# SIMULATION STUDY OF FLOW PROFILE OF AIR CHAMBER AT VARIOUS HEAT SUPPLIED

By:

**MUHAMMAD RABBANI B ABD RANI**

(Matrix No.: 115753)

SUPERVISOR:

**Dr. Mohd Azmi Ismail**

JUNE 2017

This dissertation is submitted to

Universiti Sains Malaysia

As partial fulfilment of the requirement to graduate with honors degrees in **BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)**

School of Mechanical Engineering

Engineering Campus

Universiti Sains Malaysia

# DECLARATION

I hereby declare that this dissertation is my own investigation and study. The experiment data and fabrication has originally conducted by me.

Signed..... (MUHAMMAD RABBANI B ABD RANI)

Date .....

STATEMENT 1

This dissertation is the result of my own investigation and study, except where otherwise stated. Other sources are acknowledged by giving explicit references. Bibliography/references are appended.

Signed..... (MUHAMMAD RABBANI B ABD RANI)

Date .....

STATEMENT 2

I hereby give consent for my dissertation, if accepted, to be available for photocopying and or interlibrary loan, and for the title and summary to be made available outside organizations.

Signed..... (MUHAMMAD RABBANI B ABD RANI)

Date .....

# ACKNOWLEDGEMENT

First and foremost, I am expressing my praise to God for His guidance and blessing throughout this study is carried out. I would like to express my grateful to those who are support and help me to finish this study. I would like to express my sincere gratitude and appreciation to my final year project supervisor, Dr. Mohd Azmi Ismail for his precious guidance, advice and opinions. He is very encouraging and supportive while doing this study. A highly thankful to his willingness by spending times for teaching and guiding until I had finish this project.

Besides, I would also like to express my indebted gratitude and thanks to all technical staffs in School of Mechanical Engineering for guiding and helping me while finishing this project. Lastly, I would like to give higher gratefulness to my family and fellow friends who are really supporting and helping until this project is done.

Title of thesis: Simulation Study of Flow Profile of Air Chamber at Various Heat Supplied

Date of submission (Academic year): 7 June 2017 (2016/2017)

Candidate (Matrix no.): Muhammad Rabbani b. Abd Rani (115753)

Name of supervisor: Dr. Mohd Azmi Ismail

# TABLE OF CONTENTS


DECLARATION ..... I

ACKNOWLEDGEMENT ..... II

TABLE OF CONTENTS ..... III

LIST OF FIGURES ..... V

LIST OF TABLES ..... VI

ABSTRACT ..... 1

ABSTRAK ..... 2

CHAPTER 1 ..... 3

**INTRODUCTION** ..... 3

1.1 Overview ..... 3

1.2 Problem Statement ..... 3

1.3 Objective ..... 3

1.4 Scope of the Project ..... 4

1.5 Outline of Thesis ..... 4

CHAPTER 2 ..... 5

**LITERATURE REVIEW** ..... 5

2.1 Overview ..... 5

2.2 Wind Tunnel ..... 5

2.3 Measurement of Parameters in Wind Tunnels ..... 6

CHAPTER 3 ..... 8

**METHODOLOGY** ..... 8

3.1 Overview ..... 8

3.2 Concept Design ..... 8

3.3 Simulation ..... 9

CHAPTER 4 ..... 13

**RESULTS AND DISCUSSION** ..... 13

4.1 Overview ..... 13

4.2 Volume Rendering of Air Chamber Without Heater and Without Stopper. ..... 14

4.3 Volume Rendering of Air Chamber with Heater and Without Stopper. ..... 18

4.4 Volume Rendering of Air Chamber Without Heater and with Stopper. ..... 22

4.5 Volume Rendering of Air Chamber with Heater and with Stopper. ..... 26

4.6 Velocity Profile of Air Chamber Without Heater and Without Stopper. ..... 30

4.7 Velocity Contour of Air Chamber with Heater and Without Stopper. ..... 34

4.8 Velocity Contour of Air Chamber without Heater and with Stopper. ..... 38
4.9 Velocity Contour of Air Chamber with Heater and with Stopper. ..... 42
4.10 Discussion of The Results ..... 46
CHAPTER 5 ..... 48
**CONCLUSION** ..... 48
5.1 Conclusion ..... 48
5.2 Recommendation for Further Studies ..... 48
REFERENCES ..... 49

# LIST OF FIGURES

Figure 3:1 Viewed in Solidworks ..... 8

Figure 3:2 Viewed in ANSYS Fluent ..... 8

Figure 3:3 Drawing Sheet of The Design ..... 9

Figure 3:4 ANSYS-Workbench ..... 9

Figure 3:5 Meshing file of the air chamber ..... 10

Figure 3:6 Viscous Model ..... 10

Figure 3:7 Velocity Inlet ..... 11

Figure 3:8 Energy box ..... 11

Figure 3:9 Input Source Terms ..... 11

Figure 3:10 Velocity Contour ..... 12

Figure 3:11 Draw a line across the velocity contour ..... 12

# LIST OF TABLES

Table 4-1 Types of simulation. ..... 13

Table 4-2 Volume rendering across the air chamber ..... 14

Table 4-3 Volume rendering across the air chamber ..... 18

Table 4-4 Volume rendering across the air chamber ..... 22

Table 4-5 Volume rendering across the air chamber ..... 26

Table 4-6 Graph of velocity profile across the outlet of the air chamber ..... 30

Table 4-7 Graph of velocity profile across the outlet of the air chamber ..... 34

Table 4-8 Graph of velocity profile across the outlet of the air chamber ..... 38

Table 4-9 Graph of velocity profile across the outlet of the air chamber ..... 42

Table 4-10 Maximum velocity magnitude recorded ..... 46

Table 4-11 Maximum velocity magnitude recorded ..... 47

# ABSTRACT

The wind tunnel is one of the most common experimental testing facilities used in mechanical or aerospace engineering for the testing of fluid flow behaviour. The flow profile of a fluid is vital component to gets a general comprehension of how the flow profundity changes in a chamber. The viscosity and velocity of a fluid can altogether influence the route in which it flows through a chamber. The utilization of hot air is vital in these days' application, for example, drying processes, washing processes, pasteurisation processes and so forth. Different temperature levels are connected in mechanical applications. Thus, the objective of this project is to do the flow profile of air chamber at different intake velocity so that we can determine and get overall understanding of how the flow varies in a chamber affected by the heater and stopper equipped. Next, simulation using ANSYS Workbench-Fluent software has been developed to obtain the volume rendering of the air chamber and the velocity contour at the outlet of the air chamber. From the result, it is found that the heater will affects the velocity magnitude at the outlet of the air chamber. The stopper also will affect the velocity distributed across the outlet of the air chamber.

# ABSTRAK

Terowong angin merupakan salah satu kemudahan ujian eksperimental yang paling biasa digunakan dalam kejuruteraan mekanikal atau aerospace untuk menguji kelakuan aliran bendalir. Profil aliran bendalir merupakan komponen penting untuk mendapat pemahaman umum tentang bagaimana perubahan aliran dalam ruang. Kelikatan dan halaju bendalir boleh sama sekali mempengaruhi laluan di mana ia mengalir melalui ruang. Penggunaan udara panas amat penting dalam aplikasi hari ini, contohnya, proses pengeringan, proses cuci, proses pasteurisasi dan sebagainya. Tahap suhu yang berbeza disambungkan dalam aplikasi mekanikal. Oleh itu, matlamat projek ini adalah untuk membuat profil aliran ruang udara pada halaju pengambilan yang berbeza supaya kita boleh menentukan dan mendapatkan pemahaman secara keseluruhan tentang bagaimana aliran itu berubah-ubah di ruang yang terjejas oleh pemanas dan penyambung yang dilengkapi. Seterusnya, simulasi menggunakan perisian ANSYS Workbench-Fluent telah dibangunkan untuk mendapatkan pembentukan kuantiti ruang udara dan kontur halaju di ruang pam udara. Dari hasilnya, didapati bahawa pemanas akan memberi kesan kepada magnitud halaju di ruang keluar udara. Penyumbat juga akan menjejaskan halaju yang diedarkan di seluruh ruang udara.

# CHAPTER 1

## INTRODUCTION

### 1.1 Overview

In this chapter, the problem statement, objectives, scopes of the project and thesis outline also will be included in this chapter to give a crystal clear review about this research.

### 1.2 Problem Statement

Many research has focused on wind tunnels in these decades. They are utilized for an assortment of various reasons, for example, their ability to test prototypes early in design cycles. The wind tunnel is a testing facilities utilized as a part of mechanical or aerospace engineering for the testing of fluid stream conduct. The flow profile of a fluid is very important element to helps to get an overall understanding of how the flow depth varies in a chamber. The viscosity and velocity of a fluid can significantly affect the way in which it flows through a chamber. The use of hot air is very important in nowadays application such as drying processes, washing processes, pasteurisation processes and etc. Various temperature levels are applied in industrial applications. Thus, my goal for my project is to do the flow profile of air chamber at different intake velocity so that we can determine and get overall understanding of how the flow varies in a chamber affected by the heater and stopper equipped.

### 1.3 Objective

A research will be conducted based on simulations to achieve the objectives of this projects. The objectives of this project are:

- To design an air chamber via CAD software
- To develop simulations by using software to predict the flow profile of air in the wind tunnel/air chamber
- To compare and discuss the effect of heat and stopper on air flow profile in simulation result.

## 1.4 Scope of the Project

In this research, simulations were developed by using software to know the flow profile of the air chamber. ANSYS Workbench software was used to run the simulations. Finally, make comparison of the flow profile for different aspects. Some discussions were made to clarify the difference in results obtained from the simulations.

## 1.5 Outline of Thesis

Overall, this thesis can be divided into five main chapters which are introduction, literature review, methodology, result and discussion, and conclusion.

In chapter one, it is more about the introduction to the research background, problem statement, objectives and the scopes of the project. Next, in chapter two, some literature reviews are made based on journals, books and webpages as references. Detail explanations about this research are described in this chapter.

For chapter three, it will discuss on the project methodology. Procedures on simulations will be explained briefly to give a clear view on how they are conducted. In the following chapter which is chapter four, results obtained from the experiments and simulations will be discussed and verified here. Then, results comparison will be done to see the similarities between that two results.

Last but not least, the conclusion of the project will be made in chapter five. Some suggestions and recommendations are given for improvement in future research.

# CHAPTER 2

## LITERATURE REVIEW

### 2.1 Overview

Until recently, many researchers have shown interest in the field of wind engineering. They have carried out numerous laboratory experiments and simulations regarding this field. Their findings and suggestions are reviewed in this chapter.

### 2.2 Wind Tunnel

A wind tunnel is used to study the effects of air moving past solid objects. A wind tunnel consists of a shut tubular entry with the object under test mounted in the centre. A powerful fan framework comprising of one or different fans moves air past the object; the fan must have fixing vanes to smooth the wind stream. The test object is instrumented with a delicate adjust to quantify the strengths produced via wind stream; or, the wind stream may have smoke other substances injected to make the flow lines around the object noticeable. [1]

and Scientists put a model of a plane, car or wind process in the tunnel and after that review the way air moves around the model. Wind tunnels chip away at the possibility that a stationary model with air moving around it carries on a similar way a genuine, full-scale object under test traveling through stationary air does. The models tested are stacked with many instruments and sensors that report back to the computers in the control room. Air is blown or sucked through a conduit equipped with a review port and instrumentation where models or geometrical shapes are mounted for study. Regularly, the air is travelled through the tunnel utilizing a progression of fans. The wind stream made by the fans that is entering the tunnel is itself profoundly turbulent in nature because of the fan edge movement thus it is not straightforwardly valuable for exact estimations. Different strategies are utilized to concentrate the genuine wind current around the geometry and contrast it and hypothetical outcomes, which should likewise consider the Reynolds number and Mach number for the administration of operation. [1]

## 2.3 Measurement of Parameters in Wind Tunnels

Air velocity and pressures are measured in a few routes in wind tunnels. Air velocity through the test area can be dictated by Bernoulli's principle. The course of airflow around a model can be controlled by tufts of yarn appended to the streamlined surfaces. The bearing of airflow moving toward a surface can be pictured by mounting strings in the airflow. Air pockets or smoke of fluid can be brought into the airflow upstream of the test display, and their way around the model can be shot. Shaft adjust is utilized to gauge streamlined strengths on the test display. The weight disseminations over the test demonstrate have verifiably been measured by boring numerous little openings along the airflow way, and utilizing multi-tube manometers to gauge the weight at each gap. Weight circulations can all the more helpfully be measured by the utilization of weight touchy paint, in which higher neighbourhood weight is demonstrated by brought down fluorescence of the paint by then. Weight disseminations can likewise be advantageously measured by the utilization of weight delicate weight belts, in which different ultra-scaled down weight sensor modules are incorporated into an adaptable strip. [1]

The strip is joined to the streamlined surface with tape, and it sends signals demonstrating the weight dispersion along its surface. Pressure dispersions on a test model can likewise be controlled by playing out a wake review, in which either a solitary pitot tube is used to acquire various readings downstream of the test display, or a numerous tube manometer is mounted downstream and every one of its readings are taken. All estimations of wind tunnel parameters are passed on to computers through digital instrumentation. [1]

A decent wind tunnel ought to empower estimations of precise relentless or insecure information, contingent upon the coveted application. The turbulence power in the test segment ought to be sufficiently low to encourage investigation of the physical marvels of premium. For instance, limit layer moves tests [2] frequently require to a great degree low turbulence levels. Aeroacoustics tests of trailing edge clamour require low turbulence levels to limit the impacts brought on by the impingement of freestream turbulence on the main edge of a model [3]. The test segment stream ought to be uniform and without any unintended shakiness, while optional stream impacts ought to be limited in the constriction. Stream detachment ought to be maintained a strategic distance from inside the tunnel circuit to avoid stream flimsiness and related clamour, and to limit misfortunes. Blockage impacts because of the test area display and the relating streamline shape ought to be insignificant to copy freestream conditions. Vibrations produced by the drive framework ought to be damped to limit its impact on the trial.

On account of aeroacoustics offices, acoustic treatment is fundamental to limit aggravation commotion from debasing the acoustic estimations. [4]

Obviously, uniform velocity in the test area is wanted. The inviscid centre of the test segment stream ought to have as meagre deviation from a fitting profile as could be expected under the circumstances. There are an extensive number of trial procedures accessible to gauge the mean velocity profile [5], including crossing a pitot-static or pitot test through the test segment (see Pressure and Velocity Measurements). This is a demonstrated, economical, however tedious strategy. A rake might be utilized as an option, so different spatial areas can be measured simultaneously. Single or multiple hot wires configured for constant-temperature anemometry (CTA), particle image velocimetry (PIV), and laser Doppler velocimetry (LDV) can also be used. The stream consistency is normally described as either as a min–max or a rms deviation from the mean velocity the inviscid centre stream of the test area [6].

# CHAPTER 3

## METHODOLOGY

### 3.1 Overview

In this chapter, the design and simulation procedure will be described briefly in order to find the velocity profile of the wind tunnel/air chamber.

### 3.2 Concept Design

In this project, the design of the air chamber was drawn first in Solidworks. Then, the mesh file of the design was created using GAMBIT Fluent.

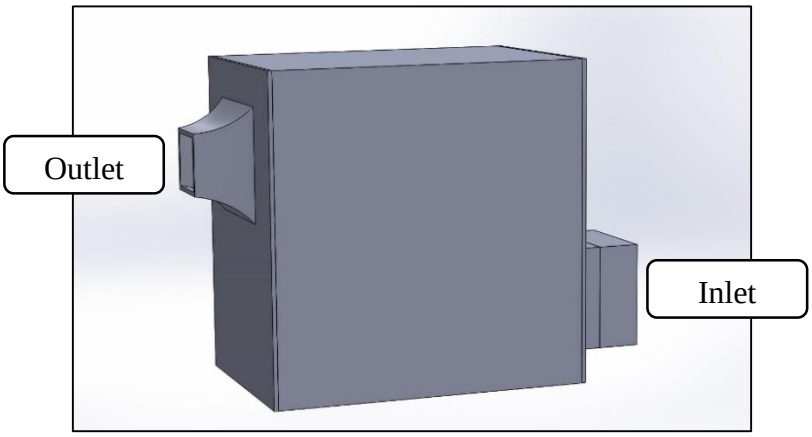



Figure 3:1 Viewed in Solidworks

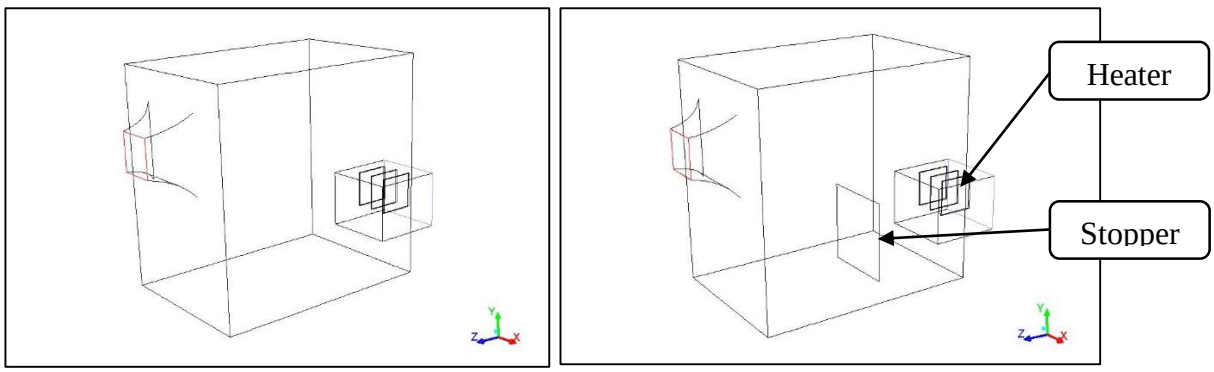



Figure 3:2 Viewed in ANSYS Fluent

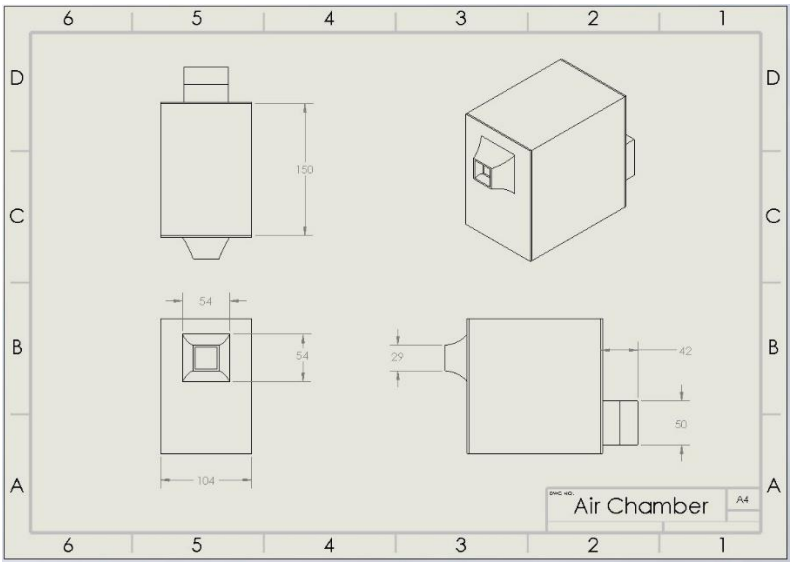



Figure 3:3 Drawing Sheet of The Design

## 3.3 Simulation

The simulation is done by using ANSYS WORKBENCH. The geometry of the air chamber is done using GAMBIT Fluent and the meshing file (.msh) was upload into ANSYS WORKBENCH-Fluid Flow (Fluent).

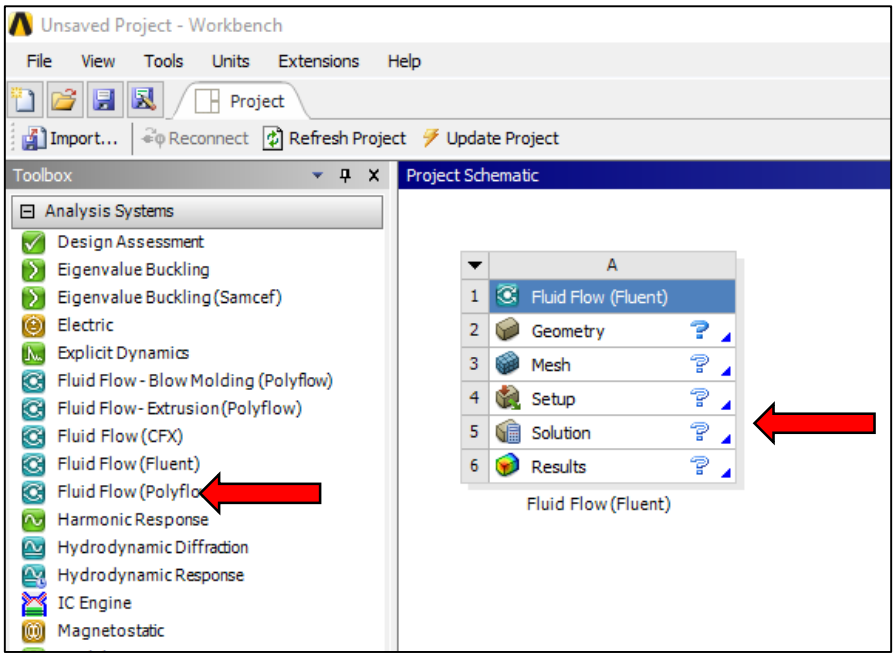



Figure 3:4 ANSYS-Workbench

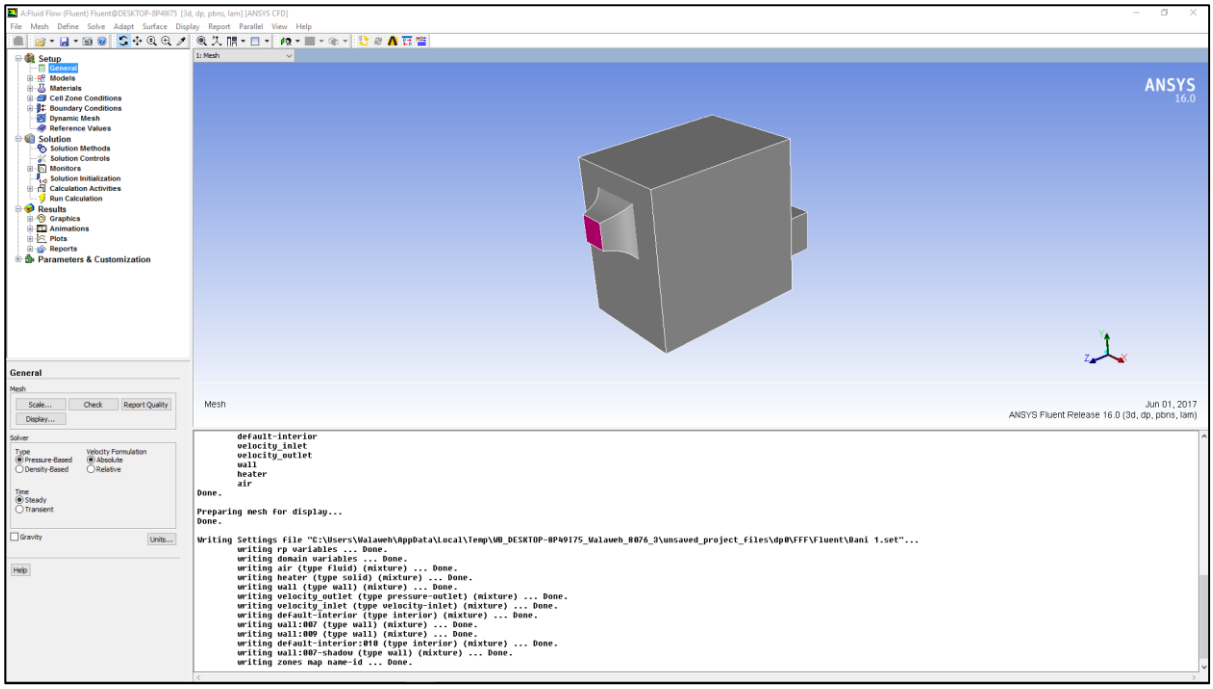



Figure 3:5 Meshing file of the air chamber

In order to perform the proper CFD simulations, the solution setup is set to pressure-based type and the time is in steady state.

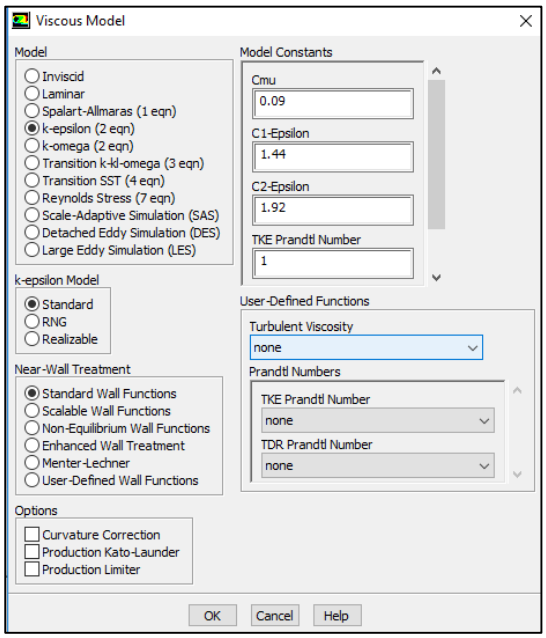



Figure 3:6 Viscous Model

For the Viscous Model, k-epsilon (2-eqn) was chosen because to simulate mean flow characteristics for turbulent flow conditions during the simulation.

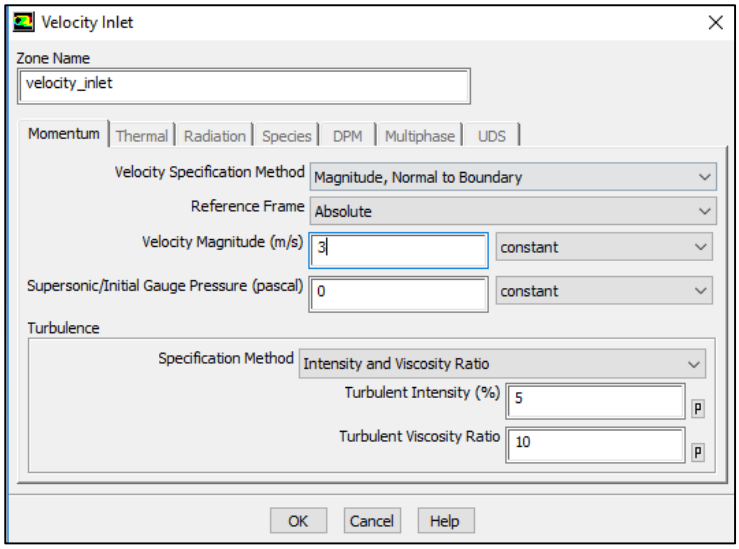



Figure 3:7 Velocity Inlet

For the boundary conditions, the velocity magnitude for the fluid at the inlet was set for 3 m/s. The simulation was repeated again by changing the velocity magnitude by increasing to 6 m/s, 9 m/s, 12 m/s, 15 m/s, 18 m/s, 21 m/s, 24 m/s, 27 m/s, and 30 m/s. And then, the simulation was calculated for about 500 iterations.

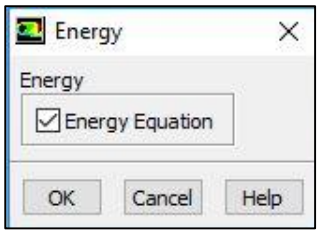



Figure 3:8 Energy box

For the simulation with heater, the Energy equation was tick to activate the relevant models, supply thermal boundary conditions, and input material properties that govern heat transfer and/or may vary with temperature.

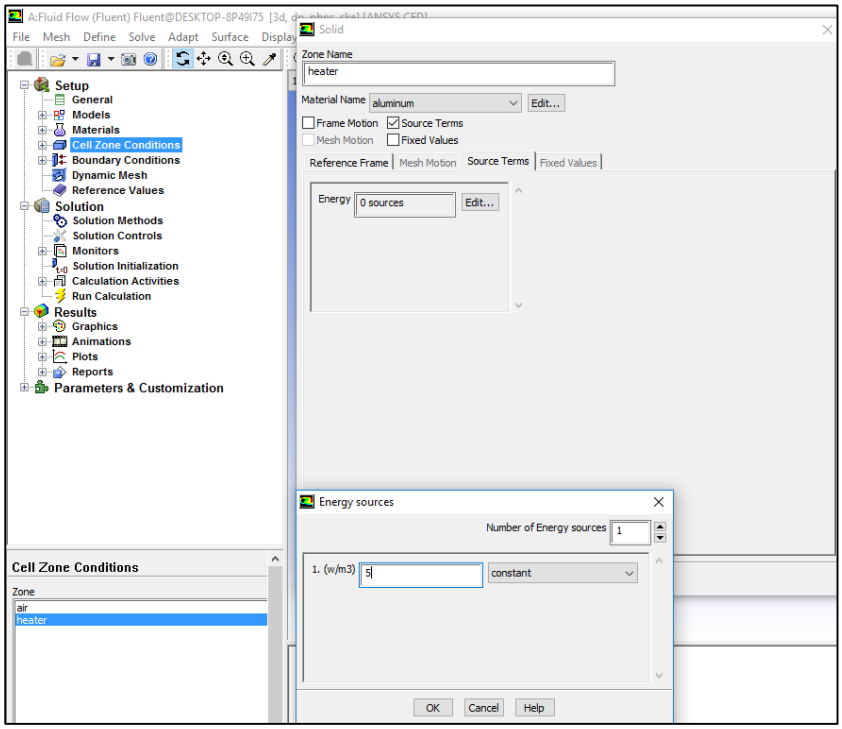



Figure 3:9 Input Source Terms

In Cell Zone Conditions, the source terms was set to 5 $w/m^3$.

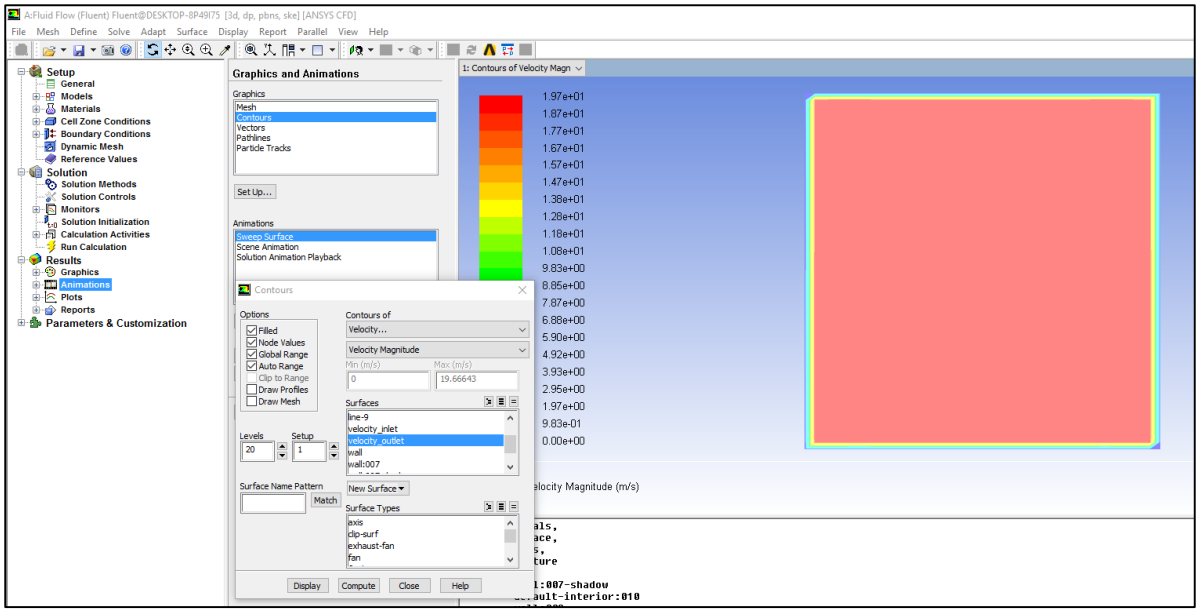



Figure 3:10 Velocity Contour

After the calculation completed, we can view the velocity magnitude contour obtained. Image above shows the velocity magnitude contour at the outlet of the air chamber. From the contour, we draw a line so that we can obtained the velocity magnitude across the air chamber outlet. All the values recorded we shown in next chapter.

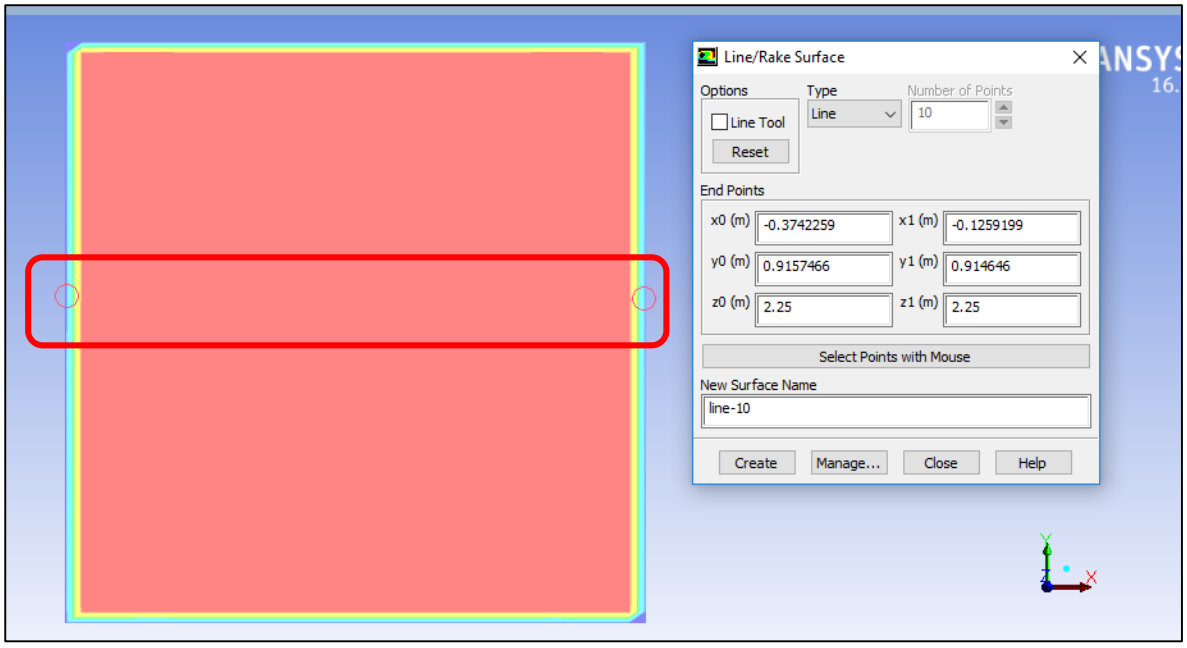



Figure 3:11 Draw a line across the velocity contour

# CHAPTER 4

## RESULTS AND DISCUSSION

### 4.1 Overview

This chapter focusing on the results and the discussion of results of the simulation that have been conducted. The simulation results including velocity contours also will be included in this chapter.

There will be four types of simulation that will discuss in this chapter:

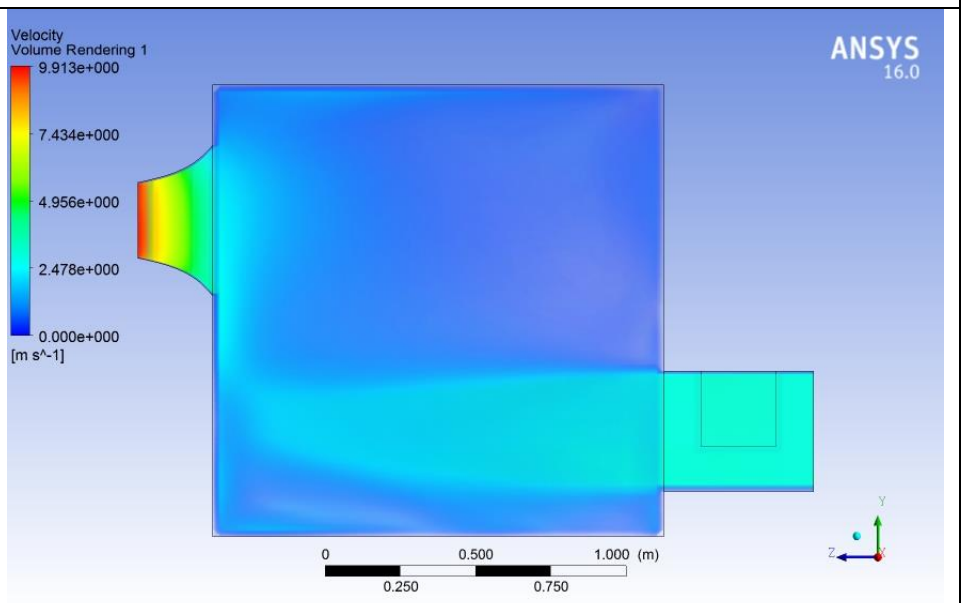
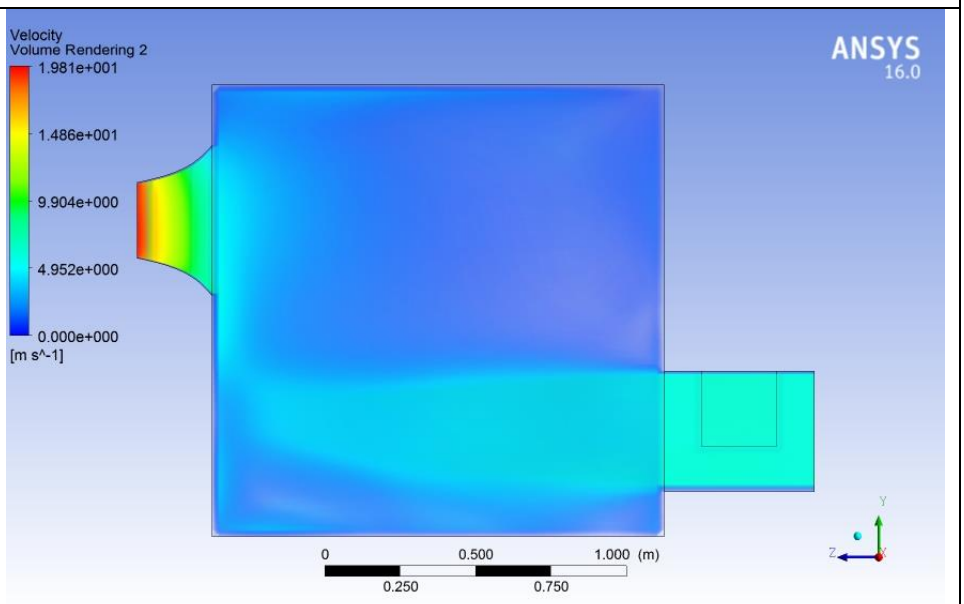
a) Air chamber/wind tunnel without heater and without stopper.
b) Air chamber/wind tunnel with heater and without stopper.
c) Air chamber/wind tunnel without heater and with stopper.
d) Air chamber/wind tunnel with heater and with stopper.

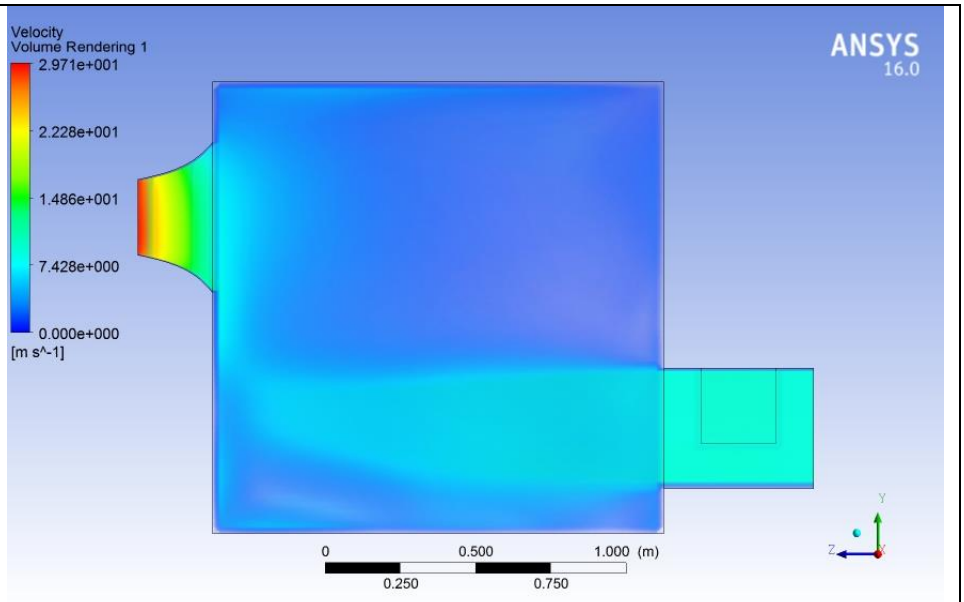
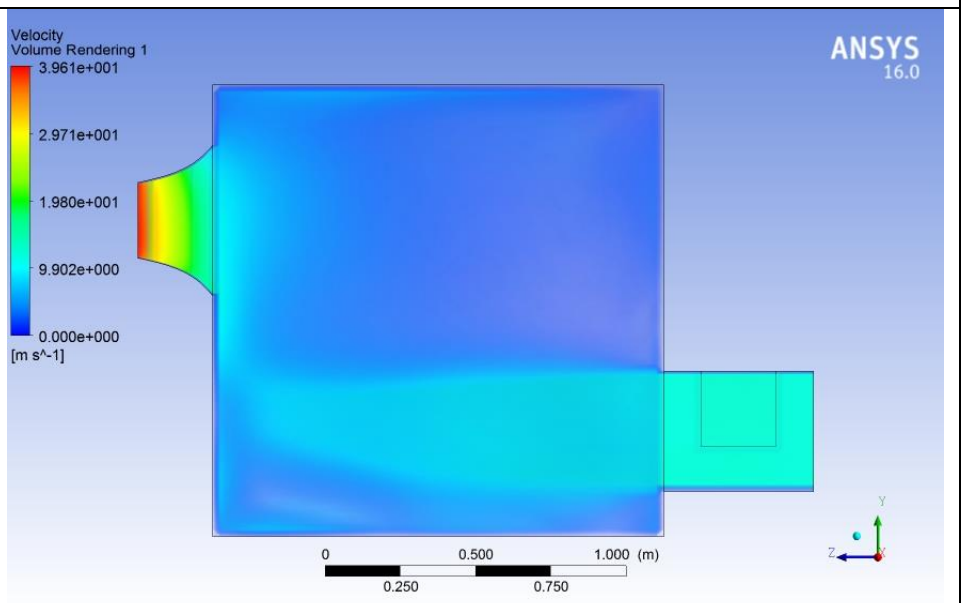
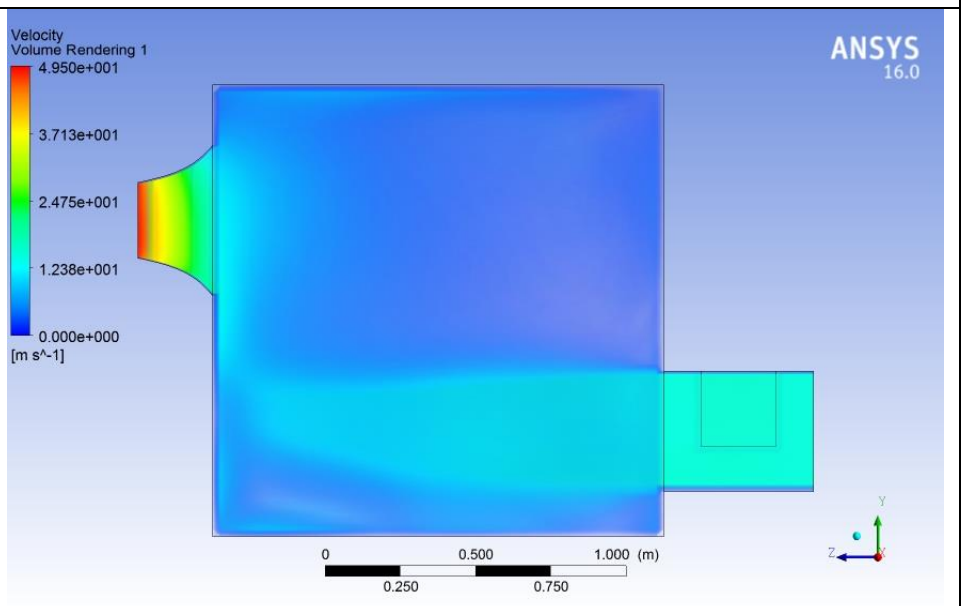
Table 4-1 Types of simulation.

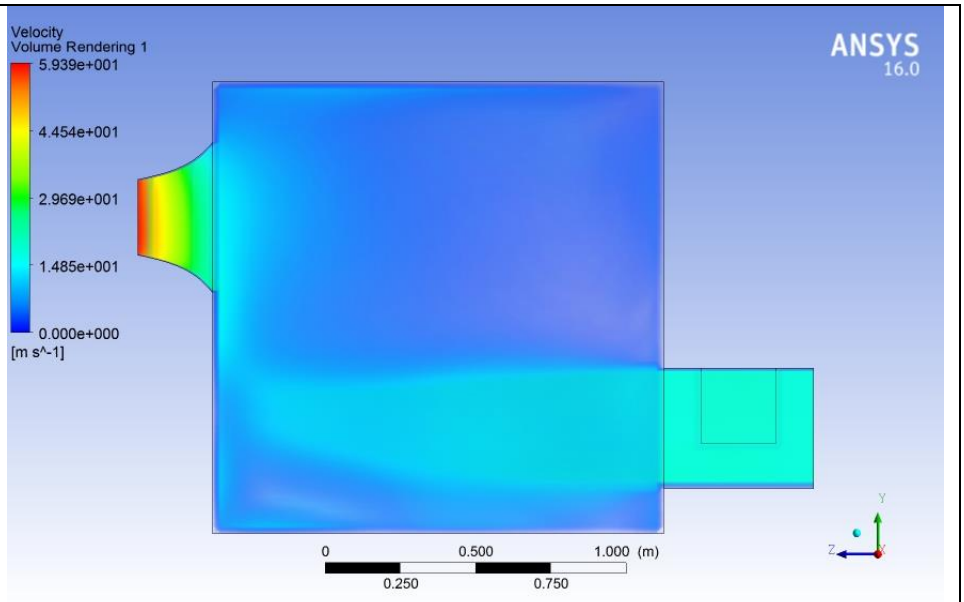
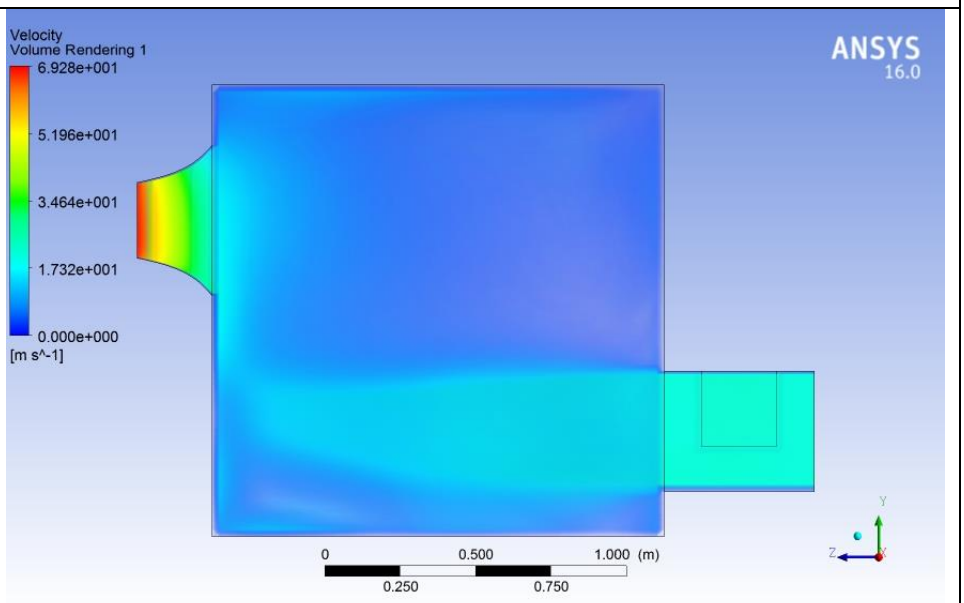
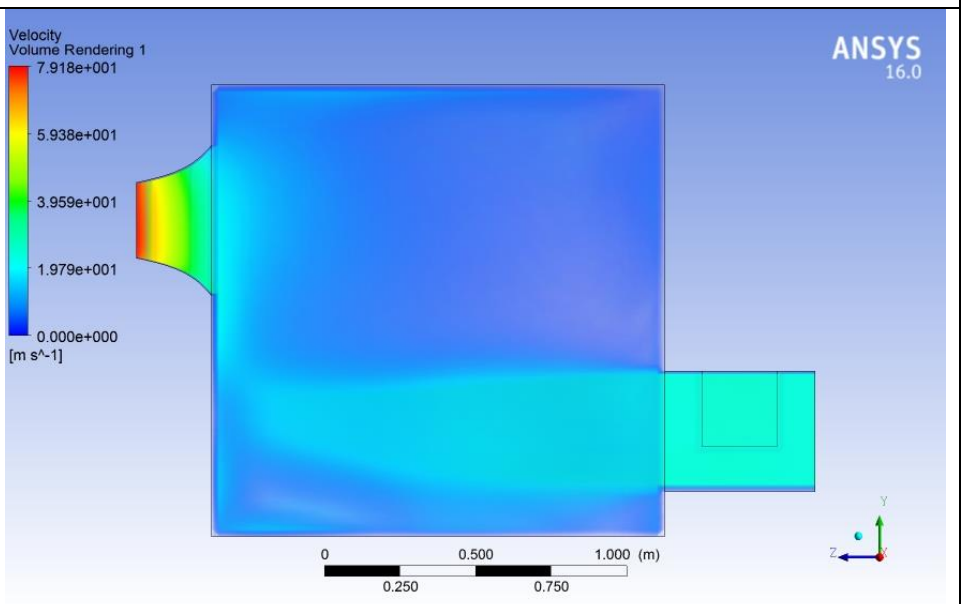
| Type | Heater | Stopper |
|---|---|---|
| A | No | No |
| B | Yes | No |
| C | No | Yes |
| D | Yes | Yes |

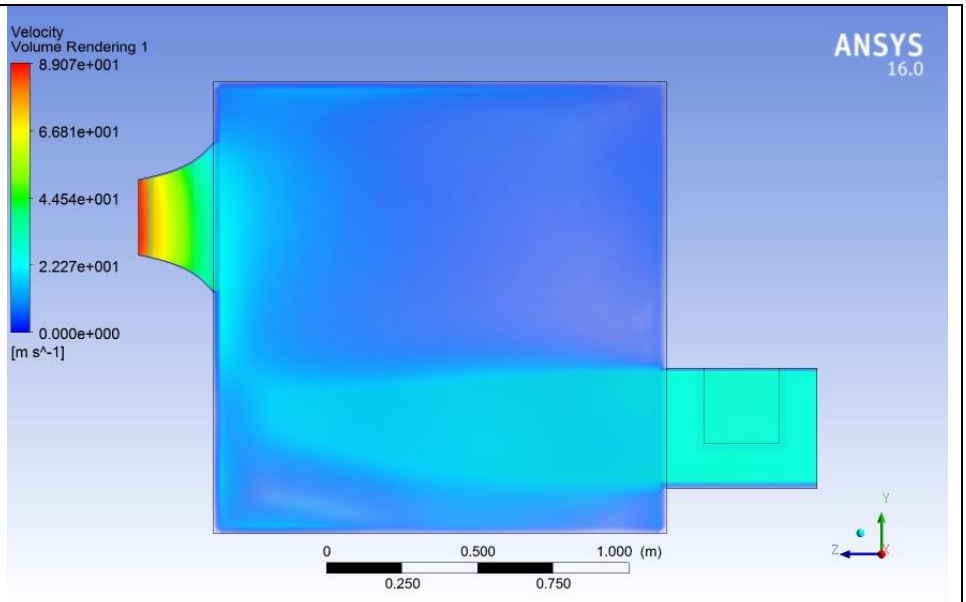
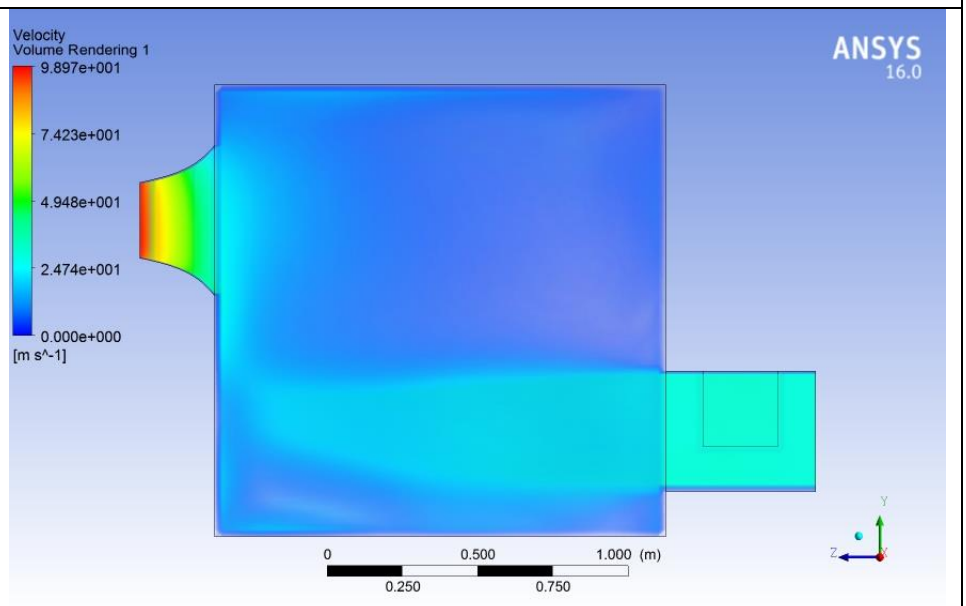
4.2 Volume Rendering of Air Chamber Without Heater and Without Stopper.

Table 4-2 Volume rendering across the air chamber

| Intake velocity (m/s) | Velocity Contour |
|---|---|
| 3 |   |
| 6 |   |

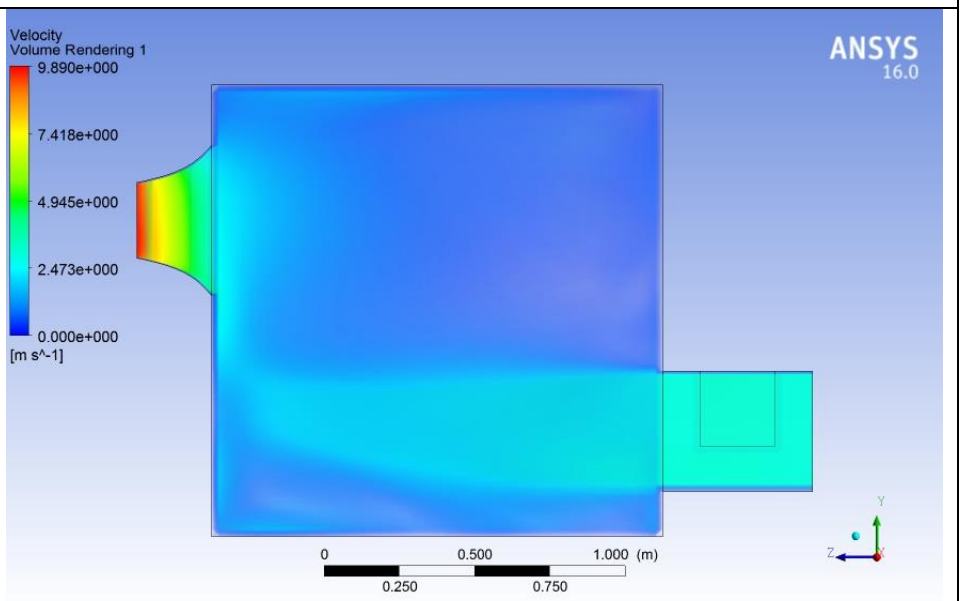
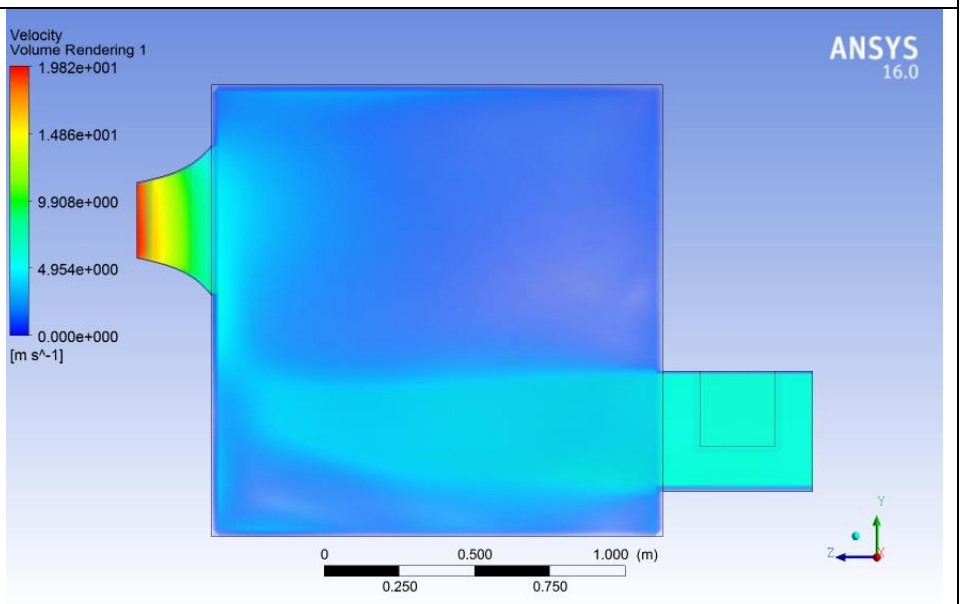
| | |
|---|---|
| 9 |  Velocity<br>Volume Rendering 1<br>2.971e+001<br>2.228e+001<br>1.486e+001<br>7.428e+000<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |
| 12 |  Velocity<br>Volume Rendering 1<br>3.961e+001<br>2.971e+001<br>1.980e+001<br>9.902e+000<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |
| 15 |  Velocity<br>Volume Rendering 1<br>4.950e+001<br>3.713e+001<br>2.475e+001<br>1.238e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |

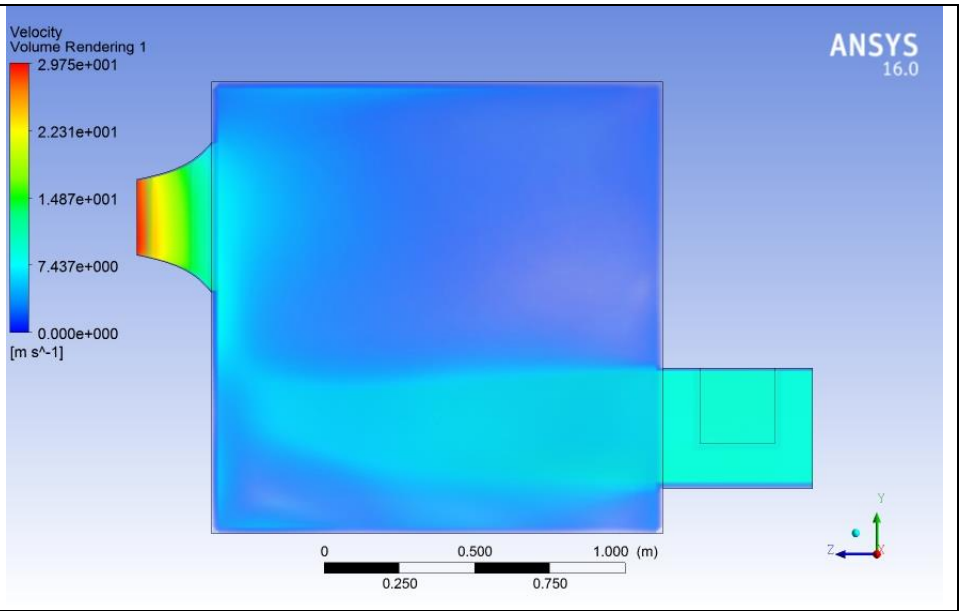
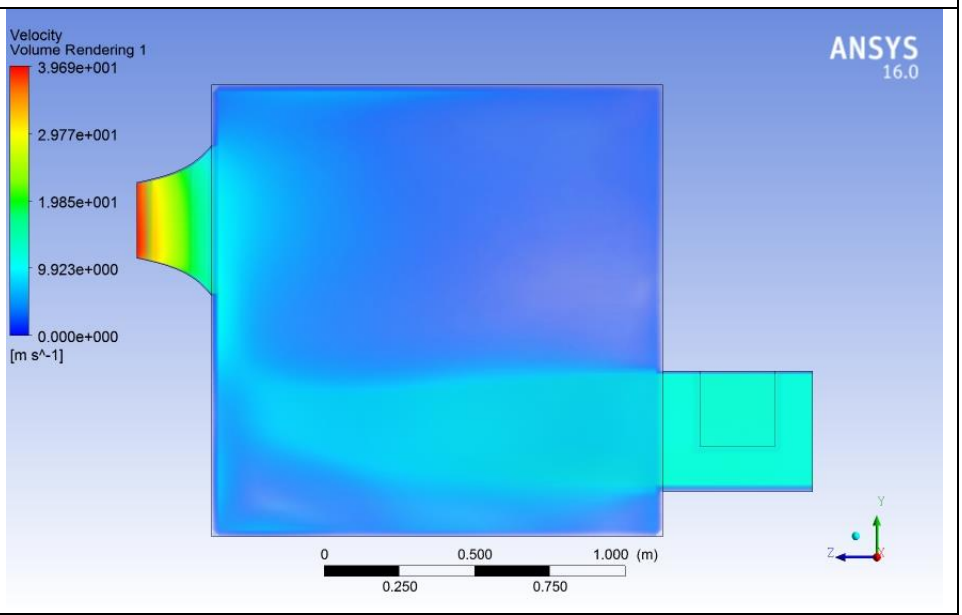
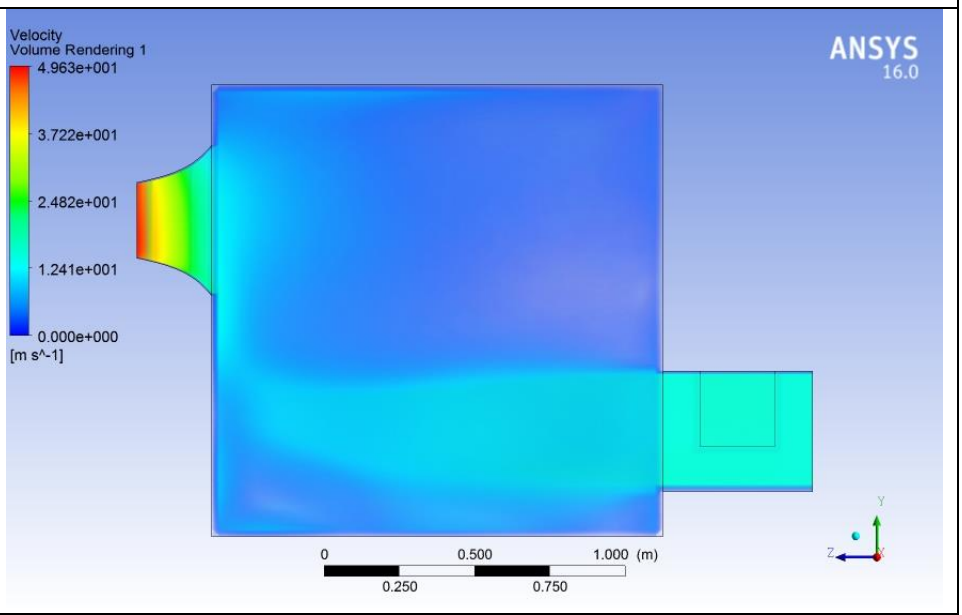
| | |
|---|---|
| 18 |  Velocity<br>Volume Rendering 1<br>5.939e+001<br>4.454e+001<br>2.969e+001<br>1.485e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |
| 21 |  Velocity<br>Volume Rendering 1<br>6.928e+001<br>5.196e+001<br>3.464e+001<br>1.732e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |
| 24 |  Velocity<br>Volume Rendering 1<br>7.918e+001<br>5.938e+001<br>3.959e+001<br>1.979e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |

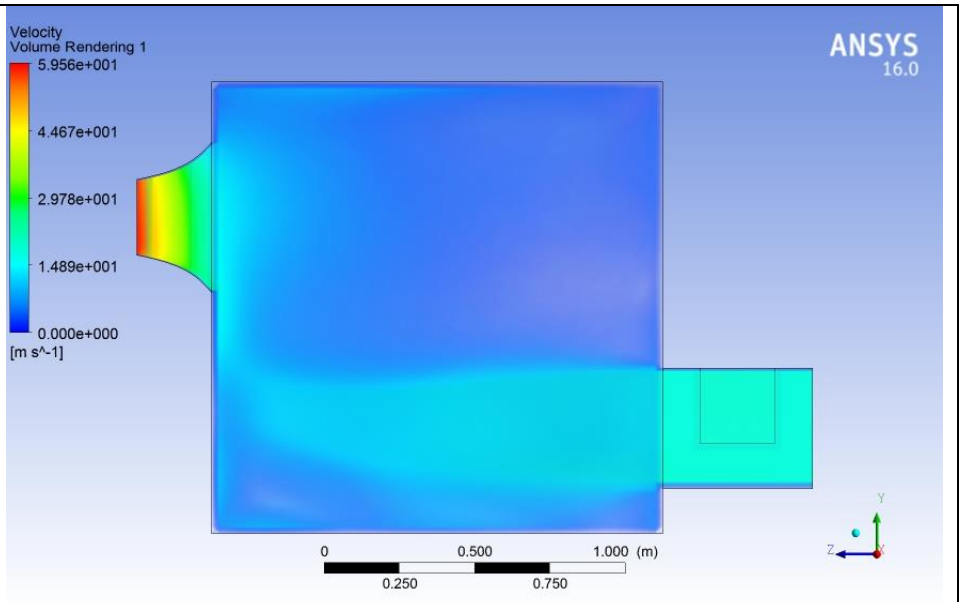
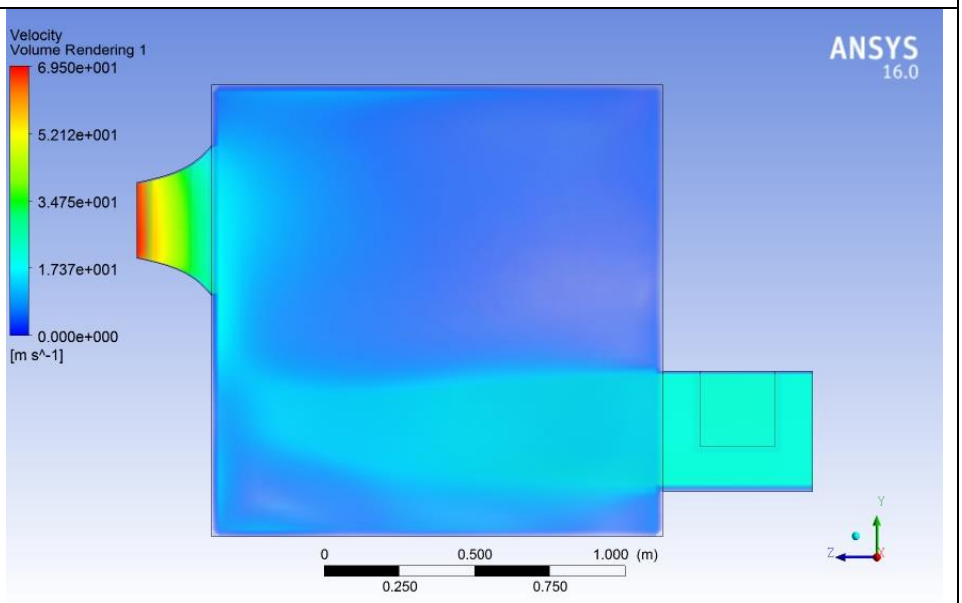
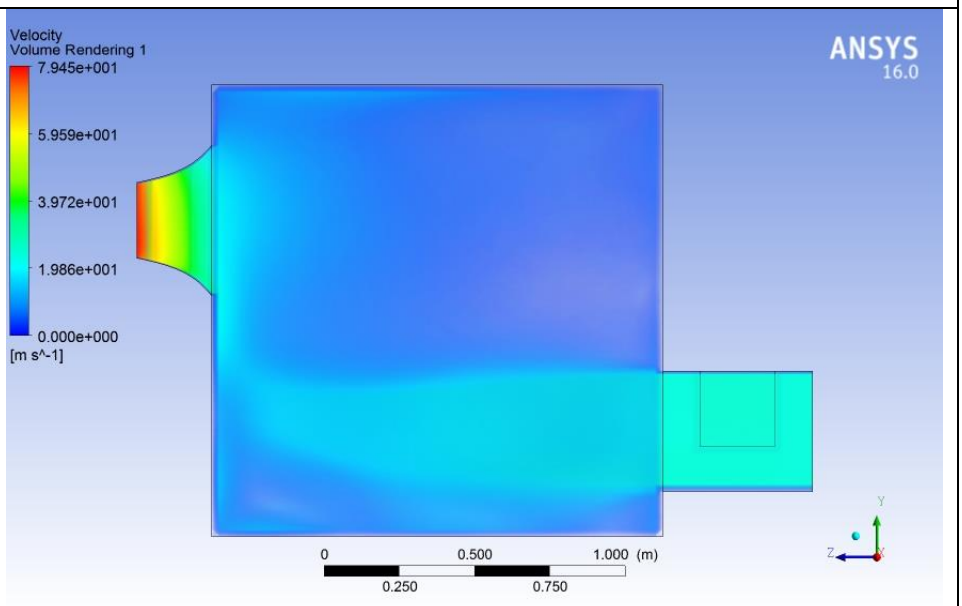
| | |
|---|---|
| 27 |  Velocity<br>Volume Rendering 1<br>8.907e+001<br>6.681e+001<br>4.454e+001<br>2.227e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |
| 30 |  Velocity<br>Volume Rendering 1<br>9.897e+001<br>7.423e+001<br>4.948e+001<br>2.474e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |

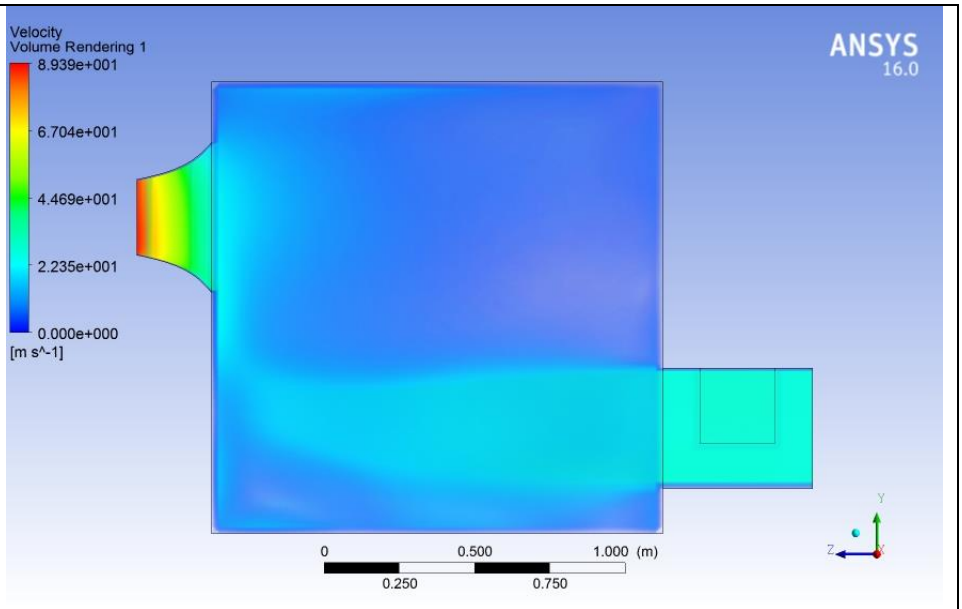
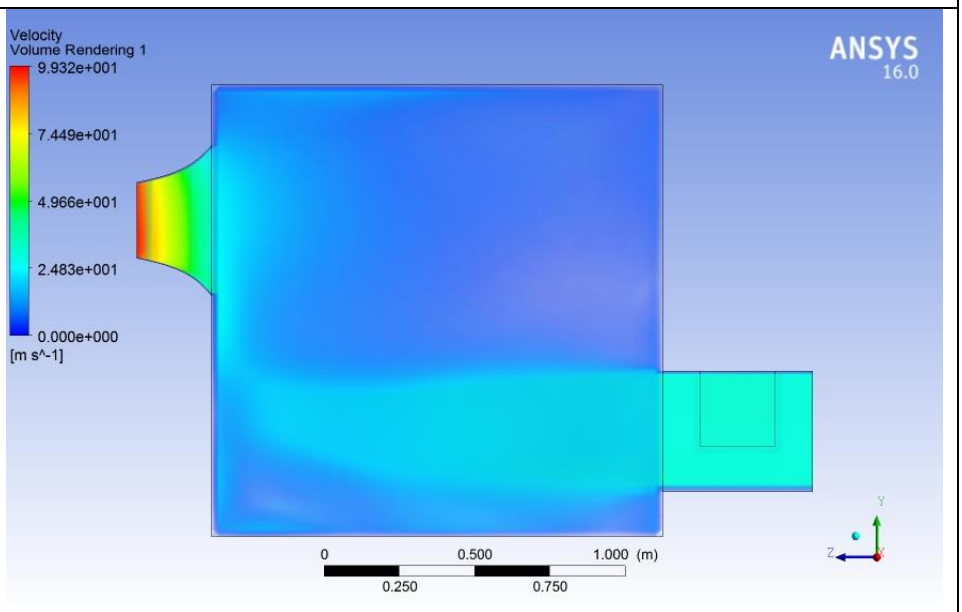
4.3 Volume Rendering of Air Chamber with Heater and Without Stopper.

Table 4-3 Volume rendering across the air chamber

| Intake velocity (m/s) | Velocity contour |
|---|---|
| 3 |   |
| 6 |   |

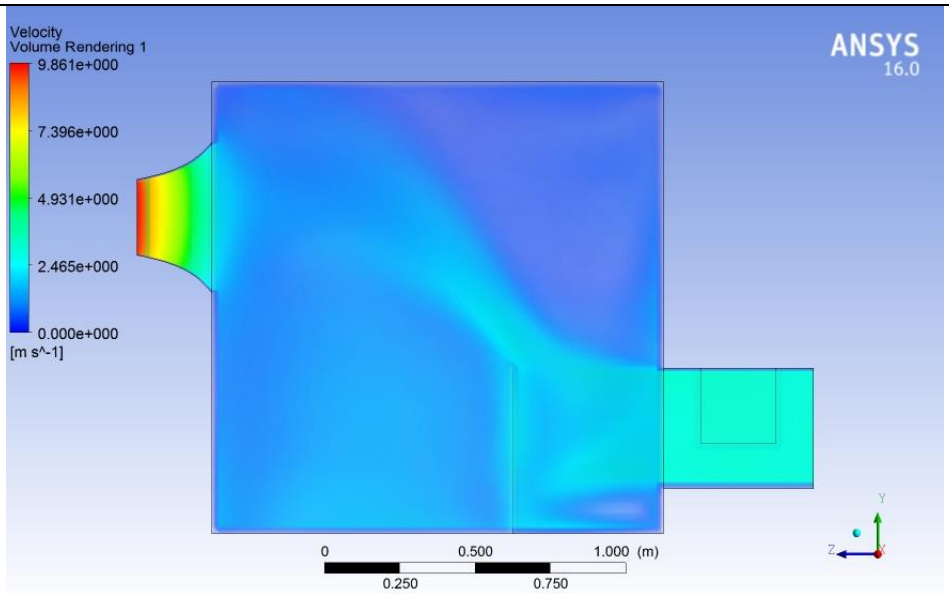
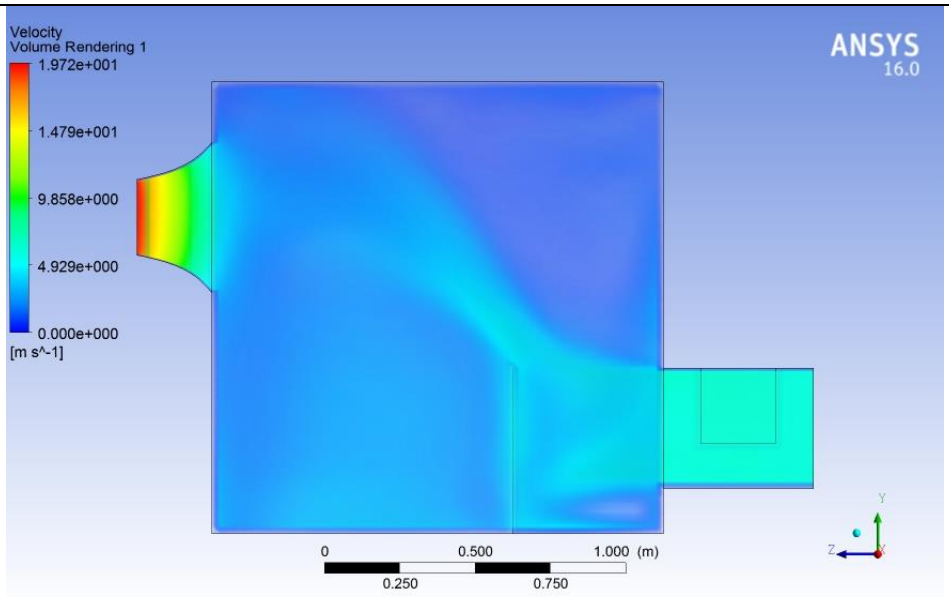
| | |
|---|---|
| 9 |  Velocity<br>Volume Rendering 1<br>2.975e+001<br>2.231e+001<br>1.487e+001<br>7.437e+000<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |
| 12 |  Velocity<br>Volume Rendering 1<br>3.969e+001<br>2.977e+001<br>1.985e+001<br>9.923e+000<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |
| 15 |  Velocity<br>Volume Rendering 1<br>4.963e+001<br>3.722e+001<br>2.482e+001<br>1.241e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |

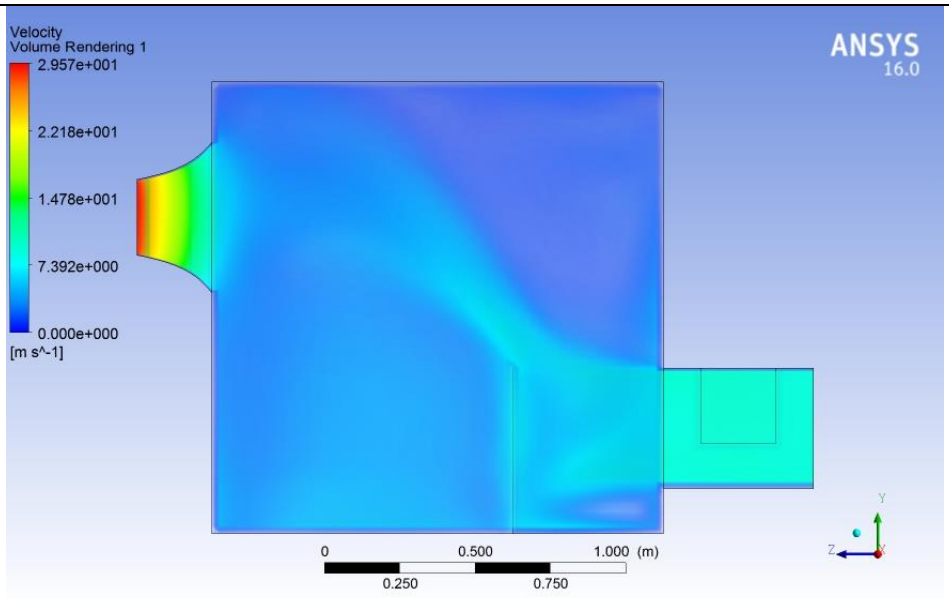
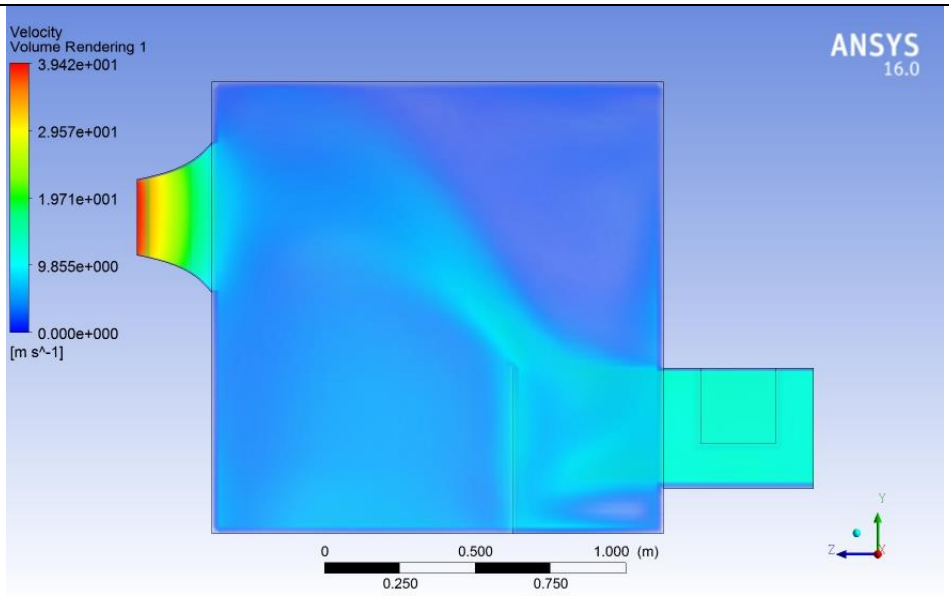
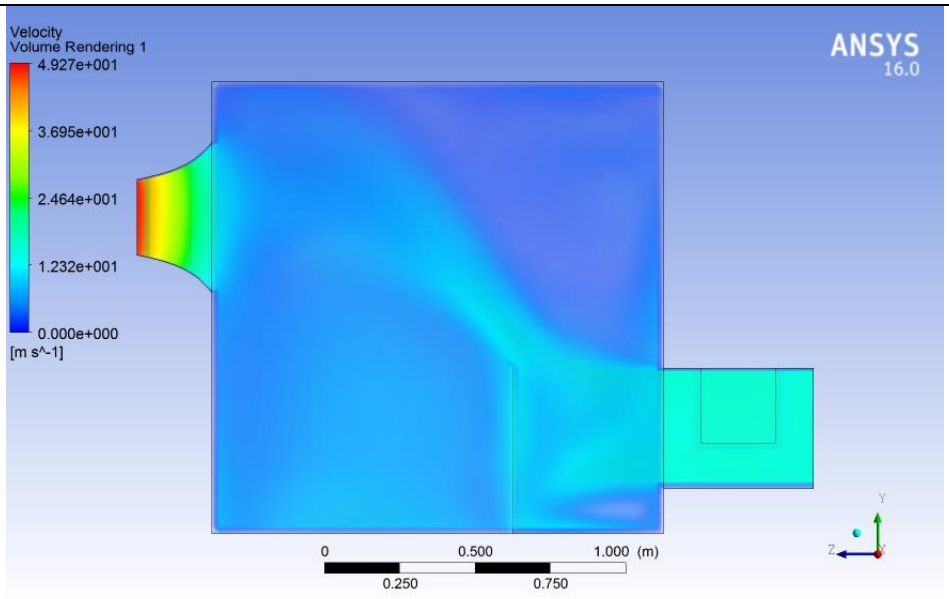
| | |
|---|---|
| 18 |  Velocity<br>Volume Rendering 1<br>5.956e+001<br>4.467e+001<br>2.978e+001<br>1.489e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750 |
| 21 |  Velocity<br>Volume Rendering 1<br>6.950e+001<br>5.212e+001<br>3.475e+001<br>1.737e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750 |
| 24 |  Velocity<br>Volume Rendering 1<br>7.945e+001<br>5.959e+001<br>3.972e+001<br>1.986e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750 |

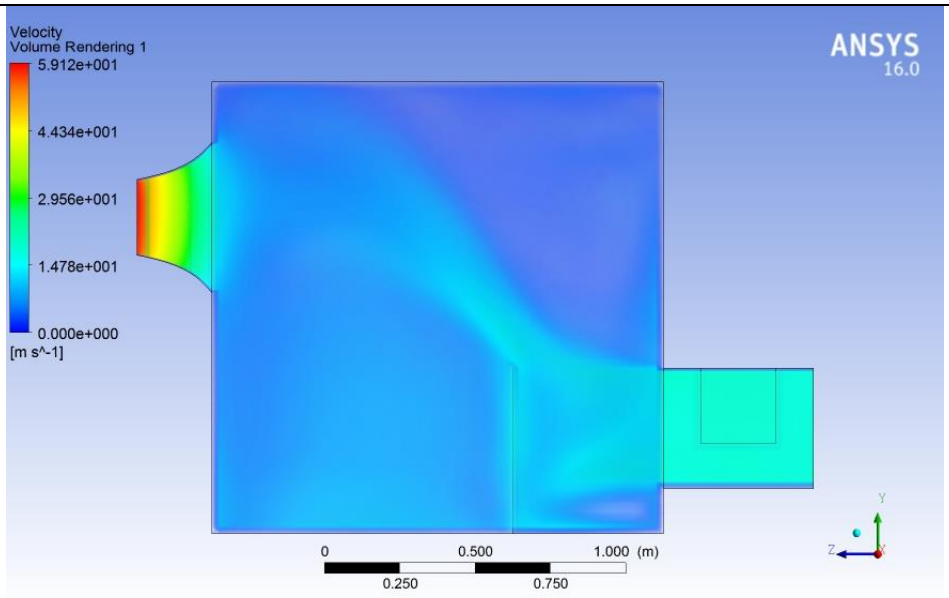
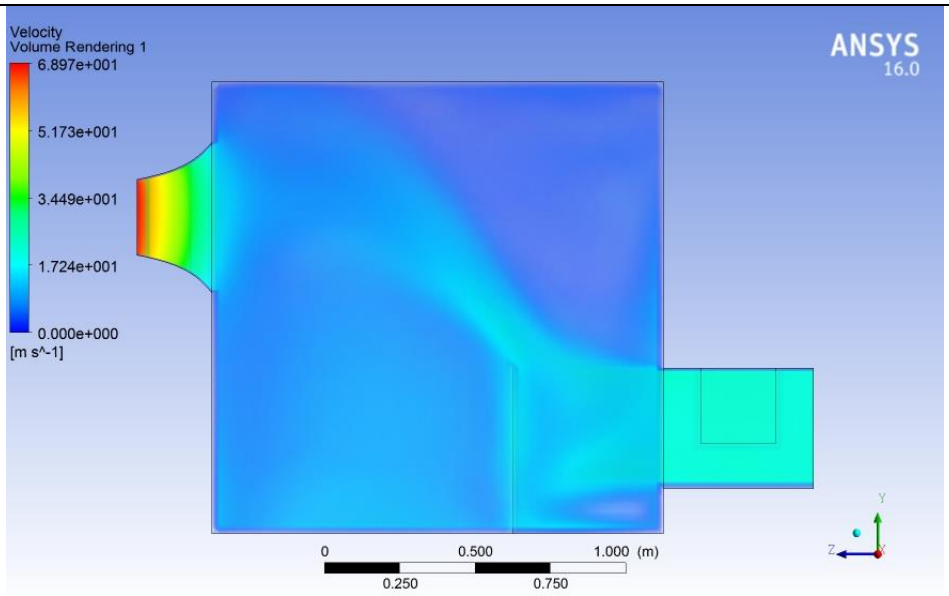
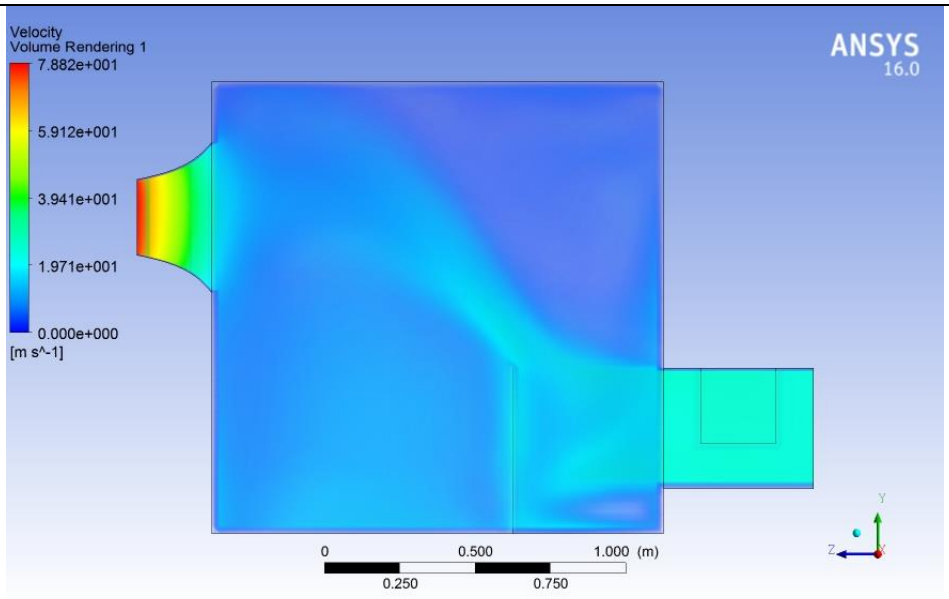
| | |
|---|---|
| 27 |  Velocity<br>Volume Rendering 1<br>8.939e+001<br>6.704e+001<br>4.469e+001<br>2.235e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |
| 30 |  Velocity<br>Volume Rendering 1<br>9.932e+001<br>7.449e+001<br>4.966e+001<br>2.483e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |

4.4 Volume Rendering of Air Chamber Without Heater and with Stopper.

Table 4-4 Volume rendering across the air chamber

| Intake velocity (m/s) | Velocity contour |
|---|---|
| 3 |  |
| 6 |  |

| | |
|---|---|
| 9 |  Velocity<br>Volume Rendering 1<br>2.957e+001<br>2.218e+001<br>1.478e+001<br>7.392e+000<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |
| 12 |  Velocity<br>Volume Rendering 1<br>3.942e+001<br>2.957e+001<br>1.971e+001<br>9.855e+000<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |
| 15 |  Velocity<br>Volume Rendering 1<br>4.927e+001<br>3.695e+001<br>2.464e+001<br>1.232e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |

| | |
|---|---|
| 18 |  Velocity<br>Volume Rendering 1<br>5.912e+001<br>4.434e+001<br>2.956e+001<br>1.478e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |
| 21 |  Velocity<br>Volume Rendering 1<br>6.897e+001<br>5.173e+001<br>3.449e+001<br>1.724e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |
| 24 |  Velocity<br>Volume Rendering 1<br>7.882e+001<br>5.912e+001<br>3.941e+001<br>1.971e+001<br>0.000e+000<br>[m s^-1]<br>ANSYS<br>16.0<br>0<br>0.500<br>1.000 (m)<br>0.250<br>0.750<br>Y<br>Z<br>X |