EXPERIMENTAL STUDY OF HEAT TRANSFER CHARACTERISTICS OF ROUND AIR JET IMPINGING ON CONVEX SURFACE

HASAN ABDUALLATIF MUHALHAL

A dissertation submitted in fulfillment of the requirement for the award of the
Master Degree of Mechanical Engineering

Faculty of Mechanical and Manufacturing Engineering
University Tun Hussein Onn Malaysia

january 2014

Abstract

An experimental study has been carried out for jet impingement cooling on a semi-circular convex surface when jet flows were ejected from single round nozzle. An experimental investigation is performed to study the effects of jet-to-surface spacing and Reynolds number on the local heat transfer distribution between an convex surface and impinging circular air jet. Experiments have been conducted with variations of nozzle exit Reynolds number is Re= 11000, 23000 and 50000, and nozzle to surface distance is L/d= 1, 2, 6, and 10 to determine the heat transfer Coefficients under a constant heat flux condition. The results showed that the heat transfer coefficient *h* is higher at the stagnation region and decrease gradually at outlet region. The heat transfer coefficients increases by increasing Reynolds number and increases with L/d and reaches a maximum of L/d = 6 at Re = 11,000 and 23,000, and at L/d = 8 for Re = 50,000, respectively. Average heat transfer rates for impingement on the convex surface are found to be more enhanced than the flat plate results due to the effect of curvature. Comparisons between the present results and the existing experimental results have also been made.

ABSTRAK

Satu kajian eksperimen telah dijalankan untuk jet pelampiasan penyejukan pada permukaan convex separa bulat apabila aliran jet telah ditolak keluar dari muncung bulat yang tunggal. Penyiasatan eksperimen dilakukan untuk mengkaji kesan-kesan jarak jet ke permukaan dan nombor Reynolds pengedaran pemindahan haba tempatan antara satu permukaan convex dan mempengaruhi jet Penyaman Pekeliling. Ujikaji telah dijalankan dengan variasi muncung keluar nombor Reynolds adalah Re = 11000, 23000 dan 50000, dan muncung ke permukaan jarak tidak L/d = 1, 2, 6 dan 10 untuk menentukan pekali pemindahan haba di bawah keadaan fluks haba yang berterusan. Hasil kajian menunjukkan bahawa pekali pemindahan haba h adalah lebih tinggi di rantau stagnasi dan penurunan secara beransur-ansur di Cawangan Wilayah. Pekali pemindahan haba meningkatkan meningkatkan nombor Reynolds dan meningkat dengan L/d dan mencapai maksimum L/d = 6 di Re = 11,000 dan 23,000, dan di L/d = 8 bagi Re = 50,000, masing-masing. Kadar pemindahan haba purata pelampiasan di permukaan convex didapati lebih dipertingkatkan daripada keputusan plat rata disebabkan oleh kesan kelengkungan. Perbandingan antara keputusan semasa dan keputusan eksperimen yang sedia ada juga telah dibuat.

CONTENTS

TITLE ii

DECLARATION iii

DEDICATION iv

ACKNOWLEDGEMENT v

ABSTRACT vi

CONTENTS viii

LIST OF TABLES xi

LIST OF FIGURE xii

LIST OF SYMBOLS AND ABBREVIATIONS xiv

LIST OF APPENDICES ix

CHAPTER 1 INTRODUCTION 1

- 1.1 Introduction 1
- 1.2 Background Of Study 2
- 1.3 Problem Statement 3
- 1.4 Project Objectives 3
- 1.5 Scopes Of Study 4

CHAPTER 2 LITERATURE REVIEW 5

- 2.1 Jet Impingement 5
  - 2.1.1 Configurations Of Impinging Jets 6
  - 2.1.2 Jet Flow Characteristics 7

2.2 Effect Of Jet To Target Plate Spacing 8
2.3 Effect Of type of coolant 9
2.4 Effect of Nozzle Geometry 9
2.5 Effect of type of nozzle 11
2.6 Effect Of jet Angle 11
2.7 Effect of nozzle diameter 11
2.8 Effect of Reynolds number 12
2.9 Previous Studies 13

CHAPTER 3 METHODOLOGY 14

3.1 Introduction 14
3.2 methodology Flow Chart 14
3.3 Experimental Apparatus 16
3.4 Experiment Equipment 17
    3.4.1 Power Supply and 3 phase inverter 17
    3.4.2 Centrifugal Blower 18
    3.4.3 Control valve and Pipe 19
    3.4.4 Thermocouple and thermocouple data logger 20
    3.4.5 Settling Chamber 21
    3.4.6 Circular Nozzle 22
    3.4.7 The convex surface 23
    3.4.8 Voltage Regulator 24
    3.4.9 Digital Multimeter 25
    3.4.10 Pitot tube 26
    3.4.11 The Air flow Meter 27
    3.4.12 Stainless Steel Foil 28
    3.4.13 Infrared Thermal camera 29
3.5 Experimental Procedures 30
3.6 Data collection 31
3.7 calculation method 32
    3.7.1 Reynolds Number 32
    3.7.2 Nusselt Number 32

3.7.3 Heat Transfer Coefficient 33
3.6.3 Heat Flux 34

CHAPTER 4 RESULT AND DISCUSSION

4.1 Introduction 35
4.2 Heat Transfer Distribution Results 36
    4.2.1. The distribution of Nusselt Number for L/d=1 37
    4.2.2. The distribution of Nusselt Number for L/d=2 37
    4.2.3. The distribution of Nusselt Number for L/d=6 38
    4.2.4. The distribution of Nusselt Number for L/d=10 39
    4.2.5. The distribution of Nusselt Number for Re=50000 39
4.3 Discussion 40
    4.3.1 Effect of Reynolds numbers on heat transfer 40
    4.3.2 Effect of nozzle to target distance on heat transfer 43

CHAPTER 5 CONCLUSION AND SUGGESTION

5.1 Introduction 46
5.2 Conclusion 46
5.3 Suggestion 47

REFERENCES 49
APPENDIXES 51

LIST OF TABLES

3.1 Specification of the inverter 18
3.2 Specification of the blower 19
4.1 Heat transfer coefficient of stagnation point 44
4.2 Temperatures of stagnation point 45

LIST OF FIGURES

2.1.a Submerged Impinging Jet 6
2.1.b Free Impinging Jet 6
2.2 Jet Impingement zones 8
3.1 Methodology Flow Chart Of Project 1 15
3.2 Schematic for experimental set-up 17
3.3 Centrifugal Blower 19
3.5 Tc-08 Thermocouple Data Logger 21
3.11 Thermocouple K-type 22
3.4 Settling Chamber 22
3.5 Nozzle geometry 23
3.9 The convex surface 24
3.8 Voltage Regulator 25
3.7 Digital Multimeter 26
3.11 Pitot Tube Theory 27
3.6 The Air Flow Meter 28
3.12 Stainless steel foil 29
3.13 Infrared Thermal camera 30
4.2 Nusselt number distribution along the vertical centre-lines with L/d=1, Re= 11000, 23000 and 50000. 37
4.3 Nusselt number distribution along the vertical centre-lines with L/d=2, Re= 11000, 23000 and 50000. 37
4.4 Nusselt number distribution along the vertical centre-lines with L/d=6, Re= 11000, 23000 and 50000. 38
4.5 Nusselt number distribution along the vertical centre-lines with

LIST OF SYMBOLS

ρ - Density

Re - Jet Reynolds number based on nozzle diameter

d - Nozzle diameter (cm)

V - Averaged velocity at the nozzle exit

γ - Kinematic viscosity (m^2/s)

h - Heat transfer coefficient ($w/m^2.c^\circ$)

L - Distance from nozzle to target plate (cm)

K - Thermal conductivity (w/m-k)

Q - Heat flux (w/m^2)

Twall - Wall temperature (c°)

Tjet - Jet temperature (c°)

A - Stainless steel area (m^2)

Nu_{st} - stagnation point Nusselt number.

CHAPTER 1

INTRODUCTION

1.1 Introduction

Impinging jets provide an flexible and effective way to transfer energy in industrial applications. It has been used for elements exposed to high temperatures and high heat flux because the efficiency is high, and the hardware is simple and easy to adjust the location of interest. The jet impingement cooling has been effectively used to eliminate excessive thermal load near the leading edge of gas turbine blade inner surface, most especially within the high pressure turbine.

There are many parameters that affect jet impingement cooling that are jet configuration, jet size, Reynolds Number, Fluid temperature, Initial surface temperature, Jet exit to surface spacing, jet diameter, Surface cooling time, Surface orientation and roughness etc. The coolant fluids used in jet impingement are mainly air and waterand sometimes fluorocarbons (FC) and Freons.

Impinging jets have been used to transfer heat in diverse applications. Some of the common areas of such applications are:

1. A single jet or a row of jets is used in spot cooling of turbine blades, especially in high performance engine.
2. A blow torch utilizes a jet of combustible gaseous mixtures to deliver the required heat released in combustion to the work.
3. A rocket is powered by the momentum change carried by a jet
4. A jet of water is used by a dentist to clean and polish the teeth of a patient.
5. In the glass industry, a matrix of jets is used for cooling intricate molds.

1.2 Background of study

A jet is defined as a quantity of fluid being discharged from a nozzle or orifice into a medium at high velocity and hence a high kinetic energy. Jet impingement is a best method for cooling or heating solid surfaces. Thus most application of jet impingement occur in industries where high cooling and heating rates are required such as cooling of stock material during material forming processes, heating of optical surfaces for defogging, cooling of critical machinery structures, drying of food products, textiles, films and paper processing of some metals and glass and due to the demand for high powered electronics has increased and we require faster, smaller, and reliable electronic components. These lead to high heat flux that must be removed. The traditional cooling techniques such as heat sink, heat sink with fan, reached their limit. Jet impingement has also become a viable for high-powered electronic thermal management solutions, and many other industrial processes. When the jet strikes the target surface it forms a very thin stagnation-zone boundary layer which offers little resistance to heat flow.

1.3 Problem statement

There are numerous studies of impingement heat transfer have been done on a jet impingement of a flat plate or concave surface. But there are not many studies concentrating on a jet impingement on convex surface.

Gas turbine has low thermal efficiency, compared to steam turbine because of the limitations in the gas turbine inlet temperature. Increasing the gas turbine's inlet temperature will greatly enhance the performance of the gas turbine, but the turbine's blade will be exposed to high gas stream. Therefore researches have endeavored to help increasing the turbine inlet temperature by researching the turbine blade cooling to reduce the surface temperature of the blade and to reduce the temperature average in the cross section of the blade thus increase the life time expectancy of the blade.

On the other hand, the advent of high-speed computers, the current method of heat removal, involving extended surfaces and fan arrays, frequently used in electronics is clearly insufficient, especially as the electronic components become more and more powerful, dissipating more heat, whereas the space around these components continues to be reduced due to miniaturization trends.

1.4 Project objectives

This study has three main objectives which are:

1. Measure the heat transfer coefficient on convex surface.
2. Define the relationship between the heat transfer coefficient with the distance between the nozzle exit and the target surface.
3. Define the relationship between heat transfer coefficient with Reynolds number.

1.5 Scope Of Study

The scopes of this project comprised the boundaries of project study. The jet impingement cooling systems are wide range of study. Many scopes should be bound in order to make this project achieve the objectives.

1. The air was used as cooling fluid that impinging from single round nozzle to convex surface.
2. The jet exit Reynolds number in range of $(11000 \leq Re \leq 50000)$.
3. The initial temperature of the surface was 50 c̊.
4. Nozzle to surface spacing ratio in ranges $L/d = 1,2, 6,$and 10.
5. The nozzle diameter was constant with 3.2 cm (inner diameter).
6. The diameter of the convex surface is 30 cm with width 30cm and thickness 5mm.
7. The surface directly heat by applying an electric current through the stainless steel foil of thickness 0.025 mm.
8. The total heat loss was neglected.
9. Using infrared camera to measure temperatures along the circumferential direction with 2°degrees separation.

CHAPTER 2

LITERATURE REVIEW

2.1 Jet Impingement

Impinging jets have attracted much research from the viewpoint of the fluid flow characteristics and their influence on heat transfer. The jet flow characteristics are highly complex and consequently the heat transfer from a surface subject to such a flow is highly variable. Numerous jet configurations have been studied and numerous experimental parameters exist that influence both the fluid flow and the heat transfer. The objective of this study is to conduct an investigation of the heat transfer mechanisms for an impinging air jet.

Much of the research presented in this chapter has been conducted as independent investigations into jet impingement fluid flow and impinging jet heat transfer. This chapter details the research concerned with the jet fluid flow characteristics. This includes all the aspects of the flow that have been shown to influence the heat transfer. Also this chapter describes the research conducted into heat transfer to an impinging jet. The variation of the heat transfer with various test parameters is discussed and related to what is known of the fluid flow.

2.1.1 Configurations of impinging Jet

There are two different flow configurations, which are submerged impinging jets and free impinging jets, when the fluid issuing from the nozzle is of the same nature as the surrounding the jet is termed submerged. When the fluids are of a different nature and the jet is not confined and the medium is at rest, the jet is termed free. When the flow of fluid of a free jet impinges upon a surface, the jet would be deflected and move outwardly along the surface. In submerged jets, a shear layer forms at the interface between the jet and the surrounding fluid. This shear layer is unstable and it generates turbulence. In free jets, the turbulent motion in the shear layer does not have a substantial effect on the flow.

In terms of geometry, there are two main types, when the jet issues from a slot, it is a two dimensional plane jet. When the jet issues from a circular nozzle or orifice, it is an axisymmetric radial jet. The present investigation deals with the latter axisymmetric type.

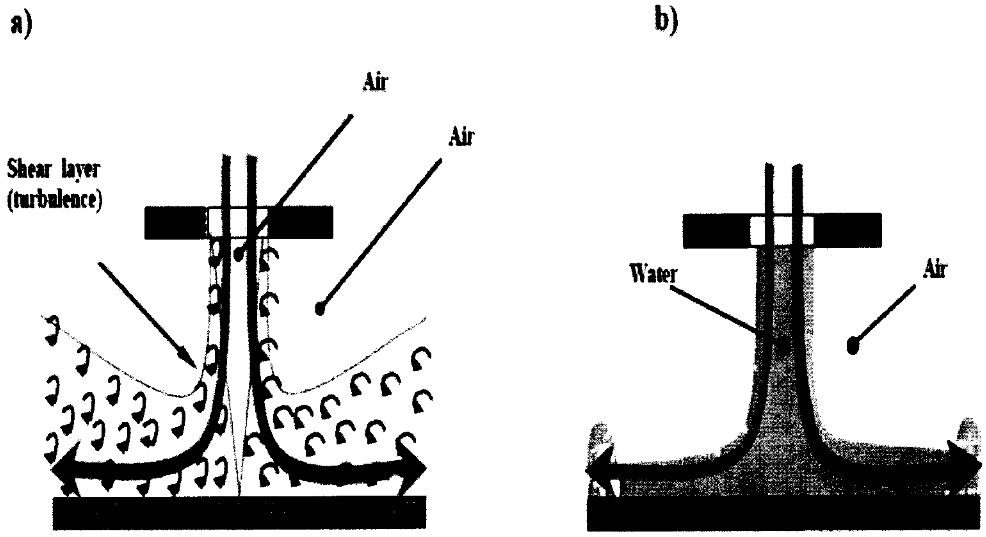

Figure 2.1: a) Submerged jet; b) Free impinging jet [15]

2.1.2 Jet flow characteristics

The impinging jet can be divided into three main regions. These are illustrated in figure 1.2. Firstly the free jet is the region that is not influenced by the impingement surface, this exists beyond approximately 1:5 diameters from the impingement surface. A potential core exists within the free jet region. In the potential core region the axial velocity remains almost equal to that at the jet entry, and the turbulence intensity level is relatively low. A shear layer exist between the potential core and the ambient fluid where the turbulence is relatively high and the mean velocity is lower than the jet exit velocity. Beyond the potential core the shear layer has spread to the point where it has moved through the centreline of the jet. At this stage the centreline velocity decreases and the turbulence intensity increases.

Secondly, the impingement region or the stagnation region is characterized by an increased static pressure as a result of the sharp decrease of mean axial velocity. Upon impingement the flow deflects and starts to accelerate along the impingement surface. The end of the impingement region is the location where the pressure gradient at impingement surface becomes negligible. for the stagnation zone. The flow field can be divided into an outer region of essentially in viscid flow and an inner viscous boundary layer region. The stagnation zone includes the stagnation point where the mean velocity is zero and within this zone the free jet is deflected into the wall jet flow.

Finally. The wall jet region is characterized by higher velocities surrounded by lower velocities on the either side, one due to the presence of the wall and the other due to the stagnant fluid. The boundary layer grows along the impingement surface. The wall jet zone extends beyond the radial limits of the stagnation zone.

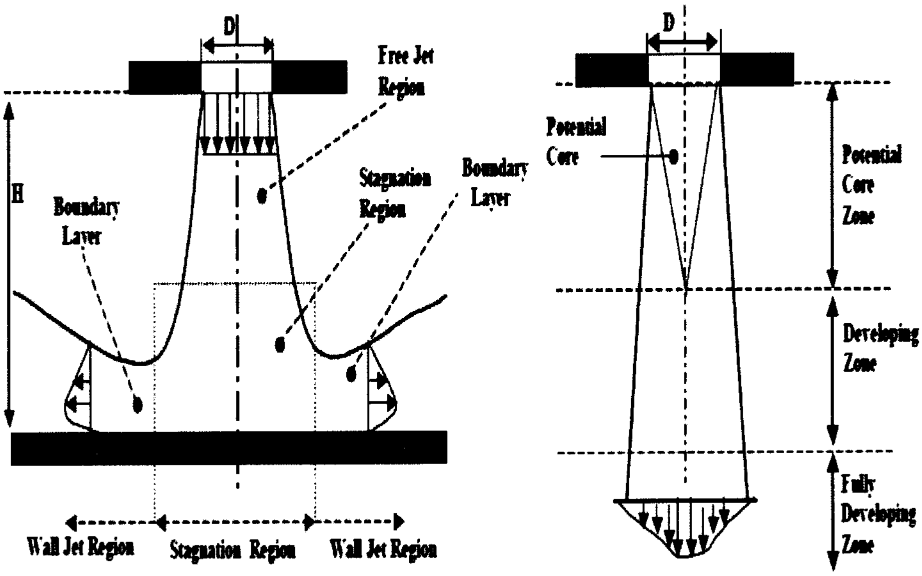

Figure 2.2: Impinging jet zones [15]

2.2 Effect of nozzle to target plate spacing

A number of studies have considered effect of nozzle exit to target plate Spacing on heat transfer characteristics.

Chitranjan agarwall, j. K. (2012) studied the effect of nozzle-to-surface spacing and Reynolds number on the heat transfer in cooling of electronic components by an impinging submerged air jet. Reynolds number based on nozzle diameter d is varied between 6000 to 23000. Distance from the tip of the nozzle-to-surface of the electronic components H varied from 2 to 10 nozzle diameters. Experiments are conducted with nozzle diameter of 5mm. It is shown that for different Reynolds numbers, the surface temperature can be significantly decreased by reducing the jet diameter. The effect of nozzle aspect ratio is less evident as the spacing nozzle- to- surface of the electronic component is increased. The jet Reynolds number does not control the role of nozzle aspect ratio on heat transfer rate. The heat transfer rate increases as the jet spacing decreases owing to the reduction in the impingement surface area [5].

M. Anwarullah, a. V. (2012) studied the transient surface cooling of a hot stainless steel surface is obtained by a water jet impingement cooling method. The surface is electrically heated up to the initial temperature of 800 °C. Water flow rate is kept constant so that jet Reynolds number remains at Re=24000. Nozzle exit to surface spacing has been changed in a range of z/d = 4-16. It is observed that for larger drop in surface temperature, the cooling time is not affected by the change in nozzle exit to surface spacing. However, for lesser surface temperature drop particularly at higher surface temperature, surface cooling time with nozzle exit to surface spacing is approximately 15 % lower [8].

2.3 Effect of type of coolant

Ali, o. Z. (2012)studied the heat transfer between a vertical round alumina-water Nano fluid jet and a horizontal circular round surface. Different jet flow rates, jet nozzle diameters, various circular disk diameters and three nanoparticles concentrations (0, 6.6 and 10%, respectively) are used. The experimental data show an increase in the Nusselt number that can reach up to 100% for some higher concentrations. This result indicates that using Nano fluid as a heat transfer carrier can enhance the heat transfer process. It was also found that presenting the data in terms of Reynolds number at impingement jet diameter can take into account the effect of jet heights and nozzle diameters. The data have also indicated that increasing heating disk diameter decreases the heat transfer coefficient [2].

2.4 Effect of nozzle geometry

The jet nozzle geometry has a significant effect on the heat transfer to the impinging air jet. This is due primarily to the influence the nozzle has on the turbulence level in the main jet flow. In addition to this, the nozzle geometry influences the entrainment of ambient fluid, the spread of the shear layer and the length of the potential core.

Luis a. Brignoni, s. V. (2000) studied the effect of changing the nozzle geometry on the pressure drop and local heat transfer distribution in confined air jet impingement on a small heat source. Heat transfer and pressure drop measurements obtained with chamfered nozzles were compared to those obtained with a square-edged (non-chamfered) nozzle of the same diameter for different turbulent Reynolds numbers. nozzle-to-target spacing, chamfer angles and chamfer lengths. The ratio of average heat transfer coefficient to pressure drop was enhanced by as much as 30.8% as a result of chamfering the nozzle, with narrow chamfering providing the better performance. Compared to square-edged nozzles, chamfering the nozzle inlet produces significant reductions in pressure drop; the average heat transfer coefficient, on the other hand, is not strongly affected [7].

Ahmed1, m. A. (2012) studied the effect of nozzle geometry on the heat transfer characteristics and local Nusselt number distributions of circular nozzle on a heated flat plate for various nozzle geometries. Experiments have been conducted with variation of exit Reynolds number, Re, is varied from 6000 to 40000 and plate surface spacing to nozzle diameter, H/d, in the range of $1 \leq H/d \leq 6$ for single nozzle with square edge (non-chamfered) and chamfered nozzles of the same diameter, 5 mm. The chamfered length, Lc is varied from 1 mm to 3.65 mm with constant chamfered angle, $\theta = 60$ for each nozzle configuration. The results indicate that the stagnation Nusselt numbers have the highest value for square edge inlet nozzle when compared with other nozzle configurations. The Stagnation point Nusselt number is highest valueat separation distance H/d = 1 for all nozzles configurations. And For a range H/d from 1 to 4, the stagnation Nusselt number values are nearly constant, and it is higher than at separation distance H/d =6 for all nozzle configurations tested [1].

2.5 Effect of jet angle

B.balakrishna1, s. I. (2013)carried out on conical nozzles with different divergence angles using supersonic gas flows though the nozzle. In this investigation the nozzles are modeled with axisymmetric condition and modeling is carried out using Gambit software. The flow characteristics investigated by considering the throat diameter and exit diameter of nozzle is same for all cases. The flow parameters, such as pressure ratio, Mach number of the flow at the nozzle exit, and the area of nozzle exit ratio are considered for the simulation studies. Based on the results obtained, the flow characteristics behave differently at different divergent angles. At 7.21° divergent angle the maximum pressure in the nozzle which is obtained to 120 Pa. Whereas at 40° divergence angle the temperature is increased to 787 °k, the turbulent intensity of the nozzle increased to 13100 m^2/s^2 and the velocity magnitude of the nozzle increasing to 1130 m/s [3].

2.6 Effect of diameter of round nozzle.

ArunJacob, L. R. (2013)has been conducted to see the effect of the geometrical parameters such as jet diameter (D), jet to target spacing (Z) and ratio of jet spacing to jet diameter (Z/D) on the heat transfer characteristics. The values of Reynolds numbers considered are in the range 7000 to 42000.The conclusions derived out of the present study are (i) the optimum value of jet spacing to jet diameter is 5. (ii). for the same value of Reynolds number the heat transfer coefficient increases by about 28 percent if the diameter of the jet is increased from 1mm to 2mm but the volume flow rate increases by 100%. (iii) Correlations are proposed for Nusselt number in terms of the Reynolds number [13].

2.7 Effect of Reynolds number.

The Reynolds number is the most important dimensionless number in fluid flow through pipes.For small *Re* numbers, viscous forces are dominant. Fluctuations in flow are damped out therefore it is a laminar flow. For large *Re* numbers, inertial forces are important, fluctuations in flow become amplified which means that the flow is turbulent. For every geometry there is a critical *Re* at which transition to turbulence The Reynolds number is a quantity which engineers use to estimate if a fluid flow is laminar or turbulent. This is important, because increased mixing and shearing occur in turbulent flow. This results in increased viscous losses which affects the efficiency of hydraulic machines. It is defined by:

$$\mathrm{Re} = \frac{Vd}{\nu} = \frac{\rho V d}{\mu} \tag{2.1}$$

ρ = Density

V = Free stream velocity

d = diameter of the pipe

μ = dynamic viscosity

ν= kinematic viscosity = μ/ρ

Agarwal, c. (2012)studied a hot stainless steel surface of 3 mm thickness was cooled with the round water jet of 2.5 mm diameter from a sharp edge nozzle. The target surface was initially heated up to 800 °C and the target surface to nozzle exit distance is kept in a range of z/d = 4-16. Water flow rate is varied such that the jet Reynolds number remains in a range of 5000-24000. In this investigation it has been observed that for the stagnation point the surface cooling is not affected by the change in nozzle exit to surface spacing. However, with the jet Reynolds number of 24000 the time taken to attain the 100 °C by the hot surface is 30% less as compare to the jet Reynolds number of 5000. And although the percentage rise in jet Reynolds number is not reduces the surface cooling time in equal proportion. The effect of rise in nozzle exit to surface spacing on surface cooling has also not witnessed with the sharp edge nozzle jet impingement cooling also.

2.8 Previous studies.

C.h. Lee a, k. L. (2007)studied the effects of concave hemi-spherical surface with an inclined angle on the local heat transfer from a turbulent round impinging jet. The liquid crystal transient method was used in this study. The Reynolds numbers 11,000, 23,000 and 50,000,and under five various distances between impinging jet and the surface (L/d) and fourtilt angles a. The correlations of the stagnation point Nusselt number according to Reynolds number, jet-to-surface distance ratio and dimensionless surface angle were also presented. In the stagnation point, in terms of Re^n, where n ranges from 0.43 in case of $2 \leq L/d \leq 6$ to 0.45 in case of $6 < L/d \leq 10$, there roughly appears to be a laminar boundary layer result. The Nusselt number at stagnation point decreases as the tilt angle is increased when Re is fixed. Among all the tilt angles, the result was maximized when L/d = 6.The 2nd peak point occurred with the inclined impinging jet on concave surface appearing far from the stagnation point, as Re value decreases or the tilt angle increases, and as L/d increases, it appeared near the stagnation point.Maximum heat transfer coefficient occurs far from the stagnation point as the tilt angle increases or the distance between the nozzle and collision surface decreases , and when L/d = 2, a = 40^0 and in this study, maximum distance moved was approximately0.7 times the diameter of the nozzle. Compared to Nu value at the stagnation point, the increasing rate of maximum Nu value rises as the tilt angle rises, and the maximum increasing rate was approximately 31.8% under the condition L/d = 2 and a = 40^0 [4].

CHAPTER 3

METHODOLOGY

3.1 Introduction

Methodology is defined as the guideline system for solving this problem, with specific components such as methods, techniques and tools. Methodology involves the design of Experiment, and detail experimental design. This chapter generally discusses methodology of the project, to ensure that the objectives and scope of the study is achieved. with a focus on measuring heat transfer coefficient, experiment and devises. Relevant data collection is done in order for further research analysis in subsequent chapter. This section contains the methodology to conduct this study.

3.2 Methodology flow chart

Flowcharts are used in designing and documenting complex processes, thereby it help to understand the process and find flaws. The methodology flowchart is illustrated in figure 3.1 A flow chart was constructed to assist the planning process for the completion of this research process smoothly.

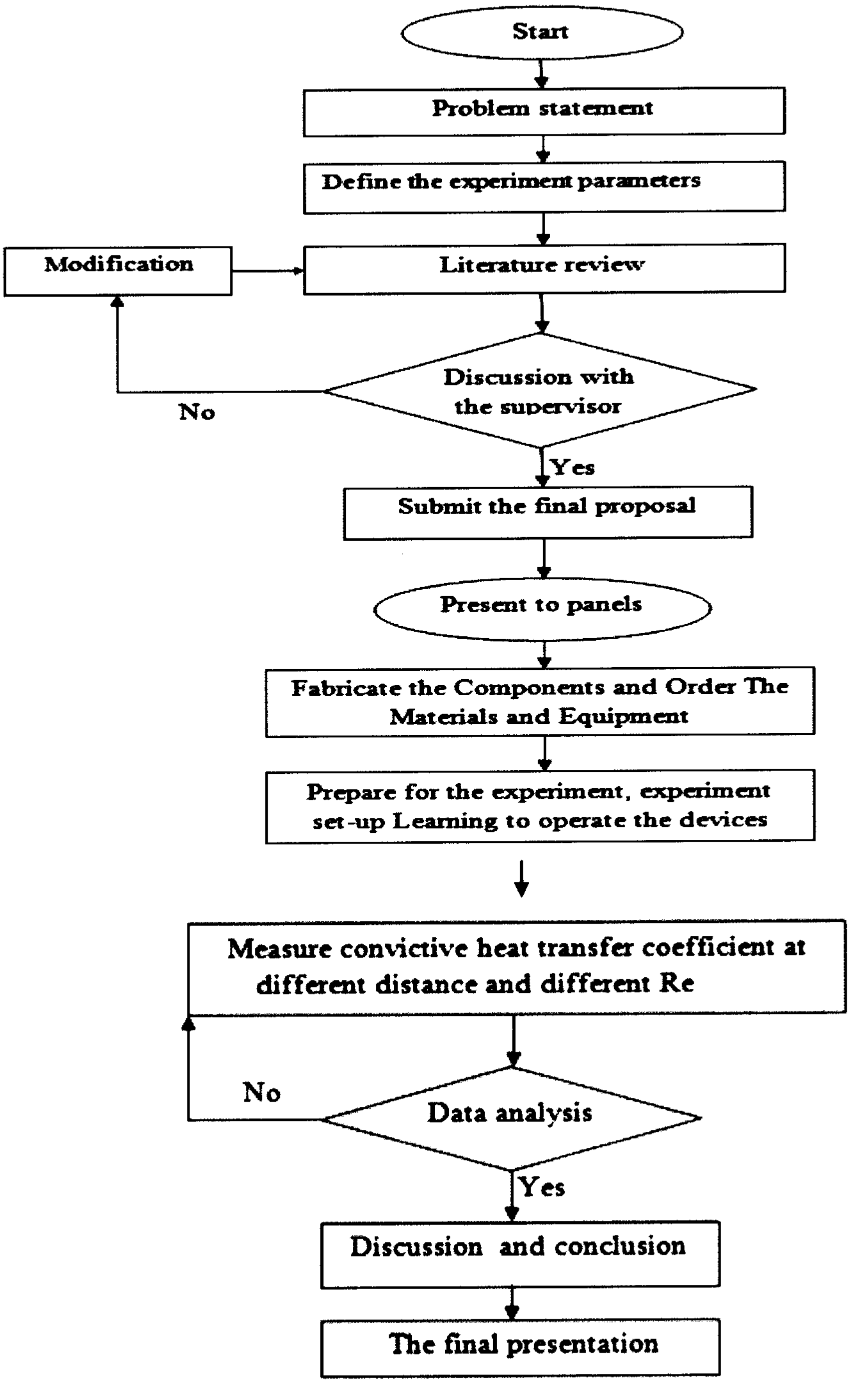

Figure 3.1: Methodology Flowchart

3.3 Experimental apparatus

The schematic diagram of experimental apparatus is shown in Fig.3.2. It consists of:

1. Power supply.
2. Centrifugal blower.
3. Control valve and pips.
4. Settling chamber.
5. Nozzle.
6. Air flow meter.
7. Stainless steel foil.
8. Voltage regulator.
9. Digital multimeter.
10. Convex surface.
11. Thermocouple data logger.
12. K-type thermocouple.
13. Infrared thermal camera
14. Pitot tube
15. Personal computer.

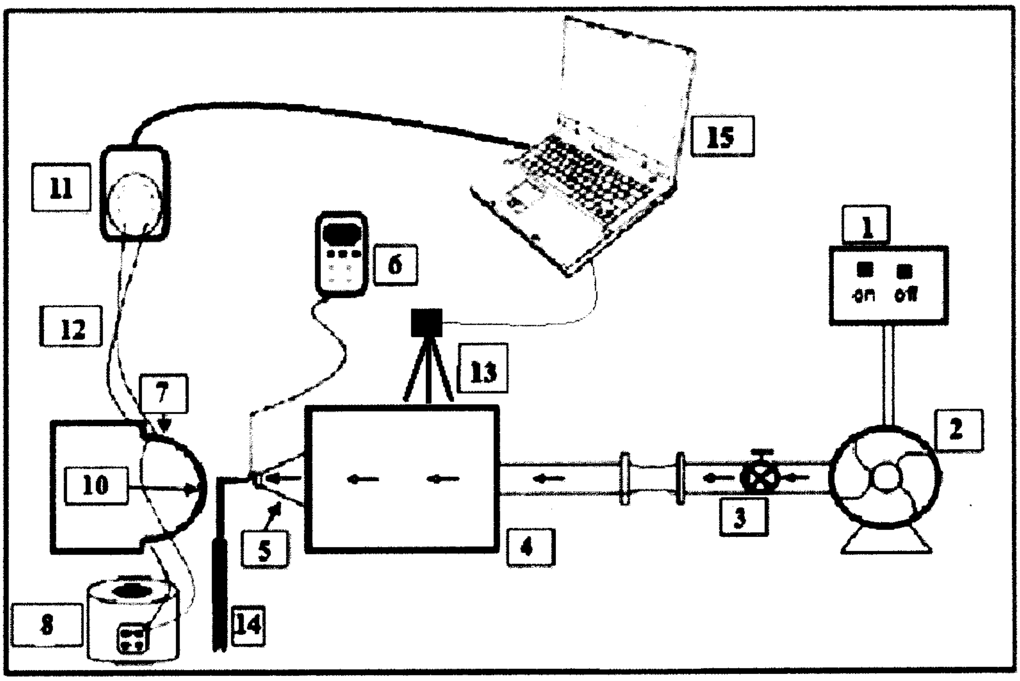

Figure 3.2: Schematic for experimental set-up

3.4 Experiment equipment

The following equipments will used in this experimental works. and it is very important to ensure that all of this equipment is in good condition, and able to give accurate results before running the experiment.

3.4.1 Power supply and 3 phase inverter

An inverter is used to control the speed of the blower motor to allow continuously regulated flow rate. The frequency of the inverter on the display screen indicates the blower speed. Table 3.1 shown the specification of 3 phase inverter.

Table 3.1: Specification of the inverter.

Type	Holip
Model	HL PA07543B
Frequency (input)	50 Hz
Frequency (output)	0.5 - 400 Hz
Voltage	415 V

3.4.2 Centrifugal blower

A centrifugal blower is a mechanical device for moving air or other gases. These fans increase the speed of air stream with the rotating impellers. They use the kinetic energy of the impellers or the rotating blade to increase the pressure of the air/gas stream which in turn moves them against the resistance caused by pipes. Centrifugal blowers accelerate air radially, changing the direction of the airflow. They are sturdy, quiet, reliable, and capable of operating over a wide range of conditions. Centrifugal blower is a constant volume device. This means that the air velocity in a system is fixed even though mass flow rate through the fan is not. Table 3.2 shown the specification of the blower. Figure 3.3 shown the air blower (http://search.nasa.gov).

Table 3.2: Specification of the blower

Type	Teco
Model	286
Motor	1.5 hp
Speed	1435 rpm
voltage	220 – 240 V

Figure 3.3: Air blower

3.4.3 Control valve and pipe

Control valves are valves used to control conditions such as flow, pressure, temperature, and liquid level by fully or partially opening or closing in response to signals received from controllers that compare a "setpoint" to a "process variable" whose value is provided by sensors that monitor changes in such conditions.

The opening or closing of control valves is usually done automatically by electrical, hydraulic or pneumatic actuators. Positioners are used to control the opening or closing of the actuator based on electric, or pneumatic signals.

3.4.4 Thermocouple and thermocouple data logger

Thermocouples are a common choice for temperature measurement due to its reliability and accuracy the Type K thermocouple is used extensively at temperature range: –270 to 1260°C, and its accuracy range ± 2.2°C or ± 0.75%, and has special limits of error in the range of ± 1.1°C or 0.4%. Type K has an exponentially increasing voltage the differences in voltages become easier to measure and more accurate at higher temperatures.

In this experiment type k thermocouple measures the temperature at nozzle exist along the convex surface by connected to the thermocouple data logger type TC-08 and the data logger connected to the laptop or PC. The TC-08 Thermocouple Data Logger used in this experiment because it has high performance and is an inexpensive, and it designed to measure a wide range of temperatures using any thermocouple that has a miniature size thermocouple connector.

The TC-08 thermocouple data logger and thermocouple type-K are shown in Figure 3.4,Figure 3.5.

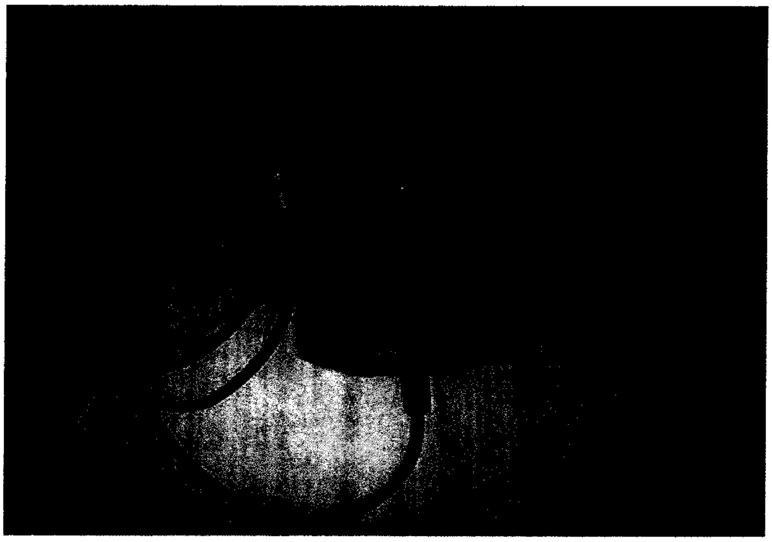

Figure 3.4: Tc-08 Thermocouple data logger

Figure 3.5: Thermocouple K-type

3.4.5 Settling chamber

There are main components to fulfill the experimental equipment are settling chamber. The settling chamber with meshes provides uniform and low turbulence intensity at the nozzle exit. The Figure 3.6has shown the equipment for settling chamber. It should avoid for any possible leakage of air. The settling chamber was made by acrylic plate and it shape rectangle.

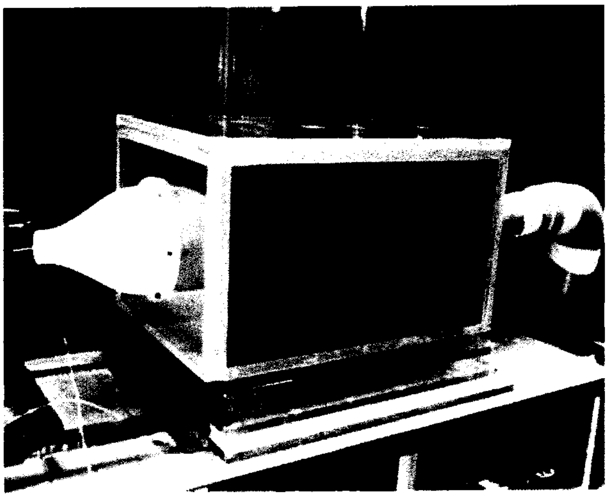

Figure 3.6: Settling chamber

3.4.6 Nozzle

A nozzle is a device designed to control the direction or characteristics of a fluid flow especially to increase velocity. A nozzle is often a pipe or tube of varying cross sectional area, and it can be used to direct or modify the flow of a fluid (liquid or gas). Nozzle is frequently used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that emerges from it (http://search.nasa.gov).

In this experiment the nozzle produces a straight jet of flow air that impinges on a convex surface. The inlet pressure of the nozzle is more than outer pressure and velocity is less than the outer velocity. The velocity of nozzle is depends upon the ending point diameter of nozzle, when the diameter of the ending point is less, the velocity of fluid is more. Figure 3.7 shown the nozzle with inner diameter 3.2 cm and outer 5.2 cm also the length of nozzle is 18 cm, and the thickness is 1 cm.

Figure 3.7: Nozzle geometry

3.4.7 The convex surface

In this experiment the target of the jet impingement has the semi circular convex surface with diameter 30 cm. The convex surface makes by cutting a circular acrylic tube and the stainless steel foil attaches to the outer surface with double sided adhesive tape. Thermocouple K-type installs from an inner acrylic wall by drilling holes. The external of the convex insulates with Styrofoam to reduce the conductive heat loss through the back surface. Figure 3.8shown Semi circular convex surface.

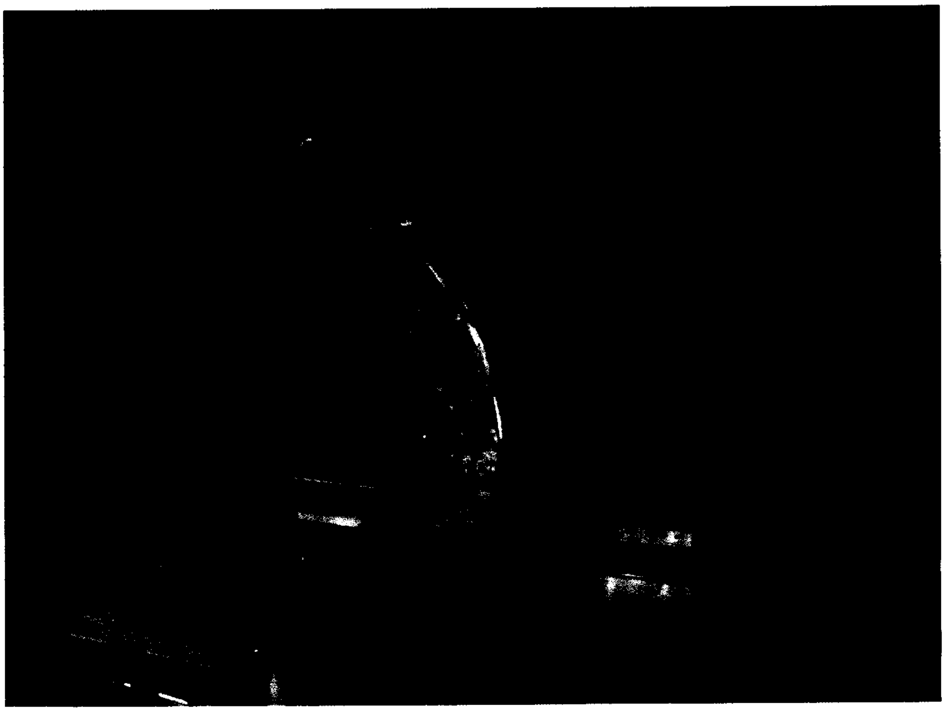

Figure 3.8: Semi Circular convex surface

3.4.8 Voltage regulator

The purpose of a voltage regulator is to keep the voltage in a circuit relatively close to a desired value. Voltage regulators are one of the most common electronic components, since a power supply frequently produces raw current that would otherwise damage one of the components in the circuit. Figure 3.9showsthe voltage regulator that use in this investigation for heating the stainless steel foil to the required temperature.

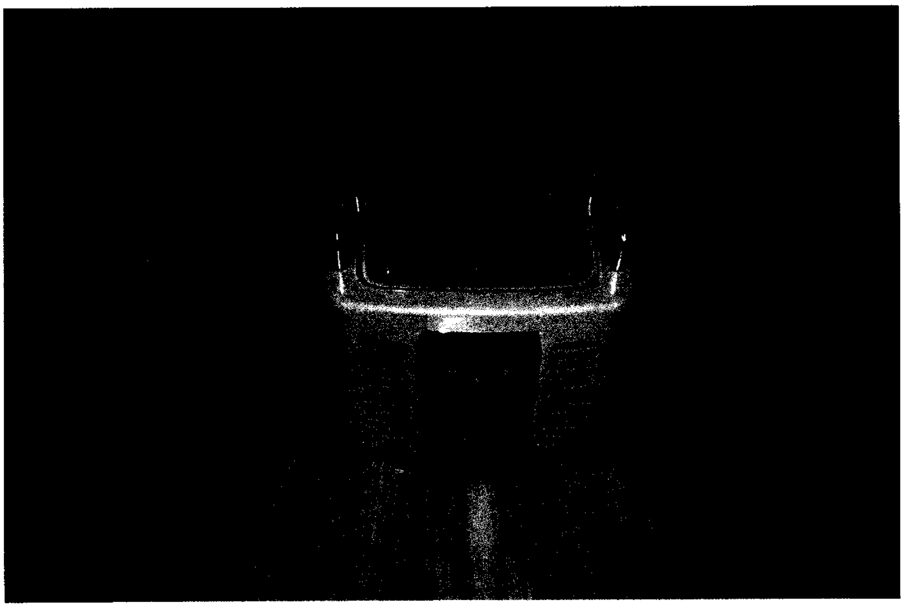

Figure 3.9: Voltage regulator

3.4.9 Digital multimeter

A multimeter is a devise used to measure voltage, resistance and current in electronics & electrical equipment. It is also used to test continuity between to 2 points to verify if there is any breaks in circuit or line. Multimeters have practically no resistance between

REFERENCES

1. Ahmed1, m. A. (2012). Influence the nozzle shape on local heat transfer in impi nging jet. International journal of systems applications, engineering & development .
2. Ali, o. Z. (2012). Nanofluid impingement jet heat transfer. Zeitoun and ali nanoscale research letters , 7-139.
3. B.balakrishna1, s. I. (2013). Investigation of supersonic flow through conical nozzle with various angles of divergence. International journal of mechanical engineering , 9-16.
4. C.h. Lee a, k. L. (2007). A study of the heat transfer characteristics of turbulent round jet impinging on an inclined concave surface using liquid crystal transient method. Experimental thermal and fluid science , 559-565.
5. Chitranjan agarwal1, j. K. (2012). Effect of nozzle exit to surface spacing on the cooling of electrically heated surface with jet impingement. International journal of advanced research in electrical, electronics and instrumentation engineering .
6. Kyo sung choo a, y. J. (2009). Heat transfer characteristics of a micro-scale impinging slot jet. International journal of heat and mass transfer , 3169-3175.
7. Luis a. Brignoni, s. V. (2000). Effects of nozzle-inlet chamfering on pressure drop and heat transfer in conFIned air jet impingement. International journal of heat and mass transfer , 1133-1139.

8. M. Anwarullah, a. V. (2012). Effect of nozzle spacing on heat transfer and fluid flow characteristics of an impinging circular jet in cooling of electronic components. Int. J. Of thermal & environmental engineering , 7-12.
9. M.a.r. Sharifl, n. R. (2010). Performance of turbulence models in the prediction of heat transfer from convex hemispherical surfaces due to round jet impingement. Proceedings of the 13th asian congress of fluid mechanics. Dhaka, bangladesh.
10. Mansoo choia, b. H. (2000). Measurements of impinging jet flow and heat transfer on a semi-circular concave surface. International journal of heat and mass transfer , 1811-1822.
11. Sopian2, r. Z. (2006). Pulsating circular air jet impingement heat transfer. Journal - the institution of engineers, malaysia .
12. Tadhg s. O'donovan, d. B. (2006). Fluctuating fluid flow and heat transfer measurements of an impinging air jet. 13th int symp on applications of laser techniques to fluid mechanics .
13. Arun Jacob, L. R. (2013). Numerical and Experimental Investigations on Jet Impingement Cooling. World Academy of Science, Engineering and Technology .
14. Agarwal, c. (2012). Jet impingement cooling of hot horizontal surface through sharp edge nozzle. International journal of applied engineering and technology issn: 2277-212x , pp.14-17.
15. Al-aqal, o. M. (1999). Heat transfer distributions on the walls of a narrow channel with jet impingement and cross flow.
16. D. E. Metzger, T. Y. (2011, Augest 25). Impingement Cooling of Concave Surfaces With Lines of Circular Air Jets. pp. 149-155.
17. D. H. Lee, Y. S. (1996, October 15). Turbulent flow and heat transfer measurements on a curved surface with a fully developed round impinging jet. Elsevier Science , pp. 160-169.