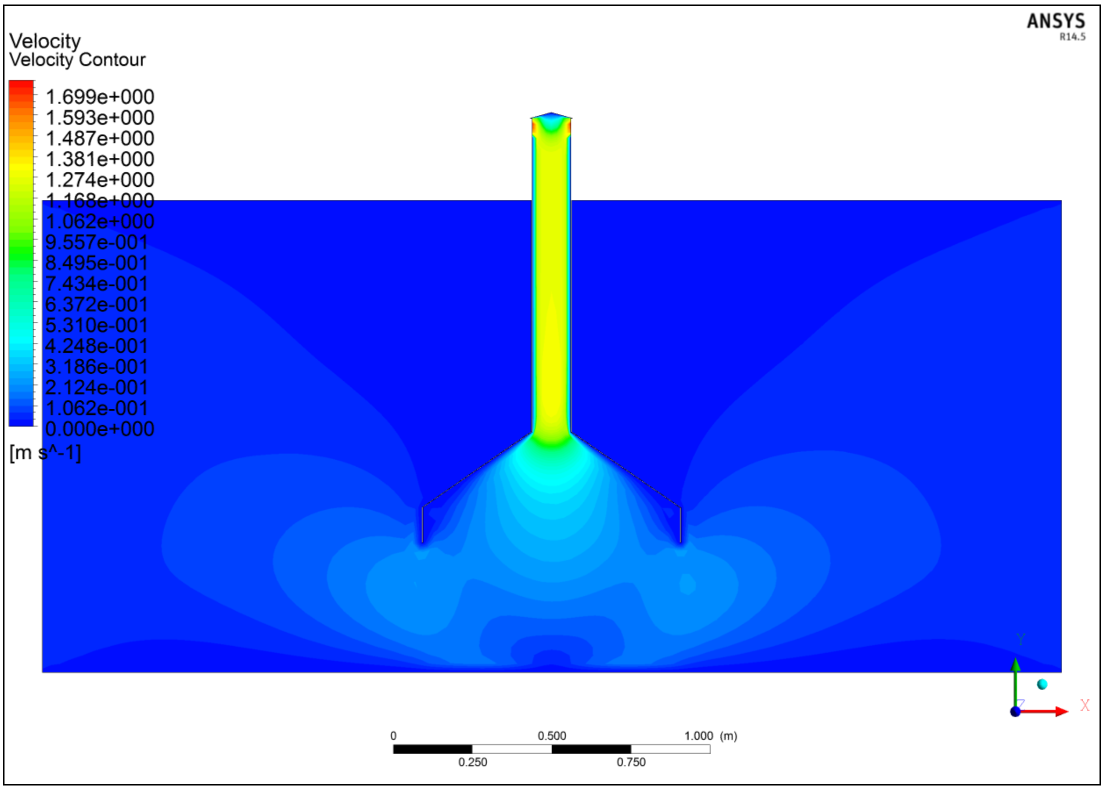


Figure 25 : Velocity Contour for 150° cup, 6 cm gap length.

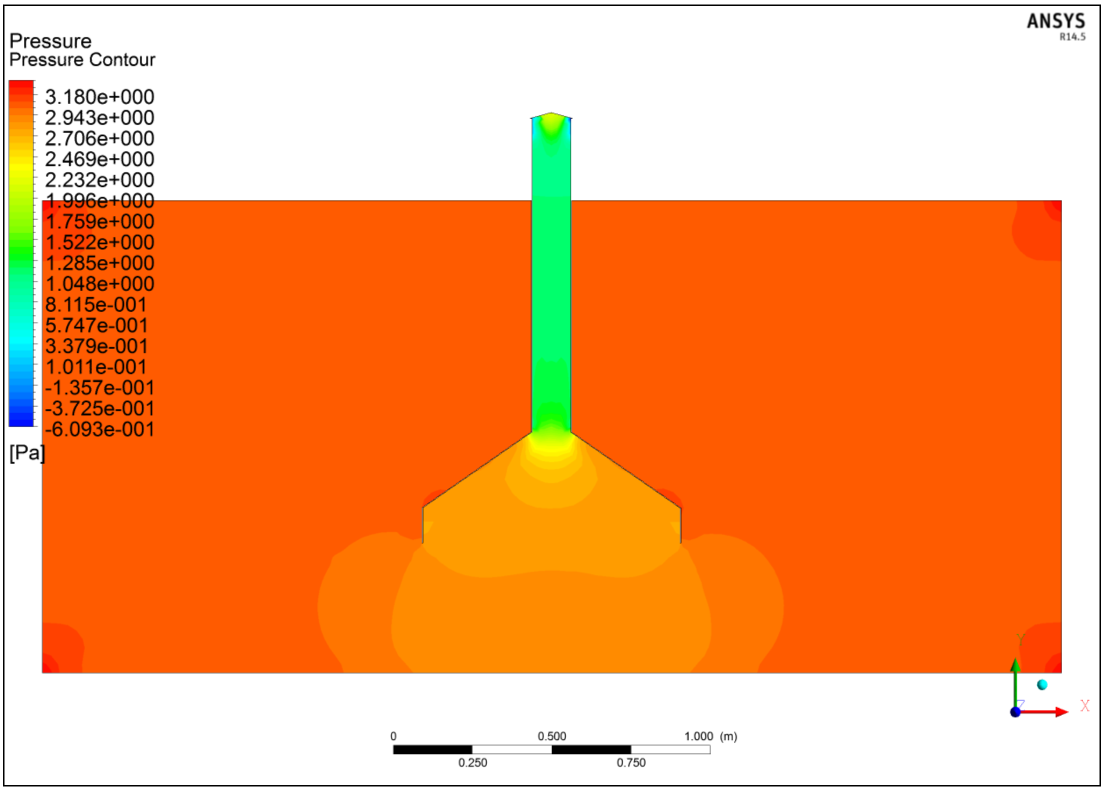


Figure 26: Pressure Contour for 150° cup, 6 cm gap length.

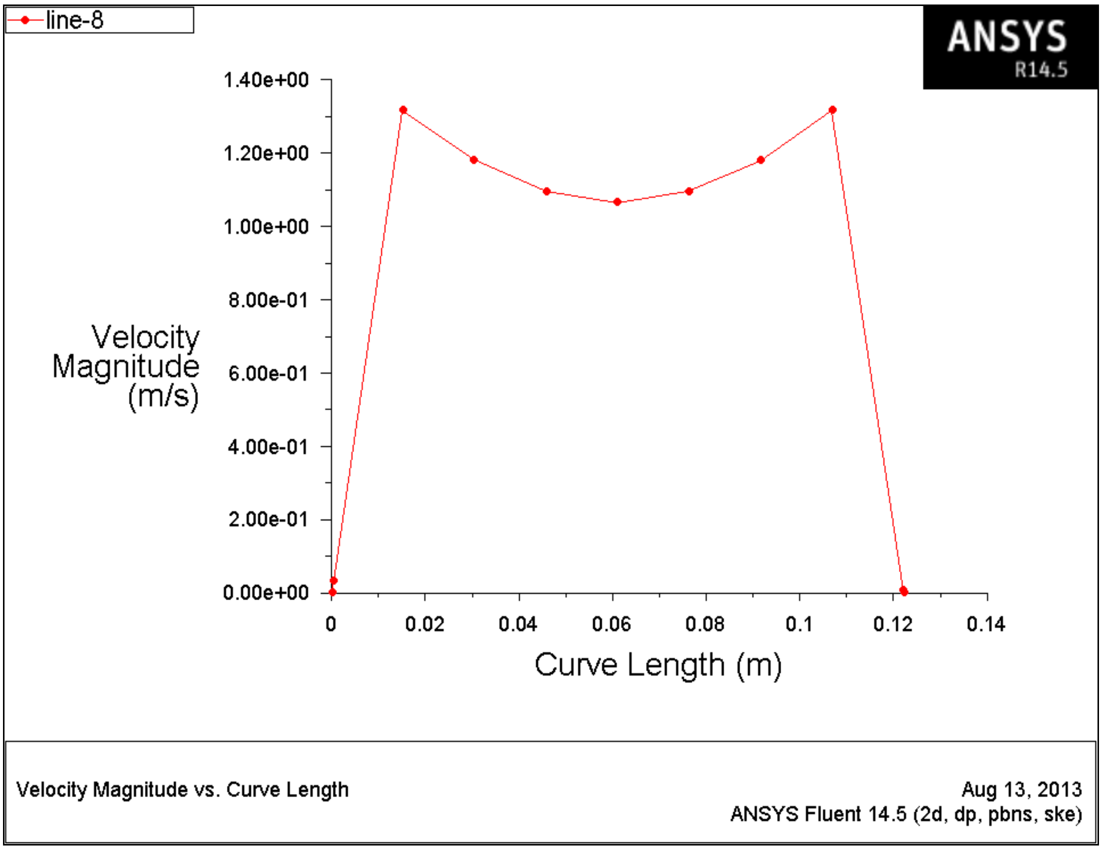


Figure 27: Velocity profile for 150° cup, 6 cm gap length.

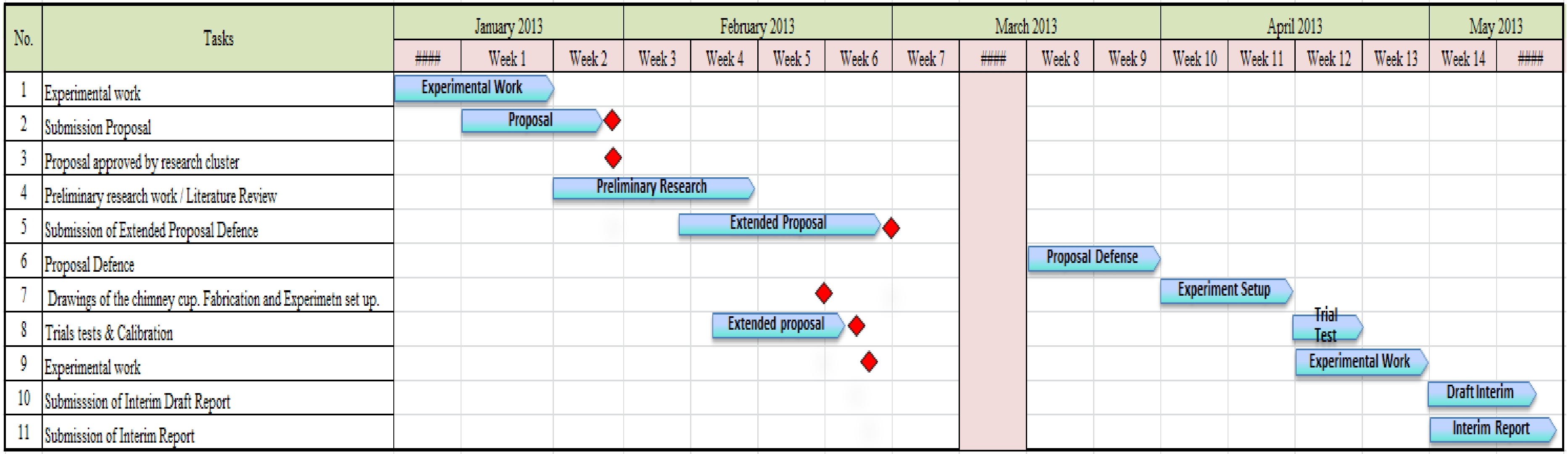


Figure 28: Gannt Chart for FYP I

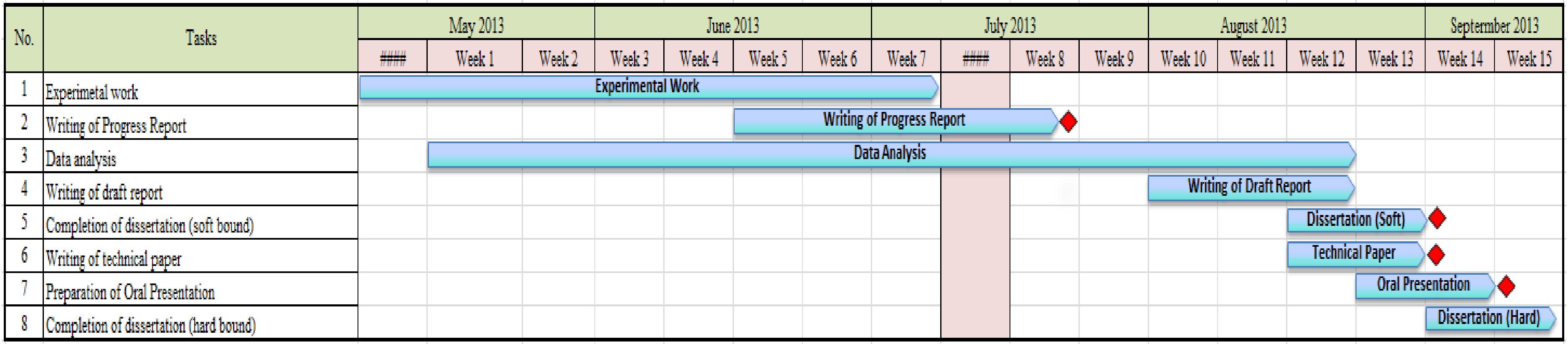


Figure 29 : Gannt Chart for FYP II