CFD Calculation of Impinging Confined Gas Jet Flames/Fires

Submitted by:

Syed Muhammad Syimir Bin Syed Mohamed Anuar

120417

Supervised by:

Dr. Yu Kok Hwa

June 2017

This dissertation is submitted to

Universiti Sains Malaysia

As partial fulfilment of the requirement to graduate with honours degrees in

Bachelor of Mechanical Engineering

School of Mechanical Engineering

Engineering Campus

Universiti Sains Malaysia

DECLARATION

This work has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any degree.

Signed...

Date...

STATEMENT 1

This thesis is the result of my own investigations, except where otherwise stated.

Other sources are acknowledged by giving explicit references.

Bibliography/references are appended.

Signed...

Date...

STATEMENT 2

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

Signed...

Date...

ACKNOWLEDGEMENT

Alhamdulillah praise to Allah for giving me the strength and guidance to me in completing my final year project as partial fulfilment of the requirement to graduate. First, I would to express my highest appreciation and thanks to Assoc. Prof. Ir. Nadiahnor Md. Yusop and Dr. Yu Kok Hwa as my project's supervisor for the encouragement and guidance throughout the entire process in the completion of this project. Both of them gave insightful suggestion, critical comments, and warm support allow me to have the opportunity to develop and improve my skills. With their knowledge, experience and patience, they really help me substantially towards the completion of this project. I am thankful to them for their contributions towards my understanding and thought. In addition, special thanks to the Universiti Sains Malaysia (USM), especially to the School of Mechanical Engineering for providing me with all the facilities needed to complete this project.

Moreover, I also want to express my sincere appreciation to my family and friends for their never-ending support and motivation especially whenever I face problems throughout my project period. Finally yet importantly, I would like to thanks those who involved directly or indirectly in helping me during the project. The help they gave not only confined to my final year project but also in my life.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

ABSTRAK

Kejadian bencana Piper Alpha berlaku pada tahun 1998 jelas menunjukkan potensi risiko yang boleh menyebabkan peningkatan bahaya apabila api jet menyantak pada struktur. Menyantak aliran jet telah digunakan secara meluas dalam banyak aplikasi industri sebagai teknik untuk meningkatkan haba dan pemindahan jisim dalam kes-kes pemanasan, penyejukan, dan pengeringan.Kaedah pembakaran jet biasanya berlaku dalam peralatan proses rumah dalam kemudahan luar pesisir di mana kebarangkalian mempunyai api menyantak pada kapal, paip, dan sokongan struktur yang tinggi. Sifat api jet yang menyebarluas disebakan interaksi aliran api yang bergolak dan tekanan tinggi boleh menyebabkan kegagalan kepada semua peralatan. Ini membawa kepada kepentingan dalam belajar ciri api jet dalam meramalkan potensi risiko kepada struktur dan siaran pencucuhan sengaja. Kajian ini memberi tumpuan kepada kajian ciri-ciri pembakaran melalui kesan menyantak api jet di kawasan yang terhad. Dalam kajian ini, siasatan berangka pada kadar pemindahan haba ke atas suhu api di dinding pembakar dibuat.Perisian SolidWorks digunakan untuk memodelkan masalah fizikal manakala perisian ANSYS Fluent digunakan untuk mensimulasikan kelakuan aliran menyantak api jet di kawasan yang terhad. Tiga faktor telah diperiksa, iaitu, ketebalan dinding pembakar, saiz muncung, dan kesan radiasi. Keputusan menunjukkan bahawa suhu adalah yang tertinggi di dinding pembakar apabila ketebalan dinding adalah 0.3 mm dan saiz muncung adalah 0.2 mm diameter, yang kedua-duanya pada 380,28 K. Keputusan juga menunjukkan bahawa jika kesan radiasi itu diambil kira, bacaan suhu yang lebih tinggi dijangka pada dinding pembakar. Keputusan juga menunjukkan bahawa kadar pemindahan haba yang tinggi berlaku apabila dinding nipis, saiz muncung besar, dan tanpa pertimbangan radiasi.

ABSTRACT

The Piper Alpha disaster incident occurred in 1998 clearly demonstrated the potential risk that could result in escalation of hazards when jet fires impinge on structures. Impinging jets flow has been widely used in many industrial applications as a technique to enhance heat and mass transfer in heating, cooling, and drying cases. Jet fires method usually occurs in house process equipment inside the offshore facilities where the probability of having flame impinging on vessels, pipework, and structural support is high. The nature of jet fires that propagate under flame-turbulence interaction flow and high pressure could cause failures to all equipment. This leads to the interest in studying jet fire characteristic in predicting the potential risk to structures and personnel of accidental ignition releases. This research focused on the study of combustion characteristics through the effect of impinging jet flames in a confined area. In this study, a numerical investigation on the heat transfer rate over the flame temperature at combustor wall is made. SolidWorks software is used to model the physical problem while ANSYS Fluent software is used to simulate flow behavior of impinging jet fire in a confined area. Three factors have been examined, i.e., the thickness of the combustor wall, size of nozzle, and radiation effect. The results show that temperature is the highest at the combustor wall when the wall thickness is 0.3 mm and the size of nozzle is 0.2 mm in diameter, which both at 380.28 K. The results also indicate that if the radiation effect is taken into account, a higher temperature reading is expected at the combustor wall. The results also show that high heat transfer rate occurs for thin wall, big nozzle size in diameter, and without consideration of radiation.

CHAPTER 1

INTRODUCTION

1.1 Overview

This chapter presents the introduction on the basic of impinging jet flow and its applications. This chapter also covers the type of flow field in impinged jet fire and properties of a jet fire. The hazardous effects of the jet fire are briefly described. Besides that, the concept and physical process in impinging jet flame on a surface are explained. In addition, problem statement, objectives, scopes of the project, and thesis outline are included in this chapter to give a clear overview about this research.

1.2 Introduction

Impinging jets flows haves been widely used in many industrial applications including drying of textiles, film and paper. This flow is typically employed when high heat transfer rates are needed. Impinging jets can be either in laminar or turbulent regime. In most cases, turbulent impinging jets would yields a higher heat transfer than laminar jets[1].

This type of flow has also been widely studied and reported frequently in journals. Still, prediction on the effects this type for flow is relatively lacking, as most part of the physics that govern the impinging jet flows have not been fully understood. Hence, more studies are needed especially the numerical studies in predicting fluid flows of the impinging jets.

Flow field in impinging jets can be divided into 3 main regions, i.e., free jet region, impinging region and wall jet region. Free jet region can be subdivided into two; i) potential core where the initial velocity of flow is maintained, and ii) shear layer where interaction with surrounding medium occur exchanging momentum and mass. Impinging region is the area where the stagnation point is located and wall jet region is the area where the flow exits the impinging region. A description physical process of impinging jets can be represented as below [2]:-

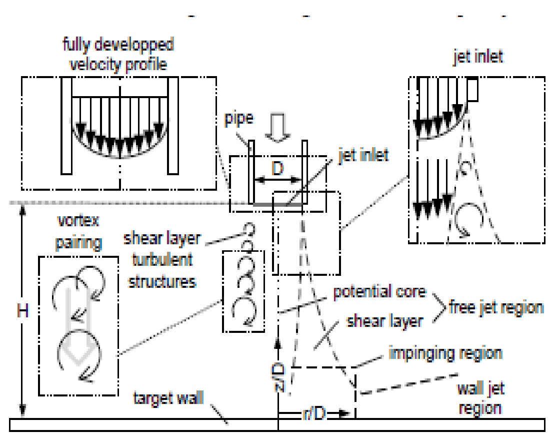


Figure 1.1 Physical processes in impinging jets [2]

An impinging jet fire is often considered as turbulent flame in real life case. It can be produced with ignition and combustion could be sustained with a continuous fuel release with significant momentum in one or numerous directions. Originally, jet fires can be formed from gaseous releases, two-phase mixture releases (i.e., liquid and gas) and releases from pure liquid inventories. Properties of a jet fire can be conditional depending on many factors including fuel composition, release conditions, release rate, ambient conditions, and geometry. These factors are important in determining the jet fire behaviour and can vary with one another [3].

Serious hazards due to fires and explosions often occur in the petroleum industry. This is because this industry holds many gas and oil installations, which are fire hazard. This means jet fires applications also involve in this category. Jet fires represent a significant risk, as it is frequently installed in house process equipment areas. This increase the probability impinging on vessels, pipework, and structural support therefore will cause failure to occur in the future.

Computational Fluid Dynamics (CFD) is a useful tool that can be employed to predict the heat transfer and fluid flow. CFD has become an essential tool to study cases involving combustion. This eases the understanding of combustion effect on plant and personnel for both industrial and consultancy context. Most researchers now uses this tool as it greatly reduces number of tests needed to be done, reduces physical labour needed to carry out experiment since no experiment is needed using CFD and most importantly can adversely reduce the cost associated with experiments.

1.3 Problem Statement

Research on impinging flame toward a surface has been performed from time to time. Installation of passive fire protection material (PFP) as an insulating barrier with fire-resistant walls is for containment for fires or slower their spread. This material can be tested from heat load in a furnace using the normal time-temperature conditions. However, this is different for an impinging jet fire case. Jet fire nature with the flame-turbulence interaction flow in it and operates under high pressure can cause wide range of failure modes. Few of the failure modes include of vessel failure, structural failure or further escalation when heat load towards the impinged surface. Efficient and precise CFD-tools analysis is therefore needed on study and make predictions the combustion characteristics through the effect of impinging flame toward a surface as it is important in validating a jet fire resistance test.

1.4 Objectives

A systematic study will be conducted by performing several simulations to achieve the objectives of this project. The specific objectives of this study are listed as below:

a) To simulate the flow behavior of impinging flames in a confined space area.
b) To determine the combustion characteristics especially the flame temperature on flame-impinged surfaces.
c) To explore the influence of the combustor wall thickness, size of nozzle and radiation on combustion characteristics of the impinging gas flames.

1.5 Scope of Project

In this project, it is all begin with reviewing the literature review to grasp what other people have done in the past related to this project research. Discovery from the literature review such as the theories, concept and idea that are collected and learnt what helps on completing this project. This research involves a numerical and simulation study on combustion characteristics through the effect of impinging gas flames in confined barrier area. Few parameters need to be determined in advance such as nozzle diameter, gas velocity and inlet temperature of gas in order to study the combustion characteristics.

Simulation is developed by using software to know the combustion characteristics through the effect of impinging gas flames in confined barrier area. This part of the research need to setup hardware and software that used for high computing simulation medium. ANSYS Workbench software is being used to run the simulation. Design of 3-D model will have to be done using SolidWorks first before meshing the model in ANSYS Fluent. Simulation can be run and then being analysed after simulation parameter setting for the 3-D model in ANSYS Fluent has been done. Discussions are then are made to clarify result obtain from the simulation.

1.6 Thesis Outline

This thesis contains five chapters that describe the details of this study. This dissertation signify completion of the final year project as partial fulfillment to graduate entitle CFD Calculation of Impinging Confined Gas Jet Flames/Fires.

Chapter 1 begins with an introduction chapter to give an overall insight on this topic. Here the problem statement, objectives and scope of project are described in detail. Next, Chapter 2 summaries the related existing works published in journals, books and reports, pertaining to the impinging confined jets. Main findings emerged from the existing studies give rise to the fundamental knowledge related to this topic and this assists the task to identify the gap for this topic.

Chapter 3 is the discussion on the project methodology. This chapter explains the overall simulation procedures flow step-by-step to give a clear view on how they are conducted. In the following chapter, which is Chapter 4, results and analysis from the simulations will be discussed and verified here. Finding from this project is being specifically discuss with the help of figure, pictures, graphs and table.

Last but not least, conclusion of the project will be made in Chapter 5. It concluded the overall project analysis into focused and concrete evidence of the accomplished objective. This chapter also provided some suggestions and recommendations that are given for improvement in future research.Flowchart of this thesis project outline is shown as in figure below:-

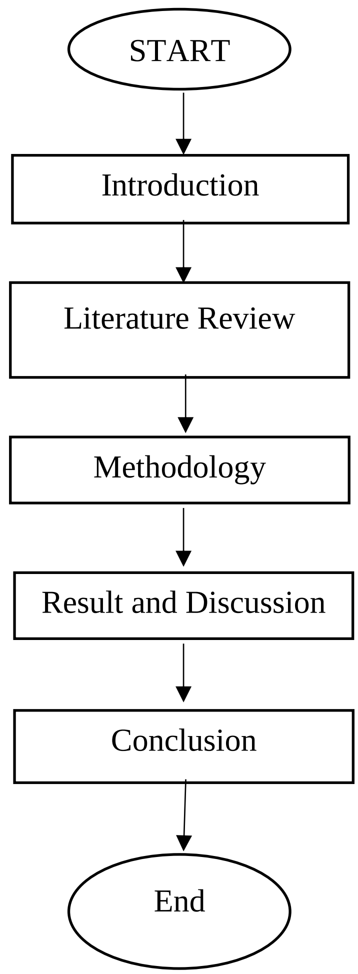


Figure 1.2 Thesis project outline

CHAPTER 2

LITERATURE REWIEW

2.1 Overview

Many researchers have shown interest in the field of combustion mainly about impingement of fire at surface characteristics during the past years especially after the Piper Alpha disaster in 1988 .They have carried out numerous laboratory experiments and simulations regarding this field. Their findings and suggestions are reviewed here. This chapter are classified into few categories such as jet impingement configuration, heat transfer study and techniques used for investigation.

2.2 Research Background

Risks of hydrocarbon jet fires on offshore installations that clearly demonstrated the potential for escalation of hazards when jet fires impinge on structures can be learned from the Piper Alpha disaster during 1998.This hydrocarbon jet fire could actually be controlled to lessen fire hazards by good design and effective protection. The method of controlled can be such as using passive fire protection system and water based system[4].

Main purpose of passive fire protection system is to contain fires or slower its spread. Material of passive fire protection is to provide solution for the purpose of thermal insulation barrier, endothermic building materials including gypsum and concrete and also novel solution based on alkali active binders. Concrete has been the always main material selection for fire protective system. There are cases sometimes where polypropylene fibres (PP) mix with concrete as reinforced material to be used as passive fire protection[5].

Jet fires usually occur at house process equipment in areas of the off-shore facilities where high probability of flame impinging on vessels, pipework and structural support to occur. Nature of fire jet behaviour can cause failures to all the equipment’s as it supplied high heat load to the equipment’s. That is why prediction of jet fire characteristic is important in reduce risk to structures and personnel of accidental ignition releases[3]. This can be achieved by the use of Computer Fluid Dynamics (CFD) as this has been the main tool in studying combustion case study in an industrial and consultancy context.

Application of CFD models for fire jet case has help solve the problem regarding turbulent flow combustion and heat transfer as it can provide any detailed information at any point in the computation domain. Such advantage can be seen research from UK Health and Safety Executive where they used CFD calculations on 5 open air and impinging natural gas and propane gas jet flames to obtained information on each study case flame behaviour[6].

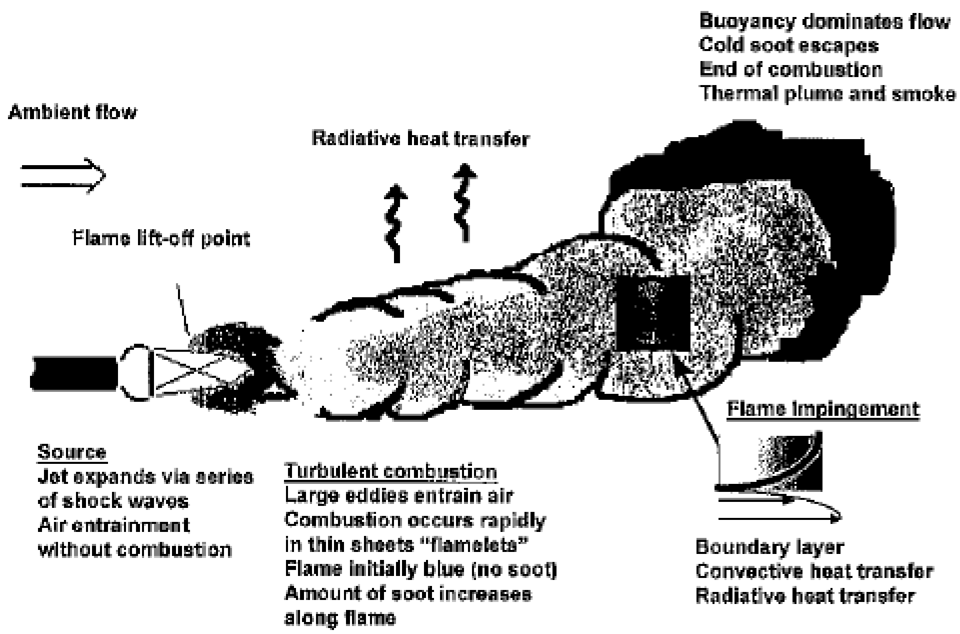


Figure 2. 1 Schematics of an impinging jet fire [6]

Validation using CFD for sub-models in determine calculation of free and impinging turbulent jet flames also has been made[7]. This research has been made to bring improvement for calculation of under-expended jet shock structures, flame lift-off and combustion in the main bulk of the flame. Simulation on a number of impinging jet fire tests with hydrogen gas has been performed by Health and Safety Laboratories (HSL) by using CFD tool[8].

2.3 Literature Review

2.3.1 Jet impingement Configuration

Impinging jet can be defined as a phenomenon in which the fluid exiting from a nozzle or orifice hits a wall or solid surface usually at normal angle .Impingement heat transfer is examine as a favorable heat transfer enhancement technique as it provides significantly high local heat transfer coefficient. It predetermines the jets impingement to be widely used in industrial applications where intensive heat transfer rates are needed such as combustor chamber wall, steam generators, electronic devices cooling and paper drying.

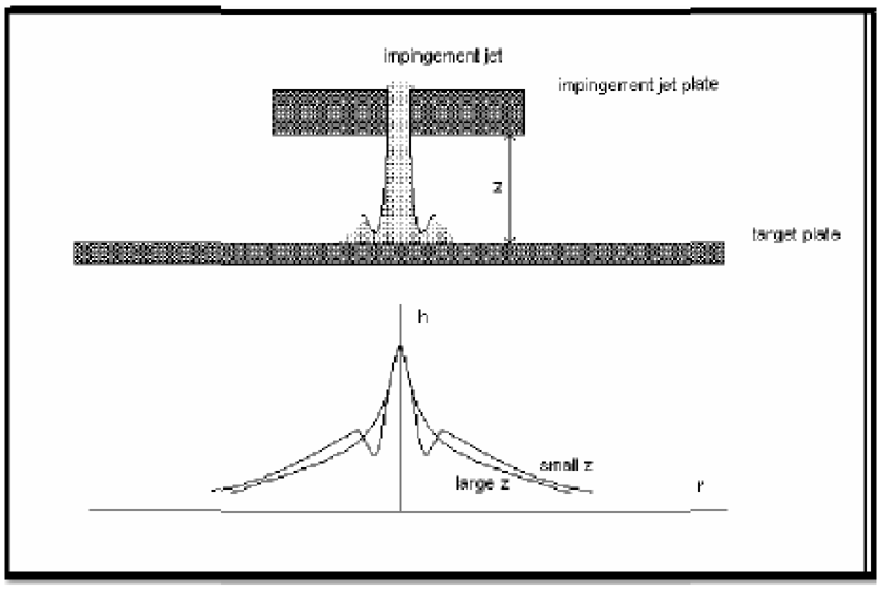


Figure 2. 2: Jet impingement heat transfer mechanism [9]

Impinging jets can be classified into two different flow configurations as either submerged impinging jets or free impinging jets. Both cases have different dynamics to it. In submerged impinging jets, the interrelationship of the jet and fluid leads to entrainment in the shear zone and growth of a potential core near the jet center line[10].This shear zone is unstable and it generates turbulence. In free impinging jets, entrainment of the surrounding fluid can be negligible as it does not have a substantial effect on the flow. This means that potential core is not relevant in this case.

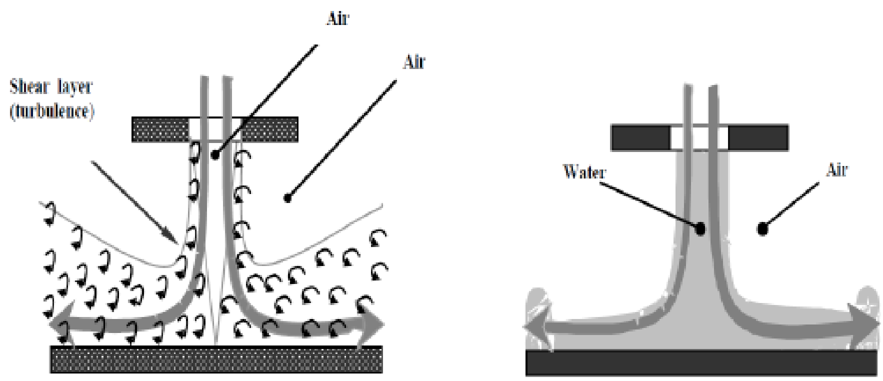


Figure 2. 3:Submerged jet; Free surface jet [11]

There is a further distinction between confined and unconfined jets. In confined jets, formation of re-circulation zone recirculation will likely to occur as fluid heated by the target plate in some instances can get recirculated and be entrained back into the impinging jet[12]. In unconfined jets, such case is not exist due to the heated fluid is not entrained back into the jet. This made the jet interacts with the ambient and otherwise quiescent surroundings that will cause higher heat transfer coefficients to achieve.

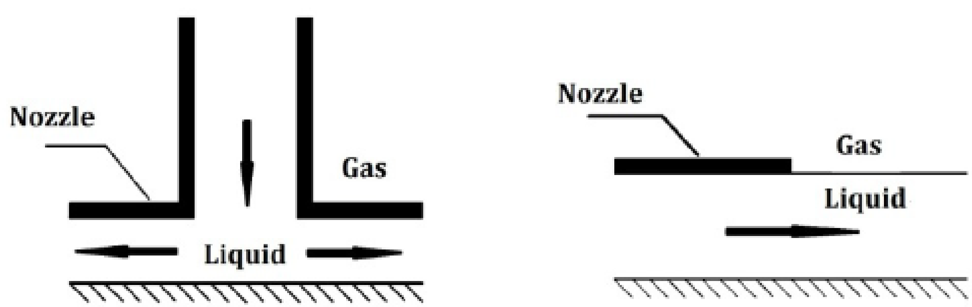
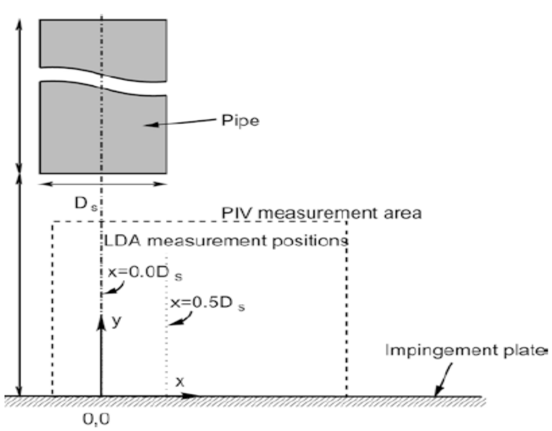


Figure 2. 4:Confined; Wall (free surface) [13]

There are two cases of configuration in terms of geometry which are a planar case with the jet issuing from a slot and an axisymmetric case with a round nozzle. Again, the dynamics of both cases are different. Take a round jet shape for example, it exhibit formation of axisymmetric vortex rings which are stretched during their convection along the wall. On the other hand for plane jets case, the vortices are formed as filaments parallel to the slot. They are created either in symmetric or antisymmetric mode on both sides of the jet causing these vortex filaments are not stretched[11]. There are also many others geometries that can take in consideration such as jets issuing from square, rectangular, elliptical nozzles and oblique jets.

In industrial applications, the jet impinge on a surface that are usually large in size and the common single jet flow is usually not sufficient for intensive cooling or heating it. In this case, an array of jets is usually more favorable. There is very little is known about the flow and turbulence structure in multiple jet configurations in comparison with single jet configurations. However, the additional factor due to interaction of neighboring jets, which, depending on their mutual distance, can have a dominant effect on heat transfer intensity, and especially on its distribution over the impinged wall surface[14].



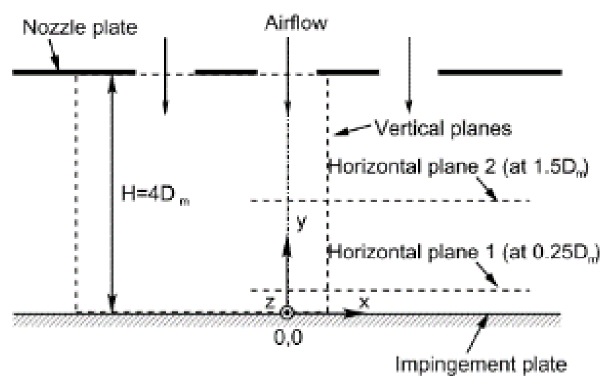


Figure 2. 5:: Flow configuration of jet [14]

(a)Single jet (b) Multiple jet

2.3.2 Heat Transfer study

Jambunathan conducted a detailed study of jet impingement cooling. Conclusion from this is that for local heat transfer coefficient correlation is a function of jet Reynolds number (Re), jet height-to-jet diameter ratio (H/d), radial distance-to-jet diameter ratio (r/d) and Prandtl number (Pr)[15]. Beitelmal research on two-dimensional impinging jets and related it with heat transfer in the stagnation point, stagnation region and wall jet region with approximate solutions, developed using simplified flow assumptions[16].

Koseoglu and Baskayabv investigate the heat transfer characteristics of confined circular and elliptical jet and the result is observed that an increase in jet to plate thickness reduces the difference between circular and elliptical flow fields[17]. Lee have research effect of nozzle diameter (1.36, 2.16, and 3.40 cm) on impinging jet heat transfer and fluid flow. Their result reported that local Nusselt numbers in the stagnation point region corresponding to $0<r/d<0.5$ are increased with increasing nozzle diameter[18]. Dong undergoes an experimental study to investigate the heat transfer and wall pressure characteristics of a pair of laminar air jet impinging vertically upon a horizontal flat plate. It can be concluded that the pressure distribution on the impingement plate and heat transfer from the jet to the plate were greatly influenced by the interference occurred between the two jets[19].

Gardon and Akfirat study on the variation in local heat transfer rate produced by impinging slot jets with changes in the free stream turbulence at the nozzle exit. In conclusion, some seemingly anomalous heat transfer phenomena could be explained in light of the turbulence intensity inherent in jets, and that circular jet with larger nozzle diameter produced higher heat transfer rate[20]. More recently, Zhou and Lee went research on heat transfer and fluid flow characteristics due to impinging air jets from sharp-edged rectangular nozzle. The local heat transfer distributions along the minor axis and average Nusselt numbers are correlated with turbulence intensity[21].

2.3.3 Techniques used for investigations

Jet impingement yield very high heat transfer coefficients. There are a lot of work documented on flow behaviour and heat transfer of impinging jet fire. Mirko Bovo [2] described the numerical modelling of impinging jets heat transfer. This thesis use to modal stationary impinging jets with different nozzle-wall distances and different Reynolds number. Effects of varying mesh density and topology also being studied. Transient simulations are to be said more suitable for studying the dynamic turbulent flow in impinging jet. Results from the modelling shows peacock grid topology and finer grids give better results with respect to convergence robustness and accuracy in result.

L.C.Shirvill, A.Ungut and A.D Johnson use CFD for calculation on impinging gas jet flame[6].They performed calculations on 5 open air and impinging natural gas and propane gas jet flames. Their main calculations is to provide information about flame, temperatures, velocities, dynamic pressure and convective and radiative heat transfer to flame-impinged surfaces. Their objective is also to develop jet fire tests of fire protection materials. Validation of the calculations were made against experimental measurements taken at test sites.

A.D. Johnson study both free and impinging turbulent jet flames by numerically and experimentally[7].He demonstrate calculations for an open air sonic 0.3 kg/s propane flame,2.5kg/s subsonic natural gas flame in open air and impinging on a 2m diameter cylindrical target. This thesis also bring improvement for calculation of under-expended jet shock structures, flame lift-off, and combustion in the main bulk of the flame. He also predict the convective heat transfer by using a practical model. Reliable predictions has been obtained as a result for under-expanded sonic jet structure, jet flame trajectory, flame lift-off position, flame temperatures,soot formation and external thermal radiation.

Deiveegan Muthusamy study modelling of hydrogen jet fires using CFD[8].This mainly to predict industrial fires efficiently and with good precision. He is using flame models for non-premixed flames, discrete transfer radiation model and soot model for this case study. Result of the study show that he face couple of issues regarding comparison of radiative fluxes due to released fire scenarios that quite transient and limitations of analysis in software use.

CHAPTER 3

METHODOLOGY

3.1 Overview

Thickness of the combustor wall, size of nozzle and radiation effect are some of the factors that affect the combustion characteristics in confined space, which in this project is the micro combustor. Thus, further investigation regarding three of the parametric study are conducted. In this chapter, numerical and simulation procedure that has been made will be described briefly in order to find the combustion characteristics in micro combustor. A flowchart is included to explain the whole progress of the project. Furthermore, description of simulation and numerical approach used for testing are also included.

3.2 Project Flowchart

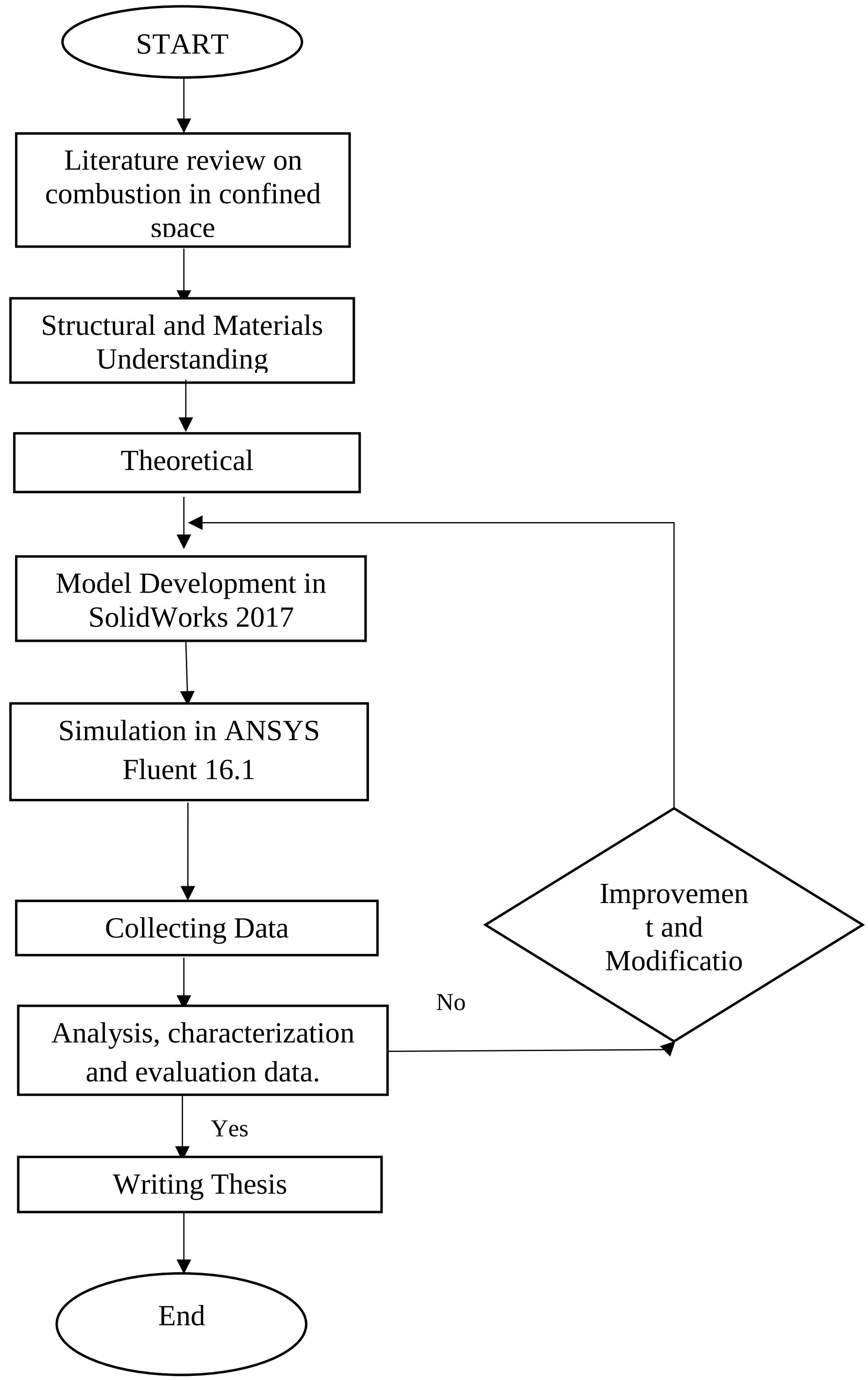


Figure 3. 1: Project Flowchart

3.3 Mathematical Models

3.3.1 Characteristics of combustion to be modelled

- **Flame lift-off**

  Combustion cannot occur at early part of impinging fire system because turbulent strain rate is too high at flammable part of shear layer. It will reduce the chance for a flame to exist if further downstream of the strain rate. This part located at the edge of the jet. Thus, flame lift-off must be consider and determined as this greatly increase air entrainment at initial part of the jet.

- **Turbulent Combustion**

  This model is required as most real life case involving impinge fire is in turbulent flow nature. Further moving in the direction in which the stream flows of the lift off point will make turbulent strain rate falls sufficiently. This cause the combustion fail to exist. Combustion that occur in the flamelets can be in two type of condition. This is either to be in premixed condition or diffusion condition. Premixed is a condition when the fuel and oxidant mixed before a reaction to occur while diffusion is a condition when fuel and oxidant come together in the flame. This is all depend on degree of mixing of the mixture in the region upstream of the flame lift of point. Furthermore, with this model consideration, some combustion model characteristics such as temperature and species concertation can be determined.

- **Radiation**

  Radiative heat transfer at an object play an important role in the modelling of a combustion system. This is in fact that all objects emit thermal radiation as long as their temperature is greater than Absolute zero but no object emits thermal radiation ideally. This model show equivalent role with convection heat transfer in turbulent diffusion flame. Radiative heat transfer mainly responsible for heat emitting from the flame to external objects.

3.3.2 Governing Equation

All of the flow equations are given according to ANSYS 15.0 Fluent Theory Guide in a coordinate notation.Study of combusting flow is solved as a steady state problem for this project.

3.3.3.1 Continuity

General form of continuity equation can be used for both incompressible and compressible flows. The source S_m is the mass added to the continuous phase from the dispersed second phase and any user-defined sources. Equation for general continuity equation can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \tag{3.1}$$

3.3.3.2 Momentum Conservation Equations

General form of momentum conservation equation can be used as an inertial (non-accelerating) reference frame. Equation for momentum conservation equation can be written as follows:

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\overline{\overline{\tau}}) + \rho \vec{g} + \vec{F} \tag{3.2}$$

Where:

p is the static pressure.

$\overline{\overline{\tau}}$ is the stress tensor.

$\rho \vec{g}$ are the gravitational body force and external body force.

$\vec{F}$ also contains other model-dependent source terms such as porous-media and user-defined sources.

3.3.3.3 Turbulence model

Navier-Stokes equations are the main equation to be consider in turbulence case.Solution variables in the instataneous Navier-Stokes equations are decomposed into the mean and fluctuating components in term of reynolds average.Velocity components of the equation and also for the pressure and other scalar quantities:-

$$u_i = \bar{u}_i + u'_i \qquad \phi = \bar{\phi} + \phi' \tag{3.3}$$

Where

$\bar{u}_i$ and u'_i are the mean and fluctuating velocity components.

ϕ scalar for pressure or energy.

This yeield the equations the esemble-averaged momentum equations which can be written in cartesian tensor form as:-

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u'_i u'_j}\right) \tag{3.4}$$

Both the equations above are called Reynolds Navier Stokes (RANS) equations.

3.3.3.4 Energy equation

Energy is be considered in this project, as there is heat transfer case included. Model use in this project has heat transfer occur within the fluid and solid regions. This is perfect for this project as ANSYS Fluent can solve problems ranging from thermal mixing within a fluid to conduction in composite solids. Other relevant physical models, supply thermal boundary conditions, and input material properties that govern heat transfer has to be setup too. General energy equation use is :-

$$\frac{\partial}{\partial t}(\rho E)+\nabla\cdot(\vec{v}(\rho E+p))=\nabla\cdot\left(k_{eff}\nabla T-\sum_j h_j\vec{J}_j+\left(\bar{\bar{\tau}}_{eff}\cdot\vec{v}\right)\right)+S_h \quad (3.5)$$

k_{eff} is the effective conductivity

$\vec{J}_j$ is the diffusion flux of species

First three terms on the right-hand side of represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively.

S_h includes the heat of chemical reaction, and any other volumetric heat sources that have being defined.

3.3.3.5 Radiative Heat Transfer Equation

Radiative heat transfer equation (RTE) for an absorbing, emitting, and scattering medium at certain position and in certain direction is:

$$\frac{dI(\vec{r},\vec{s})}{ds}+(a+\sigma_s)I(\vec{r},\vec{s})=an^2\frac{\sigma T^4}{\pi}+\frac{\sigma_s}{4\pi}\int_0^{4\pi}I(\vec{r},\vec{s}\,')\Phi(\vec{s}\cdot\vec{s}\,')d\Omega' \tag{3.6}$$

Where

$\vec{r}$ = position vector

$\vec{s}$ =direction vector

$\vec{s}\,'$ = scattering direction vector

s = path length

a = absorption coefficient

n = refractive index

σ_s = scattering coefficient

σ = Stefan-Boltzmann constant

I = radiation intensity, which depends on position and direction

T = local temperature

Φ = phase function

Ω' = solid angle

3.3.3.6 Premixed Flamelet Combustion Model

Zimont model is used in analysis for this premixed combustion case. Zimont turbulent flame speed is computed using a model for wrinkled and thickness flame fronts.

$$U_t = A(u')^{3/4} U_l^{1/2} \alpha^{-1/4} \ell_t^{1/4} \tag{3.7}$$

$$U_t = Au' \left(\frac{\tau_t}{\tau_c} \right)^{1/4}$$

Where

A =Model constant ,

u' = RMS(root-mean-square) velocity (m/s)

U_l =Laminar flame speed(m/s)

α =Unburnt thermal diffusity(m^2/s)

ℓ_t = Turbulence length scale

τ_t, = Turbulence time scale(s)

τ_c = Chemical time scale (s)

3.4 Numerical models

3.4.1 Introduction

Design of micro combustor is drawn by using SolidWorks 2017.SolidWorks is a CAD software, which is a mechanical design automation application that lets designers rapidly draw out ideas, experiment with features and dimensions and construct models and detailed drawings. Moreover, user get to sketch their own ideas and experience with different designs to create 3D models using the simple to learn graphical user interface. This software is widely used by students, designers and engineers to produce single and difficult parts, assemblies and drawings[22].

Main advantage of this software is that the graphics user interface is very easy to use and better in user-friendly term compared to other CAD solid modelling software. Apart from that, this software assist in creating 2D or 3D solid models without any complexity, faster and cost effective way. This solid modeller also contain motion, simulation, toolbox, photoview360, e-drawings and DWG editor. SolidWorks can import geometry from virtually any CAD/CAE software in Parasolid, ACIS, STEP, IGES, or native CATIA V4/V5 formats.

SolidWorks contain a variety range of applications in industries such as automotive, process plant, energy conservation, machinery, product design and many more other engineering services. This software help the industries through manage their design in various products and services. It also manage to provide testing of their products and services in cost effective manner. SolidWorks also contain few features such as parts and assembling of products, animation and rendering in 3D and simulation, which provide help for many engineering analysis case.

3.4.2 Modelling of the Geometry Design

All dimensions of the confined space area, which is the same, specified for the existing cylindrical micro combustor used to study its combustion characteristics that being conducted research by Pan Jianfeng that being used for this project[23].

Geometry of the cylindrical micro combustor chamber is 20mm long with an inner diameter of 2.4 mm. Combustor wall of the combustor has thickness about 0.3mm.This combustor wall is made of Sic material. This micro combustor include with a backward facing step as developed by W.M. Yang et al[24] that capable of enhancing mixing process and prolonging residence time. Nozzle that located at the left end of the combustor is being injected by premixed hydrogen–oxygen mixture into the combustor, which has 0.8mm in diameter size. Schematic diagram of the geometry of the cylindrical combustor is shown as in figure below:-

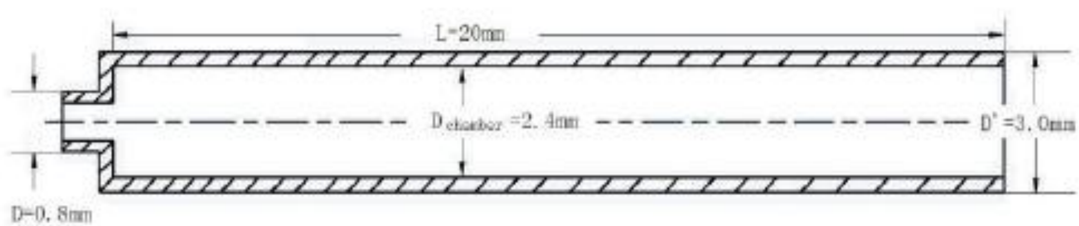


Figure 3. 2: Schematic diagram of cylindrical micro combustor [23]