SIMULATION STUDY OF DRYING CHAMBER FOR MARINE PRODUCT

NUR IZZATI BINTI MOHD AZHAR

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

SIMULATION STUDY OF DRYING CHAMBER FOR MARINE PRODUCT

NUR IZZATI BINTI MOHD AZHAR

This report is submitted
in fulfillment of the requirement for the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

MAY 2019

DECLARATION

I declare that this project report entitled "Simulation Study for Drying Chamber for Marine Product" is the result of my own work except as cited in the references

Signature : ....

Name : ....

Date : ....

APPROVAL

I hereby declare that I have read this project report and in my opinion this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering.

Signature	:.	....
Name of Supervisor	:	....
Date	:	....

DEDICATION

To my beloved mother and father

ABSTRACT

Drying chamber is one of the application in the drying process. This application is widely used in various industries such as marine, agricultural product and others. In this study, a drying chamber for drying process on marine product was selected. It able to produce high quality and hygienic of products. The propose design of drying chamber consist of trays that arranged differently for each design. However, the distribution of velocity and temperature along the trays is not uniform due to different position of trays. This uneven dried of product contributed to the low quality of product. Therefore, there are five configurations of trays has been analyzed in order to obtain the best performance of uniformity velocity and temperature distribution within drying chamber. The prediction of the distribution of velocity and temperature in the chamber along the trays has been simulated by using Computational Fluid Dynamics (CFD) software which is Ansys Fluent 16.1. Operating and boundary conditions is determined based on the literature review. A validation is performed by comparing the data obtained from the CFD simulation with the data experimental from the literature review to ensure the correct method is used. Then, the simulation conducted with 2-Dimensional geometry that sliced into two halves and the sliced face is set as a symmetry plane for the analyzation in steady state condition. Nine points are plotted at the different coordinates on the trays to obtain range of velocity and temperature on each trays for all the configurations. The data obtained is tabulated in a table based on the designs. Comparison all the configurations based on the difference values for each points. Less differences for each points contributed the more uniformity of distribution. The best configuration of trays is determined by achieved both of uniformity and proposed as the best design. From the results obtained, it concluded that design (A) and (D) are choose as the best designs for uniformity that can be developed in future study.

ABSTRAK

Kebuk pengering adalah salah satu aplikasi dalam proses pengeringan. Aplikasi ini digunakan secara meluas dalam pelbagai industri seperti produk marin, pertanian dan sebagainya. Dalam kajian ini, ruang pengering untuk proses pengeringan pada produk marin telah dipilih. Ia mampu menghasilkan produk yang berkualiti tinggi dan bersih. Reka bentuk yang dicadangkan untuk ruang pengering terdiri daripada dulang yang diatur secara berbeza untuk setiap reka bentuk. Walau bagaimanapun, pengedaran halaju dan suhu di sepanjang dulang tidak seragam disebabkan kedudukan dulang yang belainan. Produk kering yang tidak sekata ini menyumbang kepada kualiti produk yang rendah. Oleh itu, terdapat lima konfigurasi dulang yang dianalisis untuk mendapatkan prestasi terbaik keseragaman pengedaran halaju dan suhu dalam ruang pengeringan. Ramalan pengedaran halaju dan suhu di dalam kebuk yang merentasi dulang telah disimulasikan dengan menggunakan perisian Perkomputeran Dinamik Bendalir yang merupakan Ansys Fluent 16.1. Syarat operasi dan sempadan ditentukan berdasarkan semakan kesusasteraan. Pengesahan dilakukan dengan membandingkan data yang diperoleh daripada simulasi CFD dengan data eksperimen dari kajian literatur untuk memastikan kaedah yang digunakan adalah benar. Kemudian, simulasi dilakukan dengan geometri 2 dimensi yang dipotong ke dalam dua bahagian dan permukaan pemotongan ditetapkan sebagai satah simetri untuk dianalisis dalam keadaan tetap. Semua profil pengagihan halaju dan suhu dalam ruang pengeringan telah dianalisis oleh simulasi CFD. Sembilan titik diplot di atas dulang pada koordinat yang berbeza untuk mendapatkan julat halaju dan suhu pada setiap dulang untuk semua konfigurasi. Data yang diperolehi disusun dalam jadual mengikut reka bentuk masing-masing. Perbandingan semua konfigurasi dilakukan berdasarkan perbezaan nilai untuk setiap titik. Semakin rendah perbezaan untuk setiap titik, semakin mudah untuk mencapai keseragaman. Konfigurasi terbaik dulang ditentukan dengan mencapai kedua keseragaman dan akan dicadangkan sebagai reka bentuk terbaik. Dari hasil yang diperoleh, ia menyimpulkan bahawa reka bentuk (A) dan (D) dipilih sebagai reka bentuk terbaik untuk keseragaman yang boleh dibangunkan dalam kajian pada masa depan.

ACKNOWLEDGEMENT

Praise to Allah Almighty who has guided me along the journey of completing this study successfully. Without His helps and guidance, I would never been able to achieve what I have accomplished now. Every person and detail that has come along of this work is a result of His blessings. A big thanks to my family especially my beloved parents, Mohd Azhar bin Mat Ali and Hasiah binti Muhammad have supported me along the way with their countless efforts to get me through this undertaking in providing me advices and moral support, financial supports and much more than that cannot be written down in a single page. They deserve my gratitude after God and His Messanger. My deepest gratitude gores to my supervisor, Dr. Mohd Afzanizam bin Mohd Rosli for giving me this opportunity to do final year project with him. He never hesitated to give me advice and guidance whenever I confronted problems. I am thankful for his patience and advice while leading me in this project. Finally I took this opportunity to thank all the people that I might have missed mentioning in this section for your time and support in producing this final year project. Thank you to everyone especially my friends for being there for me, your support will never forgotten and shall be rewarded by Allah.

CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

CFD	Computational Fluid Dynamic
FE	Finite Element
FD	Finite Dynamic
CAD	Computer-aided Design
2D	Two dimensional
3D	Three dimensional
UV	Ultra-violet

LIST OF SYMBOL

ρ	=	Density of fluid
k	=	Turbulent kinetic energy
ε	=	Rate of dissipation
μ	=	Dynamic viscosity
μ_t	=	Turbulent viscosity
S_k, S_ε, S_h	=	User-defined source terms
G_k	=	Generation of turbulent kinetic energy due to the mean velocity gradients
G_b	=	Generation of turbulent kinetic energy due to buoyancy
σ_k	=	Turbulent Prandtl numbers for k
σ_ε	=	Turbulent Prandtl numbers for ε
C_p	=	Specific heat capacity at constant pressure
T	=	Temperature
Pr_t	=	Prandtl number
E	=	Total energy
p	=	Pressure

CHAPTER 1

INTRODUCTION

1.1 Background

Drying process are widely used in several of industries especially in food industry which have a huge impact to this industry (Fuente-blanco and Sarabia, 2006). Drying is defined as a mass transfer process resulting in the removal of water moisture or moisture from another solvent, by evaporation from a solid, semi-solid or liquid (hereafter product) to end in a solid state (Arun, 1995; Noh, Mat and Ruslan, 2018; Pravin et al., 2017). The purpose of drying process is to lessen the amount of water content in the product to make sure the environment become inappropriate for proliferation of microorganisms (Putra and Ajiwiguna, 2017; Noh, Mat and Ruslan, 2018). Therefore it can prolong the shelf-life of products and furthermore can decrease the expenses of transportation and capacity (Zhang and Fournier, 2017).

Drying process is important as the most effective and practical method for food preservation to reduce losses especially in produced marine product such as salted fish, anchovies and others since it can be safely stored for longer period of time. This method widely used at the state whereas nearby with the sea such as Terengganu, Sabah and others as their business.

Normally, people at the Malaysia are using open sun drying method to dry their marine products such as salted fish, small prawns, anchovies and others. However, this method has several disadvantages such as exposed to the dust, air pollution or the case of pest or insect attacked on the products and lack of hygienic (Pravin et al., 2017; Rizal and Muhammad, 2018; Misha *et al.*, 2014). Besides that, this method also not effective when the condition is not conducive such as rainy season, therefore artificial drying process is needed to be develop (Pravin et al., 2017; Putra and Ajiwiguna, 2017). It will took a long period of drying rate during cloudy weather and limitations of time during rainy day. Therefore, the products of drying become low quality of products (Pravin et al., 2017; Jamila et al., 2018).

There are many advantages for using this method which are the quality of the products is higher due to sun drying because it can prevent the products from the contamination and the case of pest and insect attacked on product is reduced (Pravin et al., 2017; Anna et al., 2014; Jamila et al., 2018).

Other than that, solar drying system can save energy, consume less time, use less space, enhance the quality of product, improve the efficiency of the drying process and save the environment (Misha and Yusof, 2018).

The characteristics of great drying chamber are the uniformity air flow distribution in the chamber to make sure uniform drying of product (Misha *et al.*, 2014; Paper, 2015; Jia *et al.*, 2016; Zhang and Fournier, 2017; Zhang and Long, 2017). Drying uniformity can be enhanced when air flow distribution throughout the drying chamber in a good condition (Misha *et al.*, 2013, 2014). The purpose of uniform air flow distribution is to improve the efficiency of drying chamber so that the homogeneity of product being dried become improved (Misha *et al.*, 2014). In order to get uniformity air flow distribution, the arrangement of tray dryer must be improved. The alternate arrangement of dryer tray position was embraced to guarantee that all the plate are exposed straightforwardly to drying air and to enhance airflow distribution throughout the drying chamber (Paper, 2015). Other than that, air temperature and velocity also influenced in the drying process. The range of air velocity in the drying process is 1.5 m/s to 3 m/s (Putra and Ajiwiguna, 2017). However, sometimes the air velocity is depended on moisture content of product. The evaporation rate from the product will be rise when the temperature of air increased. So that, the capacity of water vapour become larger (Misha et al., 2015).

There are many components in the drying chamber as shown in Figure 1.1. The main components of the drying chamber are:

1. The cabinet / chamber
2. Dryer tray
3. Heat Source
4. Inlet and outlet

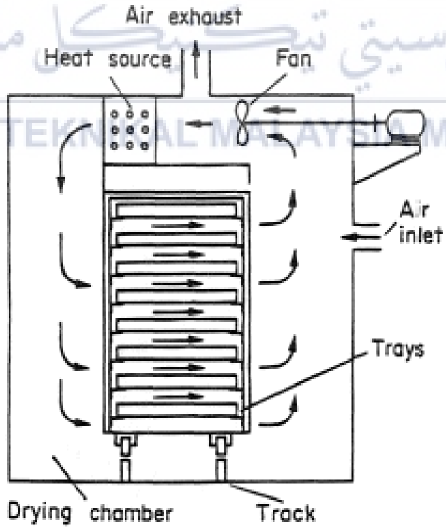


Figure 1.1 The example of drying chamber (After Sloan ,1967)

The most suitable method to evaluate that drying chamber is simulated by Computational Fluid Dynamics (CFD) due to the expensive and difficulties of the measurement for drying parameters if doing an experiment. Nowadays, the CFD simulation is used extensively because it's capable to solve the equations for many cases such as conservation of mass, momentum and energy using numerical methods to predict drying parameters which are temperature, velocity and pressure profiles (Misha *et al.*, 2014; Paper, 2015). The users of CFD simulation become increase throughout the year. They become more realized that used CFD simulation is easier than experimental. This is because it is very appropriate to use for estimation of drying parameters in the drying chamber compared to the experiment which are expensive, difficulties and consume a lot of time to install a few equipment (Paper, 2015). Besides that, by using this software, all the parameters can be adjusted freely in orders to get the best solutions.

In this study, the prediction of drying time of a marine product to dry are simulated and compared by using Computational Fluid Dynamics (CFD) analysis which is ANSYS. The study has involved two-dimensional fluid domain in steady state simulation. This state has study about the predicted air flow and temperature distribution of domain. The profiles of air velocity and temperature can be studied for analysing the uniformity distribution in the chamber.

1.2 Problem Statement

Drying process by sun drying is the most popular method used by Malaysian citizen to dry their foods especially in marine product such as salted fish, anchovies and others as food preservations. However, this method has many limitations caused of weathers at the Malaysia is not hot throughout the year since Malaysia located at the equator of the earth which are hot and rainy weather. Therefore, it took a long period of drying during cloudy weather and the product does not dry well that caused the declined quality of product and reduced time to store. Basically, sun drying needed a large space when involved a lots of quantities of dried products. So that, the products exposed the product to the contamination of dust, pollution and insect. Therefore, a drying chamber with technology must be develop to overcome this problem. However, various of drying chamber in the industries also have limitations such as the sources of heat likes fan or blower installed in the infrared drying system causing the inaccurate distribution of temperature in the chamber as well as limiting amount of product to be dried. The influence of geometry of drying chamber such as the locations of inlet and outlet and also the arrangement of trays in the chamber will also affecting the rate of drying. Furthermore, the not suitable temperature with the moisture content of product will caused it either over dried or not-fully dried. So that, the texture and freshness of product will be damaged. So, the product will lack of quality to be in food industry. Therefore, in order to improve the performance of drying chamber in industries, simulation by using Computational Fluid Dynamics (CFD) can be conducted in order to identify and predict the best design of drying chamber that can fulfil all the issues related the performance of drying chamber especially in term of uniformity drying rate.

1.3 Objectives

The objectives of this study are:

1. To determine the best temperature distribution in drying chamber using Computational Fluid Dynamic (CFD) Simulation.
2. To determine the best velocity distribution in drying chamber using Computational Fluid Dynamic (CFD) Simulation.
3. To propose the best configuration of trays using CFD simulation.

1.4 Scope of Work

The scope of works for this study are:

1) Specifically for the marine product.
2) Verification and validation results of existing drying chamber from publish journal.
3) The simulation will be carried out by using CFD simulation which is ANSYS Fluent 16.1.
4) The simulation conducted in 2 dimensional geometry.
5) To perform steady state condition of analysis.

CHAPTER 2

LITERATURE REVIEW

2.1 Definition of Drying

Drying has been used widely as the method for food preservation and one of the integral part of food processing. This method has been discovered and carried out since the early recorded history of human civilization (Henry, 2018). Drying process is defined as the process of the mass transfer that resulting in the removal of water content or moisture content from another solvent from solid, semi-solid or liquid (hereafter product) by evaporation to end in a solid state. The proper drying results in a concentrated food source which is can be handle easily and distribute due to weight and volume reduction of the product.

The purpose of drying process is to lessen the amount of the water content in the product especially in marine product to make sure the environment become inappropriate for the growth of microorganisms such as bacteria, insects larvae and others. The activity of this microorganisms and insects is inhibited in an environment where the equilibrium relative humidity is below 70%. Therefore, it can prolong the shelf life of products and can reduce the expenses of transportation and capacity. So that, the product can be safely stored for a long time. The process is very important as the most effective and practical method for food preservation to reduce losses especially in produced dried squid since it can be safely stored for a longer period of time. Therefore, the preservation of the merchandise (dry product), drying will bring on savings in storage and transport of foods will be reduced as a result of the reduction in the amount of weight and bulk that happens throughout most ways of drying (Rizal and Muhammad, 2018).

Drying process is affected by many factors such as air velocity, temperature and pressure to ensure all the process worked well. The most important factor that affected the drying rate is temperature of drying air (Krokida et al., 2003). Babalis et al. state that air velocities in drying process in greater that 2 m/s has no significant effect on the drying rate. In the conclusion, the most significantly affected by the air temperature is drying kinetics.

Putra and Ajiwiguna (2017) have conducted an experiment to analyse the effect of the air temperature and velocity in the process of drying. Based on the result obtained, it can be concluded that the effect of temperature is less significant at high air velocity. The basic principle of drying process is to transport water vapour from wet object to the air. Velocity of air is proportional to the mass flow rate of air. At higher mass flow rate, the capacity increase since a lot of amount of air.

2.2 Method of Drying

2.2.1 Natural Drying

In the history, the natural resources such as the sun and the wind were used in order to remove the water or moisture from the food likes fruits, meats, grains and herbs (Ahmed *et al.*, 2013). Natural drying as shown in Figure 2.1 is the method of drying where the product has been exposed to the sunlight directly to reduce the moisture content of product as food preservation also known as sun drying. So that, it can be stored safely in longer time of period. In the developed countries, the most common method for drying is open sun drying as the simplest and most economical method of food preservation. Open sun drying also known as traditional drying method before technology has been developed in the world.

William L. Kerr (2018) stated this is one of the oldest methods in drying process where the products are set and arrange on trays at the outside under the sun to dry the products. Basically, to achieve a successful drying, it is required the amount of air temperature is above 85 ^{o}F (29.44 ^{o}C) and the humidity is less than 60% (William L. Kerr, 2018). After technology has been discovered, the industrial drying like solar drying has been introduced to make drying process become easier. However, this method has several disadvantages and has many limitations that caused the quality of product become low.

Figure 2.1: Natural Drying (AP Mnkeni, 2016)

Firstly, the disadvantage of open drying process is not suited and effective to all regions due to the condition is not conducive and limited caused adverse weather (William L. Kerr, 2018) such as during rainy season. So that drying process will required a longer period of time during cloudy weather (Jamila et al, 2018). The product will not fully dried because of the lower temperature. This problem will be contributed to the growth of microorganisms that caused the product cannot be safely stored for a long time. Besides that, the product become low quality of product due to this weakness.

Other than that, traditional open drying is needed a big scale of space or areas when involved a lots of quantities of product during the drying process. This method cannot be developed at the limitations of areas. It will contributed difficulties to the people to find out a large areas to do this method. When the area is limited, the quantity of product also will be limited. This is caused the waste of time and longer period of time is needed to get a lot of dried product. Besides that, due to this process has been developed at the opened areas, the product will be exposed to the contamination of dust, pollution and insects. So that, the quality of product become low and not suitable to make as a food to avoid food poisoning.

2.2.2 Industrial drying

Due to the technology nowadays, the method of drying process has been improved and functionally throughout the year. Therefore, drying chamber that assisted with the solar as the energy source has been developed. This known as solar drying as shown in Figure 2.2. In developed solar drying, the most important factor to consider is the accuracy of temperature. The calculation of heat losses by drying chamber is quite challenging and tough to find numerous variable accurately (Salah et al., 2007). Basically, drying rate by solar drying is quicker than open sun drying. It also protected the product from the contamination of dust and animals or insects attacked because the drying chamber has been designed fully covered by suitable materials. William L. Kerr (2018) state that the foods which is too much exposed to the sun can harm and damage due to the UV-sensitive.

Pravin et al., (2017) have done an experiment on the drying agricultural products by design and constructed the solar dryer. The solar dryer has been designed with solar collected that function as air heater with the baffles. This drying chamber also contained rack of four net tray that assimilated together. The design based on the geographical location. From the study, they found that many advantages can be obtained from this method which are can rise the quality of products. This is because solar dryer can be prevent from contamination and reduced the case and insect attacked on product due to safely protected in the chamber. Other than that, the time of drying can be reduced and the product dried uniformly.

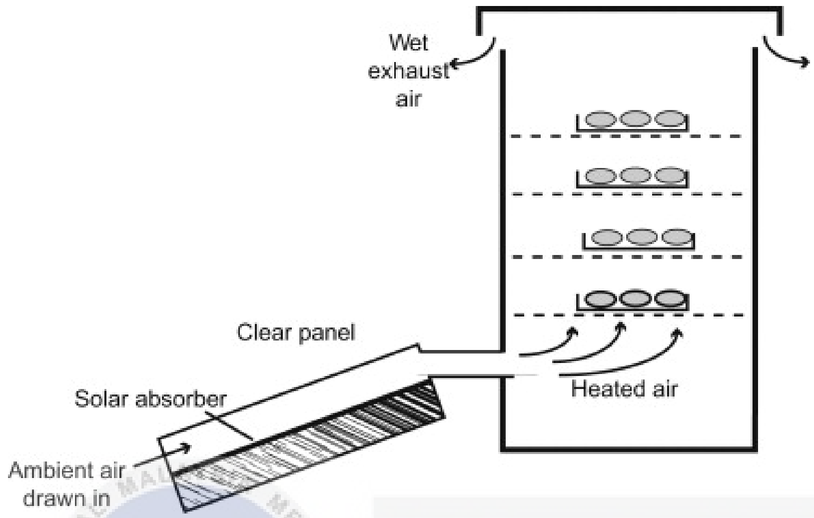


Figure 2.2: General Model of Solar Drying (William L. Kerr, 2013)

2.3 Potential in Development Drying Chamber at Malaysia.

As one of the developed country, Malaysia also has been developed a few model drying chamber for drying process. Malaysia usually received around 400-800 W/m^2 of solar radiation per day. This result is received after a few investigation and analysation at several main towns at the Malaysia (Bashria et al., 2004). Basically, Bashria et al., (2004) state that the parameters that affecting of solar drying projects in Malaysia involved the air flow, temperature, global solar radiation and others. Since the location of Malaysia in the world is at the equator which has hot and cold season. So that it suitable to develop drying chamber at the Malaysia due to able received high radiation per day.

Ahmad Fudholi et al., (2015) has conducted an experiment to investigate the drying time of agricultural and marine products by different types of dryer according the sun radiation at Malaysia. Based on the Table 2.1, there are comparison the drying time of different types of products.

Table 2.1 Comparison drying time between agricultural and marine.

(Ahmad Fudholi, 2015)

Products			Herbal tea / Green tea	Red Chillies	Salted Fish	Seaweed
Characteristics	Moisture content (%)	Initial	87	80	64	90
		Final	54	10	10	10
	Weight (kg)	Initial	10.03	1.25	0.22	-
		Final	2.86	0.25	80	-
	Temperature (^oC)		50	50	50	50
	Flow rate (kg/s)		251.67	0.07	0.0778	0.05-0.12
	Drying time (hr)		12	31	8	14

In Malaysia, marine products have been processed about 64% by fish manure and fish meal as the most two largest processed marine product as food preservations. M.S. Ismail (1981) state that carargidae fishes took about 9.5 hours to dry and obtain the moisture content of fish as in market.

2.4 Differences between Drying on Agricultural and Marine Product

Agricultural and marine products is most popular products that has been used to dry as a food preservation. Most of the agricultural and marine products require drying process as one of the method to preserve and improve the quality of the final product. This product become widely used because as a main sources at the developed country. There are many marine products has been produced to dry products. The drying rates of certain products depended on the moisture content of products. Generally, the moisture content of product increase, the drying rate of product also increase. However, this problem can be overcome due to the high of amount of air velocity flow into the drying chamber. The design of drying chamber is also one of the factors of drying rate.

The agriculture and marine products have different moisture content depends on the types of products respectively. The moisture content and weight are the most important factor in drying process. From Figure 2.3, it can be concluded the relation between moisture content and weight of marine product affected drying time. The relationship between moisture content and drying time is directly proportional. Therefore, when the moisture content increase, drying time to dry also increase as shown in the Table 2.1.

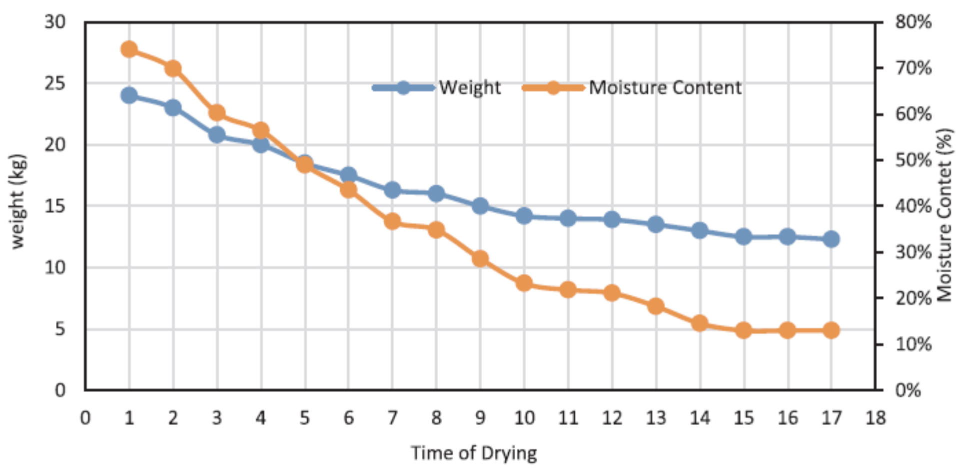


Figure 2.3 Changes in weight and moisture content of fish during the process of drying. (Hamdani et al., 2018)

As mentioned by Ganesh et al., (2011) that one of the marine product which is fish is very important foodstuff. This is because fish has high value of protein content and nutritional for the health of human body. Therefore, human encouraged to store for a longer period of time those marine products as their foods by drying method as the most effective and easier method. The research was held at India as a one of the leaders of world in aquaculture and fisheries with 8100 km coastline yielding 6.26 million tonnes of fish per year. A few methods of solar drying have been conducted to compare the best method in the drying process. Table 2.2 shows that the quality and other parameters of drying on Bombay duck and prawn dehydrated by using a several of drying methods.

Table 2.2: Quality and other drying parameters of Bombay duck and Prawn dehydrated using various drying methods (Ganesh et al., 2011)

Dryer	RR		Shrinkage, %	Density, kg m^{-3}		ΔE	Hardness, g	OA	E_{sp}, kJ kg^{-1}	D_{eff} $\times 10^9$, $m^2 s^{-1}$	DT,h
	25 °C	100 °C		True	Bulk						
				Bombay duck							
OSD	1.75^a	0.85^a	68.24^a	612.18^a	112.12^a	33.23^a	1744.3^a	5.12^a	0^{a*}	0.31	17.33
SCD	2.55^b	1.13^b	56.24^b	700.12^b	122.82^b	21.33^b	971.68^b	6.86^b	2098^a	1.01	10.00
HAD	2.45^b	1.12^b	55.15^b	760.52^c	128.18^b	19.29^b	1121.52^c	6.95^b	18950^b	1.51	7.50
FD	2.85^c	1.15^b	5.32^c	400.23^d	65.23^c	5.34^c	892.61^d	8.87^c	21200^c	0.70	9.83
				Prawn							
OSD	$2.56^{a,c}$	2.25^a	65.14^a	410.25^a	85.21^a	38.58^a	1443.33^a	5.87^a	0^{a*}	0.08	8.50
SCD	3.2^b	3.01^b	45.24^b	422.23^a	87.81^a	22.18^b	1218.22^b	6.66^a	959^a	0.21	4.00
HAD	$3.05^{a,b}$	2.85^b	43.23^b	425.41^a	89.19^a	20.58^b	1040.67^c	6.89^a	8081^a	0.31	5.00
FD	3.65^c	3.35^d	6.41^d	315.35^b	50.12^b	6.55^d	516.00^d	8.72^b	13507^b	0.10	6.00

S.H. Sengar et al., (2009) studied that the drying on salted fish and salted prawns in low cost solar dryer. The comparison between open sun drying and solar dyer has been studied based on the time required for moisture content reduced from 75% to 16%. Based on the result, salted fish needed 8 hours in order to dry prawns up to 16.15% simultaneously while unsalted fish required 15 hours to reach the moisture content until 15.15%. The graph in the Figure 2.4 shows the relation of variation of drying rate and moisture ratio with time of the products.

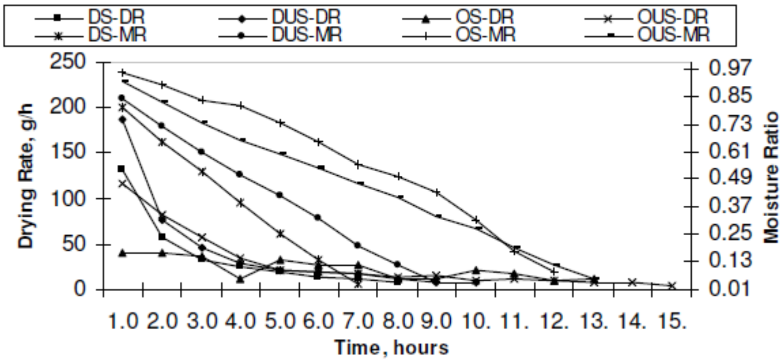


Figure 2.4 Variation of drying rate and moisture ratio with time (S. H. Sengar, 2009)

Zayana et al., (2013) have conducted an experiment on analyse the performance of the solar drying system for marine product of Oman. This project developed and presented new design of solar drying system that suitable to perform drying process for variety types of fishes as foods preservation. Method by solar drying is classified as the best method in produced high quality dried products compared to the sun drying. The design has been fabricated as considered two main factor of developed drying chamber which are the condition of heat and velocity of air flow. Both factors are most relevant things to conduct drying process in order to obtain the accurate drying time. Based on the result from the experiment conducted, it presented the reduction weight of sample fishes due to the time. After experiment, it proved that solar drying system is hygenity and effective in heating process and also able to reduce the mass of fishes up to 30%.

Table 2.3 Data temperature of chamber and mass of fish sample without salt with respect to time. (Zayana et. al. 2013)

Day	Time	Solar Dry without salt			
		Temperate C°	Sardine (g)	Skater bream (g)	Jake mackerel (g)
1	6:00	22	12	400	500
	9:00	31	12	400	480
	12:00	56	11.5	400	480
	15:00	43	11.5	370	460
	18:00	36	11.5	360	450
2	6:00	21	11.5	360	450
	9:00	40	11	350	440
	12:00	58	10	350	400
	15:00	48	10	310	370
	18:00	36	9.5	300	340
3	6:00	19	9.5	300	340
	9:00	45	9	270	300
	12:00	54	8	230	260
	15:00	49	8	210	240
	18:00	39	6	200	220

Table 2.4 Data temperature of chamber and mass of fish sample with salt with respect to time. (Zayana et. al. 2013)

Days	Time	Solar Dry with salt			
		Temperate C°	Sardine (g)	Skater bream (g)	Jake mackerel (g)
1	6:00	22	12	400	500
	9:00	31	12	400	490
	12:00	56	11.5	400	490
	15:00	43	11.5	390	480
	18:00	36	11.5	380	480
2	6:00	21	11.5	380	480
	9:00	40	11	360	470
	12:00	58	10	360	460
	15:00	48	10	350	420
	18:00	36	9.5	350	400
3	6:00	19	9.5	350	400
	9:00	45	9	350	380
	12:00	54	8	320	360
	15:00	49	8	290	320
	18:00	39	7	260	300

2.5 Drying Model Development and Simulation

The process of drying of the product is depending on a few factors such as the air velocity, temperature and pressure. There are plenty of studies have been done on the food drying process especially in agricultural products. Many drying chamber models were developed and simulated so that the dried products quality could be predicted and controlled well. The design of tray dryer is widely used in variety of applications because of the simple design and able to dry products at high condition. Basically, the simulation studied on uniformity of air flow distribution between tray dryer, the influence of air velocity due to the air drying time and many. The simulation has been simulated to get the optimal design of drying chamber for the high quality of product.

Pengfei et al., (2017) studied the effects of the conveyor positions on airflow distribution in three types of belt dryers that are simulated and compared by using Computational Fluid Dynamics (CFD) analysis. Airflow velocity in a drying chamber has been predicted by utilized CFD simulation. The governing equations of the numerical model using CFD software which is ANSYS FLUENT version 15.2 has been solved by using finite volume method. Based on the result of the simulation, the conveyor positions is significant effects on the rate of airflow in two layer belt dryer. Besides that, the simulation also shows three different profiles of airflow for three dryer models. Therefore, an optimal dryer gas is chosen as the model for the future analysis of air velocity on feed surface. The result shows that the airflow velocity in type B dryer has a better uniform pattern compared to the type A dryer and type C dryer. This is because the streamlines that crossing the conveyors are distributed uniformly.

Suhaimi et al., (2013) has developed a new design of the tray dryer for agricultural product. They studied on the prediction of drying uniformity the design by using

Computational Fluid Dynamic (CFD) simulation. The 3D simulation is done in this study as represent the actual model of tray dryer. The uniformity of drying is analysed based on the temperature and velocity profile, streamline and velocity on each tray dryer. Basically, the temperature is considered as a constant for all the trays. Based on the result, the average air velocity at the tray 1, 7, 8 and 15 are much higher than other trays. Then, the rest of tray dryer are uniform to each other. Based on the simulation result, streamline colour shows that the area nearest to the inlet and outlet has higher air velocity. The upper tray gave the highest average of air velocity since it closes to the inlet and outlet. The velocity of air along the tray is low due to the distributions of air to all the trays at the particular levels. In the conclusion, the average air velocity for this model is about 0.38 m/s.

Table 2.5: Previous Study in Drying Chamber

Author	Objectives	Method	Boundary Conditions	Conclusion
Arina et al., (2018)	To simulate and analyse the effect of product arrangement and different operating condition based on the temperature and air flow distribution inside drying chamber of newly developed solar dryer.	CFD simulation by ANSYS Fluent version 14	Inlet velocity : 3m/s Solid geometry : 3d tetrahedral mesh Inlet temperature : 50 ^{o}C	CFD able to simulate and predict the most optimum condition inside drying chamber
Misha, S. et al. (2014)	To predict and identify the uniformity of drying in development of new design of the commercial tray dryer for agricultural product.	Simulation by CFD, Ansys	Inlet 1 : 3 m/s , 0.5843 kg/s Inlet 2: 3 m/s, 0.029215 kg/s Outlet : Gauge pressure = 0 Turbulent flow	Good air flow contributed uniform drying rate and improve quality of product.
Misha.S, et. al., (2003)	To identify performance of tray dryer system between porous and solid product.	Simulation by CFD, Ansys	Inlet 1 : 8.8 m/s, 1.715 kg/s Inlet 2 : 0.8575 kg/s, 8.8m/s Temperature : 75^{o}C Outlet : Pressure gauge = 0	Arrangement of tray is important to improve drying rate which is influenced by average air velocity on tray.

2.6 Computational Fluid Dynamic (CFD) Simulation

Computational Fluid Dynamic (CFD) simulation is one of the field of fluid mechanics in order to solve and analyse problems which is involved the fluid flows by using numerical methods and algorithms. Besides that, CFD simulation is a computer-based tool for simulating the behaviour of systems involving fluid flow, heat transfer and other related physical processes. It works by solving the governing equations of fluid flow over a region of interest with specified conditions on the boundary of that region.

CFD is used to obtain the details about the flow field such as shear stresses, velocity and pressure profile of a computational domain. Besides that, CFD can handle the laminar flows easily but impossible to solve the turbulent flows without invoking turbulent models (Yunus and John, 2010).

There are plenty of studies that used Computational Fluid Dynamics as a tool to run simulation of the study. This method has been verified as the best and suitable method to evaluate the models especially drying chamber due to the expensive and difficulties of the measurement for drying parameters if conduct an experiment. Besides that, an experiment also took a longer period of time to get the result. Basically, this simulation is able to solve the equations for many cases such as conservation of mass, momentum and energy using numerical methods to predict drying parameters such as air velocity, temperature and pressure profile. Therefore, it regularly used when involved fluid flow investigation.

Francisco et al. (2009) have modelled three dimensional geometry of beef carcass. In order to decrease the time of computational, there are three step of methods that have been used in this simulation to make sure the process of heat and mass transfer simulated simultaneously. The step involved is the simulation of flow field in steady state as a first step. The second step is needed to calculate of the heat and mass transfer. Lastly, the thirdstep method is simulation of the meat carcass on process of heat and mass transfer simultaneously. All these steps only will be valid when the heat and mass transfer is only constants which is under assumptions. The simulation is done by using three different ways which are Computational Fluid Dynamics (CFD), Finite Element (FE) and Finite Dynamic (FD) to compare the best result of simulation. The comparison of the result between this three methods as shown in Figure 2.5 below. From the result, they concluded that CFD model shown that the result most accurate and better with the experiment result than other two models. CFD method is best heat load predictions can be obtained compared to the FD or FE. Besides that, CFD also can predict the centre and temperature of surface of the leg and shoulder very well which are CFD has been predicted the local temperature variations up to 9 ^{o}C and the variations of water activity up to 9% around the beef surface. These result were obtained from the CFD simulation.

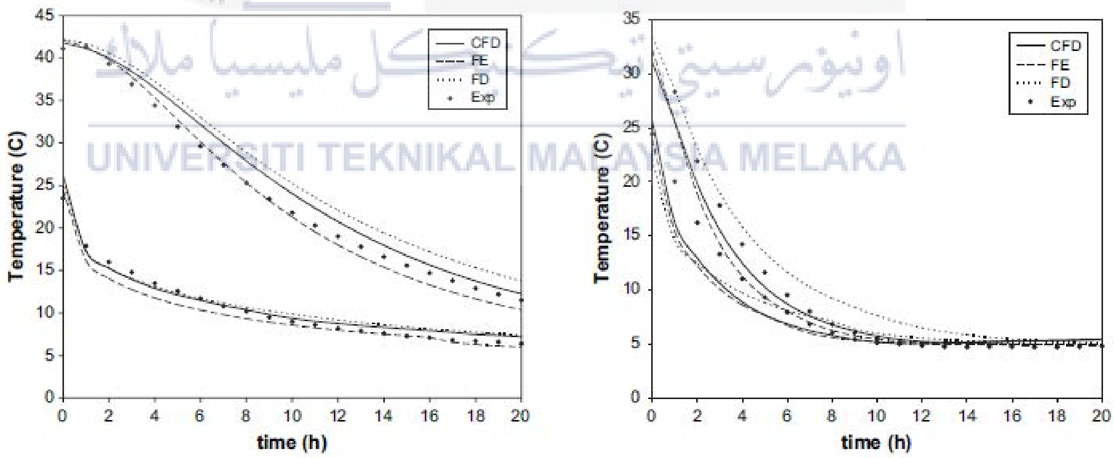


Figure 2.5: Comparison between CFD, FE and FD method in predicted temperature of drying against time of drying.

(Francisco et al. 2006)

Jung et al., (2008) have done a two dimensional simulation of silica gel drying as shown in Figure 2.6 using computational fluid dynamics to analyse the mass and heat transfer of moisture and water vapour in a humid-air dryer. Two different designs of the humid air dryers were used as the computational domains to compare their performance in moisture drying. The mathematical model was developed using the CFD software which is ANSYS FLUENT. To simplify the numerical calculation of liquid-gaseous multiphase problem, they have made some assumptions on the models, which state that the gaseous fluid is an ideal gas mixture, the flow is turbulent, and the model is non-isothermal and in unsteady state, there are no penetration of gaseous-fluid into the silica gel and the silica gel is uniform in size during drying. In the simulation, all the governing equations except the momentum conservation were solved using the user defined functions of the Fluent solver.

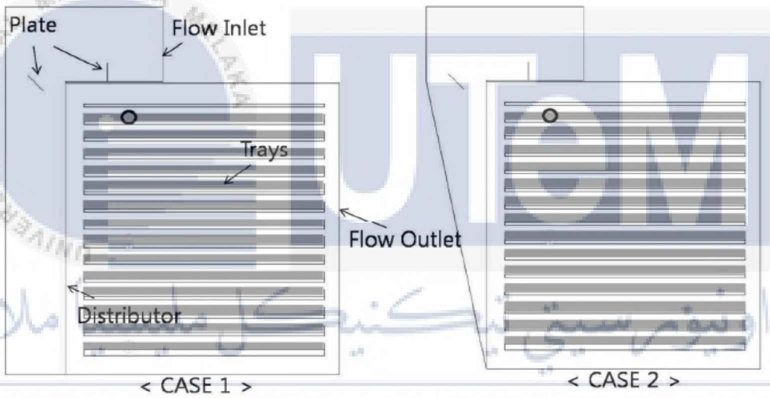
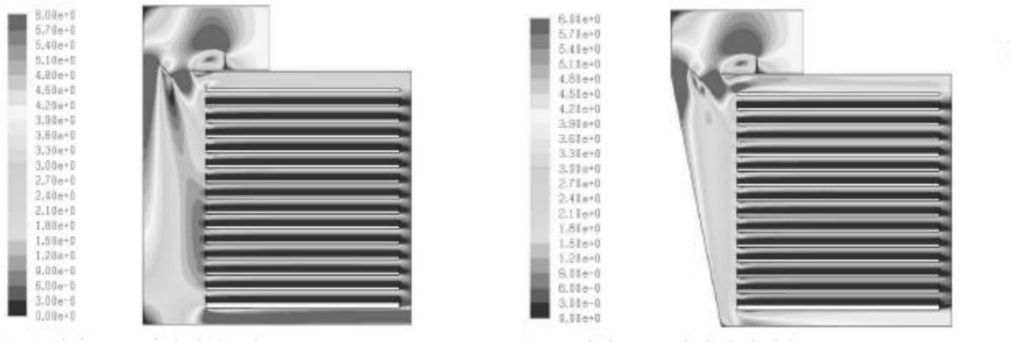


Figure 2.6: The 2D domain of silica gel drying

From the simulation result, the drying kinetics were estimated with the change of solid moisture content and temperature with respect to time elapsed, which is the drying curves. The result shows that the design in case 2 dryer has higher performance due to increase in air velocity, which could be observed in velocity contours of both cases. The velocity distribution in case 1 is not uniform compared with case 2 and there is a dead zone found in the left bottom of the case 1 dryer where the velocity is less than 0.3 m/s.



(a) Case 1 (b) Case 2

Figure 2.7: The contour of velocity profiles in the drying chamber for (a) Case 1 and (b) Case 2.

Computational Fluid Dynamic (CFD) is the most commercial code that can be used to conduct the simulation of air flow pattern and the distribution of temperature (Paola et al., 2006). Paola et al., (2006) have discovered about the two dimensional and three dimensional computational domain by CFD simulation. Two dimensional simulations is obtained from the cross-sectional of the model i.e. vertical display cabinet included the compartment of external ambient as computational domain while 3D computational domain involved in huge portion of external ambient both in transversal and longitudinal direction. The simulations consists two method either steady or transient state.

In developed drying chamber, the best ideal conditions of the drying chamber must be carefully identified in order to obtain the best design of drying chamber. These conditions is verified and simulated by using Computational Fluid Dynamics (CFD) Simulation which is ANSYS Software.

2.7 Advantages and disadvantages of Computational Fluid Dynamics (CFD) Simulation

Computational Fluid Dynamics (CFD) Simulation is one of modern analysis method that can be used by engineers to solve and simulate the characteristics of fluid flow by using high speed computers. So, that it can make engineer's especially designer engineers to analysis and simulate their models easily. There are many advantages in using Computational Fluid Dynamics (CFD) which are:

1. The behaviours and physical boundary conditions of computational domain can be more specified and precise or any impact can be deal separately.
2. Result from the CFD simulation is more accurate and faster than physical modelling. This is because CFD able to manage any complexity of model geometry caused of these three factors which are:
    i. Generally, the process of meshing of geometry model can be built quickly than model scale can be manufactured.
    ii. Parts of CFD model that can be copied and pasted for similar or symmetrical duct systems while all the parts of the physical mode must be fabricate separately.
    iii. After the CFD model has been built, the simulation can be run simultaneously on the different computers. So that a few model geometry can be simulated in one time but the real model investigated one by one and took longer period of time.
3. All the parameters of flow is compiled together which are not accessible in experiments.

4. For the starter of scheduled process, a few different types of prototypes are simulated in order to faster obtain the information for next planning.
5. Problem solving can be more understand through CFD rather than experiment caused all the fluid flow of parameters can be observed.
6. The value of CFD simulation is very cheaper than experiments which needed a lot of money when needed to repeat the experiments until success to get the correct result.

Other than that, usually all the software is not formed perfectly due to some errors same as CFD simulation. There are several disadvantages of CFD simulation occurred during analyzation which are:

1. There will be occurred some errors during the simulations because of the flow of models and the boundary conditions is too much simple.
2. Potential distrust due to the several value of calculations per cell caused the errors of interpolation in results.
3. In larger or bigger size of models, the time of computation may took a longer of time or should be extend for a certain time.
4. It also may contributed higher costly when involved the wrong consulting rather than experiments.

However, all these disadvantages can be reduced by do more practiced on this simulation. It will be easy to the user who has experienced and good in deal with the step of simulation program to get great result.

Suhaimi et al., (2013) also state that the benefit on using CFD simulation in their research on prediction of drying uniformity in tray dryer system. They managed to use this software due to able to solve all the equations such as conservation of mass, momentum and energy by using numerical methods to predict suitable air velocity, temperature and profiles of pressure in the drying chamber caused considered all the parts eventough integral part of geometry design. Therefore, the airflow and performance of drying chamber can be assumed.

2.8 Basic Governing equations for CFD simulation

The conservation of the mass, energy and momentum in drying air resulted in the continuity, Navier- Stokes and equation of energy, respectively (Amanlou and Zomorodian, 2010). For the turbulent model, the turbulent kinetic energy, k, and its dissipation rate, ε, are calculated from the following equations of the transport (Misha et al., 2015)

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] G_k + G_b - \rho\varepsilon - Y_M + S_k \quad \ldots (1)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \ldots (2)$$

The following equation is mentioned for modelling of the convective heat and mass transfer in the k-ε models (Misha et al., 2015):

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i}[u_i(\rho E + p)] = \frac{\partial}{\partial x_i}\left[\left(k + \frac{C_p \mu_t}{Pr_t}\right)\frac{\partial T}{\partial x_i} + u_i(\tau_{ij})_{eff}\right] + S_h \quad \ldots (3)$$

2.8.1 Moisture Content

The percentage of moisture content is determined by using the following formula:

$$\text{M.C (wet bulb) \%} = \frac{(W_1 - W_2)}{W_1} \text{ x } 100 \quad \ldots (4)$$

$$\text{M.C (dry bulb) \%} = \frac{(W_1 - W_2)}{W_2} \times 100 \qquad \ldots (5)$$

Where W_1 = Weight of sample before drying

W_2 = Weight of sample after drying

2.8.2 Drying Rate

The drying rate is determined by using the following formula:

$$\text{Drying rate (D.R.)} = \frac{\Delta W}{\Delta T} \qquad \ldots (6)$$

Where ΔW = Weight loss in one hour interval

ΔT = Difference in reading of time (h)

CHAPTER 3

METHODOLOGY

3.1 Introduction

In order to systematically satisfy the research objectives, the following research methodology is applied. The project is started on searched a few related journal to gain related information to conduct the project. As the starter of the project, a lot of effort is put on the literature review on the drying chamber simulation by Computational Fluid Dynamics (CFD) especially in marine product industry. After that, the project proceeds with the study about the simulation that contain drawing and meshing of the drying chamber. Then, CFD simulation is performed by using ANSYS Fluent to completed mesh and fabric model. The percentage of error is calculated to verify and validate the correct method has been used. The project is continued with the designing of the drying chamber model. The behaviour and compartment in the drying chamber is studied to improve it become better for future usage. The suitable and accurate parameter related to the project is determined. The detail flow of methodology for this project as shown as in Figure 3.1 below:

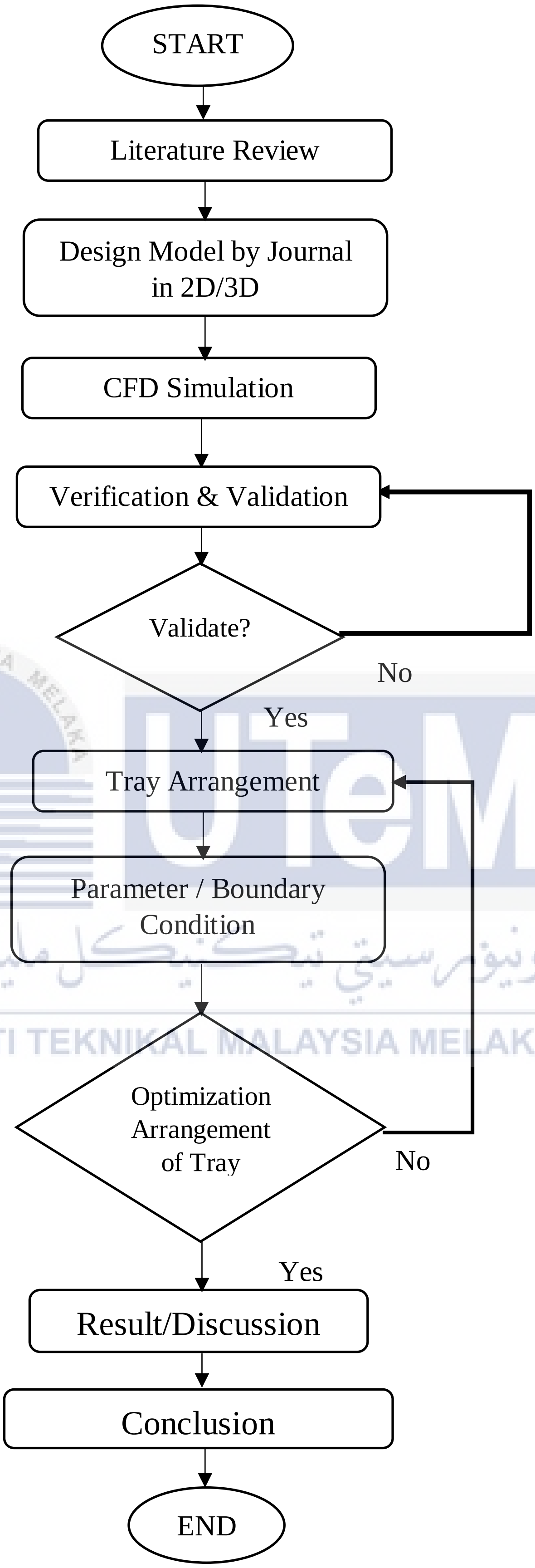


Figure 3.1 Flow Chart of the Methodology

3.2 Problem Identification

Before performing a CFD analysis, it is importance to understand and identified the main issues in developing drying chamber and input the problem of fluid flow to the CFD program based on the literature study from the previous researcher. For the first step of the study, the goal of modelling is defined based on the previous researcher. In this study, the performance of drying chamber was determined and analysed by using two-dimensional (2D) CFD simulation in steady state condition.

3.3 Pre-processing

3.3.1 Geometry Creating

The following step is to create the geometry of the computational domain, i.e. the drying chamber with trays that is needed to be analysed. Two dimensions approach is chosen rather than the three dimensions due to the high requirement of computer resource. 2D geometry is easier to analyse because less time consuming in calculation every single nodes of geometry. For the analysis, the drying chamber was sliced into two parts in order to get efficient of computational. Then, the sliced faces was taken as a symmetry plane for further analysis. Figure 3.2 shows the geometry of the symmetry face of the drying chamber with trays. The dimension of the drying chamber is predicted based on the a few model in the literature study. Therefore, in this project, the dimension of the drying chamber was set to 1150 mm x 625 mm.

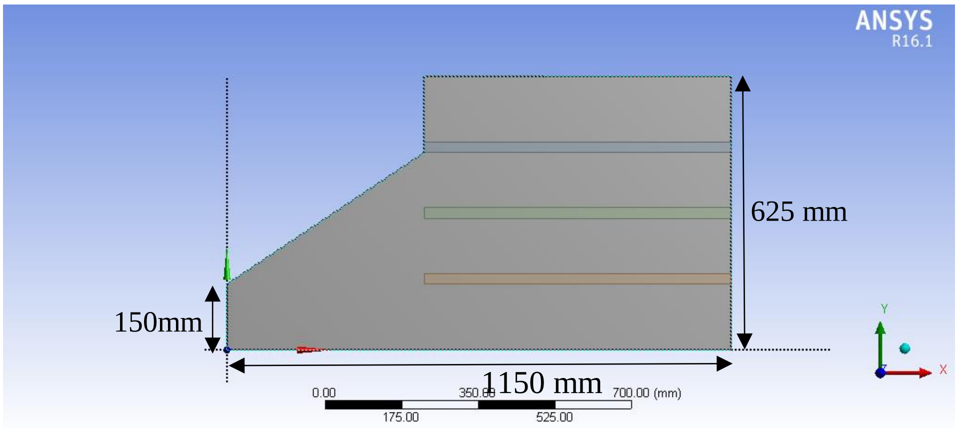
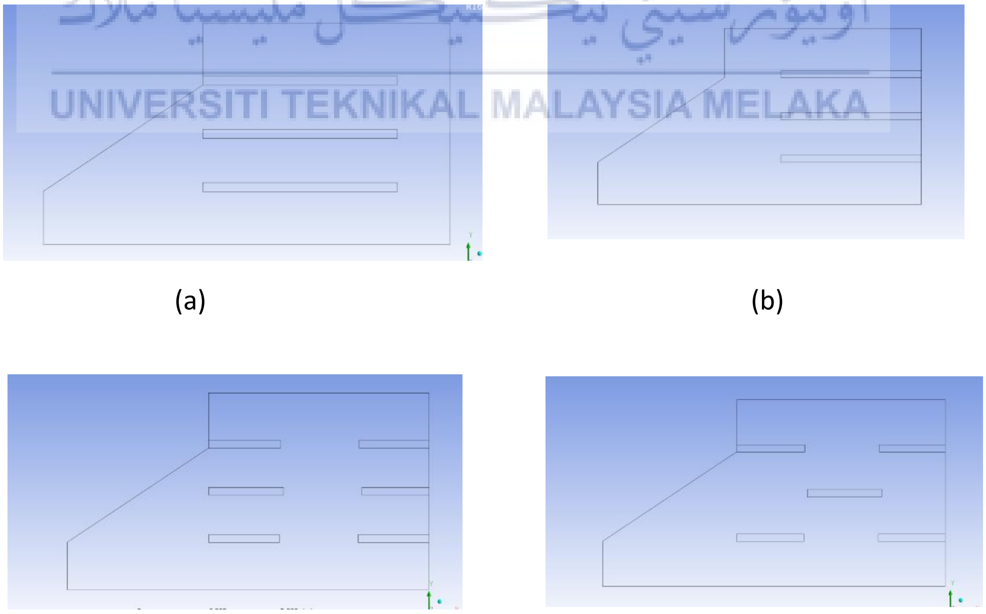


Figure 3.2: Symmetrically cut 2D representation of the flow domain

Besides that, there are another expected four designs of variation tray configurations in the chamber going to be produced as shown in Figure 3.3. The parameters and guides in designing the position of trays based on the literature review done. The purpose of this section is to design the best configuration of trays in order to obtain the best uniformity of velocity and temperature distribution. All the designs of tray configurations have been done in by using Design Modular provided by ANSYS.



(a) (b)

(c) (d)

Figure 3.3: Variation of tray configurations

3.3.2 Grid Generating

In process of grid generating, all the surface of fluid domain is performed meshing to analyse the numbers of nodes and elements. The domain is meshing into the unstructured 2D mesh by tetrahedral cell with nodes 3630 and elements 7008. If the grid generated is satisfying, then the pre-processing is proceeded with setting boundary conditions of the computational domain. Otherwise, the meshing process is repeated until the satisfied grid is obtained. A few face and edge sizing were applied in order to achieve the best grid. Figure 3.4 shows the corresponding meshes of the front view case.

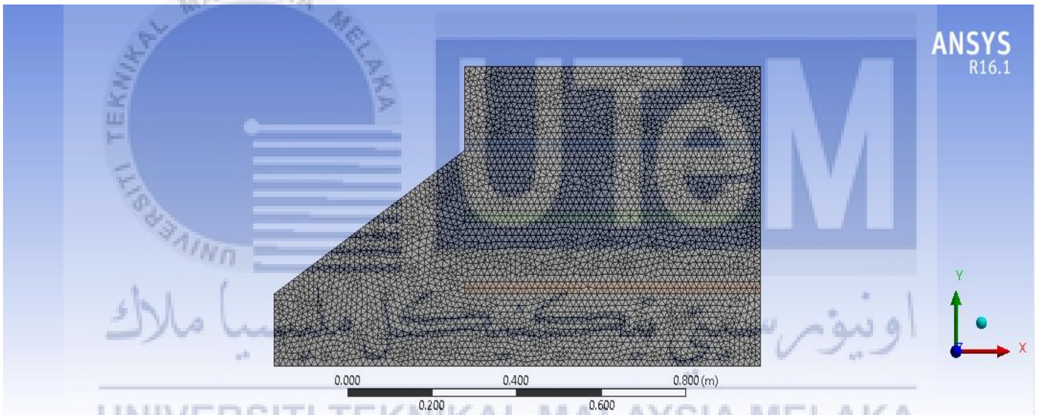


Figure 3.4: Unstructured Grid of Flow Domain

3.3.3 Defining of boundary conditions specifications and fluid properties.

In the continuum regions defining step, the fluid region inside the drying and the fabric surface are considered as air and porous media respectively. The boundary conditions of the domains consist of velocity inlet, pressure outlet and wall as shown in Figure 3.5. Anyway, problem arises in defining the boundary condition and the properties of the wall of chamber because Fluent package provided general properties of materials. Therefore, the properties of chamber wall is determined based on the literature study by identify the common material used for the chamber wall.

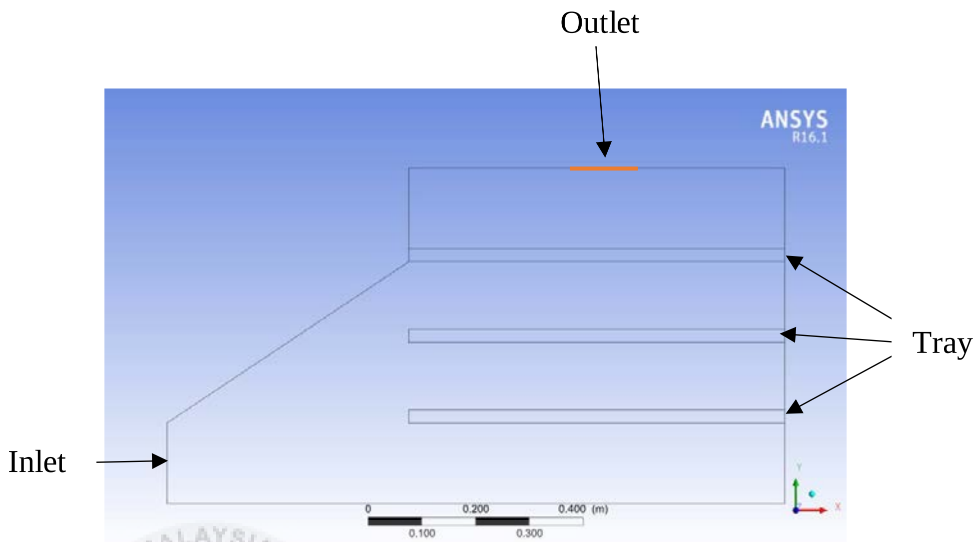


Figure 3.5: Boundary Conditions of the Drying Chamber

3.4 Solving

After all the boundary conditions of the domain is set up, the setting of the Fluent solver is adjusted according the suitability for solving the problem. The steps included in setup the solvers includes choosing solver, specifying fluid material, operating condition, boundary condition choosing the equation to be solved, selecting the discretization schemes to solve the equation, initializing the solution variables, activating the solutions monitors and finally solving the problem by running the iteration until the solution is convergence.

3.4.1 Choosing the suitable solver

In the drying chamber, the heated air flows from the inlet through the surface of trays and finally flows out through exhaust. Obviously, it is an internal flow domain. To perform the steady simulation on the air flow, the double precision solver and segregated method were chosen. Energy equation was also activated since the drying process is related to heat transfer process. Finally, the standard k-ε with standard wall treatment model was selected to solve the turbulent flow. The standard k-ε model is a semi-empirical model based on model transport equations for the turbulent kinetic energy (k) and its dissipation rate (ε).

3.4.2 Setting Materials Properties

The fluid material in this problem is obviously the air. Thus, the properties of air were input based on the working temperature. Table 3.1 shows the properties of air.

Table 3.1: Properties of air and wall.

Properties	Air	Wall
Density (kg/m^3)	1.225	720
Specific heat (J/kg. K)	1006.43	1255
Thermal Conductivity (w/mK)	0.0242	0.16
Viscosity (kg/ms)	1.7894x10^{-5}	-

3.4.3 Setting Operating Condition and Boundary Condition

After set up the material of fluid, the next step is to input the operating conditions and boundary conditions to the solver. In the drying chamber, the operating temperature is 44^{o}C while operating temperature is remain constant as the atmosphere pressure. Thus, the setting of the operating pressure was set as a default. Since the air velocity at the inlet is vertically in the drying to the exhaust, the gravitational acceleration is activated which is -9.81m/s at the Y-coordinate and the gravity option in the solver was activated.

For the boundary condition set up, the boundaries of the velocity and temperature inlet and also pressure at the outlet were set up which is the air velocity is 3 m/s with temperature 40^{o}C. The gauge pressure at the outlet for all the cases are assumed to be zero. The trays was assumed as a porous media with 10% porosity. The others values that are not specified in the table were remain constant as default.

3.4.4 Configuring Solution Controls

For solving the steady state simulation of the air flow in the drying chamber, the equations to be solved were the flow, turbulence and energy equations. Then, the discretization for the momentum, turbulence kinetic energy, turbulence dissipation rate and energy were set as Second Order Upwind so that the solution would be more accurate compared to the first order upwind option.

The convergence of solution is one of the important in CFD simulation because the lesser the residual remain in the iteration, the more accurate result will be obtained from the solution. Thus, the convergence criterion of the equations to be solved were set up 0.001 while for energy convergence criterion was set as $1x10^{-6}$. This set up means that the iteration will be calculated and would stop once all the numerical residuals of the equations being solved until as lower as the convergence criterion set up.

3.4.5 Initializing the solution variables and activating solution monitor.

At this stage, all the information required by the solver was setup. The solution variables were initialized by computing from the velocity inlet, the residual monitor was turn on to show graph of numerical calculation to reach convergence. So that, the convergence of solution can be observed to identify all the set up were correct.

3.4.6 Solving

Finally, as the last step before obtained the result, the simulation was performed by setting the number of iterations and reporting interval required and started to calculate the iteration. Make sure the solution has been initialized before proceed to the calculation of iteration. While waiting the solution reached convergence, the residual monitor was observed to check the residual change for iteration done. The solution convergence depends on the number of grids of the domain. Basically, 2D domain will take a short time consuming for convergence compared to the 3D domain. If the solution is not reached convergence, the parameter of solution or quality of mesh should be investigated and modified the necessary boundaries to improve the solution convergence and accuracy of results.

3.5 Post-Processing

After the Fluent solver finished the calculation of iteration means that the solution reached the convergence, the results of simulation can be analysed in both numerically and graphically. By using the Fluent solver, the contour, vector and streamline of the velocity air flow and temperature distribution in the drying chamber were plot. Thus, the result can be analysed according to the objectives of study. Average value for the velocity and temperature for the selecting points to measure also can be determined in this Fluent solver.

3.6 Validation

To compute results from the post-processing are validated by comparing existed data either from the experiment or previous researchers based on literature study. For this step, it is importance to convince the result obtained from the simulation are correct. The error of validation should be less than 5%. The lesser of error, the most accurate results were obtained.

CHAPTER 4

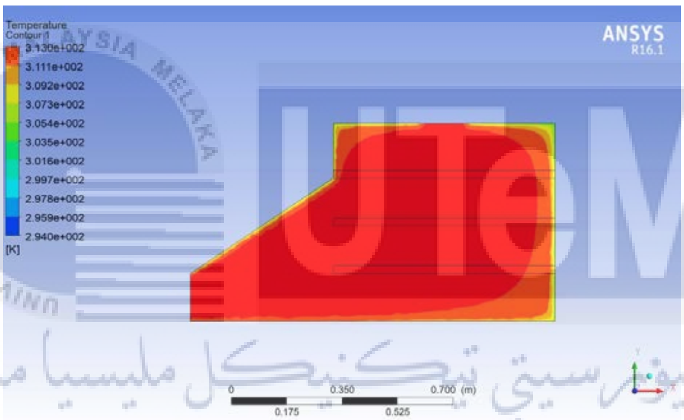
RESULT ANALYSIS AND DISCUSSION

4.1 Model Prediction

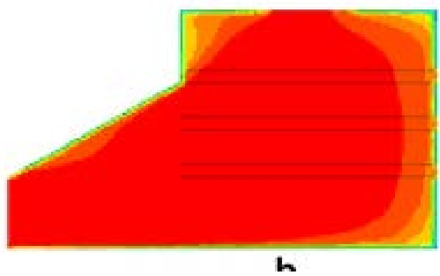
The best performance of drying chamber is predicted by the uniformity distribution of temperature and velocity. Based on the study on previous journal as part of validation process, uniformity studied by variation geometry of drying chamber with different of size of inlet, position of outlet and angle of divergence. Therefore, in this study has been studied variation of configuration of tray arrangement for improvement of uniformity in drying chamber. In the application of drying, temperature, velocity and drying air humidity condition contributed larger significance effect to the process of drying. However, for this simulation study, only velocity and temperature distribution that can be analysed since the equation used for the simulation is not related to the humidity. Therefore, the best performance and high efficiency of drying chamber can be assessed by comparing and analysing the uniformity of air flow and temperature distribution in the drying chamber. Moreover, the non-uniformity distribution caused by poor design and arrangement of trays in the chamber. However, this issues can be solved by predicted the uniformity by using CFD simulation.

4.2 Result Validation

In order to validate the simulation result, a comparison between result and experiment data obtained from the literature study was performed. The purpose of this validation to ensure that the steps and properties used for the drying is corrected. By fitting both experimental and simulation drying curves in the graph, the variation between these results could be analysed. Figure 4.2 shows the correlations between the predicted data obtained from the CFD simulation and experimental data from the literature review even though there are a little bit differences between these results. However, this correlation still accepted and the results is validate.



(a)



(b)

Figure 4.1: The contour for temperature profiles within drying chamber (a) Results of CFD Simulation; (b) Results of CFD Simulation from the Literature Review.

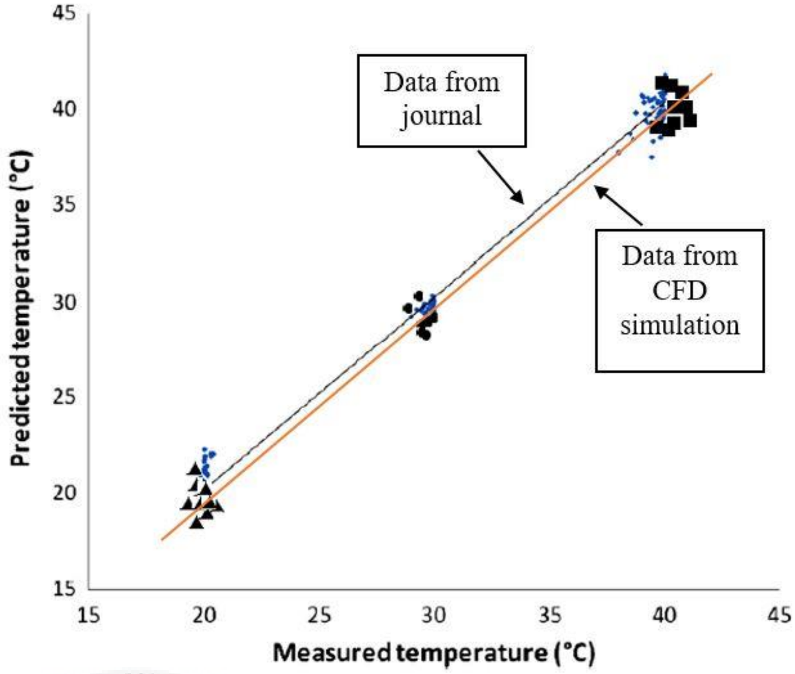
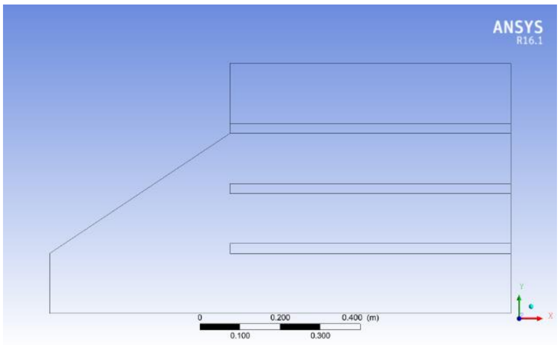


Figure 4.2: Comparison of the results of experimental and CFD predicted drying air temperature.

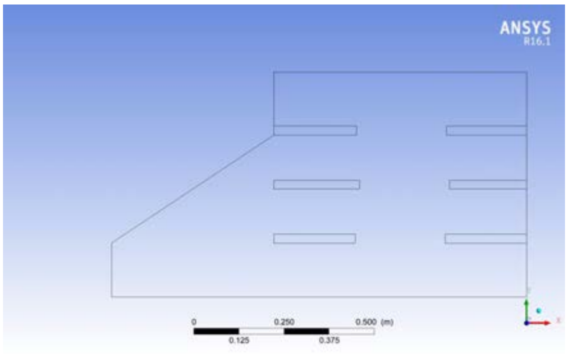
4.3 Design of Drying Chamber

In this study, there are five design of drying chamber has been proposed with different configuration of trays. The purposed of variation of configurations of trays is to determine the best design in order to achieve the best velocity and temperature distribution in the drying chamber. Table 4.1 shows the different types of configuration of trays in drying chamber.

Table 4.1: Five designs of Drying Chamber with different arrangement of Trays.



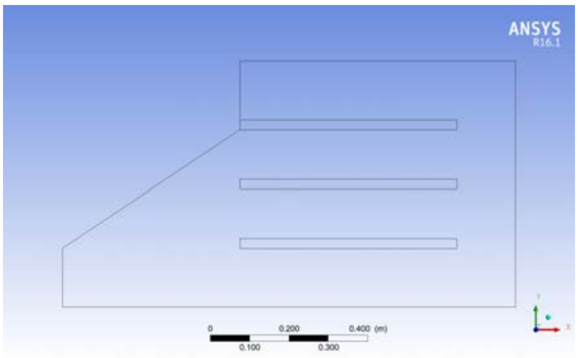
Design A



Design B

Design C

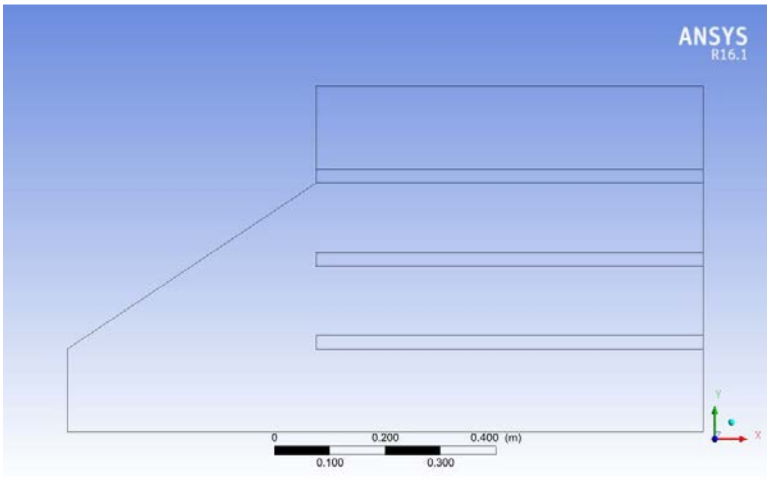
Design D



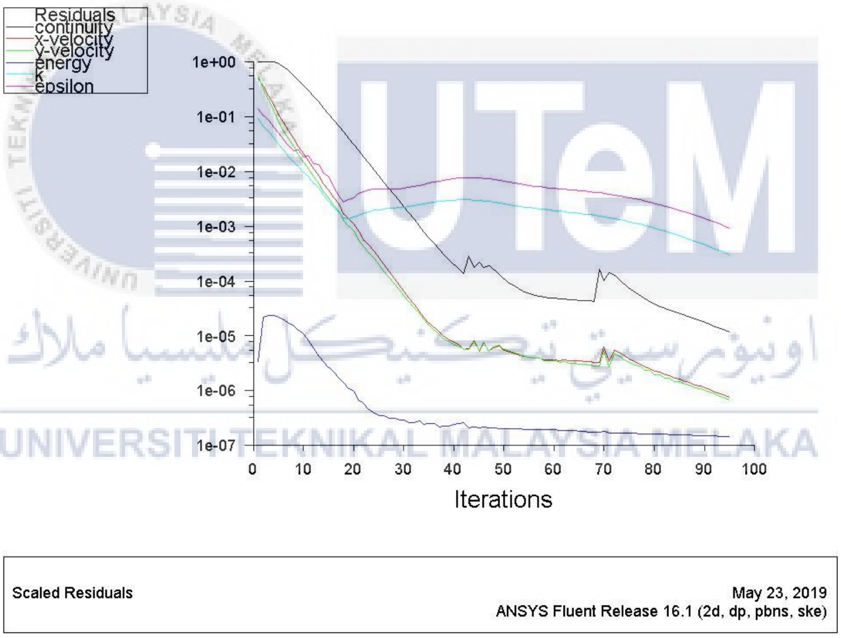
Design D

4.4 Result of Convergence Solution

The number of iteration calculated at the solver must be converged to make sure all the nodes and elements of the flow domain done to be calculated. This solution of convergence is important to ensure the data obtained from the simulation is more accurate. The time consumed for the solution convergence depended on the numbers of nodes and elements. However, the value of convergence can be set up during simulation. For this study, the values of convergence are 1×10^{-6} for the energy equation and 0.001 for the rest related equation. The following graph in Figure 4.3, 4.4, 4.5, 4.6 and 4.7 shows the convergence solution for all the trays configurations.

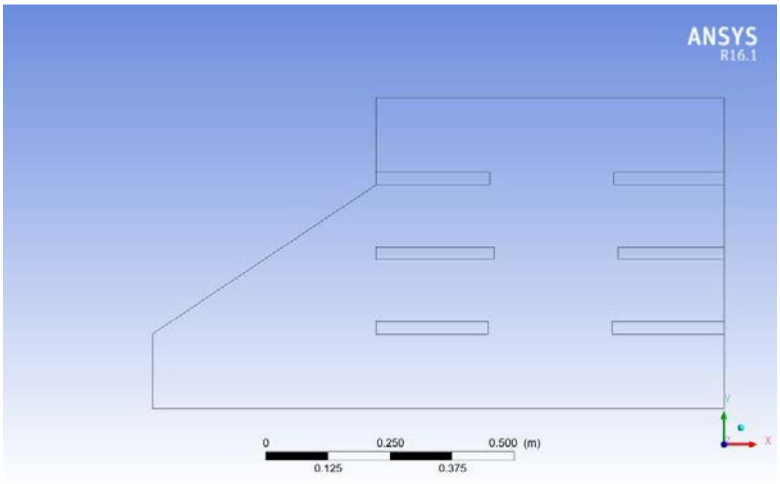


(a)

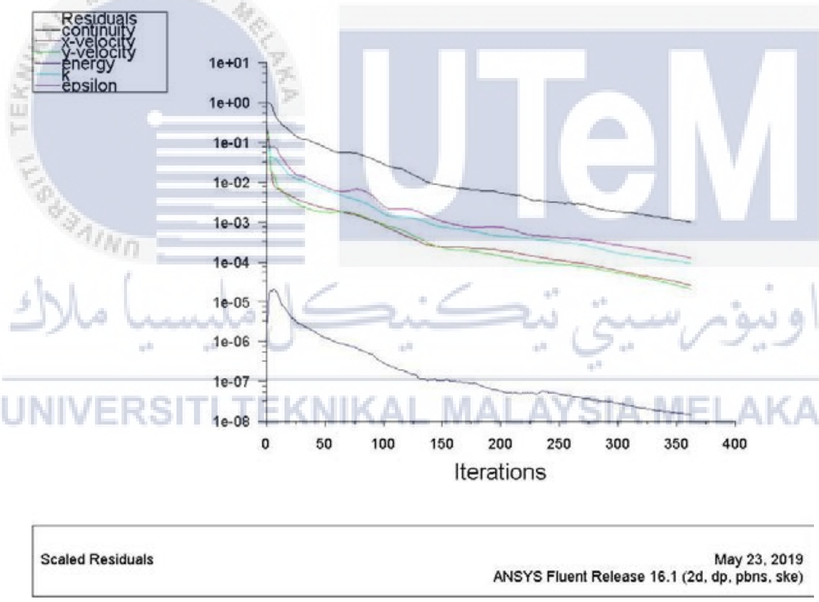


(b)

Figure 4.3: Configuration of trays for Case 1 (a) Design (a); (b) Convergence Solution for Design (a)

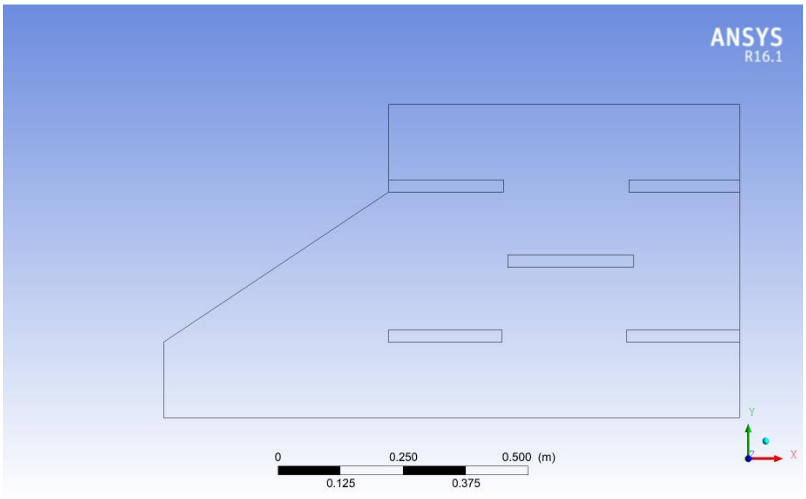


(a)

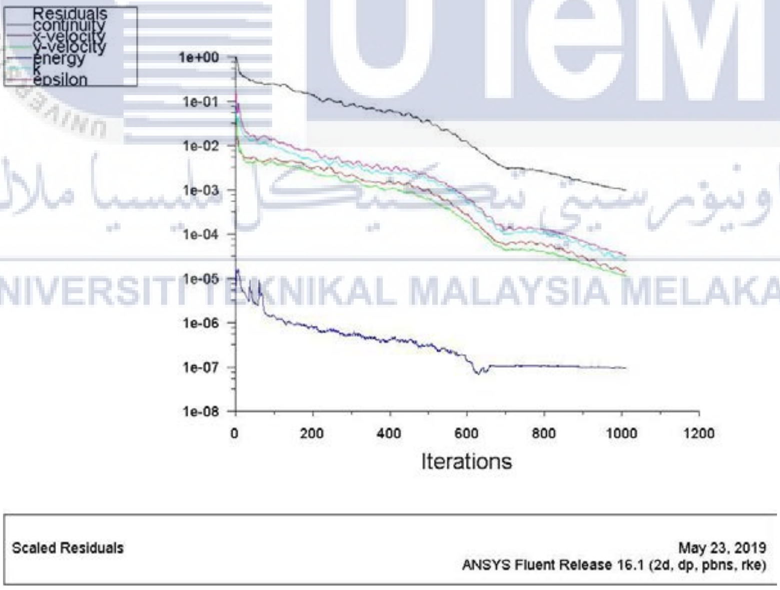


(b)

Figure 4.4: Configuration of trays for Case 2 (a) Design (b); (b) Convergence solution for Design (b)

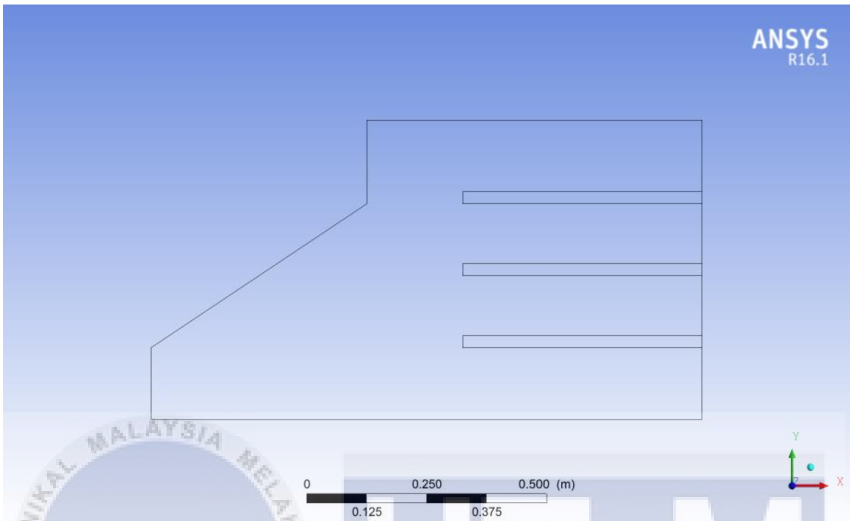


(a)

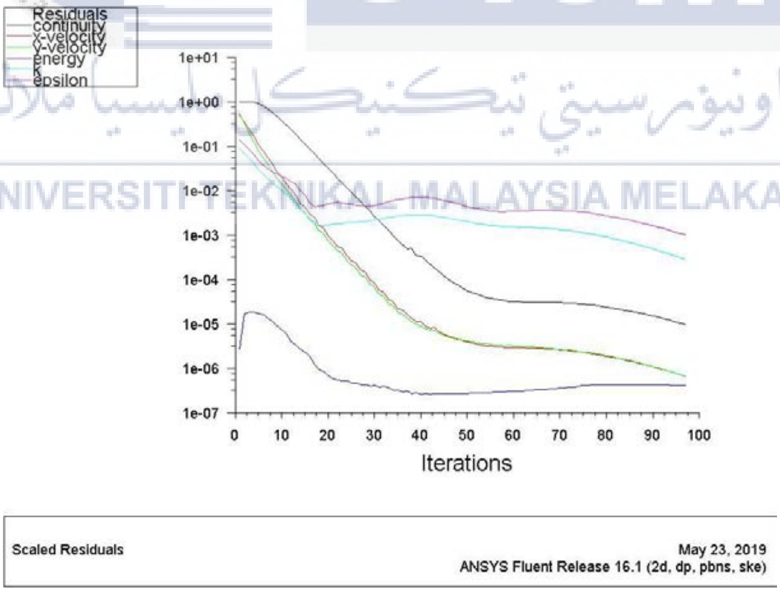


(b)

Figure 4.5: Configuration of trays for Case 3 (a) Design (c); (b) Convergence solution for Design (c)

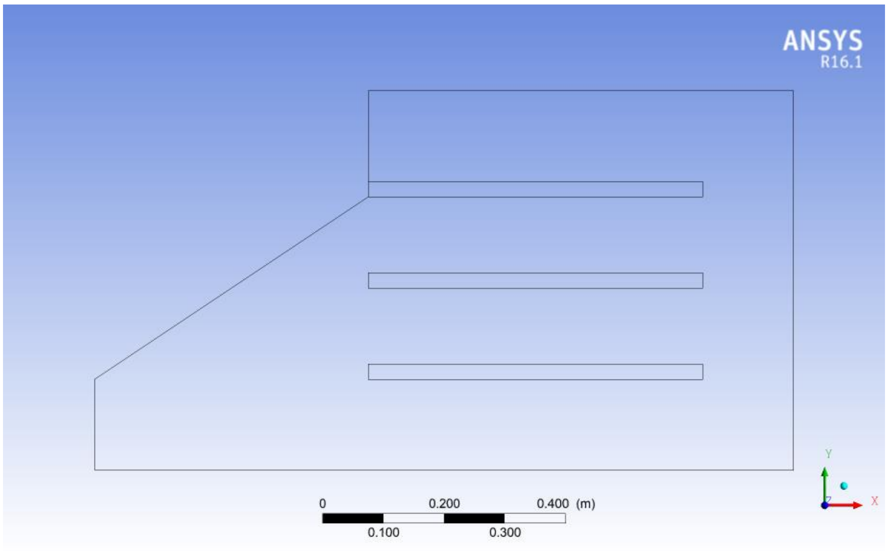


(a)

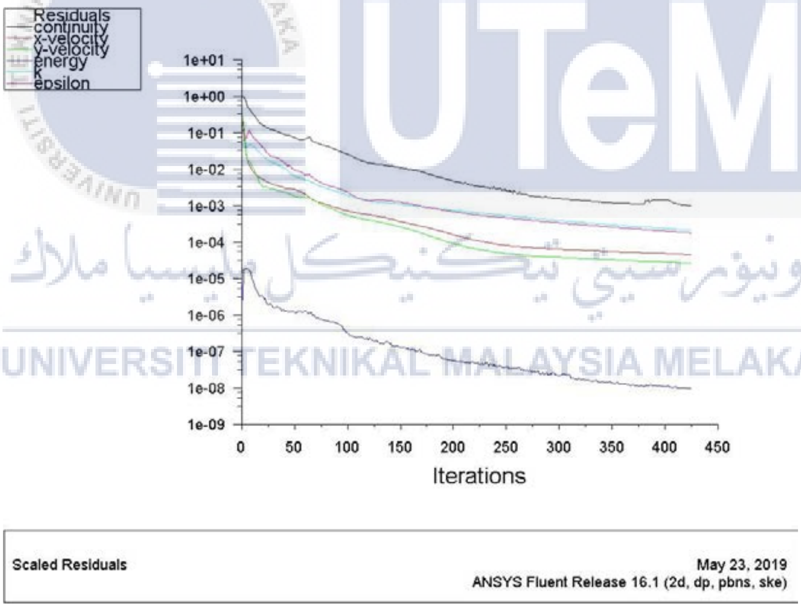


(b)

Figure 4.6: Configuration of trays for Case 4 (a) Design (d); (b) Convergence solution for Design (d)



(a)



(b)

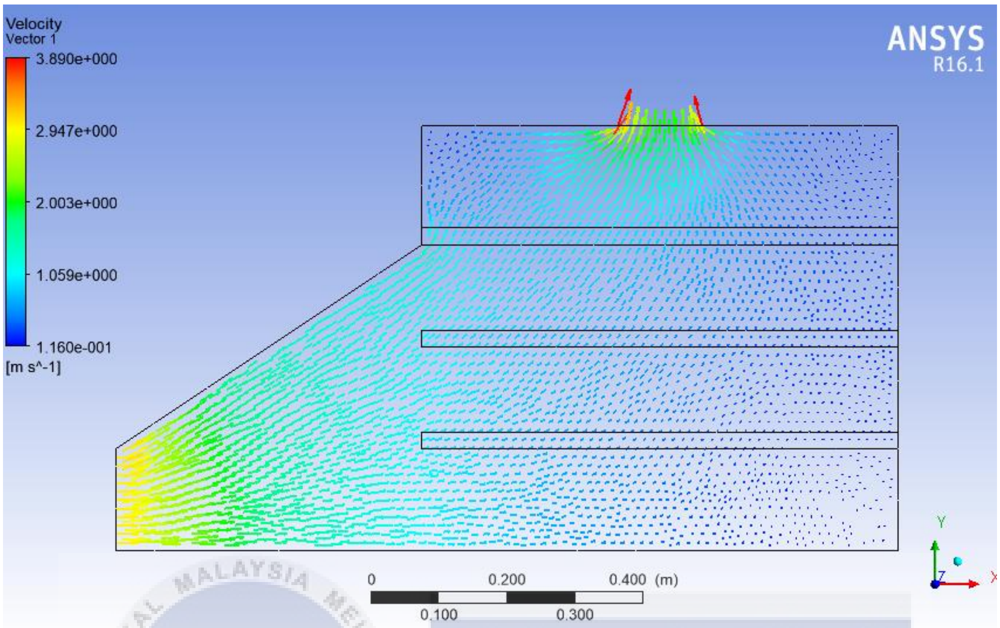
Figure 4.7: Configuration of trays for Case 5 (a) Design (e); Convergence solution for Design (e)

4.5 Air Flow Simulation Predictions

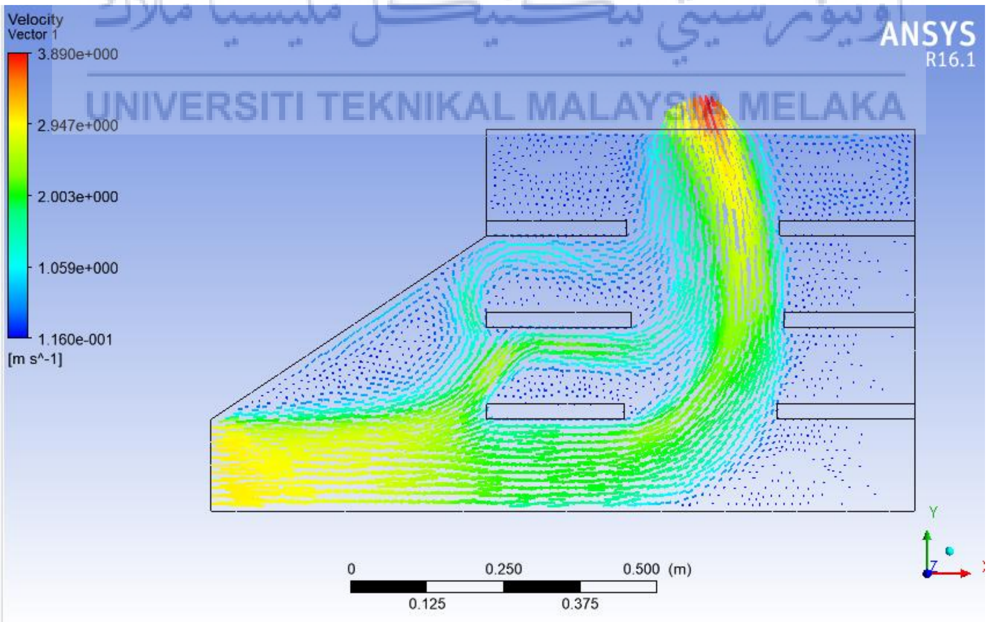
The steady state simulation was performed to solve the continuity, momentum, energy, turbulent kinetic energy and turbulence dissipation rate equation. In this study, the drying airflow velocity distribution within the drying chamber was done in two-dimensional, steady state condition and the analysis is studied at the symmetry faced of the domain. Figure 4.8 below shows the vector of air flow velocity distribution in the drying chamber with the variation of tray configurations by CFD simulation. It found that there are different flow for each design. There are five designs of drying chamber with different arrangement of trays. The purpose this variety is to analyse and determine the best configuration of trays in the best performance and most uniform distribution of velocity.

Based on the vector colour, it can be concluded that the airflow velocity is higher at the areas that close to the inlet and outlet zones. Therefore, the distribution of velocity is distributed differently along the trays depends on it levels of arrangement. Basically, the trays near to the inlet and outlet zones will get higher air velocity in average. From the simulation result obtained, it shows that design (b) and (c) are the poorest of tray configurations because of zigzag pattern is not suitable for these case since the position of inlet and outlet nearest. For both design (a) and (d) show reasonable uniform air velocity distribution in the chamber as shown in Figure 4.8.

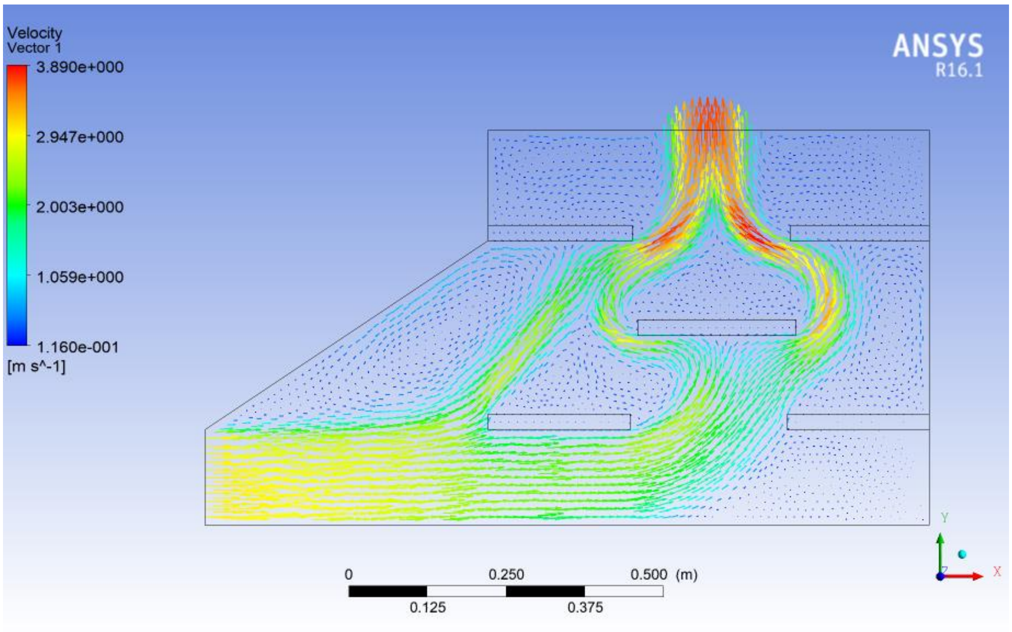
\



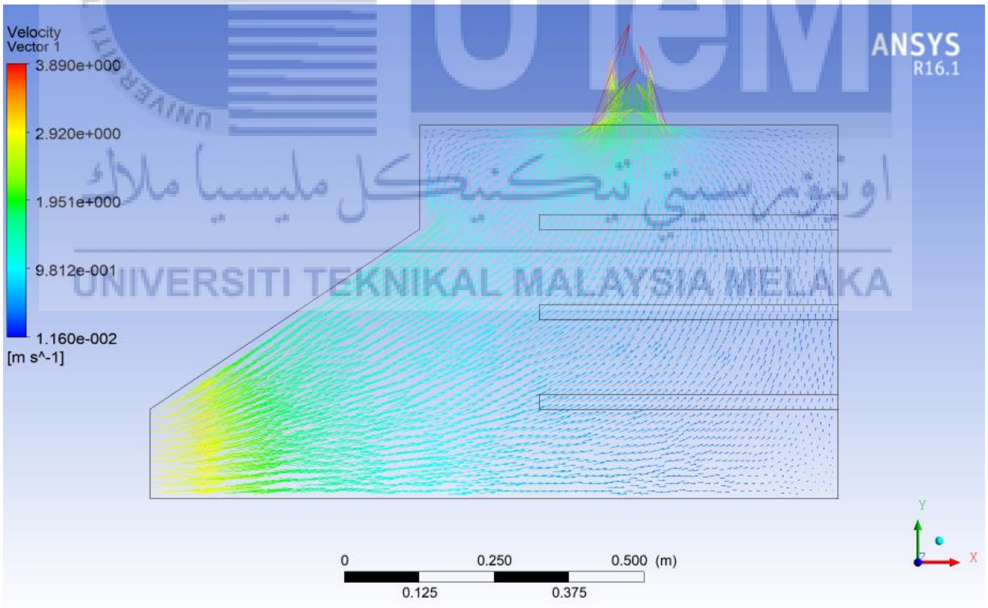
(a) Design A



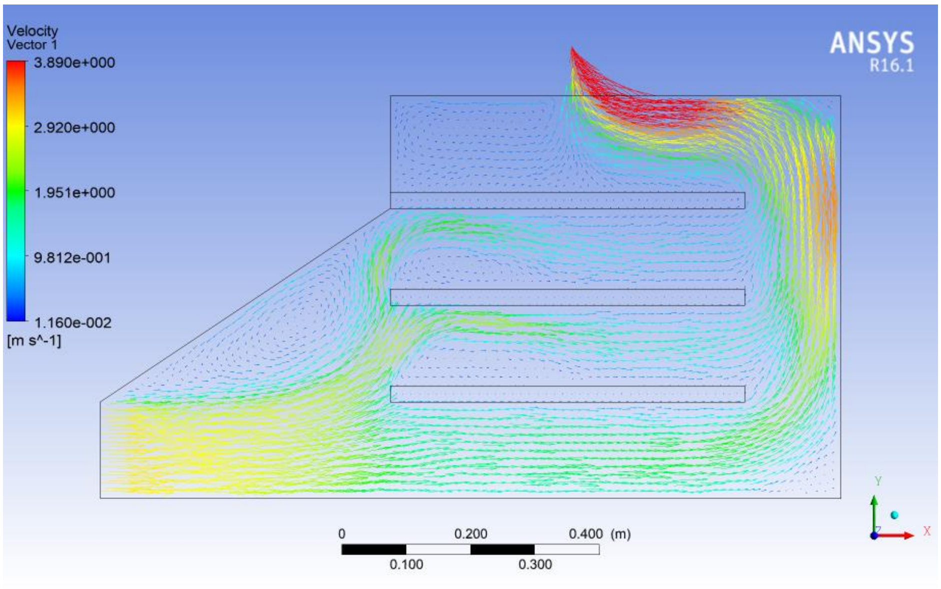
(b) Design B



(c) Design C



(d) Design D



(e) Design E

Figure 4.8: Design of drying chamber for five variation configuration of trays and vector profiles of airflow velocity distribution using CFD Simulation

Based on the literature review, the most important parameter in studying the uniformity of drying within a drying chamber application is velocity distribution. Therefore, nine points are located at the tray's zone as shown in Figure 4.9 in purpose of prediction the exact values along the trays to determine the uniformity distribution in the chamber. All the position of points as shown as in Figure 4.8 are remain constant for all the design.

The results of simulation for the predicted velocity distribution on nine different points for all the designs are shown in Table 4.2. Based on the table, it can be concluded that the best performance in uniformity distribution within drying chamber along the trays can be assessed by design (A) and (D). Although design (D) shows that at the point 3, 6 and 9 are highest values than others points caused no trays located at that placed. So that, there are no resistance in distribution of velocity in the chamber to the outlet. All the trays are set up in porous media in order to improve the uniformity of distribution.

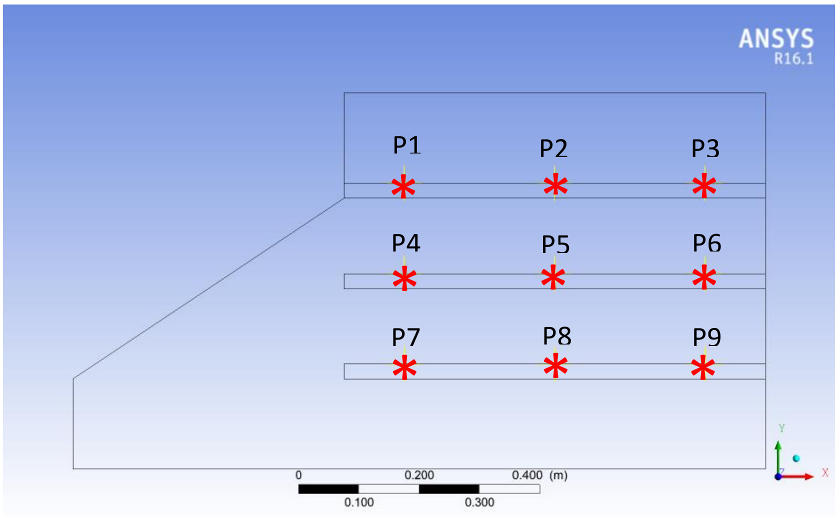


Figure 4.9: The Location of Measured Points for Predicted Velocity on Trays

Based on the table below, the average of velocity at the point 9 is the poorest point of velocity distribution. This is because that point is far from inlet and outlet. Therefore, that zone is the lowest zone that received air flow velocity and poor performance of drying process. From the graph in Figure 4.12, it clearly displayed that the most uniformity distribution of velocity for all the design are Design A and D.

Table 4.2: Predicted Velocity Distribution by CFD Simulation on 9 Different Points for Variation Configuration of Trays.

Design	Velocity (m/s)								
	P1	P2	P3	P4	P5	P6	P7	P8	P9
A	0.8586	0.9394	0.3284	0.9198	0.6083	0.3192	0.8406	0.4780	0.2141
B	0.0318	1.9950	0.1096	0.0406	1.4660	0.2875	0.1076	2.1790	0.0235
C	0.1152	1.3980	0.1248	2.1730	0.0976	0.7042	0.0934	2.0940	0.0747
D	0.8720	0.9322	0.3288	0.9504	0.6012	0.3175	0.8793	0.4713	0.2131
E	0.0894	0.1272	2.5920	0.1094	0.1654	1.9040	0.0945	0.1130	1.5520

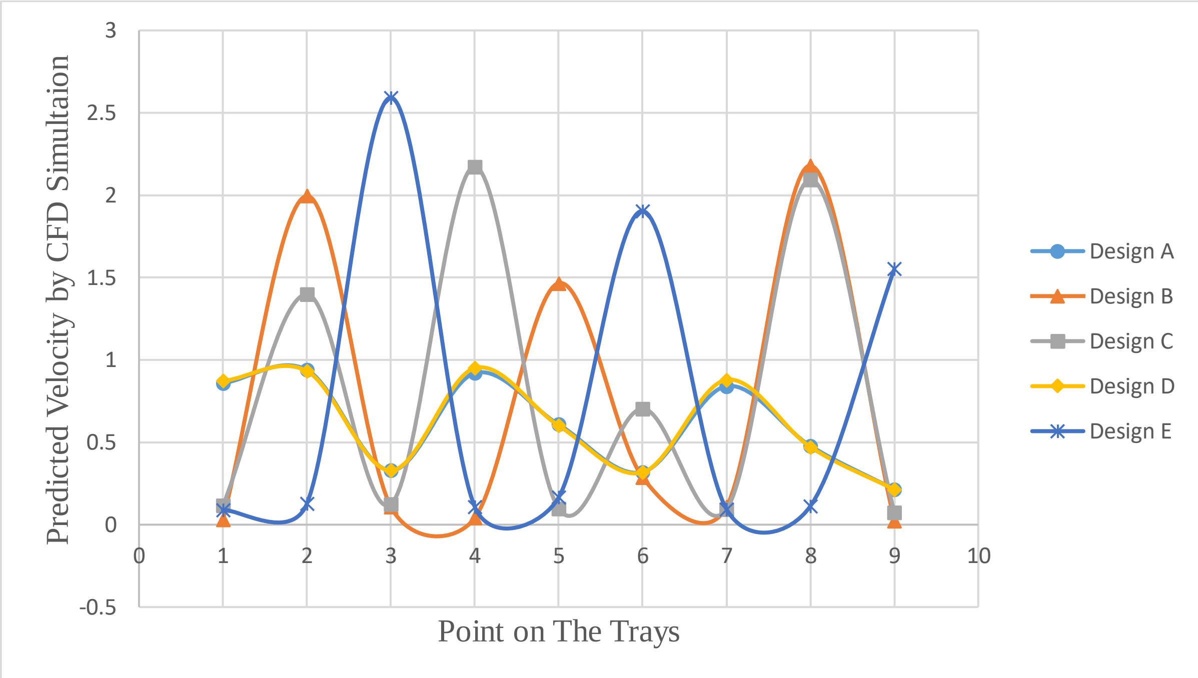
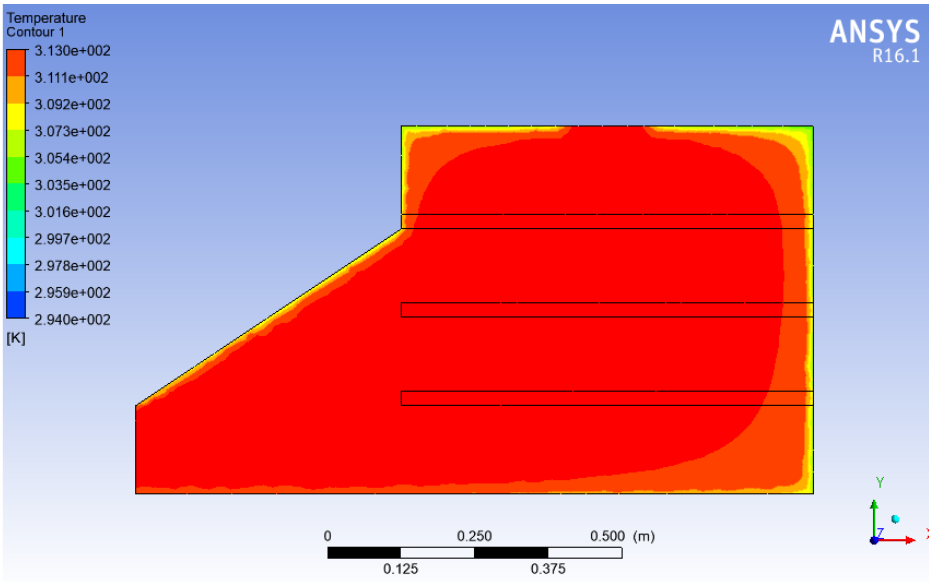


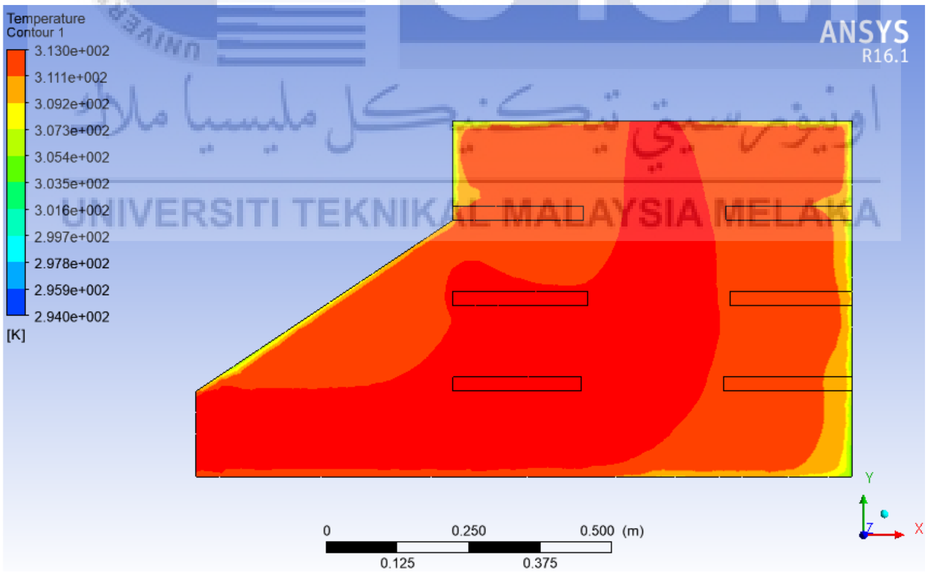
Figure 4.10: Predicted Air Velocity by CFD Simulation against Nine Points Located on Trays for Each Design

4.6 Temperature Distribution

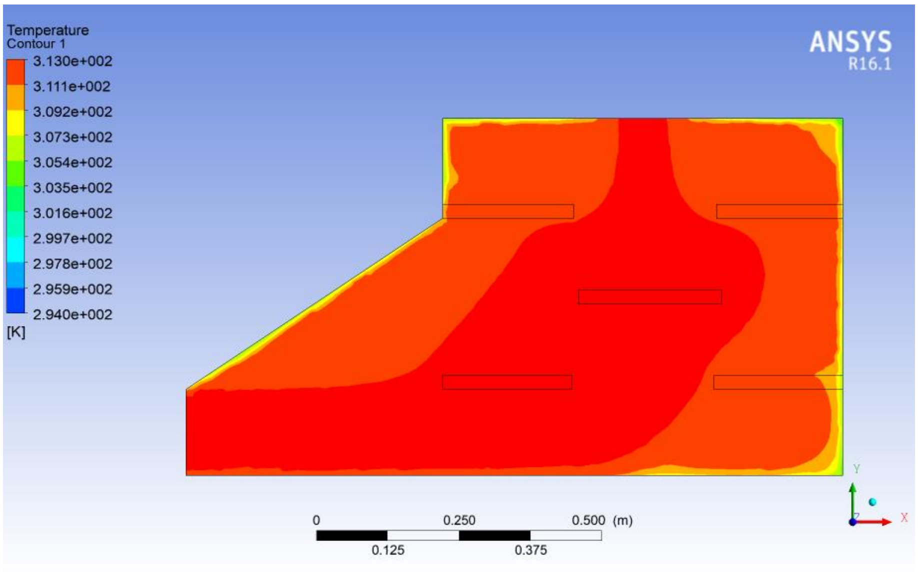
Firstly, the hot air temperature from the inlet was set as 40℃ and operating temperature is 44℃. Based on the contour profile of temperature distribution for the variation of trays configuration in Figure 4.11, temperature distribution within the drying chamber for all the designs are more or less a similar trend. However, from the results CFD simulation in Figure 4.10 shows that the best performance of temperature distribution assessed by design (A) and (D). This is because the temperature distribution profiles for both designs distributed similar evenly for each trays. Based on the results obtained for all the configurations as in Figure 4.10, it displayed that changing either the geometrical shape of drying chamber or variation of tray arrangement only gave a little bit influence to the temperature distribution compared to the velocity distribution.



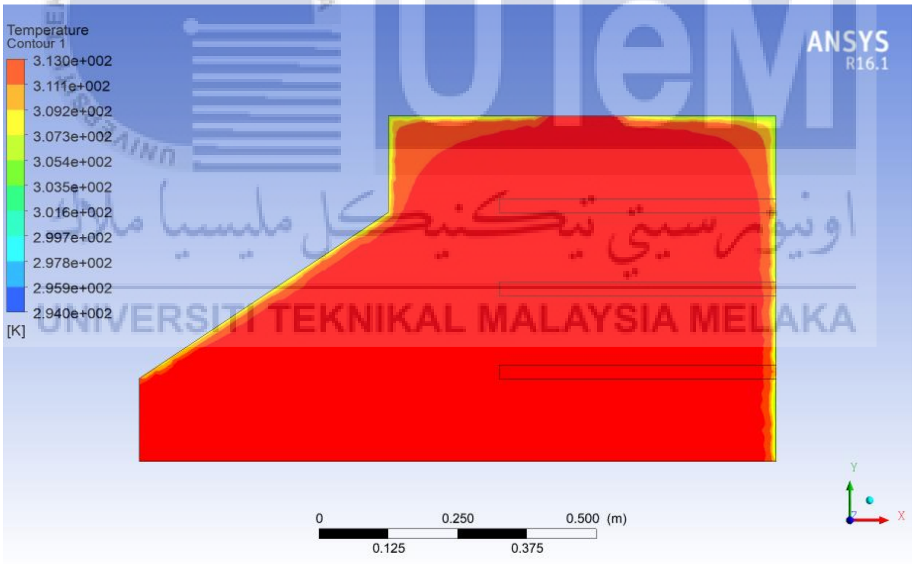
(a) Design A



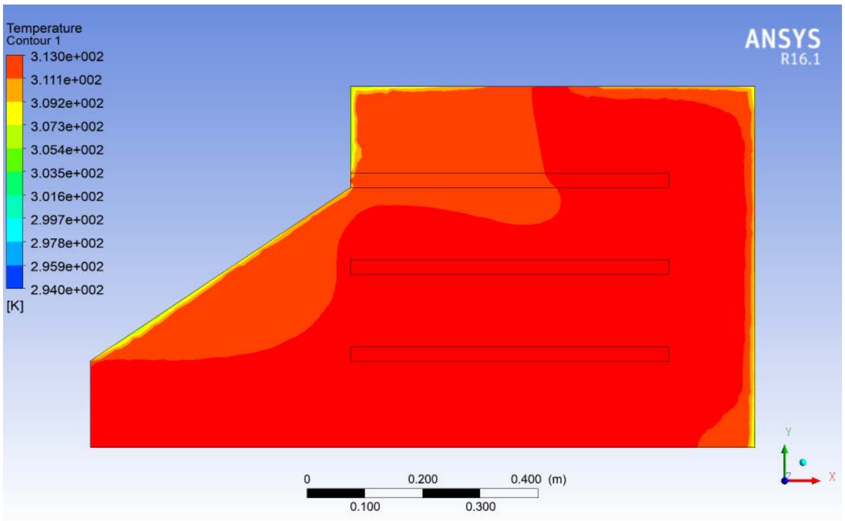
(b) Design B



(c) Design C



(d) Design D



(e) Design E

Figure 4.11: Design of drying chamber for five variation configuration of trays and temperature profiles using CFD Simulation

For the measured temperature distribution on the trays, the plotting point is sane when measured average velocity as shown in Figure 4.9. Based on the table 4.3 below, the data shows that only small differences of temperature distribution among the trays for all the designs. From the graph in Figure 4.12, it clearly displayed the most uniformity distribution of temperature for all the design. For the design (A) and (D), it can be concluded that both design approached the uniformity of temperature distribution.

Table 4.3: Predicted Temperature Distribution by CFD Simulation on 9 Different Points for Variation of Trays Configuration.

Design	Temperature (K)								
	P1	P2	P3	P4	P5	P6	P7	P8	P9
A	313.1	313.1	313.1	313.1	313.1	313.1	313.1	313.1	313.0
B	312.0	313.1	311.4	313.0	313.1	312.0	313.1	313.1	311.8
C	312.6	313.1	312.5	313.1	313.1	312.4	313.1	313.1	311.6
D	313.1	313.1	313.1	313.1	313.1	313.1	313.1	313.1	313.1
E	312.6	313.1	312.5	313.1	313.1	313.1	313.1	313.1	313.1

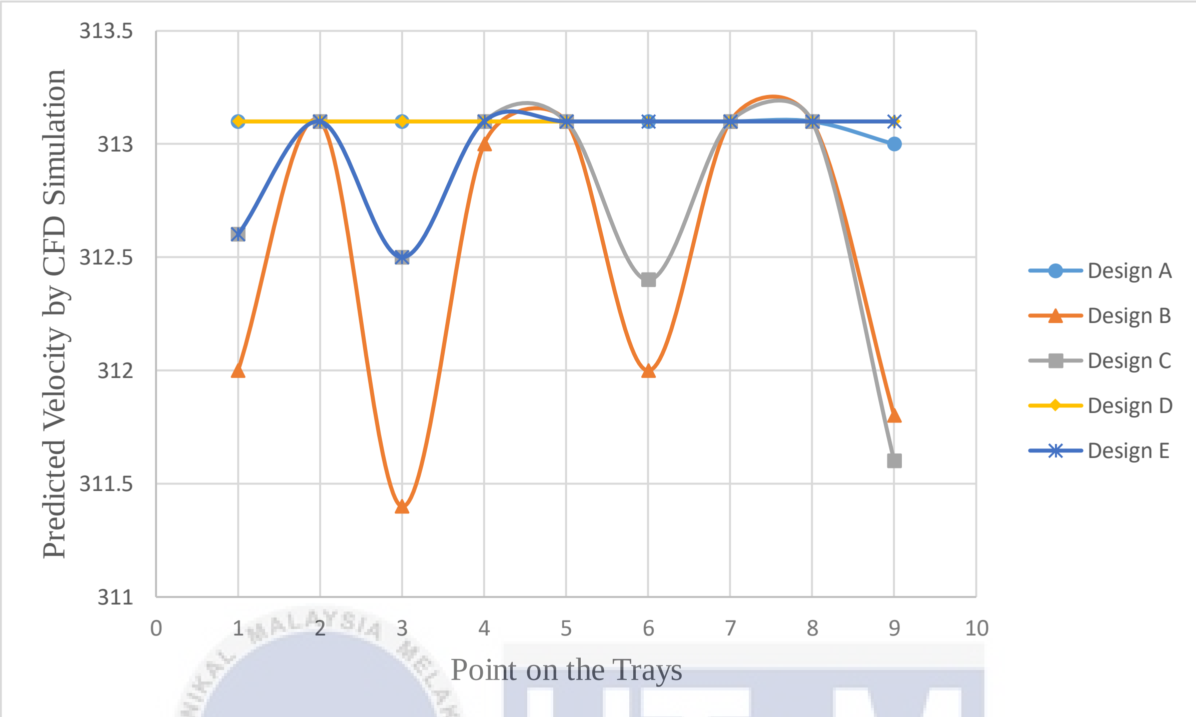


Figure 4.12: Predicted Temperature by CFD Simulation against Nine Points Located on Trays for Each Design

4.7 The Best Configurations of the Trays.

Last but not least, based on the data obtained from the CFD simulation as tabulated in Table 4.1 and 4.2, it can be concluded that the design with the best configurations is Design A and D. This is because both of the designs achieved uniformity in terms of velocity and temperature distribution simultaneously. Therefore, the propose design for the best performance in term uniformity distribution is design A and D.

CHAPTER 5

CONCLUSION

5.1 Conclusion

In this study, the literature review on the application of the drying chamber in the various industries was done and identified the main issue of this application. From the finding during literature review, non-uniformity velocity and temperature distribution within the drying chamber being main issue of this application. However, prediction the flow distribution in the chamber for the new design and configuration was done by using CFD software which is ANSYS Fluent 16.1 in order to save cost and time. The model of drying chamber as a computational domain has been identified and created using the pre-processor, Designer Modular. The properties and boundary conditions of the drying chamber were not stated specifically in the Fluent. Therefore, all the boundary conditions related to the simulations were identified and determined based on the literature review. In order to ensure all the properties and boundary conditions are right, a result validation is performed by comparing data obtained by experiment in the literature review with data from the CFD simulation. The less differences of percentages comparison, the more accurate predicted data obtained. In addition, there are five different of configurations of trays in the chamber. The purpose of variety configuration to study the best performance among the five designs. Therefore, 2D simulation in steady state is performed to analyse the uniformity of velocity and temperature distribution within variety of trays arrangement. Nine points are located on the trays to identify the range of velocity and temperature distribution among the trays in order to determine the best performance in distributed uniformly. The designs that uniformly in both distributions is determined as the best configurations which is design (A) and (D).

The determination of the design based on the CFD simulation on the behaviour of velocity and temperature among the trays. All these profiles of temperature and velocity distribution were shown and predicted by the post-processing of the CFD simulation result.

5.2 Recommendation for Future Work

There are several recommendations that can be performed in the future study for improvement of uniformity distribution of velocity and temperature within drying chamber which are:

1. Improve the source code developed for the clothes drying process from 2D to 3D model so that the result of simulation is more accurate and able to display the real situation of air flow in the chamber.

2. Conduct an experimental works on various configurations of trays in drying chamber to validate the result of simulation.

3. Variation of materials used for the chamber wall and trays to obtain the best performance in heat transfer for drying.

4. Improve the source code so that the parameters of the drying i.e. dimensions, porosity, moisture content, thermal conductivity, specific surface area and etc. for the purpose edited as needed.

REFERENCES

Abhay L., Chandramohan V.P., V.R.K. Raju., "Design, development and performance of indirect type of solar dryer for banana drying", Engineering Procedia 109, pp. 409-416.

Ahmad Fudholi, K. Sopian, M.H. Ruslan, M.A. Alghoul, M.Y. Sulaiman, Review of solar dryers for agricultural and marine products, Renewable and Sustainable Energy.

Ahmad Fudholi, K. S. (2010). Review of solar dryers for agricultural and marine products. *Renewable and Sustainable energy Review*, 1-30.

Ahmad Fudholi, M. Y. (2015). Prospect and Future of Solar Dryer for Agricultural and Marine: Perspective Malaysia.

Ahmed, N. *et al.* (2013) 'Different Drying Methods : Their Applications and Recent Advances', 4(1), pp. 34–42.

Amanlou, Y. and Zomorodian, A. (2010) 'Applying CFD for designing a new fruit cabinet dryer', *Journal of Food Engineering*. Elsevier Ltd, 101(1), pp. 8–15. doi: 10.1016/j.jfoodeng.2010.06.001.

Anna H., Iva K., Rithy C., Petra C., Jan B., "Development of solar drying model for selected Cambodian fish species", The Scientific World Journal, 2014, pp. 1-9.

AP Mnkeni, P Soundy and MO Brutsch, "Solar Drying of Fruit and Vegetables", Department of Agriculture.

Arina M.N, Sohif M., Mohd H.R., “CFD Simulation of temperature and air flow distribution inside industrial scale solar dryer”, Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 45, Issues 1 (2018), pp. 156-164.

Arun S. Mujumdar, “Handbook of Industrial Drying”, Vol. 1, Marcel Dekker Inc., New York (1995)

Bashria, A. N. (2004). Criteria for Solar Drying Projects in Malaysia. *Journal of Industrial Technology*, 39-48.

Chao J., Li W., Wei G., Chuanping L., “Effect of swing temperature and alternating airflow on drying uniformity in deep-bed wheat drying”, Applied Thermal Engineering 106 (2016), pp. 774-783.

Chi-Young J., Jong-Ryul K., Sung-CHul Y., “Two-dimensional simulation of silica gel drying using computational fluid dynamics”, Journal of Ceramic Processing Research, Vol.9, No.2, 2008, pp. 184-188.

Francisco Javier Trujillo, Q. T. (2009). A Computational fluid dynamic model of the heat and moisture transfer during beef chilling. *International Journal of Refrigeration*, 998-1009.

Fuente-blanco, S. De and Sarabia, E. R. De (2006) ‘Food drying process by power ultrasound’, 44, pp. 523–527. doi: 10.1016/j.ultras.2006.05.181.

Ganesh Visavale, B. T. (2011). Comparative study on drying of selected marine products: Bombay Duck (Herpodon nehereus) and prawn (Penaeus indicus). *International Journal of Food Engineering*, 1-17.

Hamdani, T. R. (2018). Fabrication and testing of hybrid solar-biomass dryer for drying. *Case Studies in Thermal Engineering*, 489-496.

Henry Sabarez, “Drying of Food Materials”, Reference Module in Food Science, 2016.

Jamila P., Saritha K., Hermina G., Jeyasanta K.I., “Effect of drying technologies on the biochemical properties of Stolephorus Commersonnii”, Food Quality and Safety, vol.3, 2018, pp. 153-158.

Jia, C. *et al*. (2016) ‘Effect of swing temperature and alternating airflow on drying uniformity in deep-bed wheat drying’, *Applied Thermal Engineering*. Elsevier Ltd, 106, pp. 774–783. doi: 10.1016/j.applthermaleng.2016.06.056.

J.G.Brennan. (2003). *Drying theory of air drying.* Encylopedia of food sciences and nutrition (Second Edition).

Lyes B., Azzeddine B., “Design and simulation of a solar dryer for agriculture products”, Journal of Food Engineering 59 (2003), pp. 259-266..

M. I. Fadhel, K. Sopian, W.R.W Daud, M.A. Alghoul, “Review on advanced of solar assisted chemical heat pump dryer for agriculture produce”, Renewable and Sustainable Energy Reviews, Vol. 15, Issue 2, February 2011, pp. 1152-1168.

Mohd Y.H.O., Ahmad F., Kamaruzzaman S., Mohd. H.R., Muhammad Y., “Drying Kinetics Analysis of Seaweed Gracilaria changii using solar drying system”, Sains Malaysiana 41(2) (2012): 245-252.

M. S. Ismail, “Solar Energy and Agricultural Wastes in the Drying Fish of Fish Products, Solar Energy Conversion II, 1981, pp. 406.

M. Y. H. Othman, K. Sopian, B. Yarim, W.R.W. Daud, “Development of advanced solar assisted drying systems”, Renewable Energy 31, 2006, pp 703-709.

Misha, S. *et al.* (2013) ‘Comparison of CFD Simulation on Tray Dryer System Between Porous and Comparison of CFD Simulation on Tray Dryer System Between Porous and Solid Product’, (April).

Misha, S. *et al.* (2014) ‘The Prediction of Drying Uniformity in Tray Dryer System using CFD The Prediction of Drying Uniformity in Tray Dryer System using CFD Simulation’, (November 2013).

Misha, S. and Yusof, A. A. (2018) ‘Experimental Investigation on Indirect , Natural and Forced Convection Mixed Mode Solar’, (July).

Misha. S, M.A.M. Rosli, M.Z. Akop, “Design and simulation of an optimized mixed mode solar dryer integrated with desiccant material”, International Journal of Mechanical and Mechatronics Engineering, Vol.17, No.6, 2017, pp. 65-73.

Misha.S, S. Mat, M.A.M. Rosli, M.H. Ruslan, K. Sopian and E. Salleh, “Simulation of Air Flow Distribution in a Tray Dryer by CFD, Recent Advances in Renewable Energy Sources, pp. 29-34.

Mustafa A., Ataollah K., Burak A., Ali A., “Development of a new drying chamber for heat pump mint leaves dryer”, Renewable Energy Systems, 2016, pp. 805-812.

Noh, A. M., Mat, S. and Ruslan, M. H. (2018) 'Akademia Baru CFD Simulation of Temperature and Air Flow Distribution inside Industrial Scale Solar Dryer Akademia Baru', 1(1), pp. 156–164.

Prof.Pravin M.G., Amit S.D., Ranjit C.B., Sagar C.P., Vishal G.P., "Design and construction of solar dryer for drying agricultural products", International Research Journal of Engineering and Technology, Vol.4, No.3, 2017, pp. 1946-1951.

Putra, R. N. and Ajiwiguna, T. A. (2017) 'Influence of Air Temperature and Velocity for Drying Process', *Procedia Engineering*. The Author(s), 170, pp. 516–519. doi: 10.1016/j.proeng.2017.03.082.

O. Yongson, I.A Badruddin, Z.A. Zainal, P.A.A Narayana, "Airflow analysis in an air conditioning room", Building Environment, Vol.42, 2007, pp. 1531-1537.

Rizal, T. A. and Muhammad, Z. (2018) 'Case Studies in Thermal Engineering Fabrication and testing of hybrid solar-biomass dryer for drying fi sh', *Case Studies in Thermal Engineering*. Elsevier Ltd, 12(July), pp. 489–496. doi: 10.1016/j.csite.2018.06.008.

Ruichao Z., Jibo L., "Study on drying uniformity of static small-sized drying box for fruits and vegetables", Procedia Engineering 205 (2017), pp. 2615-2622.

Salah A.E., M.H. Ruslan, B. Yatim, "Drying chamber performance of V-groove forced convective solar dryer", Desalination 209 (2007), pp. 151-155

S.H. Sengar, Y. K. (2009). Low cost solar dryer for fish. *African Journal of Environmental Science and Technology*, 265-271.

Shyam S. Sablani, M. S.-M. (2002). SUN AND SOLAR DRYING OF FISH SARDINES. *Proceedings of the 13th International Drying Symposium*, 1662-1666.

William L. Kerr, “Handbook of Farm, Dairy and Food Machinery Engineering”, Food Drying and Evaporation Processing operations, 2013.

Yunus A. Cengel, John M. Cimbala , “ Fluid Mechanics: Fundamental and Applications, Second Edition”, Mc Graw-Hill Companies Publishers, New York, Americas, 2010, pp. 853-907.

Zayana Nasser Ahmed Al Rawahi, A. M. (2013). Performance analysis of solar drying system for marine product of oman. *International Journal of Students Research in Technology and Management*, 610-613.

Zhang, P. and Fournier, J. (2017) ‘ScienceDirect ScienceDirect Computational fluid dynamic analysis of airflow in belt dryer : effects of conveyor position on airflow distribution Assessing the temperature function for a long-term district heat demand forecast’, *Energy Procedia*. Elsevier B.V., 142, pp. 1367–1374. doi: 10.1016/j.egypro.2017.12.521.

Zhang, R. and Long, J. (2017) ‘ScienceDirect Study on Drying Uniformity of Static Small-sized Drying Box for Fruits and Vegetables’, *Procedia Engineering*. Elsevier B.V., 205, pp. 2615–2622. doi: 10.1016/j.proeng.2017.10.201.

GANTT CHART PSM 2 (SEMESTER 1)

TASK NAME / WEEK	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Discussion with supervisor														
Research project														
Writing Chapter 1														
• Introduction/ Background														
• Problem statement														
• Objectives														
• Scope of Study														
• General Methodology														
Progress Report Submission														
Writing Chapter 2														
Writing Chapter 3														
Presentation Preparation														
Proposal Submission														

GANTT CHART PSM 2 (SEMESTER 2)

TASK NAME / WEEK	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Discussion with Supervisor														
Result Validation														
Simulation by CFD														
Optimization of Design														
Literature Review														
Writing Chapter 3														
Writing Chapter 4														
Progress Report Submission														
Writing Chapter 5														
Presentation Preparation														
Report Submission														
Seminar														