NUMERICAL STUDY OF JET IMPINGEMENT COOLING ON A SMOOTH CURVE SURFACE

ALI MOHAMMED HAYDER

A thesis submitted in fulfillment of the requirement for the award of the Degree of Masters of Mechanical Engineering

Faculty of Mechanical Engineering and manufacturing
University Tun Hussein Onn Malaysia

December 2014

ABSTRACT

Impinging jets are a best method of achieving particularly high heat transfer coefficient and are therefore employed in many engineering applications. In this study we seek to understand the mechanism of the distributed heat on the curve surface with the goal of identifying preferred methods to predicting jet performance.

The goals that have been achieved in the numerical results displayed are determine the influence of impingement jet characteristics on thermal and flow field on a curve surface, determine the variation of Nusselt numbers (Nu_D) along the curve surface in order to understand the heat transfer characteristics and study the effect of position (in the center, in the mid and in the end) and angle (α=90°, 60° and 30°) of jet impingement on curve surface, different Reynolds numbers (Re_D) in range of (5000, 6000, 7000, 8000 and 9000). The program, which was extracted results it is (GAMBIT 2.4.6) and (FLUENT 6.3), simulation is (2-D) in submerged jet flow and the continuity, momentum and energy equations were solved by means of a finite volume method (FVM).

This study covers the effect of different Reynolds numbers (Re_D) on average Nusselt numbers (Nu_{avg}) and local Nusselt numbers (Nu_D). From the result, the average Nusselt numbers (Nu_{avg}) increased with the increase of Reynolds numbers (Re_D) for all cases, in comparison between different positions (center, mid and end), of nozzle on curve surface at angle (α=90°) the maximum value of average Nusselt numbers (Nu_{avg}=388.3) is found when the nozzle locate in the end followed by the mid position and smallest value of average Nusselt numbers (Nu_{avg}=182.25) in the center of curve surface. In case of slant angle (α=60º) the maximum value of average Nusselt numbers (Nu_{avg}=387.47) is found when the nozzle locate in the end followed by the mid position and smallest value of average Nusselt numbers (Nu_{avg}=308.3) in the center of curve surface.

ABSTRACT

In case of slant angle (α=30º) the maximum value of average Nusselt numbers (Nu_{avg}=323.8) is found when the nozzle locate in the mid followed by the center position and smallest value of average Nusselt numbers (Nu_{avg}=185) in the end of curve surface.

The effect of jet impingement in the center, mid and end of curve surface at different slant angles (α=30°, 60° and 90°) on the contours of velocity and temperature are also presented, it is clearly found that the strong mixing or high turbulence flow beside the stagnation point in the right and left sides in case of the nozzle in the center and in the mid, and also noted that the flow beside the stagnation point in the one sides only in case of the nozzle in the end which leading to provide the many vortices and this vortices increases with increase the Reynolds numbers (Re_D). The highest mixing flow and vortices was noted at highest Reynolds number (Re_D=9000) the flow was found weak at smallest Reynolds number (Re_D=5000). The variations of local Nusselt numbers (Nu_D) with (s/B) for different Reynolds numbers (Re_D) were also presented. It is clearly seen that the local Nusselt numbers (Nu_D) increases with the rise of Reynolds numbers (Re_D) for the nozzle for all cases.

ABSTRAK

Jet hentaman adalah kaedah terbaik terutamanya bagi mendapatkan pekali pemindahan haba yang tinggi. Oleh itu ia banyak diguna pakai diaplikasikan didalam bidang kejuruteraan. Dalam kajian ini, kami berusaha untuk memahami mekanisma taburan haba diatas permukaan lengkung dengan matlamat untuk mengenalpasti kaedah yang sesuai untuk mengukur prestasi jet hentaman.

Matlamat yang telah tercapai dalam keputusan melalui kaedah berangka yang dipaparkan adalah pertama, pengukuran pengaruh ciri-ciri hentaman jet terhadap pemindahan haba dan aliran pada permukaan lengkung. Kedua, kesan kedudukan jet hentaman (di tengah-tengah, pertengahan dan hujung) dan sudut jet hentaman (α=90, 60 dan 30) terhadap nombor Nusselt (Nu_D) sepanjang permukaan lengkung. Ketiga, adalah kesan perubahan nombor Reynolds (Re_D) dalam julat (5000, 6000, 7000, 8000, dan 9000), perisian yang digunapakai untuk mendapatkan keputusan ialah (GAMBIT 2.4.6) dan (FLUENT 6.3) simulasi yang dijalankan adalah dalam (2-D), aliran jet adalah submerged jet, dan persamaan continuity, momentum serta tenaga diselesaikan dengan keadah isipadu terhingga (FVM).

Kajian ini meliputi kesan nombor Reynolds (Re_D) yang berbeza terhadap nombor Nusselt (Nu_{avg}) dan juga nombor Nusselt (Nu_D) yang telah dipuratakan. Dari hasil yang didapati, nombor Nusselt (Nu_{avg}) yang telah dipuratakan meningkat dengan peningkatan nombor Reynolds (Re_D) bagi semua keadaan. Dalam perbandingan antara kedudukan "nozzle" pula bagi nozzle dengan sudut jet hentaman (α=90), nilai maksimum (Nu_{avg}=388.3) diperolehi pada kedudukan nozzle di hujung lengkung, ia diikuti pada kedudukan "nozzle" di pertengahan dan nilai minimum purata nombor Nusselt (Nu_{avg}=182.25) di pusat lengkung. Bagi " nozzle" dengan sudut jet hentaman (α=60) ia menunjukan nilai maksimum (Nu_{avg}=387.47) pada kedudukan nozzle di hujung lengkung, diikuti pada kedudukan pertengahan dan nilai minimum (Nu_{avg}=308.3) dikedudukan tengah lengkung. Seterusnya bagi sudut

ABSTRAK

jet hentaman (α=30), nilai maksimum purata nombor Nusselt (Nu_{avg}=323.8) diperolehi apabila "nozzle" berada pada kedudukan pertengahan, diikuti pada kedudukan pusat dan nilai minimum (Nu_{avg}=185) pada kedudukan nozzle dihujung permukaan lengkung.

Kesan kedudukan jet hentaman bagi sudut "nozzzle" berbeza terhadap kontur halaju dan suhu juga dipersembahkan. Ia jelas menunjukan bahawa keadaan aliran jet hentaman akan mempengaruhi nilai nombor Nusselt (Nu_D). Kesan nombor Reynolds (Ru_D) yang berbeza terhadap nombor Nusselt (Nu_D). Kesan nombor Nusselt (Nu_D) juga dibentangkan. Ia jelas dapat dilihat bahawa nombor Nusselt (Nu_D) akan meningkat dengan peningkatan nombor Reynolds (Ru_D) untuk semua keadaan kedudukan dan sudut "nozzle".

CONTENTS

CHAPTER 2 LITERATURE REVIEW

CHAPTER 3 METHODOLOGY

LIST OF FIGURES

LIST OF ABBREVIATION AND SYMBOLS

Re_D	Reynolds number
ρ	Density of air
v	Inlet velocity of the jet
μ	Viscosity of air
B	Outlet diameter of nozzle
D	Inlet diameter of nozzle
H	Distance between the nozzle and surface
L	Length of nozzle
R_C	Radius of concave surface
Nu_D	Nusselt number
Nu_{avg}	Average Nusselt number
h	Convection heat transfer coefficient
k	Thermal Conductivity
α	Slope of nozzle
q_w	Heat flux on the heating surface
τ	Shear stress
P	Static Pressure
s	Circumferential distance from the stagnation point
δ	Boundary layer thickness

λ	Dimensionless system parameter
Γ	Circulation
c_p	Specific heat capacity
q	Heat transfer rate rate
ϕ	Scalar property
ε	The turbulent dissipation rate
A	Heating surface area
a	Coordinate perpendicular to the curve surface
u	Cartesian velocity x-direction
v	Cartesian velocity y-direction
i, j	Indices for coordinate notate
x, y	Coordinates
X_{sim}	Numerical data
X_{exp}	Experimental data
n	Number of data

CHAPTER 1

INTRODUCTION

1.1 Introduction

Impinging jets are known as a method of achieving particularly high heat transfer coefficients and are therefore employed in many engineering applications. Impinging jets have been used to transfer heat in diverse applications, which include the drying of paper and the cooling of turbine blades, and to de-ice aircraft wings in severe weather, and to cool sensitive electronic equipment Hollorth et. al. (1992). We seek to understand the mechanism of the distributed heat on the surface with the goal of identifying preferred methods to predicting jet performance. The directed liquid or gaseous flow released against a surface can efficiently transfer large amounts of thermal energy or mass which demonstrated the capability of cooling high heat flux surfaces while maintaining a low thermal resistance Ferrari et. al. (2003). Jet impingement experiments attempt to measure the average heat transfer coefficient, even though it is known that heat transfer coefficients are known to change as a function of distance from the impact zone. So jet impingement with high velocity jets has become an established method for surface cooling or heating in a wide variety of processes and thermal control applications. The use of impingement jets for the cooling of modern aero-engine components is widespread, especially within the hot stationary parts Han et. al. (1985). Extensive research has been conducted on steady impinging jet to understand their heat and mass transfer characteristics. Numerous studies and reviews on the subject of steady jet heat transfer have been published over the last several decades in order to further enhance the heat transfer Kondjoyan et. al. (2002).

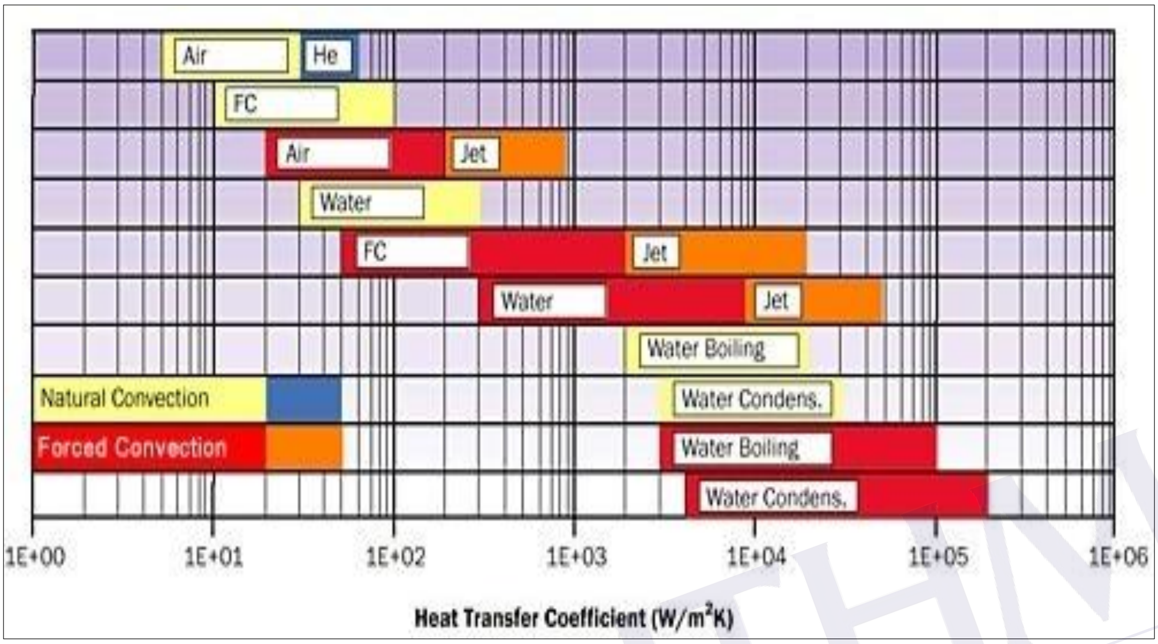

Figure1.1: Heat transfer coefficient attainable with natural convection, single-phase liquid forced convection and boiling for different coolants Lasance et. al. (1997)

1.1.1 Background of Study

The application of impinging jets for cooling, drying, and cleaning can be found in a wide variety of industries by virtue of their high heat and mass transfer on and around stagnation points. They are therefore a vital subject for many researchers in both the academic and industrial fields. Since the understanding of free jets is essential to reveal the mechanism of impinging jets and improve technical methods, the effects of nozzle configurations, jet velocity, and ambient conditions have been well addressed jet impingement has also been extensively studied, with investigations of how such parameters as single and multiple jets, nozzle and impingement wall configurations, and the angle between the jet and impingement wall affect the heat transfer performance investigations of the heat transfer performance of impinging jets on both convex and curve semi-circular surfaces were conducted Chan et. al. (2003). Over the past 30 years, experimental and numerical investigations of flow and heat transfer characteristics under single or multiple impinging jets remain a very dynamic research area Thakare et. al. (2000).

1.1.2 Jet Impinging on Curved Surfaces

When jet impingement cooling is applied to curved surfaces such as a turbine blade surface, the curvature effect should be taken into consideration. For flows on a surface with curve curvature, for sufficiently high flow speeds, the centripetal force due to the curvature usually makes the flow unstable and a Taylor-Gortler type vortex is produced. The velocities are low near the wall and are large away from the wall. This means that the centrifugal forces on the faster moving fluid particles are higher and there is a tendency for these fast moving particles to be pushed outward near the surface which causes instability. This vortex has its axis parallel to the flow direction and is known to enhance momentum and energy transfer Mayle et.al. (1979).

1.2 Problem Statement

A lot of equipments or appliances need to have high heat transfer performance to guarantee the quality and also to increase the capability; old cooling systems cannot be used anymore as it does not cool sufficiently. This makes it necessary to develop new techniques to meet the demand, so researchers are moving toward the technology of jet impingement cooling systems.

In gas turbine systems, turbine blades inside gas turbine engine are exposed directly to hot gas from combustion chamber. To meet the demand for high thermal efficiency, gas turbine engines are operated at high turbine inlet temperatures. However, high temperature creates large, non-uniform thermal loads on gas turbine components. In the design of an impinging-jet system for a given thermal application, a large number of geometric and flow parameters like jet type, nozzle to target spacing, angle of impingement, nozzle design, jet-inlet Reynolds numbers etc. are involved. So purely experimental approach to the problem is unlikely to lead to a satisfactory solution at reasonable cost and time, so should hold numerically accurate simulations of the flow and heat-transfer through internal cooling passages achieve efficient cooling develop cooling method to protect gas turbine components, to find the best heat transfer coefficient for these problems.

1.3 Project Objectives

The objective of this study is:

1) To determine the influence of impingement jet characteristics on thermal and flow field on a curve surface.
2) To determine the variation of Nusselt numbers (Nu_D) along the curve surface in order to understand the heat transfer characteristics.
3) To study the effect of position and angle of jet impingement on curve surface.

1.4 Scopes of Study

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

1) Working in this project using (ANSYS) software.

2) Simulation is (2-D) in submerged jet flow.

3) This project study using one type of nozzle is a slot nozzle.

4) Investigate the effect of different angles α of jet impingement (30º, 60º and 90º) on thermal and flow filed with different position of jet impingement.

5) The effect of different values of Reynolds numbers (Re_D= 5000, 6000, 7000, 8000 and 9000) on the thermal and flow filed of curve surface.

1.5 Research Significance

Jet impingement study is of important because of its various applications in different industries, ranging from to cool sensitive electronic equipment and drying of paper and the cooling of turbine blades, and to de-ice aircraft wings in severe weather, so advanced approach yields low cost and accurate prediction of heat transfer processes on a curve surface by impinging jet, which may help the design of above applications, with relative ease.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Heat transfer enhancement forcurvesurface by using jet impingement has been a subject of interest for sciences and researchers in the past decades. Numerical and experimental studies have been reported in order to increase the amount of heat transferred by jet impingement. Jets impinging on a surface give rise to high heat transfer rates, which may be exploited in many applications for drying, heating or cooling purposes. For this reason, impinging jets have been subject matter of investigation by both scientists and technicians around the world for long time. This topic has been addressed in both numerical and experimental ways entailing proliferation of scientific and technical documents.

Herein, we cite mainly literature dealing with curved targets, which complies with the topic of the present project.

2.2 Fluid Flow Characteristics

An impinging jet is consisting of three characteristics regions namely free jet region, impingement region and wall region as illustrated in Fig.2.1. The free region is the region where the velocity profile is not influenced by the impingement. The impingement or the stagnation region is the region where adverse pressure gradient attenuates the mean axial velocity, subsequently deflecting the fluid flow into the wall jet region. In wall jet region, the fluid flow velocity is low at the wall due to the viscous friction. Laterally from the wall the shear strain rate reduces gradually but increases again as the higher velocity fluid mix with the surrounding fluid Martin (1977).

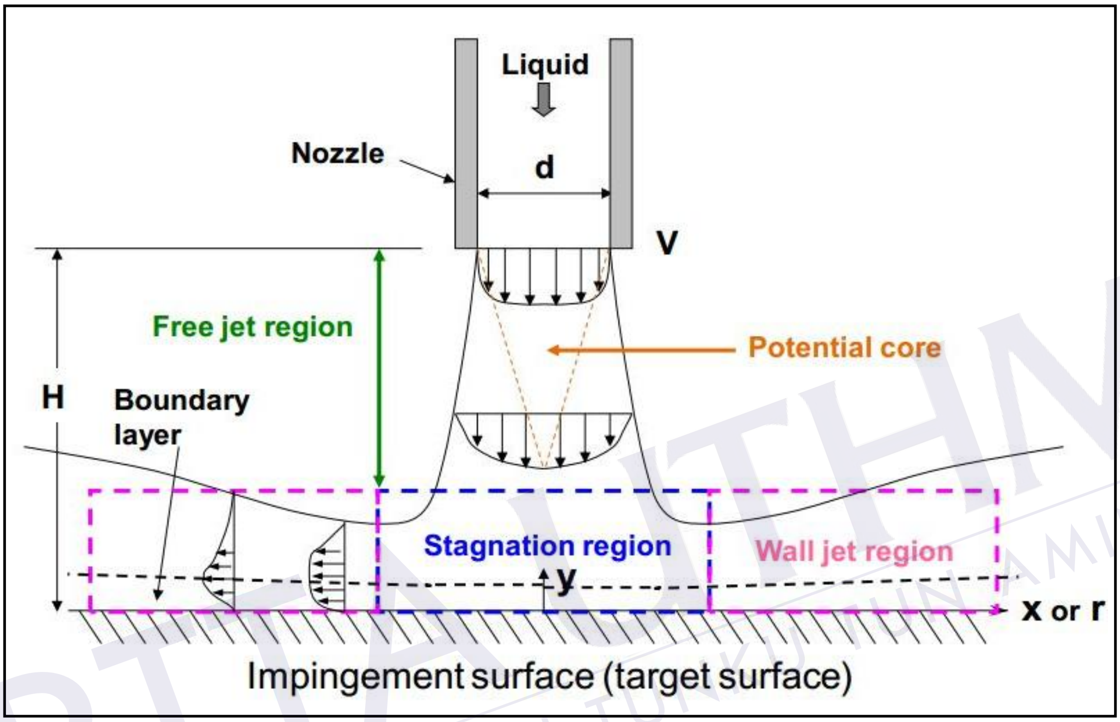

Figure 2.1: jet impingement regions Jambunathan et. al.(1992).

The free jet region is further separated into three characteristics regions; the developing flow region, the potential core region and developed flow region. The developing flow region is where the surrounding fluid entrained into the jet starting from the nozzle edge.

The mixing or shear zone causing the jet to spread and reduces the flow velocity. The potential core region is a part of the developing flow region and is the region where the velocity is not less than (95 %) of the velocity at the nozzle exit. Beyond the potential core region, the lateral profiles of the axial velocity are a bell shape. In the developed flow region similar axial velocity profiles exist at different jet lengths Jambunathan et. al. (1992).

2.3 Previous Studies of Jets Impinging on a Curve Surface

Öztekin et.al. (2012) studied experimentally and numerically the hydrodynamic characteristics of a slot jet flow impinging on a curve surface. Six different curve plates with varying surface curvature and a flat plate were used. Air was used as the impinging coolant. In the experimental work, the slot nozzle used was specially designed with a sixth degree polynomial in order to provide a uniform velocity profile at its exit. The experiments were carried out for the jet Reynolds numbers (Re_D) in the range of ($3000 \leq Re_D \leq 12500$), the dimensionless nozzle-to-surface distance range of ($1 \leq H/W \leq 14$) for dimensionless value of the curvature of impinging surfaces in the range of (R/L=0.5, 0.5125, 0.566, 0.725, and 1.3). The pressure coefficient, (Cp) for each test case was obtained across dimensionless arc length (S/W). Numerical computations were performed by using the (k-ε) turbulence model with enhanced wall functions for the curve surface with (R/L=0.725) and for the flat plate. An increase in the dimensionless nozzle-to-surface distance (H/W) decreases the pressure coefficient (Cp) at the stagnation point. From the result, it was clearly seen that the local variation of the pressure coefficient shows that the (k-ε)turbulence model with the enhanced wall functions is in excellent agreement with the experimental results.

Öztekin et. al. (2013) examined the turbulent slot jet impingement cooling characteristics on curve surface with varying surface curvature experimentally and numerically. Air was used as the impingement coolant. In the experimental work, a slot nozzle specially designed with a sixth degree polynomial in order to provide a uniform exit velocity profile was used the experiments were carried out for the jet Reynolds numbers (Re_D) in the range of ($3423 \leq Re_D \leq 9485$), the dimensionless nozzle-to-surface distance range of ($1 \leq H/W \leq 14$) for dimensionless values of the curvature of impinging surfaces in the range of (R/L = 0.5, 0.725), and 1.3 and a flat impingement surface. Constant heat flux was applied on the plates. Numerical computations were performed using the (k-ε) turbulence model with enhanced wall functions. For the ranges of the governing parameters studied, the stagnation, and local and average Nusselt numbers (Nu_{avg}) have been obtained both experimentally and numerically. It was clearly found that the average and stagnation point Nusselt numbers (Nu_D) decrease as (H/W) increases for all the parameters studied. The average and stagnation point Nusselt numbers (Nu_D) increase as Reynolds numbers (Re_D) increases. The local Nusselt numbers (Nu_D) is observed to decrease monotonically from its stagnation value along the circumferential direction. Generally, while Reynolds numbers (Re_D) increases, an increase in the average Nusselt numbers (Nu_{avg}) onto the impingement surfaces with lower curvature radius is observed. It was also observed that the surface curvature increases the average Nusselt numbers (Nu_{avvg}) for($R/L \geq 0.725$). The best cooling performance was obtained for ($R/L = 1.3$).

Chang et. al. (2009) studied a new cooling scheme for electric rotor machines; the impinging jets issued from the armature onto the stator with the spent flows directed toward two annular exits at both ends can convect the Joule heat out of the rotor machinery effectively. An experimental study is accordingly devised to investigate the heat transfer performances over the outer cylinder of a concentric annulus with an impinging jet-array issued from the rotating inner cylinder. Intermittencies of impinging jets and spent flows in the Taylor-Couette-Poiseuille annular flow feature the dominant flow physics that affect the heat transfer performances. A set of selected experimental data illustrates the isolated and interdependent influences of jet Reynolds numbers (Re_D), Taylor number (Ta) and rotating Grashof number (Gr_ω) on local and area-averaged Nusselt numbers (Nu_D and $\overline{N}u_{avg}$). With the present parametric conditions examined, the coupled (Re_D), (Ta) and (Gr_ω) effects have led the ratios of rotational and non-rotational ($\overline{N}u_{avg}$), in the range of (0.75–1.48). In conformity with the experimentally revealed heat transfer physics, the heat transfer correlation that permits the evaluation of (Nu_D) over the outer cylinder of the concentric annulus subject to jet-array impingement from the rotating inner cylinder is generated.

Choi et. al. (2000) studied experimentally the fluid flow and heat transfer for jet impingement cooling on a semi-circular curve surface. The distributions of mean velocity and velocity fluctuation on the curve surface had been measured in free, impinging and wall jet flow regions by using a Laser Doppler Anemometer. Local Nusselt numbers (Nu_D) had also been measured. Variations of jet Reynolds numbers (Re_D), the spacing between the nozzle and the target and the distance from the stagnation point in the circumferential direction had been considered.

Craft et. al. (2008) studied computations the flow and heat transfer from arrow of round jets impinging onto a curve semi-circular surface, designed to reproduce the important flow features found in internal turbine blade cooling applications. Linear and non-linear eddy-viscosity models were applied, with wall-functions to cover the near-wall layer. These are shown to capture the overall flow characteristics, including the wall jets created by impingement on the curved surface and the down washes caused by the collision of these wall jets. Whilst the non-linear model performs slightly better than the linear, both under predict the turbulence levels close to impingement and in the down washes. The standard, log-law based, form of wall-function is found to be inadequate in predicting the heat transfer, and a more advanced form developed at Manchester the (AWF) was also tested. The exact way in which convective terms are approximated in this latter approach is shown to be crucial, and a for mis presented which leads to stable and reasonably accurate solutions that capture the overall pattern and impingement Nusselt numbers($\mathrm{Nu_D}$) levels shown in measurements, but under predict heat transfer levels around the jet down washes.

Eren et. al. (2006) studied experimentally the nonlinear flow and heat transfer characteristics for a slot jet impinging on a slightly curved curve surface. The effects of jet Reynolds numbers ($\mathrm{Re_D}$) on the jet velocity distribution and circumferential Nusselt numbers($\mathrm{Nu_D}$) were examined. The nozzle geometry was a rectangular slot and the dimensionless nozzle-to-surface distance equals to (L=8). The constant heat fluxes were accordingly applied to the surface to obtain an impingement cooling by the air jet at ambient temperature. The measurements were made for the jet Reynolds numbers of ($\mathrm{Re_D}$=8617, 13350 and 15415). New correlations for local, stagnation point, and average Nusselt numbers ($\mathrm{Nu_{avg}}$) as a function of jet Reynolds number ($\mathrm{Re_D}$) and dimensionless circumferential distance were reported. The local circumferential, the stagnation point, and the average Nusselt numbers ($\mathrm{Nu_{avg}}$) are determined. The new correlations for slightly curved curve surface are reported here for ($\mathrm{Nu_D}$, $\mathrm{Nu_{st}}$ and $\mathrm{Nu_{avg}}$) Reynolds numbers ($\mathrm{Re_D}$) dependence of local Nusselt numbers ($\mathrm{Nu_D}$) with the exponent of (0.58) for the curve surface was found to be significantly higher than the exponent of (0.5) corresponding to the jet impinging on the flat target surface under laminar boundary condition.

Fenot et. al. (2008) investigated experimentally the heat transfer due to a row of air jets impinging on a curve semi-cylindrical surface. Heat transfer characteristics were measured using a heat thin foil technique and infrared thermograph. Adiabatic wall temperatures and local heat transfer coefficients were determined by means of a linear regression method. The effect of high relative curvature (d/D) was investigated by changing the jet tube diameter (impinging surface diameter remaining constant). Reynolds numbers (Re_D), injection temperature, spacing between adjacent jets and jet exit to surface spacing was also made to vary. Curvature has different effects over the adiabatic wall temperature and Nusselt numbers (Nu_D)distributions. First, the curvature increase provokes a small growth of Nusselt numbers (Nu_D) in the impingement region. On the other hand, curvature produces a confinement of the jet's flow that has two consequences: stagnation of the adiabatic wall temperature and decrease of Nusselt numbers (Nu_D) distribution. Nusselt numbers (Nu_D) distribution for a curve surface resembles such distribution over a flat plate except for local minima along the impinging line and the lack of local extremes for (s/d=2). Local minima situated just in front of the injection could be the results of a "dead fluid" area that prevent jets from coming into impact. Lack of local minima around (s/d=1.4) and of local maxima around (s/d=2.1) are probably due to centrifugal forces that perturb flow of the boundary layer. More precisely, single jet Nusselt numbers (Nu_D) distribution presents these extremes along the y-axis (where the plate corresponds to a flat plate) but not along the curvilinear abscissa. Lastly, relative curvature seems to have two opposite effects over the Nusselt numbers (Nu_D) distributions. The first involves overall reduction of the Nusselt numbers (Nu_D). Since (Nu_D) is also reduced for a semi-confined flat plate, this effect is probably of the confinement variety. As for the second effect, it enhances heat transfer near the impinging zone and is probably directly due to the increase in relative curvature.

Imbriale et. al. (2014) investigated the heat transfer between a curve surface and a row of air jets impinging on it. Measurements were performed with the heated thin foil heat transfer sensor using (IR) thermograph as temperature transducer. Experimental tests were carried out by varying jets inclination, pitch, impinging distance, mach and Reynolds numbers (Re_D).Data is reduced in dimensionless form in terms of the Nusselt numbers (Nu_D) and analysed by separately accounting for the influence of the main relevant parameters. As main results, (3-D) nusselt maps show presence of stream-wise vortices driven by surface curvature, jet inclination and jet pitch. At last, the Nusselt numbers (Nu_D), averaged over the impinged area, was represented as a function of the Reynolds numbers (Re_D) following a data correlation present in literature.

Lee et. al. (2007) examined experimentally the effects of curve 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. This method suddenly exposes reheated wall to an impinging jet and then a video system recorded the response of the liquid crystals to measure the surface temperature. The Reynolds numbers (Re_D=11,000, 23,000 and 50,000), were used nozzle-to-surface distance ratio was from (2 to 10) and the surface angles were (α=0°, 15°, 30°, and 40°). The correlations of the stagnation point Nusselt numbers (Nu_D) according to Reynolds numbers (Re_D), jet-to-surface distance ratio and dimensionless surface angle were also presented. In the stagnation point, in terms of Reynolds numbers (Re_D) 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 maximum Nusselt numbers (Nu_D) in this experiment occurred in the upstream direction. The displacement of the maximum Nusselt numbers (Nu_D) from the stagnation point increased with increased surface angle or decreasing nozzle-to-surface distance. Under this condition, with surface curvature at (D/d=10), the maximum displacement is about (0.7) times of the jet nozzle diameter.

Choia et. al. (2000) an experimental study of fluid flow and heat transfer has been carried out for jet impingement cooling on a semi-circular curve surface. The distributions of mean velocity and velocity fluctuation on the curve surface have been measured in free, impinging and wall jet flow regions by using a Laser Doppler Anemometer. Local Nusselt numbers (Nu_D) have also been measured. Variations of jet Reynolds numbers (Re_D), the spacing between the nozzle and the target and the distance from the stagnation point in the circumferential direction have been considered. Emphasis has been placed on measuring turbulent jet flow characteristics including impinging and evolving wall jets and interpreting heat transfer data, particularly, the occurrence and its location of secondary peak in connection with data of measured mean velocity and velocity fluctuations on the curve surface.

Mohammad pour et. al. (2014) investigated numerically the effects of square waveform (intermittent) and sinusoidal waveform pulsation on the heat transfer rate from a slot jet impinging to a curve surface. A numerical analysed of turbulent flow and heat transfer in a two-dimensional jet is performed using the (RNG k−Ɛ) model. The affects of jet Reynolds numbers(Re_D), pulsation frequency, and nozzle to target surface spacing in both types of waves and the effect of the amplitude of sinusoidal waves on distribution of the surface time-averaged local Nusselt numbers(Nu_D) were studied. The numerical results showed that in the pulsed jets, the increased of frequency in the range of (20-80) H/z and the Reynolds numbers(Re_D) in the range of (4740-7200) caused the increase of the time average Nusselt numbers(Nu_{avg}) compared to steady jet cases. In the pulsed jets, reducing the nozzle-to-surface distance caused the increased of heat transfer from the target surface. Moreover, the increased of pulse amplitude from (0.2 to 1.0) in the sinusoidal waves increased the time-average Nusselt numbers (Nu_{avg}). Finally, the comparison of results indicates a considerable increased of the heat transfer rate for the square form waves than sinusoidal waves compared to the steady state data.

Sharif et. al.(2010) investigated numerically the convective heat transfer process from curve cylindrical surfaces due to turbulent slot-jet impingement is performed. Constant heat flux condition was specified at the curve surfaces. The flow and thermal fields in the vicinity of the surfaces were computed using the (RNG k-Ɛ) turbulence model with a two-layer near wall treatment. Parametric studies were carried out for various jet-exit Reynolds numbers (Re_D) ,surface curvature, and nozzle-to-surface spacing. Results presented include streamlines, isotherms, velocity and temperature profiles in the wall-jet region, and the local Nusselt numbers (Nu_D). Distribution on the impingement curve wall for various parameter values in the study. The results indicate that while the jet-exit Reynolds numbers(Re_D),And the surface curvature had a significant effect on the heat transfer process; it was relatively insensitive to the jet-to-target spacing. A correlation for the average Nusselt numbers (Nu_{avg}). At the curve surface as a function of the parameters considered in the study was also derived.

As results, it was clearly found that the Reynolds numbers (Re_D) ,had a significant effect on the heat transfer process. The local Nusselt numbers (Nu_D) at the heated surface significantly increased at any particular circumferential location with increased Reynolds numbers (Re_D), for any set of jet-to-target spacing and relative curvature. It was also observed that the local Nusselt numbers (Nu_D) distribution starts with a high value at the stagnation point with a peak at a slightly offset location from the stagnation point and then quickly and monotonically decreases along the heated surface. No secondary peak is noticed in the local Nusselt numbers $(\mathrm{Nu_D})$ distribution even for the case when the surface is within the jet potential core (h/B=3 and 6).The local Nusselt numbers$(\mathrm{Nu_D})$distribution is not very sensitive to the jet-to-target spacing for higher relative curvature (D/B) values. This is due to the fact that the surface approaches to the planar limit as (D/B) increases. The surface curvature has significant effect on the local and average Nusselt numbers $(\mathrm{Nu_{avg}})$ when the curvature is strong, i.e., at lower (D/B) values.

Sharif et. al. (2013) investigated numerically the effect of surface roughness on the convective heat transfer from a heated convex hemispherical surface due to a turbulent round cold air jet impingement had been investigated numerically using the (ANSYS FLUENT CFD) code. Initially, the performance of a few turbulence models, namely, the (RNG k-ε) model, the realizable (RNG k-ε)model, the (standard k-ω) model, and the (SST k-ω) model, in the prediction of the convective heat transfer for such a flow configuration, was evaluated against experimental data. Based on this evaluation, the (SST k-ω) model is chosen and employed to further investigate the surface roughness effect on the jet impingement heat transfer process for the above mentioned configuration. The flow and geometric parameters for this study are the jet exit Reynolds numbers$(\mathrm{Re_D})$ the jet diameter (d), the diameter of the hemispherical surface (D), the distance of the impingement surface from the jet exit (L), and the equivalent average sand grain roughness height of the hemispherical surface (K_s). Computations have been performed for various combinations of these parameters for a range of the disvalues. It is observed from the results that, the local and average Nusselt numbers$(\mathrm{Nu_D}, \mathrm{Nu_{avg}})$ at the hemisphere surface is enhanced with increasing surface roughness height. Thus it was concluded that the impinging jet heat transfer will be augmented if roughness is added to the hemispherical surface.

Terekhov et. al. (2009) study experimentally of flow characteristics and heat transfer for jet impingement cooling of obstacles in the form of single spherical cavities was reported. The distributions of flow velocities between the nozzle and the obstacle, and also the fields of pressure and heat-transfer coefficients inside the cavity were measured. It is found that, at a value of depth the cavity generates the large-scale toroidal vortex, essentially influencing on the heat transfer. The cavity flow becomes unstable, exhibiting low-frequency pulsations of local heat fluxes. In the examined ranges of Reynolds numbers (Re_D=1.2–5.8$*10^4$), and cavity depths (equal to or smaller than 0.5D_c) the local heat-transfer intensity in the cavity is lower than that on a flat obstacle; yet, this reduction is almost fully compensated by increased area of the heat-exchanging surface.

Yang et. al. (1999) study experimentally has jet impingement cooling on a semi-circular curve surface when jet flows were ejected from three different slot nozzles--round shaped nozzle, rectangular shaped nozzle and (2-D) contoured nozzle. Experiments had been conducted with variations of nozzle exit Reynolds numbers (Re_D) in the range of (4819 ≤Re_D≤ 14499) and nozzle-to-surface distance (Z_n) in the range of (½ ≤ Z_n/B ≤ 20) to determine the heat transfer coefficients under a constant heat flux condition. The developing structures of free jets were measured by hot-wire anemometer to understand the characteristics of heat transfer in conjunction with measured jet flows. Markedly different flow and heat transfer characteristics had been observed depending on different nozzle shapes. Average heat transfer rates for impingement on the curve surface were 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 had also been made.

Yang et. al. (2011) investigated numerically the flow field and heat transfer characteristics of a slot turbulent jet impinging on a semi-circular curve surface with uniform heat flux. The turbulent governing equations were solved by a control-volume-based finite-difference method with a power-law scheme and the well-known (k-ε) turbulence model and its associate wall function to describe the turbulent structure. In addition, a body-fitted curvilinear coordinate system is employed to transform the physical domain into a computational domain. Numerical computations have been conducted with variations of jet exit Reynolds numbers (5920≤ Re_D ≤23,700), dimensionless jet-to-surface distance (0.5≤ H/B ≤12), dimensionless jet width (0.033≤ B/D ≤0.05) and the heat flux (1663 W/m²≤ q″≤ 5663 W/m²). The theoretical model developed was validated by comparing the numerical predictions with available experimental data in the literature. The variations of local Nusselt numbers (Nu_D) along the semi-circular curve surface decrease monotonically from it is maximum value at the stagnation point. The numerical the local Nusselt numbers(Nu_D)is reasonably predicted with a maximum discrepancy within (15%).

CHAPTER 3

METHODOLOGY

3.1 Introduction

One of the aims of the present project is to produce a numerical model capable of predicting heat transfer characteristics of a slot jet impinging from a (2-D) contoured nozzle on to a smooth curve surface with various parameters, on the basis that simulation with the least computational cost is the most desired. The focus of the present work is to get good agreement between (CFD) result and experimental result of study on heat transfer characteristics. Furthermore, the present work also aims that the numerical simulation can be easily duplicated by others for further improvement in future work.

(FULENT) solver is based on the finite volume method. In this case, the domain of the model to be examined is discretized into a finite set of the control volumes, which are termed meshes and cells. Then, the general conservation equations for mass, momentum and energy are solved on this set control volumes. Other transport equations such as the equations for species may also be applied. Subsequently, partial differential equations are discretized into a system of algebraic equations. All algebraic equations are solved numerically to render the solution field (CFD) theories, the (CFD) modeling process, physical model and assumptions such as physical model, governing equations, finite volume method (FVM), and fluid flow computations by (SIMPLE) algorithm are presented in the following section.

3.2 CFD Theories

In the current study (CFD) technique was used to investigate the effect of geometrical parameters of jet impingement in a smooth curve surface. The interest region where flow simulation is to be done is the computational domain of the problems. (CFD) is computer based tool for simulating the behavior of system involving fluid flow, heat transfer, and other related physical processes. It works by solving the equations of fluid flow in a special form over a regain of interest, with specified known conditions on the boundary of that region. (CFD) has facilities in industry as it can solve several system configurations in the same time and at a cheaper cost compared with another experimental study.

3.3 The CFD Modeling Process

Basically (CFD) problem could be tackled by following the general procedure as shown below. (GAMBIT 2.4.6) and (FLUENT 6.3), commercial software was used in the current study which was used the following steps: The general modeling process Venturino et. al. (1995). Fig. 3.1 shows below.

The (CFD) modeling region could be classified into few major steps: Pre-processing stage, models step and solving, and followed by the post-processing stage. In pre-process stage, the geometry of (CFD) region was constructed and computational mesh generated in (GAMBIT). It is then followed by physical model, boundary conditions, initial conditions, and other appropriate parameters were defined in models step and solving. Finally, the results could be obtained throughout the computational domain in the post-processing stage.

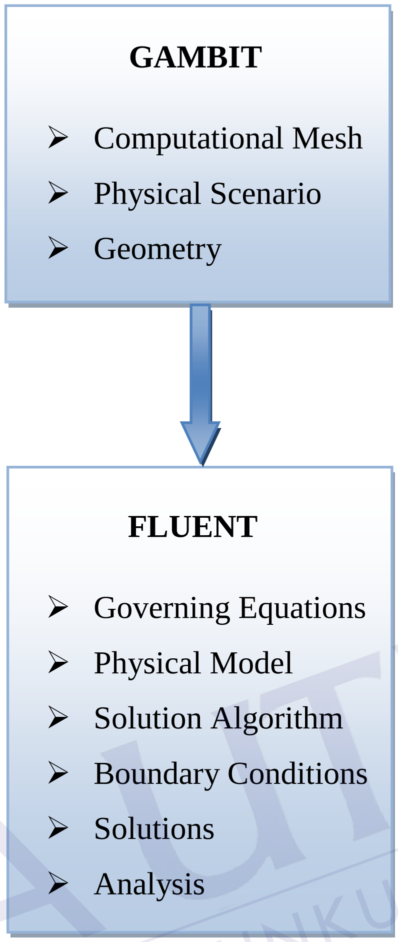

Figure 3.1: general modeling process

3.4 Physical Model and Assumptions

3.4.1 Physical model

The diagram in Fig. 3.2 and Fig. 3.3 shows of the jet impingement in a smooth curve surfaces the radius of curve surface is (R_C = 75mm), the inlet diameter is (D=61mm), the outlet diameter of nozzle is (B=5mm), the length of nozzle (L=104mm) and the thickness of nozzle is (t=2mm). The ranges of angles of jet impingement α are (α 30°, 60° and 90°).

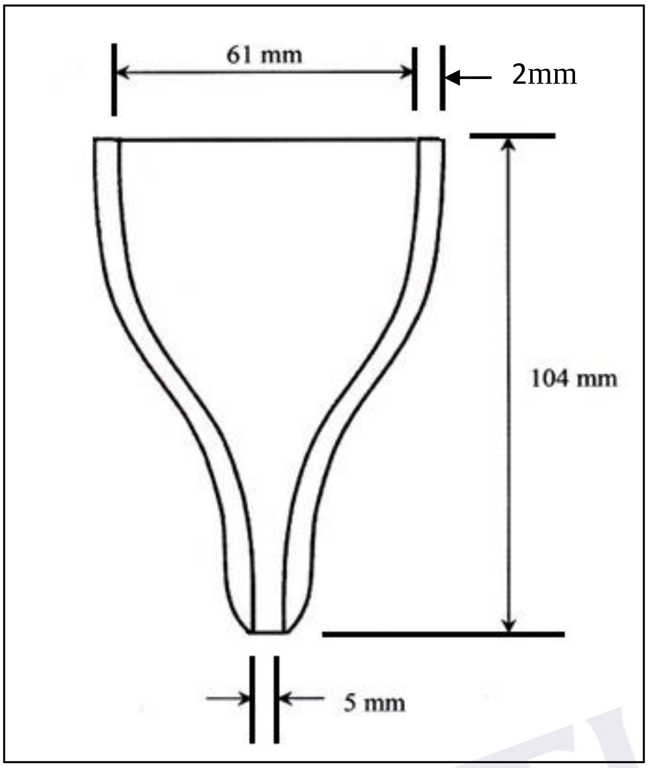

Figure 3.2: Nozzle geometry

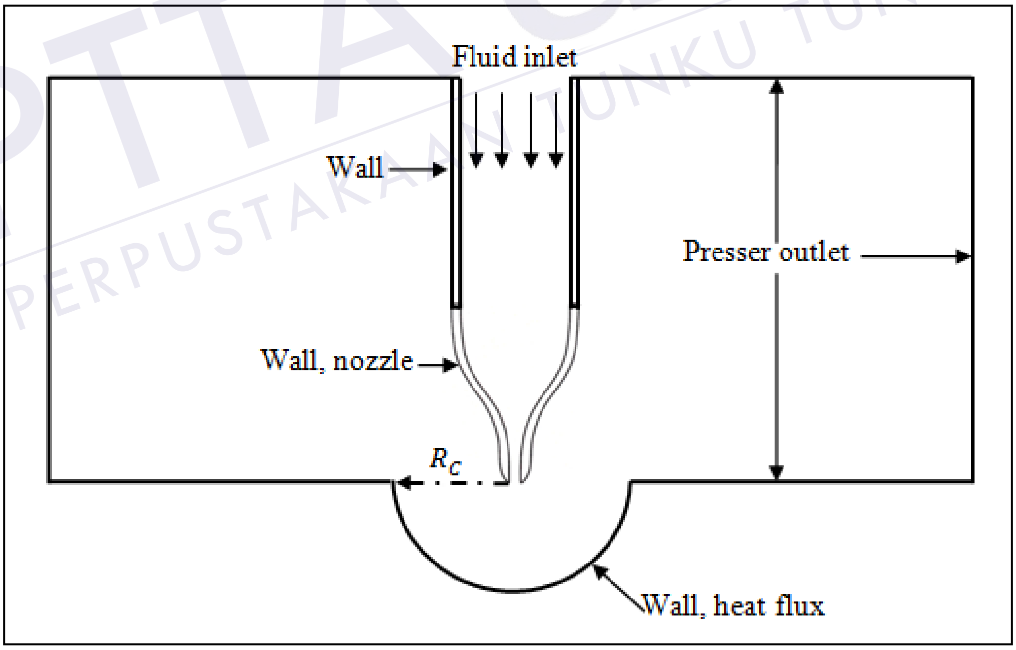

Figure 3.3: Numerical domain drawing

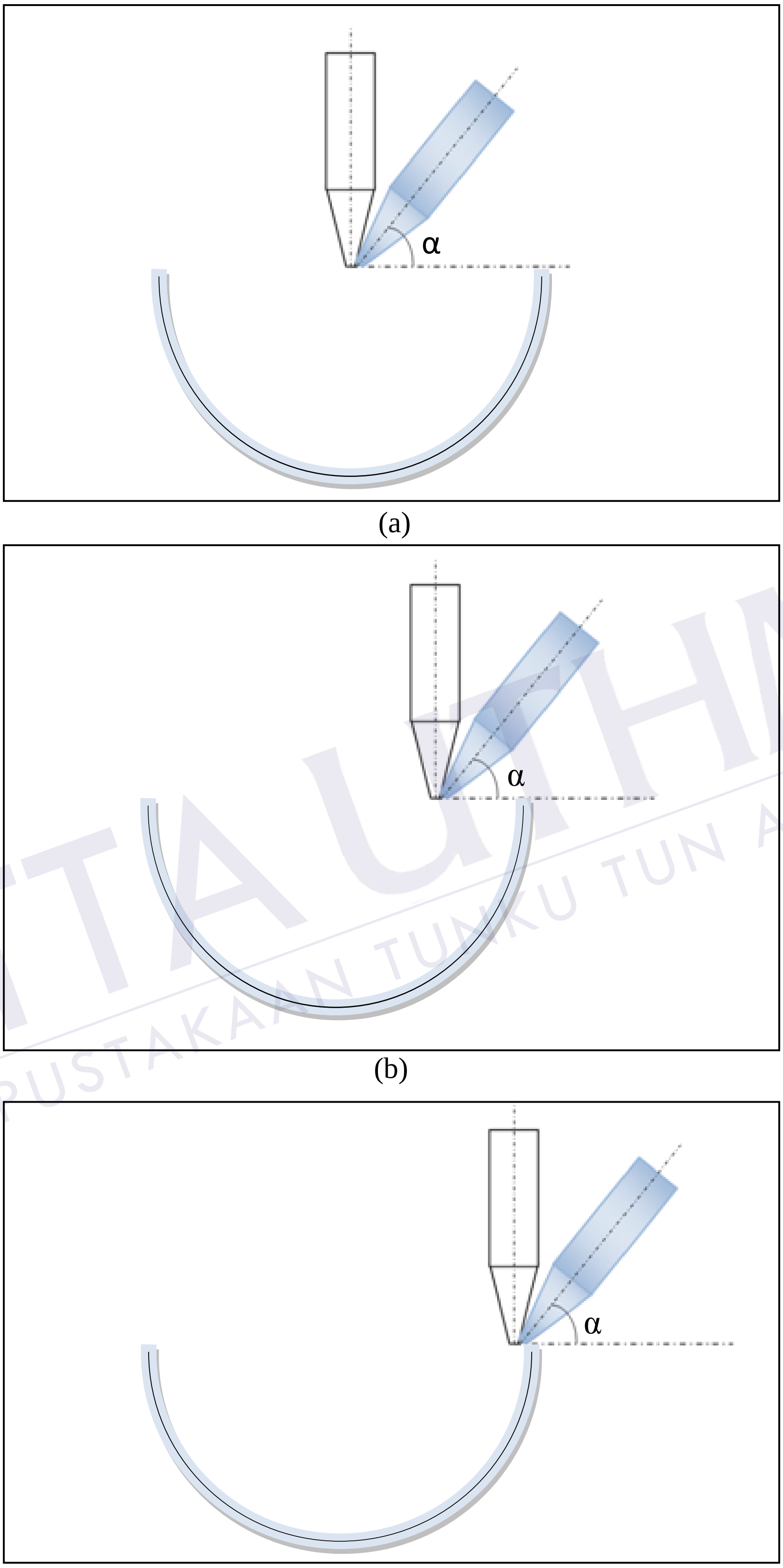

Figure 3.4: Different angles of Jet impingement with different location of jet a. (in the centre of the curve) b. (in the mid of the curve) c. (in the end of curve)

3.4.2 Boundary conditions and numerical setup

The boundary conditions for the present problem are specified for the computational domain as shown in Fig. 3.3. These figures show the numerical domain for jet impingement in curve surface this study, the curve surface subjected to uniform heat flux while the top side is subjected to velocity inlet and the curve is subjected to pressure outlet. The value of heat flux of (5000 $\mathrm{W/m^2}$) is subjected at curve surface and the temperature of inlet jet is (300 K). The turbulence model is very important to accommodate the flow behavior of each application. The standard turbulence model and the renormalized group (RNG k-ε) turbulence model were selected. The independent is incompressible Navier-Stokes equations and the turbulence model analysis was solved using finite volume method. To evaluate the pressure field, the pressure-velocity coupling algorithm (SIMPLE), (Semi Implicit Method for Pressure-Linked Equations) was selected. At the inlet, fully developed velocity profile was imposed. The turbulence intensity was kept at (5.5%) at the inlet. The solutions are considered to be converged when the normalized residual values reach (10^{-5}) for all variables.

3.4.3 Governing equations

To describe forced convective heat transfer, to complete the (CDF) analysis of jet impingement on curve surface with louvered strips inserts. The (continuity, momentum and energy equations) are required. The phenomenon under consideration is governed by the steady (2-D) from of the continuity; the time averaged incompressible Navier-Stokes equations and energy equation. In the Cartesian tens or system these equations can be written as follows Quatember and Muhlthaler (2003).

Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (3.1)$$

Momentum equation:

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right)=-\frac{\partial p}{\partial x}+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y} \tag{3.2}$$

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}\right)=-\frac{\partial p}{\partial y}+\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y} \tag{3.3}$$

Where:

$$\tau_{xy}=\tau_{yx}=\mu\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right) \tag{3.4}$$

$$\tau_{xx}=2\mu\frac{\partial u}{\partial x} \tag{3.5}$$

$$\tau_{yy}=2\mu\frac{\partial v}{\partial y} \tag{3.6}$$

Energy equation:

$$\frac{\partial}{\partial x_j}\left(\rho u_j C_p T-k\frac{\partial T}{\partial x_j}\right)=u_j\frac{\partial p}{\partial x_j}+\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\mu\frac{\partial u_k}{\partial x_k}\delta_{ij}\right] \tag{3.7}$$

Where:

C_P = The specific heat at constant pressure, (kJ/kg. K).

K = The turbulent kinetic energy, ($\mathrm{m^2/s^2}$).

ε = The turbulent dissipation rate,($\mathrm{m^2/s^3}$).

μ = The Dynamic viscosity, ($\mathrm{N.s/m^2}$).

The Reynolds-averaged approach to turbulence modeling requires that the Reynolds. Stresses $(-\rho\overline{\acute{u}_i\acute{u}_j})$ need to be modeled.

For closure of the equations, the (k-Ɛ) turbulence model was chosen. A common method employs the Boussinesq Hypo thesis to relate the Reynolds stresses to the mean velocity gradients:

$$(-\rho\overline{\acute{u}_i\acute{u}_j})=\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \tag{3.8}$$

The turbulent viscosity term (μ_t) is to be computed from an appropriate turbulence model. The expression for the turbulent viscosity is given as:

$$\mu_t=\rho C_\mu\frac{K^2}{\varepsilon} \tag{3.9}$$

In the present study, (k-ε) Renormalized Group (RNG) turbulence model is used as follows:

$$\frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho\varepsilon \qquad (3.10)$$

Similarly the dissipation rate of Turbulence Kinetic Energy (TKE)ε is given by the following equation:

$$\frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \qquad (3.11)$$

Where:

$G_k =$ The rate of generation of the Turbulence Kinetic Energy (TKE) while $(\rho\varepsilon)$ is the destruction rate (G_k) is written as:

$$G_k = (-\overline{\rho \acute{u}_i \acute{u}_j})\frac{\partial u_j}{\partial x_i} \qquad (3.12)$$

3.5 Finite Volume Method (FVM)

The finite volume method is a discretization method which was well suited for the numerical simulation of various types (elliptic, parabolic, and hyperbolic) of conservation laws. Some of the important features of finite volume method were similar to these of finite element method. It may be used on arbitrary geometries, using structured or unstructured meshes and it leaded to robust schemes. Feature is the local conservatively of the numerical fluxes that was the numerical flux was conserved from one discretization cell to its neighbor. Finite volume method is locally conservative because it was based on a balance approach: local balance was written on each discretization cell which often called “control volume” by divergence formula an integral formulation of the fluxes over the boundary of the control volume was then obtained. The fluxes of boundary were discredited with respect to the discrete unknown. The principle of the finite volume method was given a number of the discretization points which may be defined by mesh, to assign one discrete unknown per discretization point and to write one equation per discretization point at each discretization point the derivatives of the unknown were replaced by finite volume through the use of Taylor expansions Versteeg et. al. (2007).

Quantity (ϕ) as generic conservation is considered and the velocity field and all of the fluid properties are informed to be known. The governing equations of steady convection diffusion could be presented as general transport equation for property (ϕ) as:

$$div(\rho\phi u) = (\Gamma\nabla\phi) + S_{\phi} \tag{3.13}$$

Eq. 3.13 is adopted as a starting point for computational procedures in (FVM) for fluid flow and heat transfer processes. The computational domain of rib-groove channel is discredited into finite set control volumes or cells. This process is known as the geometrical discretization of the domain Patankar et. al. (1972). The sequence of the general finite volume method approximation which is used for steady convection diffusion problems in the present study are introduced as follows:

1) The general transport equation for continuity, momentum, and energy are applied to each cell, computational domain and discretized as illustrated in Fig. 3.6.

   Grid independent test has been performed for the physical model to obtain the most suitable mesh faces size. In this study, after testing a number of different mesh faces at case of the nozzle in center with slant angle is (α=90º) of jet impingement. All mesh faces are used to plot the average Nusselt numbers (Nu_{avg}) at the curve surface. The discretization grid is hybrid unstructured and non-uniform and the number of Cells=66758, number of Faces=101699 and number of Nodes=34942. In this case, mesh faces with 101699, is used as it's the best in terms of both the accuracy and computational time. In fact, better accuracy can obtained in grid size with more nodes in terms of results but with increasing the density of the cells would make the computational time longer. For the curve surface the boundary layer mesh was used in order to get better result with high accuracy. A boundary layer mesh is a mesh with dense element distribution in the normal direction along specific boundaries.

REFERENCES

Hollorth, M, Durbin, BR. (1992). *"Impingement Cooling of Electronics"*. ASMEJ of Heat Transfer. Vol.114.607-613.

Ferrari, J., Lior, N., and Slycke, J. (2003). *"An evaluation of gas quenching of steel rings ringsby multiple -jet impingement"*. J.Mater.Process.Technol.Vol.136.190–201.

J. C. Han, J. S. Park, and C. K. Lei, P.P. (1985) *"Heat transfer enhancement in channels withturbulencepromoters,"*. JournalofEngineeringforGasTurbines and Power.Vol. 107. 628 - 635.

Kondjoyan, A., Peneau, F. and Boisson, H.C.(2002) *"Effect of High Free Stream Turbulence on Heat Transfer between Plates and Air Flows"*.A Review of Existing Experimental Results. Int. Journal of Thermal Science.Vol. 41. 1-16.

Lasance, C., Technical Data column, January (1997) *"Electronics Cooling"*.

T.L.Chan, Y.Zhou, M.H.Liu, C.W.Leung, P.P. (2003) *"Mean flow and turbulence measurements of the impingement wall jet on a semi-circular convex surface,"*. Experiments in Fluids. Vol.34.140-149.

Thakare S.S. and Joshi J.B., (2000) *"CFD modeling of heat transfer in turbulence pipe flows."*. Aiche Journal. Vol.46. P.P. 1798 -1812.

Mayle,R.E.,Blair,M.F.and Kopper,F.C.(1979*). "TurbulentBoundary Layer Heat Transfer on Curved Surfaces."*. ASME Journal of Heat Transfer.Vol.101.521– 525.

Martin, H. (1977). *" Heat and Mass Transfer between Impinging Gas Jets and Solid Surface."*. Advances in Heat Transfer. Vol. 13. 1 – 60.

Jambunathan, K., Lal, E., Moss, M., A., and Button, B. L. (1992). *"A Review of Heat Transfer Data for Single Circular Jet Impingement."*.Int. J. Heat and Fluid Flow. Vol. 13. 106 - 115.

Ebru Öztekin, Orhan Aydin, Mete Avc (2012) *"Hydrodynamics of a turbulent slot jet flow impinging on a concave surface"*.International Communications in Heat and Mass Transfer 39 pp. 1631–1638.

Ebru Öztekin, Orhan Aydin, Mete Avc (2013) *"Heat transfer in a turbulent slot jet flow impinging on concave surfaces"*. International Communications in Heat and Mass Transfer 44, pp. 77–82.

Shyy Woei Chang, Tussling Yang, Dar-Wei Shih (2009) *"Jet-array impingement heat transfer in a concentric annular channel with rotating inner cylinder"*. International Journal of Heat and Mass Transfer 52, pp.1254–1267.

Mansoo Choia, Han Seoung Yooa, GeunyoungYanga, JoonSikLeea, Dong KeeSohn (2000) *"Measurements of impinging jet flow and heat transfer on a semi-circular concave surface"*. International Journal of Heat and Mass Transfer 43, pp.1811-1822.

T.J. Craft, H. Iacovides, N.A. Mostafa (2008) *"Modelling of three-dimensional jet array impingement and heat transfer on a concave surface"*. International Journal of Heat and Fluid Flow 29, pp.687–702.

Hayder Eren, Nevin Celik, Bulent Yesilata (2006) *"Nonlinear flow and heat transfer dynamics of a slot jet impinging on a slightly curved concave surface"*.

M.Fenot, E. Dorignac, J.-J. Vullierme (2008) *"An experimental study on hot round jets impinging a concave surface"*. International Journal of Heat and Fluid Flow 29, pp. 945–956.

Michele Imbriale, Andrea Ianiro, Carosena Meola, Gennaro Cardone (2014) *"A Convective heat transfer by a row of jets impinging on a concave surface"*. International Journal of Thermal Sciences 75, pp.153-163.

C.H. Lee, K.B. Lim, S.H. Lee, Y.J. Yoon, N.W. Sung (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 31, pp. 559–565.

Tadhg S. O'Donovan *, Darina B. Murray (2007) *"Jet impingement heat transfer – Part I: Mean and root-mean-square heat transfer and velocity distributions"*. International Journal of Heat and Mass Transfer 50 (2007) 3291–3301.

Javad Mohammad pour, Mehran Rajabi-Zargarabadi, Arun S. Mujumdar, HadiAhmadi (2014) *"Effect of intermittent and sinusoidal pulsed flows on impingement heat transfer from a concave surface"*. International Journal of Thermal Sciences 76, pp. 118-127.

M.A.R. Sharif*, K.K. Mothe (2010) *"Parametric study of turbulent slot-jet impingement heat transfer from concave cylindrical surfaces"*. International Journal of Thermal Sciences 49, pp. 428–442.

M.A.R. Sharif, N.M. (2013) *"Ramirez Surface roughness affects on the heat transfer due to turbulent round jet impingement on convex hemispherical surfaces"*. Applied Thermal Engineering 51, pp.1026-1037.

V.I. Terekhov, S.V. Kalinina, Yu. M. Mshvidobadze, K.A. Sharov (2009) *"Impingement of an impact jet onto a spherical cavity"*. Flow structure and heat transfer International Journal of Heat and Mass Transfer 52, pp. 2498–2506.

Geunyoung Yang, Mansoo Choi, Joon Sik Lee (1999) *"An experimental study of slot jet impingement cooling on concave surface: effects of nozzle configuration and curvature"*. International Journal of Heat and Mass Transfer 42, pp. 2199-2099.

Yue-Tzu Yang, Tzu-Chieh Wei, Yi-Hsien Wang (2011) *"Numerical study of turbulent slot jet impingement cooling on a semi-circular concave surface"*. International Journal of Heat and Mass Transfer 54, pp. 482–489.

M.Venturino, P. Rubini, (1995) *"Coupled fluid flow and heat transfer analysis of steel reheat furnaces"*.School of Mechanical Cranfield University.

B.Quatember, H.Muhlthaler, (2003) *"Generation of CFD Mesh from Biplane Angiograms: an example of Image-Based Mesh Generation and simulation"*. Applied Numerical Mathematics, Vol.46, , pp379-397.

H.K Versteeg, alalasekera, (2007) *"An Introduction to Computational Fluid Dynamics"*. The Finite Volume Method:Prentice Hall.

S.V. Patankar, D.B. Spalding, (1972) *"A Calculation procedure of heat, mass and momentum transfer in Three-Dimensional parabolic flows"*. Heat and Mass Transfer, Vol.15, pp 1787-1806.

S.K. Das, N. Putra, P.Thiesen, W.Roetzel,(2003) *"Temperature Dependence of Thermal Conductivity Enhancement for Nan fluids"*.Heat Transfer,Vol.125, pp 567–574.