MARMARA UNIVERSITY
INSTITUTE FOR GRADUATE STUDIES
IN PURE AND APPLIED SCIENCES

NUMERICAL STUDY OF JET IMPINGEMENT COOLING ON A FLAT PLATE

MOHAMED SALEH KADDAFI

MASTER THESIS

Department of Mechanical Engineering

THESIS SUPERVISOR

Assoc. Prof. Dr. MUSTAFA YILMAZ

THESIS CO- SUPERVISOR

Assoc. Prof. Dr. MURAT ÇAKAN

ISTANBUL , 2017

MARMARA UNIVERSITY
INSTITUTE FOR GRADUATE STUDIES
IN PURE AND APPLIED SCIENCES

NUMERICAL STUDY OF JET IMPINGEMENT COOLING ON A FLAT PLATE

MOHAMED SALEH KADDAFI

(524612921)

MASTER THESIS

Department of Mechanical Engineering

THESIS SUPERVISOR

Assoc. Prof. Dr. MUSTAFA YILMAZ

THESIS CO- SUPERVISOR

Assoc. Prof. Dr. MURAT ÇAKAN

ISTANBUL , 2017

MARMARA UNIVERSITY
INSTITUTE FOR GRADUATE STUDIES
IN PURE AND APPLIED SCIENCES

Mohamed Saleh KADDAFİ , a Master of Science student of Marmara University Institute for graduate studies in Pure and Applied Sciences, defended his thesis entitled "**Numerical study of jet impingement cooling on a flat plate**", on July 8, 2017 and has been found to be satisfactory by the jury members.

Jury Members

Assoc. Prof. Dr. Mustafa YILMAZ (Advisor)

Marmara University ....

Assist. Prof. Dr. Hasan KÖTEN

Istanbul Medeniyet University ....

Prof. Dr. Mustafa ATMACA

Marmara University ....

APPROVAL

Marmara University Institute for Graduate Studies in Pure and Applied Sciences Executive Committee approves that Mohamed Saleh KADDAFİ be granted the degree of Master of Science in Department of Mechanical Engineering Program on 22.01.2018 (Resolution no: 2018/02-02).

Director of the Institute
Prof. Dr. Bülent Ekici

ACKNOWLEDGEMENTS

During the studies of my master thesis several people have contributed academically, practically with my researches. My studies were easier thanks to them. That is why i wish to dedicate this section to recognize their support.

I would like thank my head supervisor Assist. Prof. Dr. Mustafa YILMAZ and my co-supervisor Assist. Prof. Dr. Murat ÇAKAN who shared their knowledge, experience and support during whole thesis process.

Besides my advisors, my sincere thanks also goes to Assist. Prof. Dr. Hasan KÖTEN for him support and experiences.

I am grateful to my beloved wife, Kübra Aydin whose constant encouragement, patience, love, and care has made it possible for me to carry the research to the end. I would also like to thank my two children, Sara and Rana without their love and smiles I would have never been able to get to this point.

Finally, I would like to thank my mum, for her support and encouragement even though she is half way across the world. I am also grateful for my brothers and sisters for their encouragement and love. I would like to thank the spirit of my dad for the determination and the strength that he gave me and would like to tell him, that your encouragement is a whisper in my ears for all the times I thought I wouldn’t make it, this work is a small piece of my appreciation for everything you’ve done for me and allowed me to do. Thank you for always being there.

Mohamed Saleh

June, 2017

Istanbul. TURKEY

TABLE OF CONTENTS

NUMERICAL STUDY OF JET IMPINGEMENT COOLING ON A FLATE PLATE

ÖZET

Jetler, özellikle yüksek ısı transfer katsayısına ulaşmanın en iyi yöntemlerinden biridir ve bu nedenle birçok mühendislik uygulamasında kullanılır. Bu çalışmada, jet performansını belirlemek amacıyla düz plakada dağıtılmış ısının mekanizmasını anlamaya çalıştık. Görüntülenen sayısal sonuçlarda elde edilen hedefler, bir yüzey plakasındaki jet çarpma özelliklerinin etkisini, yüzey plakası boyunca Nusselt sayılarının değişimini, düz plakadaki ısı transfer katsayısını ve düz bir plakadaki H/D etkisini belirlemektedir. Farklı Reynolds sayıları (500,1000,1500,2000) sabit bir ısı akısı altında (3000m/w2) ısı transfer katsayısını belirler. Programdan çıkarılan sonuçlarda, jet akışındaki FLUENT simülasyonu (2-D) süreklilik ve momentumun korunumu ve enerji denklemleri çözülmüştür. Bu çalışma farklı Reynolds sayılarının ve H/D'nin Nusselt sayıları ve ısı taşınım katsayısı üzerindeki etkisini kapsar. Sonuç, tüm olgularda Reynolds sayılarının artmasıyla Nusselt sayılarının arttığını ve ısı taşınım katsayısı h'nin H/D ile azaldığını ve bir Reynolds sayısı 2000 iken ısı taşınım katsayısı durma noktasında daha yüksek olduğu ve çıkış bölgesinde yavaş yavaş azaldığı için H/D maksimum değeri olan 3'e ulaştığını gösterdi.

.

Haziran, 2017 **Mohamed Saleh**

NUMERICAL STUDY OF JET IMPINGEMENT COOLING ON A FLATE PLATE

ABSTRACT

Impinging jets are one of the best methods 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 flat plate with the goal of identifying preferred methods to predicting jet performance. The goals that have been achieved in the numerical results displayed determine the influence of the jet impingement characteristics on a surface plate, determine the variation of Nusselt numbers (Nu) along the surface plate and heat transfer coefficient on flat plate, and study the H/D effect on the/a fate plate .Different Reynolds numbers (Re) of (500, 1000, 1500, and 2000) determine the heat transfer coefficient under a constant heat flux 3000m/w^2.The results of the program, which was extracted, are FLUENT , simulation (2-D) in the jet flow and the continuity, momentum and energy equations were solved. This study covers the effect of different Reynolds numbers (Re) and H/D on Nusselt numbers and heat transfer coefficient. The result showed that Nusselt numbers increased with the increase of Reynolds numbers (Re) for all cases, and heat transfer coefficient h decreased with H/D and reached a maximum of H/D =3 at a Reynolds Number = 2000, as heat transfer coefficient is higher at the stagnation point and decreases gradually at the outlet region.

June, 2017 Mohamed SALEH

LIST OF SYMBOLS

T	: Temperature	(^{0}C)
ΔT	: Temperature Difference	(^{0}C)
P	: Pressure	(kPa)
k	: Thermal conductivity	(W/m K)
h	: Local heat transfer coefficient, (W/m^2. K)	
Nu	: Local Nusselt number,	(dimensionless)
Pr	: Prandtl number,	(dimensionless)
Re	: Reynolds number,	(dimensionless))
q″	: Heat flux,	(W/m^2))
D	: Jet diameter	(m)
Cp	: Specific heat,	(J/kg. K)
V	: Velocity	(m/s)
u	: Velocity in x direction	(m/s)
v	: Velocity in y-direction	(m/s)
A	: Wall Area	(m^2)

Greek symbols

Φ	: Viscous dissipation function	
δ	: Boundary layer thickness	(m)
τ_w	: Wall shear stress	(N/m^2)
ρ	: Fluid density	(kg/m^3)

μ : Newtonian dynamic viscosity (kg/m.s)

Subscripts

a : Ambient

w : Wall

j : Jet

2D : Two Dimensional

x : local value of parameter on target wall

i, j : Indices for coordinate notate

x, y : Coordinates

ABBREVIATIONS

CFD	: Computational Fluid Dynamics
SCRUB	: Secure Computing Research for Users' Benefit
RIT	: Rotor Inlet Temperatures
FEMLAB	: Finite Element Method LABoratory
FIDAP	: Fluid Dynamics Analysis Program.

LIST OF FIGURES

LIST OF TABLES

1. INTRODUCTION

Jet impingement is used in many industrial operations to achieve high rates of heat transfer. A liquid or gas jet can be used for industrial cooling and heating purposes. Cooling jet impingement is used to cool turbine blades walls of combustion chambers and electronic components. The parameters which possibly govern jet impingement are nozzle to plate surface distance (H), nozzle diameter (D), target surface geometry and the flow rate of the liquid or gas its Reynolds number Re. The gas turbine is an engine, which produces a large amount of energy based on its size and weight. Gas turbines are used for aircraft propulsion. Where power output of gas turbines increase with increasing turbine inlet temperature (TIT) defined as the temperature of the combustion chamber exhaust gases as they enter the turbine unit. Gas temperature is measured by the thermocouples to organize in the exhaust. Today there are many gas turbines which run on types of different fuels such as natural gas, diesel fuel, methane, crude, and biomass gases. In the past 20 years have seen significant growth in gas turbine technology, mainly due to the growth of materials technology, new coatings, new cooling schemes. In a simple gas turbine cycle (Figure 1), low air is drawn into a Compressor (state 1) where it is compressed to a higher pressure (state 2). Fuel is added to the compressed air and the mixture is burnt in a combustion chamber. The resulting hot products enter the turbine (state 3) and expand to state 4 and the air exhausts.

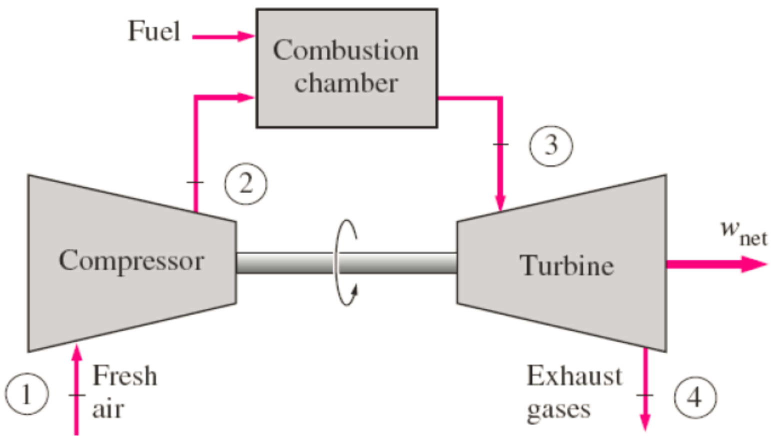


Figure 1.1 Schematic diagram of open gas turbine cycle

Most of the work produced in the turbine is used to run the compressor and the rest is used to run auxiliary equipment and to produce power. Figure 2 shows the schematic of cross section of a small gas turbine.

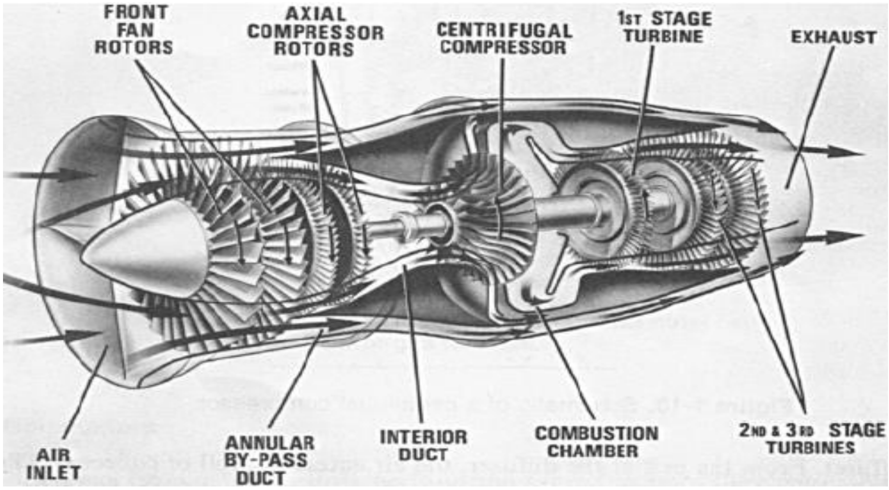


Figure 1.2 Sketch of a small gas turbine

1.1. Turbine blade

The gas turbine engines operate at high temperatures (1200-1600 °C) to improve thermal efficiency and power output. However, the heat transferred to the turbine blade is substantially increased as the turbine inlet temperature is continuously increased. The above operating temperatures are far above the permissible metal temperatures. Therefore, there is a need to cool the turbine blades for safe operation. The blades are cooled by extracted air from the compressor of the engine. Gas turbine blades are cooled internally and externally. Internal cooling is achieved by passing the coolant through several enhanced serpentine passages inside the blades and extracting the heat from outside the blades as see in the Figure 1.3.The cooling system must be designed to ensure that the maximum blade surface tempertures during operation are compatible with the maximum blade thermal stress.

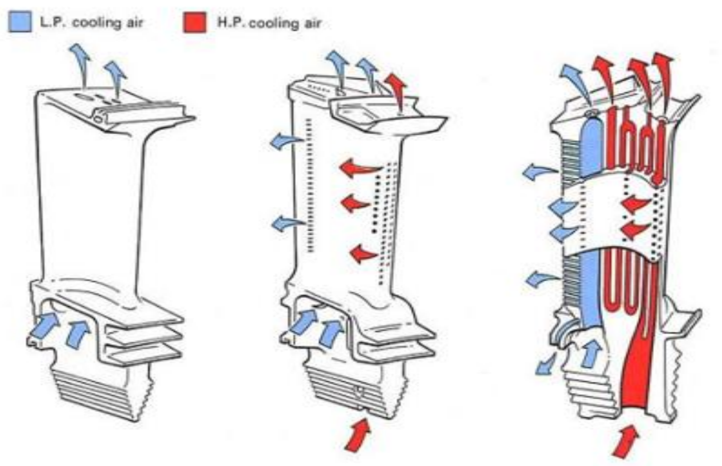


Figure 1.3 Cross section of a turbine blade with the low and high-pressure cooling airflow

In the Figure 1.4 shows a method of cooling the blade in the turbine, where the cooling air is issuing from the compressor and is orientation to the stator for cooling turbine rotor and casing to provide sufficient cooling.

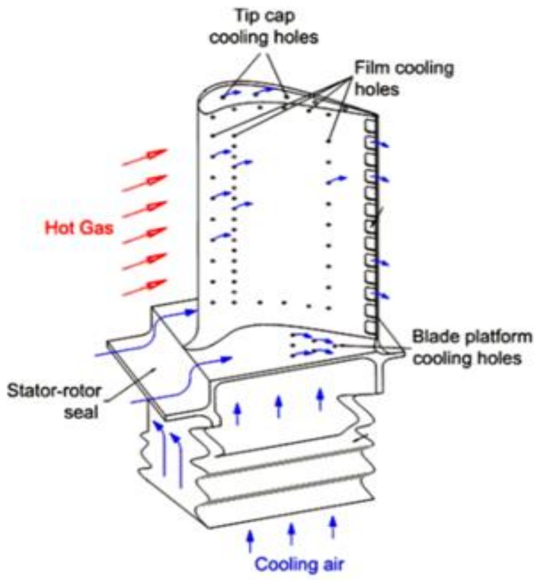


Figure 1.4 Schematic diagram of the modern gas turbine blade with common cooling techniques

1.2. Problem Statement

A lot of equipment or appliances need to have high heat transfer performance to guarantee the quality and to increase the capability old cooling systems cannot be used anymore, as they do not cool sufficiently. This makes it necessary to develop new techniques to meet the demand, 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 4 etc. 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 to achieve efficient so researchers are moving toward the technology of the jet impingement cooling systems. In gas turbine systems, turbine blades inside gas turbine engines are exposed directly to hot gas from the combustion chamber. To meet the demand of 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 cooling and to develop a cooling method to protect gas turbine components and to find the best heat transfer coefficient for these problems.

1.3. Jet impingement cooling

Jet impingement cooling is an enhanced heat transfer method capable of cooling a combustor liner without injecting cool air directly into the combustion chamber. Cooling the liner from the backside enables engineers to dissipate the heat load and maintain more uniform temperatures in the combustion region needed for efficient combustion. Figure 1.5 shows a typical combustor liner using backside jet impingement cooling.

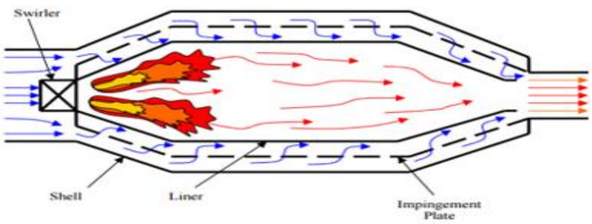


Figure 1.5 Jet Impingement Cooling of Combustor Liner

An impingement array is composed of a jet plate typically having round holes, which produces the impinging jets. The jets strike the surface to be cooled, referred to as the target plate. Traditionally, the structure of an impinging jet is broken down into three parts, the potential core, shear layer, and the wall jet. Figure 1.6 shows the impinging jet structure and the associated regions.

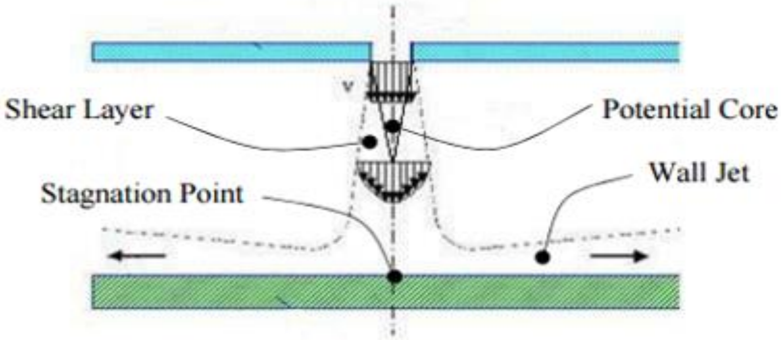


Figure 1.6 The structure of the impinging jet

At the discharge of the jet plate, the velocity profile of the jet is relatively uniform. As the jet discharges, viscous forces acting in the shear layer cause the jet velocity profile to develop and expand. The potential core of the jet is defined as the region where the viscous forces have little or no effect on the velocity profile where shear layer exists between the potential core and the ambient fluid. Once the jet strikes the target plate, the wall jet is formed as the fluid travels along the wall.

1.4. Configurations of Impinging Jets

There are two different flow configurations can distinguish submerged impinging jets and free impinging jets as see in Figure1.7. In the submerged impinging jets case, the fluid flowing from the nozzle is of the same nature as the surrounding. While In the free impinging jets the fluids are of a different nature (e.g., a water jet issuing in air). The dynamics of submerged impinging jets and free impinging jets are different, Where 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. While in free jets, the turbulent motion in the shear layer does not have a substantial amount effect on the flow.

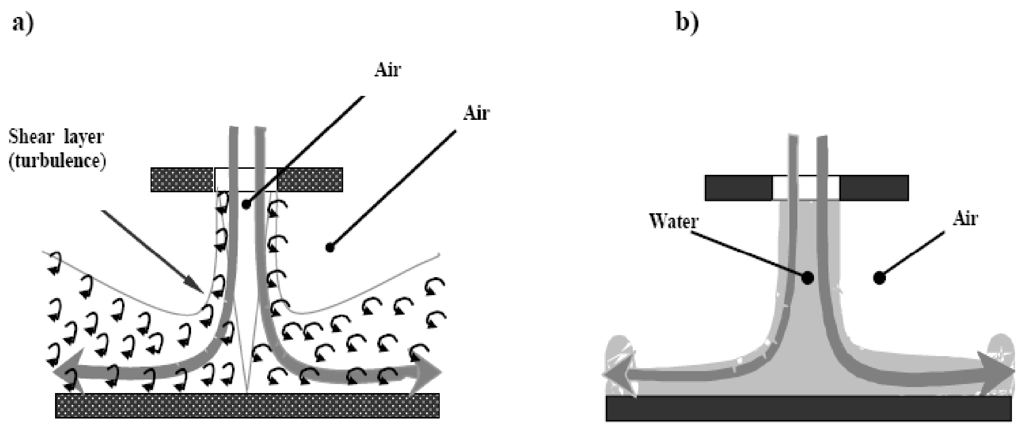


Figure 1.7 Comparison between a) Submerged jet; and b) Free impinging jet

1.5. Project Objective

This study has three main objectives, which are:

- Measure the heat transfer coefficient on plate surface.
- Define the relationship between the heat transfer coefficient with the distance between the nozzle exit and the target surface system increases with rising refrigerated space.
- Define the relationship between heat transfer coefficient and Reynolds number.

1.6. Project Scopes

This experimental study was carried out using air as the coolant medium that impinge with laminar flow region from the nozzles to the heat source which has Reynolds number in the range of 500 – 2000 (500, 1000, 1500, and 2000). The diameter of nozzle was (13 mm). The heat source will be at constant heat flux with 3000 w/m^2, which used aluminum. The dimensionless parameter that was defined in this study is the dimensionless jet to heat spacing, H/D (1, 1.5, 2, and 3).

1.7. Research Significance

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

1.8. Literature Review

Gardon and Akfirat, presented a method for the calculation of the variation of local heat transfer coefficients, velocity and turbulence distributions produced by impinging submerged jets. They found that increase in velocity also enhances the local heat transfer coefficient **[1]**.

Incropera and DeWitt, For laminar flow, they used the following expression for the local Nusselt number, $Nu_x = 0.332\ Re^{-1/2}\ Pr^{\ 1/3}$ **[2]**.

Hoffman et al, (2007) Correlations for local heat transfer coefficients are reported by Hoffman et al. (2007). Where conducted an experimental investigation on flow structure and heat transfer from a single jet impinging on a flat plate. The influences of nozzle to plate spacing and Reynolds number on local heat transfer coefficient are investigated **[3]**.

Mohamad Nor Mus, studied the effec of jet impingement cooling system on the turbine blade pressure side where blade is made from mild steel. Experiment has been conducted with various Reynolds number, Re, is varied from (646, to 2637); and nozzle exit to surface distance, H/D = (4 ,8, and12) The effectiveness of jet impingement cooling is determined depending on the heat loss from the turbine blade pressure side to the surrounding. The maximal of heat loss to the surrounding means a more effective jet impingement cooling system. The heat loss in this experiment is determined by the heat transfer rate and the Nusselt number. In terms of Reynolds number, the highest Nusselt number is produced when Reynolds number, Re = 2637, followed by 1970, 1322 and lastly 646. From the results, when Reynolds number increase, the Nusselt number will increase as well. It can be said that they are directly proportional together, it is noticed also when the heat transfer rate increases the Nusselt number would also an increase. From the experiment, the heat transfer rate increase as the nozzle exit to surface distance H/D decrease. **[4]**.

O'Donovan and Darina B. Murray, they have been measurement and comparison of heat transfer to submerged and confined air and water jets. The present paper reports on surface heat transfer measurements for a range of test parameters. These parameters include jet diameter (d from 0.5mm to 1.5mm), Reynolds number Re from (1000 to 20000) and jet to target spacing H from (0.5d to 6d). The results are presented in the form of local and full-field heat transfer coefficients for the range of conditions indicated. For low air jets the result was with decreasing H/d and increasing Reynolds number **[5]**.

Mark Michael Dobbertean, it is obsirved that increasing the Reynolds number increases the heat transfer coefficient due to the increased fluid velocity along the surface of the plate.and The maximum heat-transfer coefficient is present inside the stagnation zone **[6]**.

Nizam Bin Memth, Experiments were conducted to determine the effect of nozzle diameter on the heat transfer coefficients from a small heat source to a jet impingement cooling system, submerged and confined air. The experiment were carried out with a single jet with three different nozzle diameter, D; (0.5, 1.0, 2.0 cm) and four dimensionless jet to heat source spacing, H/D (6, 8, 10, 12) were tested within the laminar jet Reynolds number ranging from (500-2300). The results indicate tha by the increasing of the heat transfer coefficient, h the local Nusselt number, Nu also will be increase **[7]**.

Nakod, The maximum heat transfer coefficient is observed at the stagnation point, the heat transfer decreases in the horizontal direction of all the nozzles to the plate spacing values. The highest Nusselt (Nu) decreases with increasing H / D. The reason of this difference with H / D is that the fluid acceleration through a small distance between the surface plate and the nozzle outlet is effective more than with a increase distance [**8]**.

Farial A. Jafar, in this study is focused on a liquid jet impinging a flat and curved surface. This investigation comprises a study of the flow field and heat transfer within the liquid film. jet Reynolds number, from 100 to 1900. The inlet water jet temperature is 25 ºC and the heat flux = 1000 W/m^2. The objective is to determine the distribution of the local Nusselt numbers on the surface of the target surface for different Reynolds number (Re), jet to plate distance and the surface configuration (flat and curved) surfaces. The results indicate that Re has a significant effect on the flow field and heattransfer. The heat transfer rate in the stagnation and wall jet regions is enhanced with an increase in Reynolds number (Re). Increasing the Reynolds number increases the diameter of the cooling zone around the stagnation point significantly and causes a decrease in temperature at the stagnation point. When the Reynolds number increases, the consequence is an increase in the velocity of the flow and an increase in the convective heat transfer which results in better surface cooling **[9]**.

Ali Mohammed, 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).he is found, 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) **[10]**.

Yunus Cenegel Heat Transfer, The viscosities of liquids decrease with temperature, whereas the viscosities of gases increase with temperature **[11]**.

Chitranjan Agarwall, he studied the effect of nozzle to surface spacing and Reynolds number on the heat transfer in cooling of electronic components 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 electronic components H varied from 2 to 10 nozzle diameter. Experiments are conduits with nozzle diameter of 5 mm.it is shown that for different Reynolds number, the surface temperature can be significantly decrease by reducing the jet diameter. The effect of nozzle aspect ratio is lees evident as the spacing nozzle to surface of electronic component is increase. The jet Reynolds number dose not control the role of nozzle aspect ratio on heat transfer rate. The heat transfer rate increase as the jet spacing decrease owing to the reduction in the impingement surface area **[12]**.

Viskanta, Jet behavior is typically categorized and correlated by its Reynold number Re =U D/v defined using initial flow speed U, the fluid viscosity (v) and the characteristic length that is that is the nozzle exit diameter D, At Re <1000 the flow field exhibits laminar flow properites .at Re >3000 the flow has fully turbulent features. A transition region occurs with 1000<Re<3000 **[13]**.

Huang and El-Genk, have investigated the heat transfer between a uniformly heated flat plate and a circular impinging jet. They employed different Re along with different nozzle to plate distances. They found that the maximum Nusselt number occurred at the stagnation point where x = 0. **[14]**.

Narayanan et al, **(2004),** The highest stagnation Nusselt number is observed at the highest Reynolds number and at the lowest nozzle to plate spacing. As stated before the stagnation Nusselt number is maximum at the lowest nozzle to plate spacing, because the impinging cooling area at a lower nozzle to plate spacing is larger than in higher nozzle to plate spacing, these results indicate that the heat transfer coefficient decreases monotonically with increasing distance from the jet nozzle **[15].**

Han, and Ekkad, The impinging jet flows may be used to cool several different sections of the engine such as the combustor case (combustor can walls), turbine case/liner, and the critical high temperature turbine blades. The gas turbine compressor offers a steady flow of pressurized air at temperatures lower than those of the turbine and of the hot gases flowing around it. The blades are cooled using pressurized bleed flow, typically available at 6001° C. The bleed air must cool a turbine immersed in gas of 1400℃ total temperature **[16]**.

Chougule N.K, Effects of jet Reynolds number (Re), target spacing-to-jet diameter ratio (H/D) on average Nusselt number (Nu_a) of the target plate are examined. These numerical results are compared with the available benchmark experimental data. It is found that Nu_a increases from 40 to 50.1 by increasing Reynolds number from 7000 to 11000 at H/D =6. By increasing H/D ratio from 6 to 10, Nu_a decreases from 50.1 to 36.41 at Re11000. It is also observed that in multi-jet impingement, spacing between the air jets play important role. A value of nozzle pith of 3 to 5 was recommended to reduce adjacent jet interference. **[17]**.

Jambunathan and Button, Jets impinging on a smooth surface have been studied by Jambunathan et al. (1992). They presented literature surveys of jet impingement that reported enhanced rates of heat transfer, with non uniform radial distribution of the local and surface average Nusselt numbers. **[18]**.

Siba et al and Leel, have investigated the relationship between the heat transfer to the jet flow and the fluctuations of the free stream velocity **[19]**.

Lee et al, in this study is focused on effect of nozzle diameter (D) (1.36, 2.16, and 3.40 cm) on impinging jet heat transfer and fluid flow. The results indicate that the Nusselt numbers in the stagnation point region corresponding to $0 \leq X/D \leq 0.5$ increases with increasing nozzle diameter, where X is the distance of the flat plate from the stagnation point. This is attributed to the increase in the jet momentum with the larger nozzle diameter, which results in increased the heat transfer at the stagnation point **[20].**

K Marzec1 **and A Kucaba-Pietal**, An impingement cooling system is defined as an array of jets of high velocity which is made to strike a target surface. An impinging jets can be classified as a submerged impinging jet or a free impinging natural, daynamics of both case deffrence. In submerged jet, the fluid issuing from the jet is of the same density and nature as that of the surrounding fluid. While in the free jet, the fluid issuing from the nozzle is of a different density and nature than that of the surrounding fluid. **[21]**.

D.P De Witt, In this the experiment confirm that the laminar boundary layer will become unstable and turbulent. This instability occurs across different fields and with various fluids, usually Rex ≈ 5×105, where x as the characteristic flow dimension, and it's measure distance from the leading edge of the flat plate, while the flow velocity defined as free stream velocity of the fluid outside the boundary layer **[22]**.

Taylor,R.A, Heat transfer is classified into various mechanisms, such as thermal conduction, thermal convection, thermal radiation, and transfer of energy by phase changes **[23]**.

Clancy, L.J, In fluid dynamics, a stagnation point is a point in a flow field where the local velocity of the fluid is zero **[24]** .

Similarly to investigations of free jets, the use of CFD for characterizing the flow field associated with the impingement of high speed jets provides clarity and extends the capacity of traditional experimental testing. The first numerical investigations to address issues related to jet impingement were conducted in the 1980s. These included the development of turbulent models to accurately predict flows in the near wall region. A high Reynolds number form of the k_ε viscous model developed by Lam and Bremhorst was used to predict fully developed pipe flow **[25].**

Free single-phase jet impingement cooling is affected with many variables such as:

- Jet diameter (d)
- Fluid velocity (v)
- Jet to heated surface distance (H)
- Coolant properties **[26]**.

Due to their highly localised heat and mass transfer rates, impinging jets are widely used in many industrial applications, whether cooling or heating of a surface is required. Examples include, drying of clothes, cooling turbine blades, cooling of electronic equipment, use in aircraft anti-icing systems, etc. The study of impinging jets is also important from a turbulence modelling point of view. Models of momentum and heat transport in turbulent shear ows are developed by reference to ows parallel to walls. Furthermore, turbulence models for separated and recirculating ows are by no means satisfactory **[27].** In order to provide data for these models to be improved and/or developed, many researchers have studied impinging jets ows, both numerically **[28] [29] [30]**.

Asem Nabadavis* and Dipti Prasad Mishra, In this study is focused on studying the heat transfer characteristics when a high velocity air jet impinges upon a flat plate having constant heat flux by numerical analysis. Numerical analysis has been conducted by solving conservation equations of momentum, mass and energy with two equations based k- ε turbulence model to determine the wall temperature and Nu of the plate considering the flow to be incompressible. It was found from the investigation that the heat transfer rate increases with the increase of Reynolds number of the jet (Rej). It was also found that there is an optimum value for jet distance to nozzle diameter ratio (H/d) for maximum heat transfer when all the other parameters were kept fixed. Similar results as above were found when two jets of air were used instead of one jet keeping the mass flow rate constant. For a two jets case it was also found that heat transfer rate over the surface increases when the jets are inclined outward compared to vertical and inward jets and also there exists an optimum angle of jet for maximum heat transfer. It is noticed that heat transfer rate over the surface increases when the jets are inclined outward compared to vertical and inward jets and also there exists an optimum angle of jet for maximum heat transfer **[31-32]**.

K. Siva Satya Mohan1 and Dr. S. K. Bhatti, **A** numerical analysis has been performed to study the heat transfer performance of a hot fluid in a confined impinging jet on a flat heated surface. The tests were realized for the following ranges of the governing parameters: the jet thickness is 2m and the distance of horizontal jet to heated surface was set to 1 to 3 m. Three different cases are considered in this analysis. They are H/D = 0.5,1 and 1.5. Fluids like Acetylene and Acetyl chloride are compared in this analysis. Turbulent models considered for this analysis are Spallart Almaras, k-ω and k-ε. Out of these three k- ω model gives more heat transfer characteristics. It is found that when H/D = 0.5vertical Jet has more heat transfer rate than Horizontal Jet. [**33]**.

Table 1.1 Analytical results of heat transfer for jet impingement. Correspondence

Equations	Geometry	Flow type	References
$Nu = \frac{h\,D}{K}$	Impinging jet on flat and convex surfaces	Laminar	Chung et al. (2002) and Olsson (2004)
$Nu = 0.797\ Re^{1/2}\ Pr^{0.4}$	Impinging jet (circular liquid jet)	Laminar Pr >3	Liu et al. (1991)
$Nu = 0.332\ Re^{-1/2}\ Pr^{1/3}$	Flow on a flat plate	Laminar, local, $0.6 \leq Pr \leq 50$	Incropera and DeWitt (1990

Table 1.2 Summary of relevant of jet impingement

Investigation	Impingement surface	Investigation	Comments)
Slayzak et al (1994)	Flat	Heat transfer	Experimental – liquid
Wen and jang 2003	Flat	Flow field – Heat transfer	Experimental – Air
Gopper et al (2004)	Flat	Flow field – Heat transfer	Experimental – Air
Fento et al (2005)	Flat	Heat transfer	Experimental – Air
Dirita et al (2007)	Flat	Flow field – Heat transfer	Numerical – air jet
Farial. A. Jafar (2011)	Flat and Curved	Flow field – Heat transfer	Numerical- Air jet
Ali Mohammed (2014)	Semicircular convex	Heat transfer	Experimental – Air jet

2. MATERIALS AND METHOD

2.1 CFD Theories

In the present study simulations of two phases (gas and solid) using computational fluid dynamics (CFD) are carried out to investigate the flow field and heat transfer characteristics of a slot air jet impingement. The goal is to determine the distribution of the local Nussle numbers on the target surface with various Reyonda number, Re, and and nozzle exit to surface distance, (H/D). Numerical predictions are obtained using the power of computational fluid dynamics (CFD) Code - FLUENT for 2D. The flow field and heat transfer processes that occur in such configurations are generally calculated by means of experimental relationships amongst dimensionless groups. By using computational fluid dynamics (CFD) can understand the heat transfer processes that occur in over flat plate. In this case, single slot air jet is used on flat plate using a (CFD) for (2 -D) to study the effects of Reynolds number (Re), and nozzle to plate spacing (H/D) on the flow and heat transfer characteristics .

2.2. Brief about Fluent

A programming package about the fluid flow and transport of heat. It started as an independent package as an American/British collaboration, and later on was merged with ansys.

It is characterized by its high ability to accomplish acceptable results about the very complexed computing sector. As well as it can deal with different kinds of organized and non-organized networks of complexed forms in 2 and 3 dimensions. This package guarantees a good control to get the solution, and is characterized by its ability to work in different clusters for the single operation with high flexibility and workability between all the different clusters.

Also, all the required functions to get the solution and display the results are easily available in the lists and windows of the program.

2.3 Application of the computation methodology

In the Figure 2.1, The application of the computation methodology will be based upon the steps of the program (Ansys_fluent) to facilitate the sequences of information, and the following figure explains this sequence.

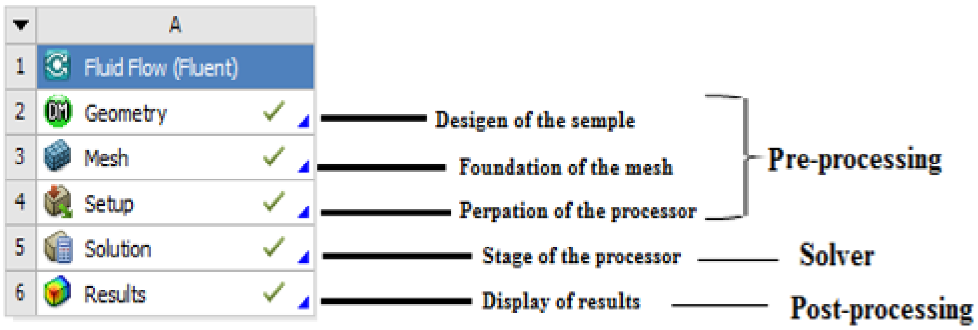


Figure 2.1 The sequences of the study used by the considered program (Ansys fluent)

2.3.1 Computational methods

The methodology contains three stages as follows:

2.3.1.1 Pre-processing:

In according to this phrase, it is clearly recognized all the inputs that is related to the problem of the program, which processed by the input tools and windows, just after, these inputs are converted to the set that suits the processor, and this phrase contains.

2.3.1.1.1 Creating the Geometry in ANSYS Design Modeler

The computational domain geometry was built using Ansys Workbench 14 as see in the Figure 2.2. Where the domain was chosen as small as possible to minimize the number of mesh elements as see in the Figure 2.3.

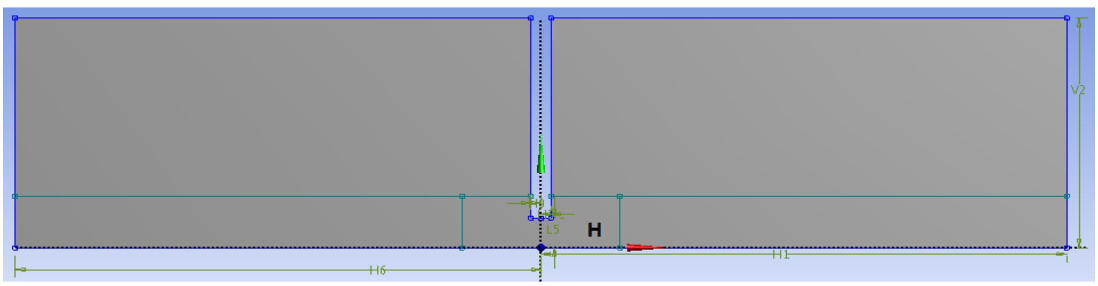


Figure 2.2 Geometry of the 2D jet model

2.3.1.1.2 Mesh

In Figure 2.3, the modelled grid is shown for flat surface. The meshing defined as important part in Ansys program, because it can show drastic changes in results this based on mesh size, where it is the process of dividing the domain into points and each point represents by an algebraic formula. Meshing means create a mesh of some grid-points called nodes. The accuracy of the results depends mainly on the size grid. Where if decrease size of the mesh can have obtained better accuracy but based on time, where the density of the cells would make the computational time longer, while if increase the size grid will be low accuracy and the density of the cells would make the computational time short.

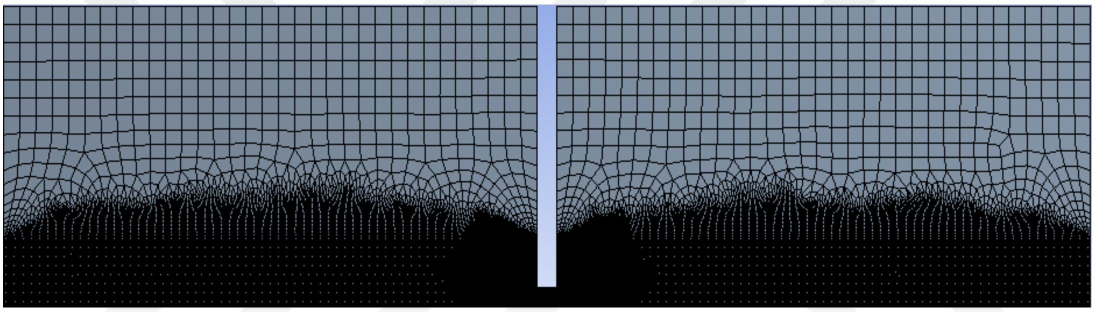


Figure 2.3 Generated mesh for the 2D jet model

2.3.1.1.2.1 Mesh Quality

The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. To study mesh quality has been study three different grids for the 2D geometries, the first simulation is with 51,020 elements, the second for 46,264 elements and the third with 39,194 elements as shown in Figure 2.4. The results show that using different grid elements has significant impact on identify simulation results. It is noticed that in the grid size of 51 ,020 elements can be get high accuracy results. Based on the previous results have been studied of the parameters under size mesh elements 51.020. The reason is because if decrease size of the mesh can have obtained better accuracy but based on time.

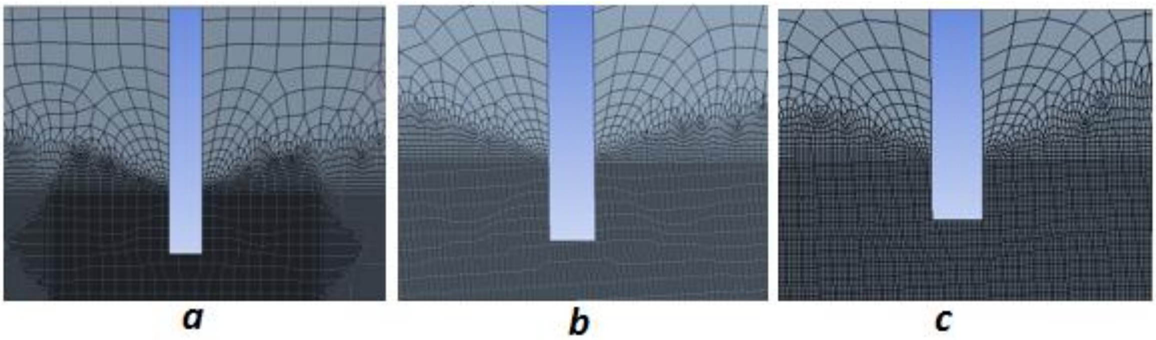


Figure 2.4. Zooming in at the jet with wall showing different grid elements: a) 51.020, b) 46,264 and c) 39,194.

2.3.1.1.3 The definition of boundary conditions

In the Figure 2.5, the boundaries condition is which surrounding the computational domain, where these boundaries assist effectively to recognize and describe problems and the correct method to create solutions.

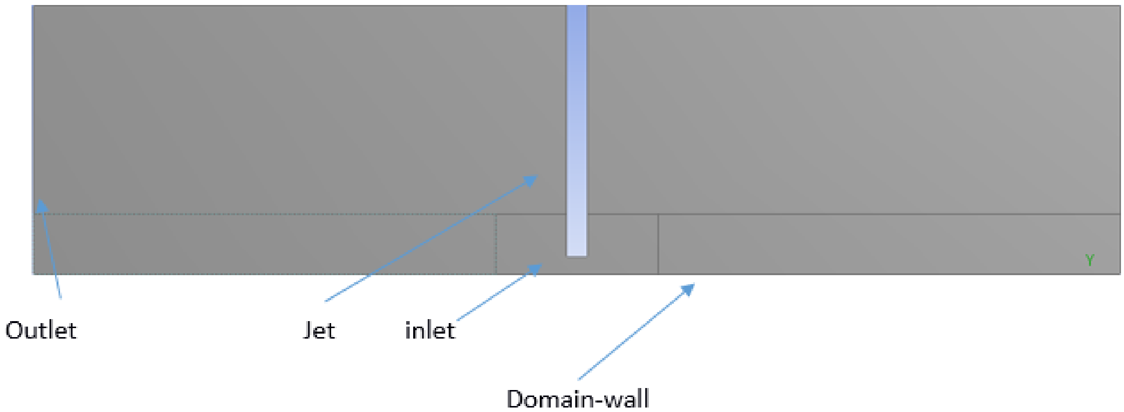


Figure 2.5 The studied geometry with the boundary conditions

2.3.1.1.4 Identify the properties of the materials

In Figures 2.6, 2.7 and 2.8, Create a new material is shown. where using the option Create/Edit Materials dialog box can define the materials such as air, solid, aluminum, in this the experiment the material is air.

After Select the material, according to temperature can change the properties of the air, where from air properties – Chart can identify density, specific heat capacity, thermal conductivity, and viscosity.

Property	Value
Density	1.225 kg/m3
C_P	1006.43 J/kg- K
Thermal Conductivity	0.0242 W/m - K
Viscosity	1.7894e-05 kg/m - s

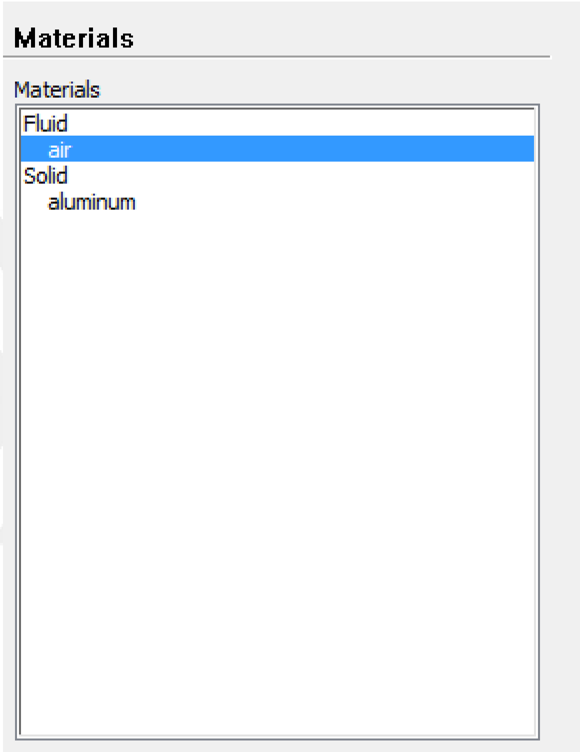


Figure 2.6 Selecting the working fluid (from materials as air)

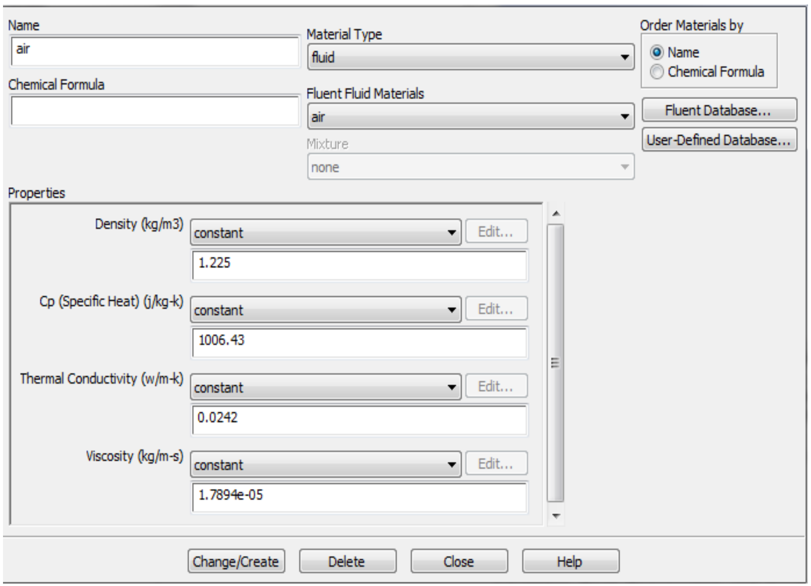


Figure 2.7 Define the type of material with its parameters (air)

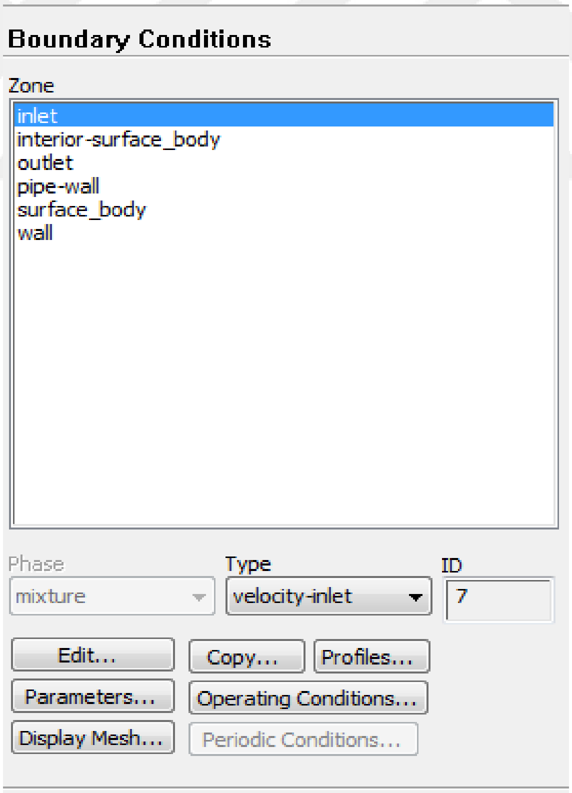


Figure 2.8 Defining the boundary conditions

In Figure 2.9, defined velocity inlet into the system is shown, where the velocity inlet (v_{in}) in jet region by using boundary condition. The velocity inlet at nozzle regions of v_{in} = (0.58, 1.15, 1.73, 2.31 m/s), respectively.

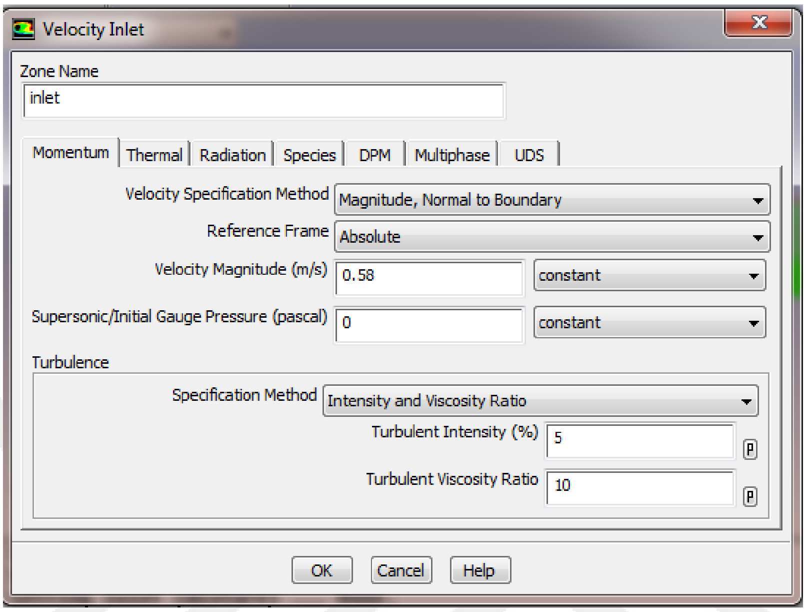


Figure 2.9 Define inlet velocity into the system

.

In Figure 2.10, defined heat flux into the system is shown, where the heat flux (q”) at wall region by using boundary condition.

- Click the Thermal tab and select heat flux re from the thermal conditions list.
- Enter 3000 w/m^2 for heat flux w/m^2.
- Click OK to close the Wall panel.
- . Close the Boundary Conditions panel

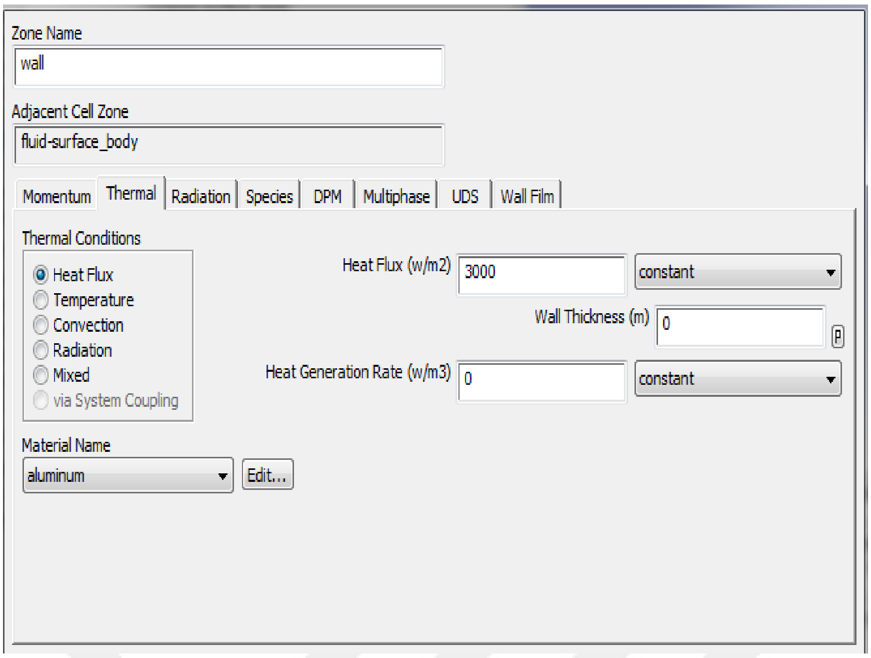


Figure 2.10 Define heat flux into the system

2.3.1.2 Processing

In this stage particularly, there is need to solve the problem, whereby the features of the stage based on following points

- In this, the stage transfers the equations (Algebraic equations) which have been getting from mesh into the system.
- Solve this the Algebraic equations by iteration, where it identifies particular number of the iteration, which the process stop carryin on as soon as reach such particular Number.
- With each iteration are the values account for the properties of the fluid at each point, and to find out whether The solution is correct or not, thus, it needs to be determined by the permittivity of the solution accuracy, which means to be an error ratio calculated between each iteration and the other, access to this percentage be a sign Access to resolve anticipated, and this is what is known as the convergence of the solution.

2.3.1.2.1 Solution initialization

In the Figure 2.11, Initialize the flow field is shown, by using the boundary conditions at the inlet velocity ($v_{in)}$ as a starting point.

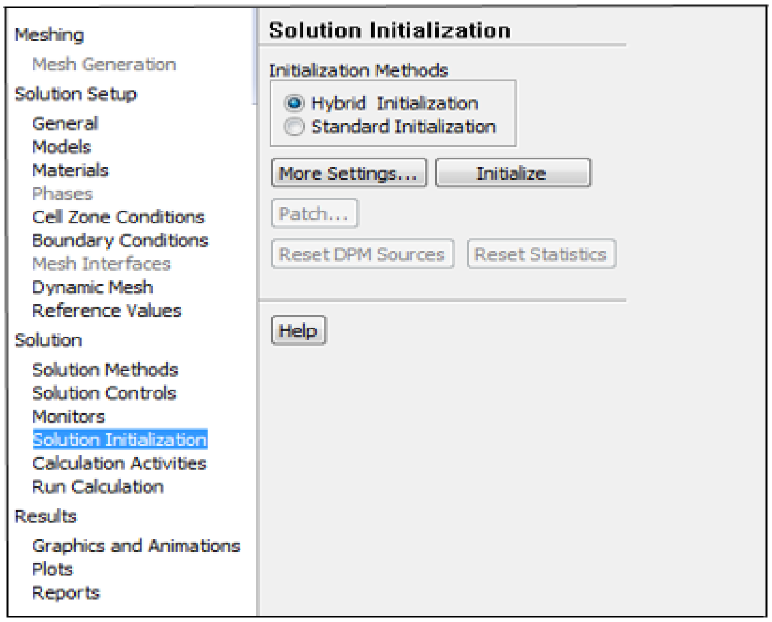


Figure 2.11 initialize the studied parameters for solution

2.3.1.2.2 Calculate a solution.

In the Figure 2.12, calculation by requesting 1000 iterations.

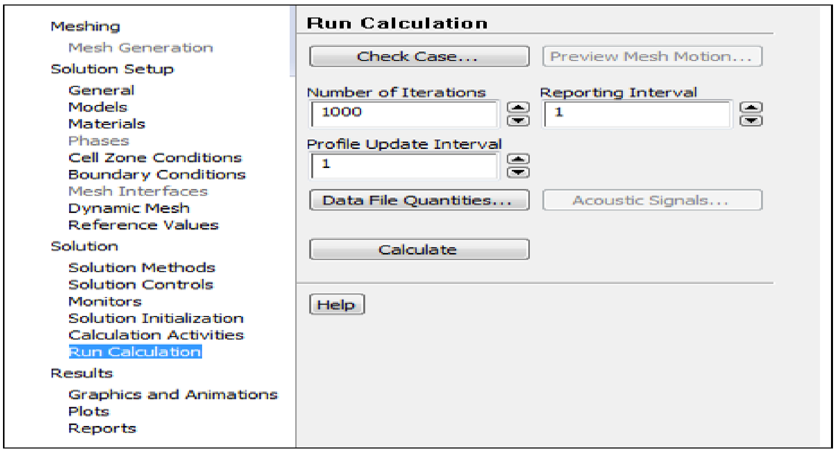


Figure 2.12 Determining the number of iterations for each calculation.

2.3.1.2.3 processing stage

The programs starting to solve the algebraic equations which must be done through requesting iterations, and a series of operations not appear to the user but its indicate to convergence solution when increase the number of iterations and Figure (2.13) illustrate the convergence solution. It can be noticed that the solution will be paused after approximately 1000 iterations when the residuals reach their specified values.

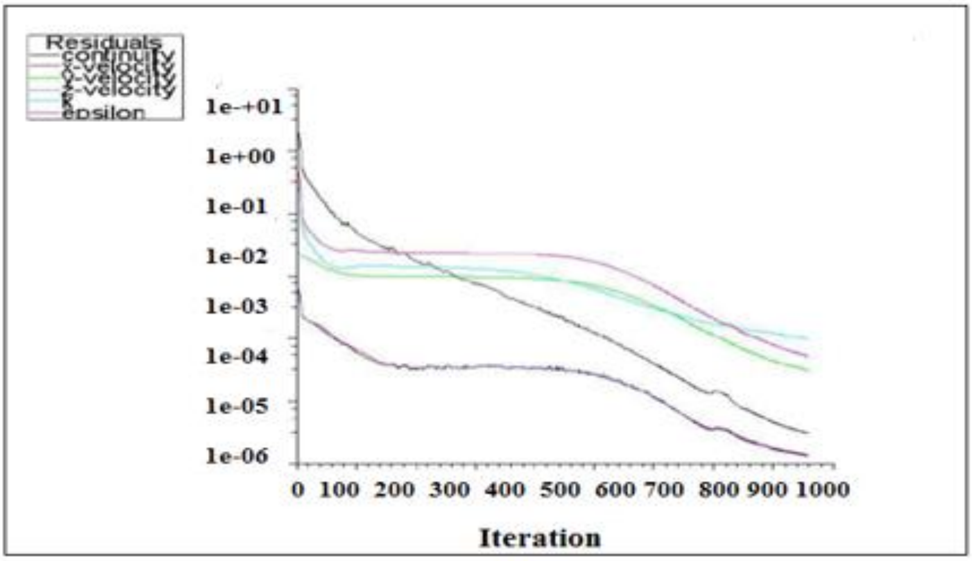


Figure 2.13 Residuals for the Converged Solution

2.3.1.3 post- processing:

In this, the stage has been shows and analysis results after received from processing, and this the stage shows:

- It is Show the values and direction of the flow.
- It is Shows distribution pressure, temperature and velocity.
- It is shows Animations from Images for model.
- It is shows charts for the properties and relations each other.

The velocity profile along the x is different in the impinging region, the velocity decreases quickly because of the wall friction. It can be seen from Figure 2.14 that near the stagnation region the x-velocity U increases, while the temperature will be high far from the stagnation region as see in the Figure 2.15.

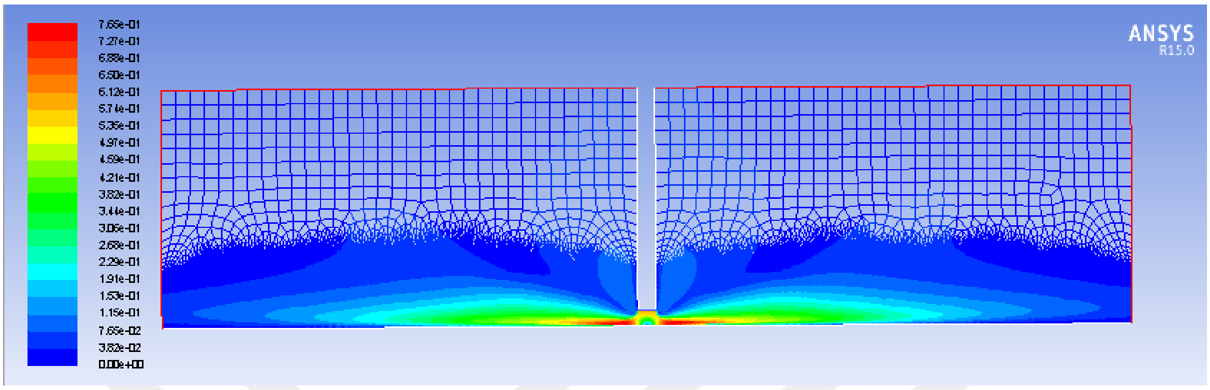


Figure 2.14 Distribution of velocity field over the flat plate

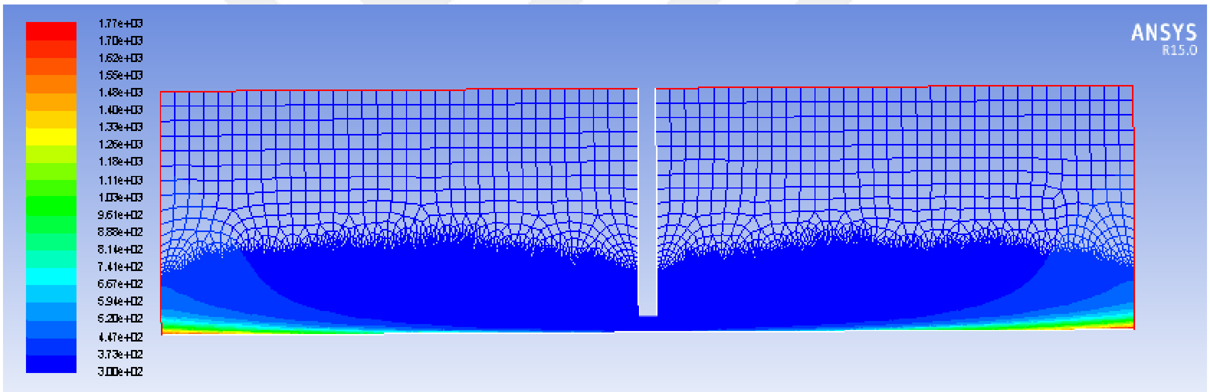


Figure 2.15 Distribution of the temperature over the flat plate

In the Figure 2.15 shows distribution of the temperature and velocity over the fate plate.

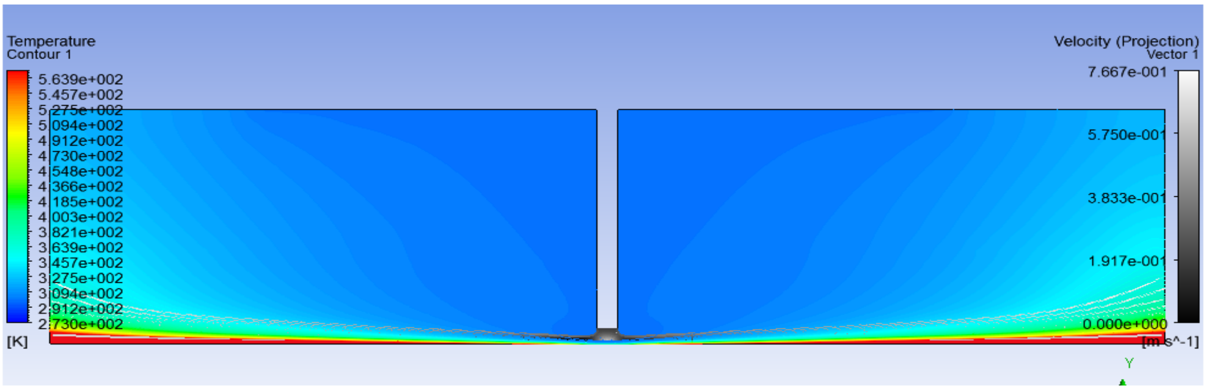


Figure 2.16 Velocity and temperature fields over the flat plate

In Figure 2.16, the flow behavior at the stagnation zone can be observed from the velocity vector plot of the stagnation zone, where the impinging point is directly beneath the jet nozzle center with highest pressure and at the same time the velocity vanishes a stagnation point, because of the accelerates. Most of the physical changes happen in the stagnation zone. The velocity gradient creates shear stress, which leads to high heat transfer according to the Reynolds analogy between shear and heat transfer.

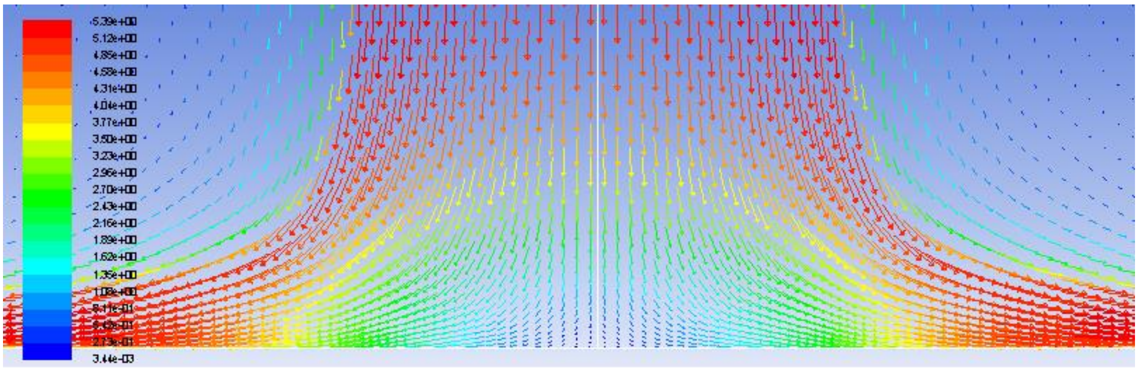


Figure 2.17 Velocity vector plot of the stagnation zone

2.4 Basic equations in Fluid Mechanics

There are three fundamental laws to this fluid "mass, conservation of momentum, and conservation of energy to obtain the continuity, momentum, and energy equations for laminar flow in boundary layers

Mass conservation equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{2.1}$$

Momentum conservation equations

$$u\frac{\partial u}{\partial x} + v\frac{\partial v}{\partial y} = \frac{\partial}{\partial x}\left(\upsilon\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(\upsilon\frac{\partial u}{\partial y}\right) - \frac{1}{\rho}\frac{\partial P}{\partial x} \tag{2.2}$$

Energy conservation equation

When the viscous shear stresses are not negligible, their effect is accounted for by expressing the energy equation as:

$$\rho\, C_p \left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}\right) = k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) + \mu\Phi \qquad (2.3)$$

Where the viscous dissipation function is obtained:

$$\Phi = 2\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2\right] + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2 \qquad (2.4)$$

Where u and v are velocity components in x and y-directions respectively, T is temperature, p is pressure. ρ, υ and ϕ are kinematic viscosity and viscous dissipation function of the fluid respective.

The boundary conditions are:

- At the jet exit

  $v = v_{in}; \quad u = 0$

- At the solid wall

  $u = v = 0$

Systematic methodology is function to transform the equations that describe the flow to a set of algebraic equations, thereafter, solving these equations numerically methods. Since there is special nature of frequency solution, also using algebraic equations, they are Outsourcing some software to gain time and to avoid the mistakes of mathematical. These programs can be divided into two majors:

- Programs are designed and written various programming languages, called In-house codes name. Examples of programming languages, Matlab, C ++, FORTRAN, et.
  - Ready programs, known as Commercial packages, which is also designed Programming languages, but different from the first category to be used here is dealing with windows to enter input, and orders to output the results, while the user is the first category (Who designs) writes (enter input system, a systematic solution, the system output the results) and then they have been starting to development these the

programs after appearance of the commercial packages , for example for these the programs :

- Fidap
- Polyflow
- Phoenix
- Star CD
- Ansys/CFX
- Flow 3D
- ESI/CFDRC
- SCRYB
- Femlab
- Fluent

The company produced the Ansys CFX contains many programs and packages to its groups, where it is Including the Polyflow and Fluent and other packages.

3. RESULTS AND DISCUSSIONS

In the present study, simulations of two phases (gas, and solid) using FLUENT code are carried out to investigate the flow field and heat transfer characteristics of a slot air jet impingement. The objective is to determine the distribution of the local Nusselt numbers on the surface of the target surface for different Re, jet to plate distance, and the surface configuration (flat surfaces).

In this chapter, the study is focused on an air jet impinging on the flat Surface. This investigation comprises a study of the flow field and heat transfer. This study presents the numerical results of a laminar flow for four cases as shown in Tables 3.2. The four cases are;

- Effect of Reynolds number on the heat transfer coefficient
- Effect of Reynolds number on Nussle number
- Effect of nozzle to plate spacing on the heat transfer coefficient
- Effect of nozzle to plate spacing on Nussle number

The jet Reynolds number, based on H/D varied from 500 to 2000. The inlet air jet temperature is 300 k and the heat flux = 3000 W/m2. The flow is assumed to be two dimensional, steady, and laminar. The fluid density, viscosity and the thermal conductivity are assumed to be constant.

In this chapter, the problem geometry is show in the Figure 3.1 for a flat at two-dimensional jet with velocity Vj issues from a nozzle on the flat surfaces. The two sides of the geometry represent pressure outlet at atmospheric pressure. The boundary layer technique has been employed on the velocity inlet region, and on top of the solid face.

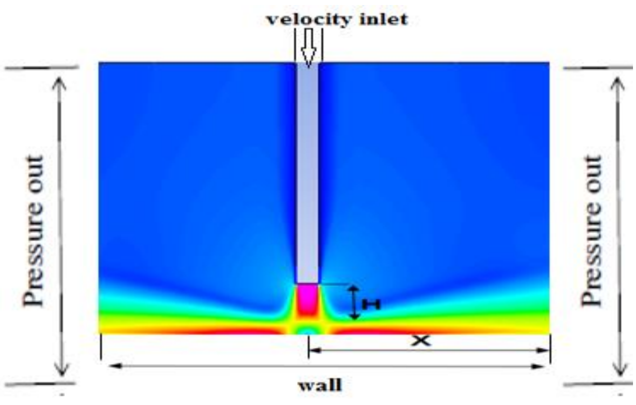


Figure 3.1 The geometry of the studied impinging jet system

3.1 No-slip condition

In the Figure 3.2, Flow field of air jet over surface plate and at the stagnation point is shown. It observed that when a fluid is forced to flow over a solid surface, the fluid motion comes to a complete stop at the surface. When a fluid is in contact with a solid surface, there can be no relative motion between the fluid in contact with the solid surface and the surface itself, in this case assumes a zero velocity this phenomenon is known as the no-slip condition, result of the no-slip condition it's mean all velocity profiles must have zero values at all the points of contact between a fluid and surface.

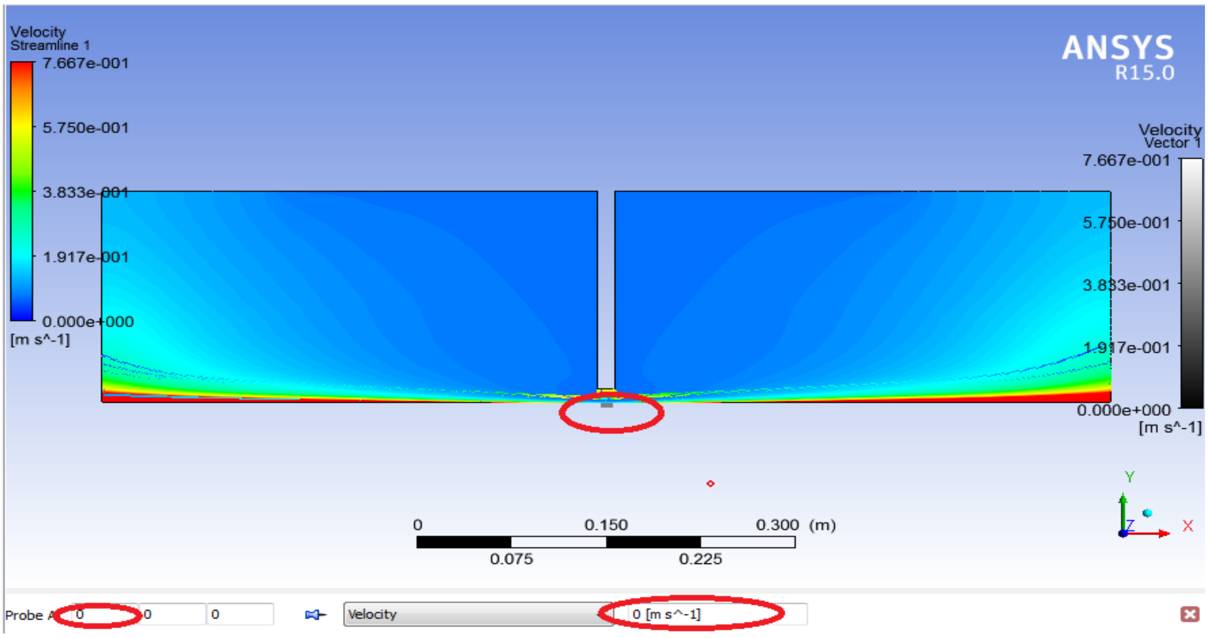


Figure 3.2 Flow field of air jet over heated surface and at the impingement region (stagnation point)

3.2 Fluid over a flat plate

The current research is study the parallel flow of a fluid over a flat plate of length x in the flow direction, as shown in Figure 3.3. The x-coordinate is measured along the plate surface from the leading edge in the direction of the flow. The fluid approaches the plate in the x-direction with uniform upstream velocity and temperature. The flow in the velocity boundary layer starts out as laminar, but if the plate is sufficiently long the flow will become turbulent at a distance x_{cr} from the leading edge where the Reynolds number reaches its critical value for transition.

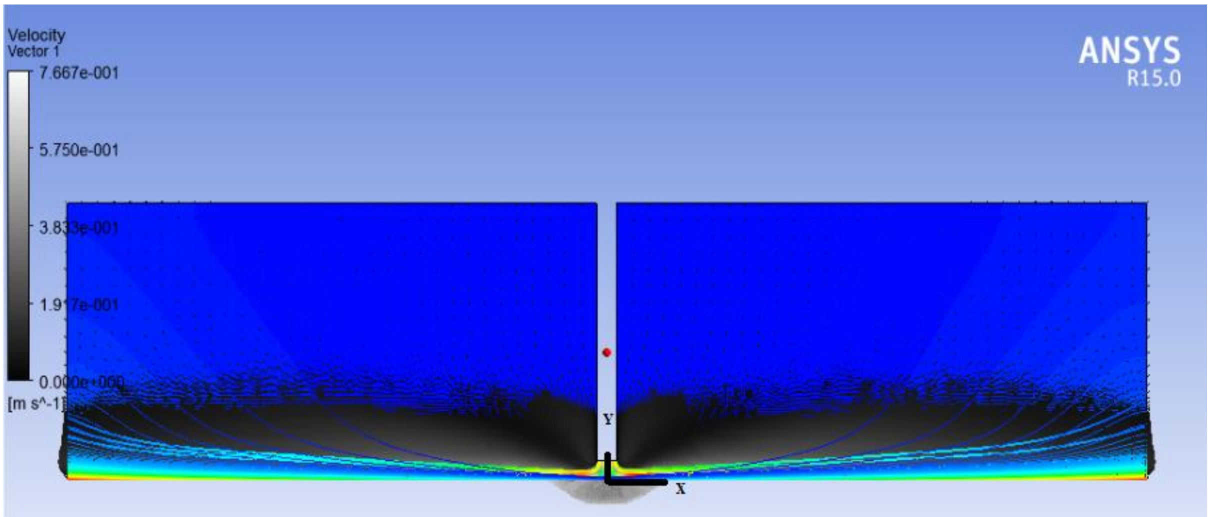


Figure 3.3 Parallel flow of a fluid over the flat plate

3.3 Flow Field Analysis

In the following two cases are presented to study the flow field:

3.3.1 Effect of Reynolds Number Re (case study 1)

Numerical simulations have been conducted with a slot jet falling vertically on a heated horizontal surface to study the effect of Reynolds number. A non-dimensional nozzle to plate spacing of H/D = 1, 1.5,2, and 3 are studied when Re is between 500 and 2000. The inlet air jet temperature was 27 ºC, and the nozzle diameter is 0.013m. When the air jet impinges on a flat surface, three different regions are identified as shown in Figure 3.4.

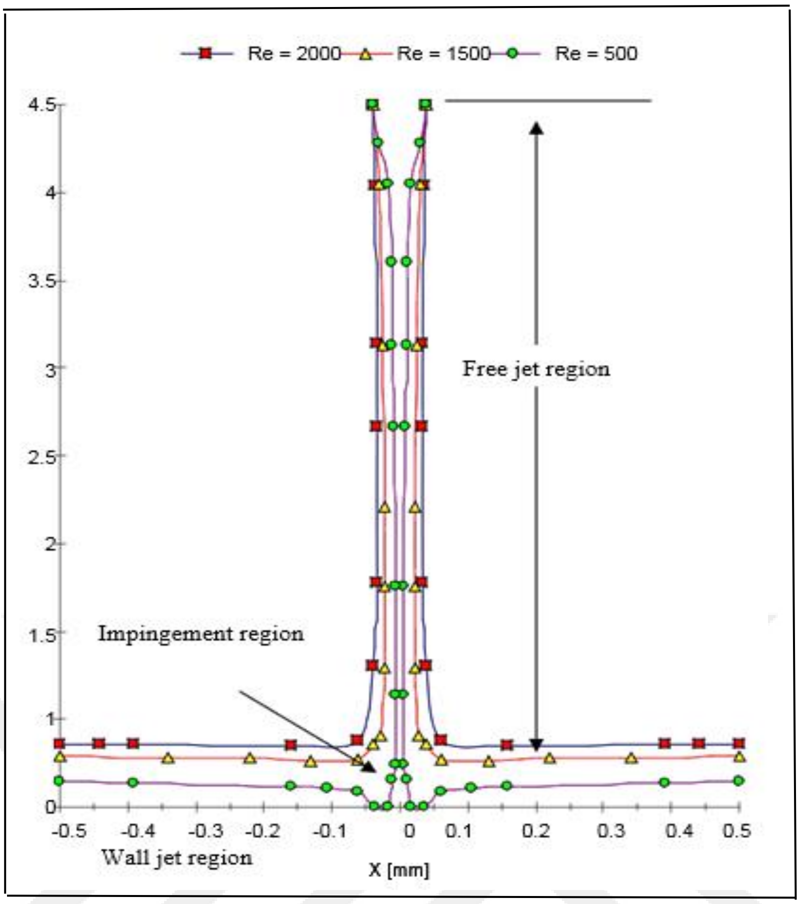


Figure 3.4 The development of the flow field from a nozzle to the plate for a range of Reynolds numbers

The first zone is the free jet region where the flow accelerates. The second is the impingement region where the interaction between the jet and the surface produces a strong deceleration of the flow, and then the airflows in a direction parallel to the surface called the wall jet region.

3.3.1.1 Reynolds Number

Reynolds discovered that the flow regime depends mainly on the ratio of the inertia forces to viscous forces in the fluid. This ratio is called the Reynolds number, which is a dimensionless quantity, and is expressed for external flow.

$$\mathrm{Re} = \frac{\text{inertia forces}}{\text{Viscous forces}} = \frac{VL}{v} = \frac{\rho VL}{\mu} \tag{3.1}$$

Where ρ is the air density, V is the inlet velocity of the jet of air with values of 0.58, 1.15, 1.37, and 2.31 m/s corresponding to Re5nolds numbers of 2000, 1500, 1000, and 500, respectively. *D*, the diameter of the nozzle, is equal to 0,013 m, and μ is dynamic viscosity for air is equal to $1.98\text{x}10^{-5}$ kg/m., whereas Reynolds number is over a flat plate is calculated $u_\infty\, \rho\, x/\mu$.

Where x is the distance from the leading edge of the plate at laminar u_∞ is free-stream velocity for a flat plate.

It can be seen from Figures (3.5,3.6,3.7, and 3.8) that near the stagnation region, the x-velocity is high in x direction, while the temperature at stagnation point so low but will increase when distance from stagnation point increase. An analysis by Schlichting (1968) with increasing distance from the jet nozzle clearly depict the stagnation point and the absolute velocity variation along the plate. The flow accelerates from the stagnation point along the plate but soon after this, the velocity decays. Because of the viscosity where the viscosity of a fluid is a measure of its resistance to flow, and it is a strong function of temperature. The velocity profile along the x-axis is different because of the effect of the wall friction, where the wall friction coefficient is calculated from the following equations:

$$C_f = \frac{\tau_w}{0.5\,\rho\, V^2} \qquad (3.2)$$

Where ρ is density and V is jet inlet velocity, and τ_w is wall shear stress from an inspection of the pressure.

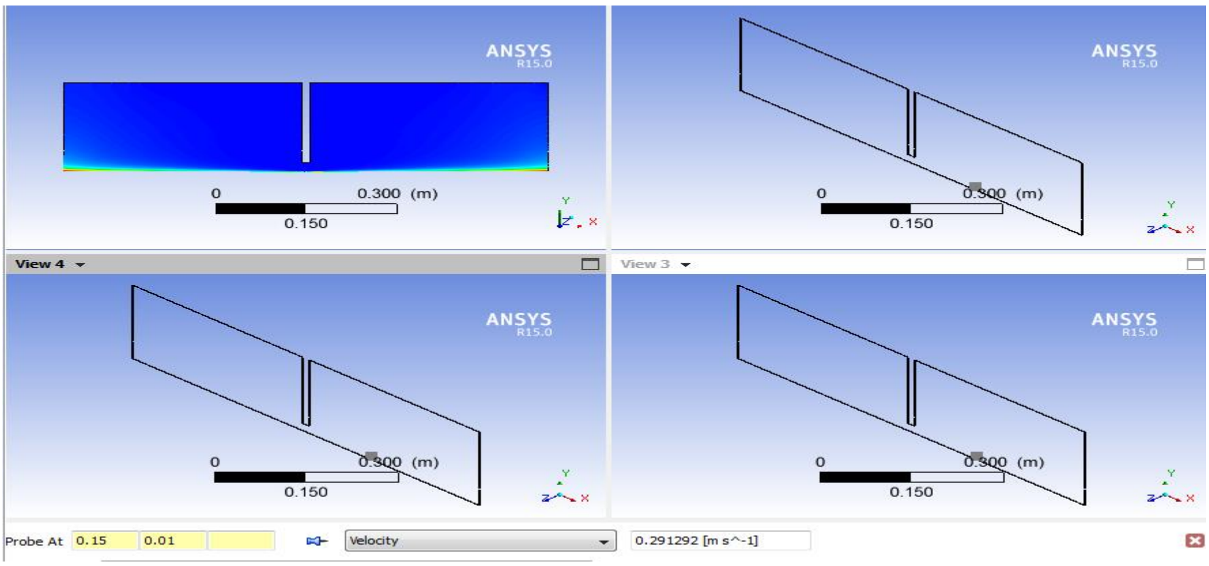


Figure 3.5. Velocity in the x-direction at 0.15 m from the wall jet region

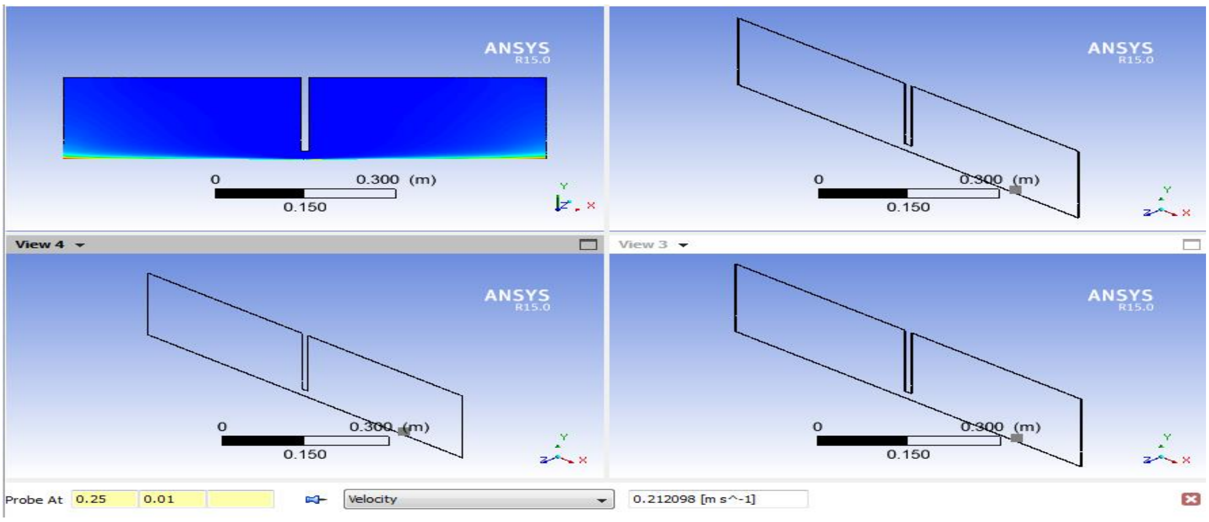


Figure 3.6. Velocity in the x-direction at 0.25 m from the wall jet region

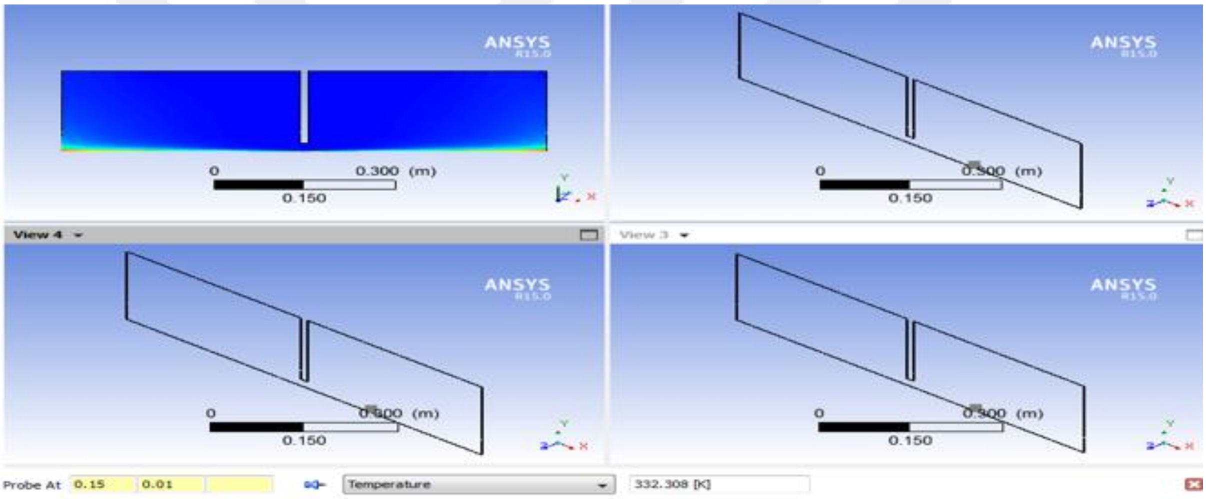


Figure 3.7. Temperature in the x-direction at 0.15 m from the wall jet region

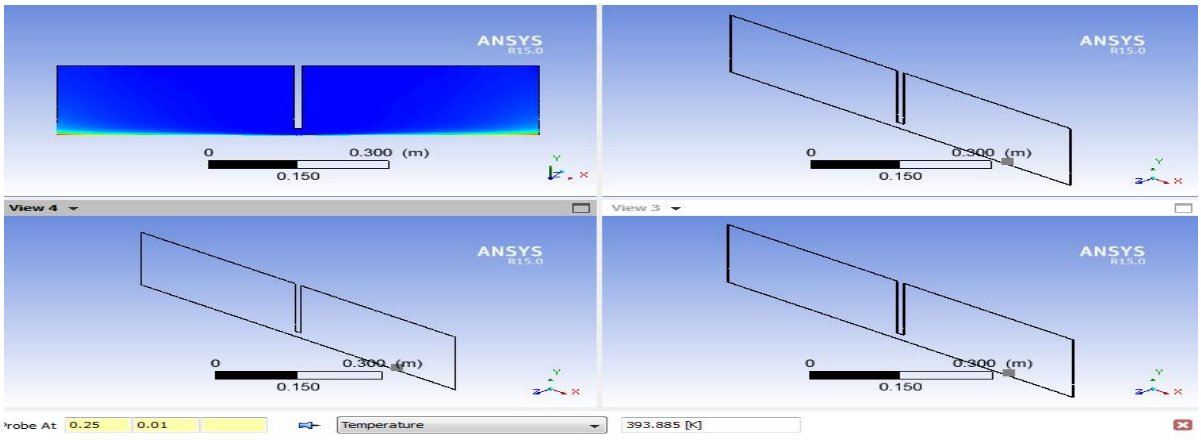


Figure 3.8 Temperature in the x-direction at 0.25 m the wall jet region

3.3.1. Effect of Reynolds Number Re on Nusselt number Nu

In Figure 3.9, the variation is presented in the Nusselt number at the impinging wall region at Reynolds numbers of 500, and 2000 and H/D of 1 and 3. It is observed that with the increase in Re the Nusselt increases ,where noticed at Re_j = 500 and different H/D =1, 1.5 ,2 and 3 the Nussle number increases from 13.47 to 14.74 (increasing by 9.5 %) ,while at Re = 2000 noticed the value of Nusselt number is the highest 31.36 at H/D=1 but values of Nusselt number starting to decrease to reach Nu = 29.77 at H/D = 3 that is mean the Nusselt number increases with decreasing H/D the highest Nu is for H/D = 1 and it decreases with increasing H/D. The reason is a friction between the particles of fluid layers, after that this fluid layer then slows down the molecules of the next layer while the local Nusselt number in impinging jets show a strong dependence on various parameters like Reynolds number and H/D.

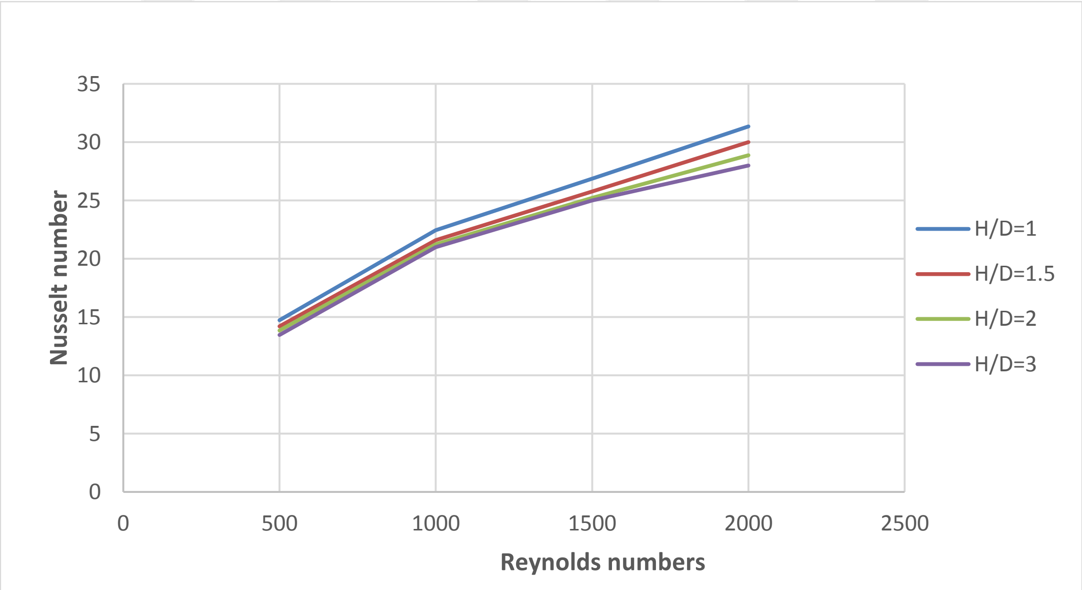


Figure 3.9 Nusselt number at several Reynolds numbers for different nozzle-plate spacing H/D

3.3.1.3 Effect of Reynolds Number (Re) on Heat Transfer (h)

In the Figure 3.10,there is clear relation between the non-dimensional Reynolds numbers Re = 500, 1000, 1500 and 2000 with heat transfer coefficient shows the relation non dimensional Reynolds Numbers Re = 500, 1000, 1500 and 2000 with heat transfer coefficient , where with use the ansys programs could find the relation between Reynolds number and heat transfer coefficient (especially at the time of operating ansys programs we could find the relation between Reynolds Number and heat transfer coefficient) ,this result inducts that the Re value increase is subject to the increase of the heat transfer coefficient, and it has been steady effect horizontal location on the heat transfer coefficient , its noticed when increase the distance from stagnation point the heat transfer coefficient decrease for example at Re = 2000 and x = 0.15 m , H/D = 1. It can be seen that the heat transfer coefficient is high h= 5.58 w/m2, while at Re = 500 at same x= 0.15m, and H/D = 1 , its observed the drop in the value of the heat transfer coefficient h=3.18 w/m2 ,this the result is indicate to the highest heat transfer coefficient is observed at the highest Reynolds number ,while when increase the location horizontal x = 0.25 m ,its observed the at Re = 2000, h = 4.39 w/m2. It can be seen that the heat transfer coefficient is decrease. These results indicate that the heat transfer coefficient decreases monotonically with increasing distance horizontal. These results indicate that the stagnation point heat transfer is a stronger function of Reynolds number than that within the wall jet region.

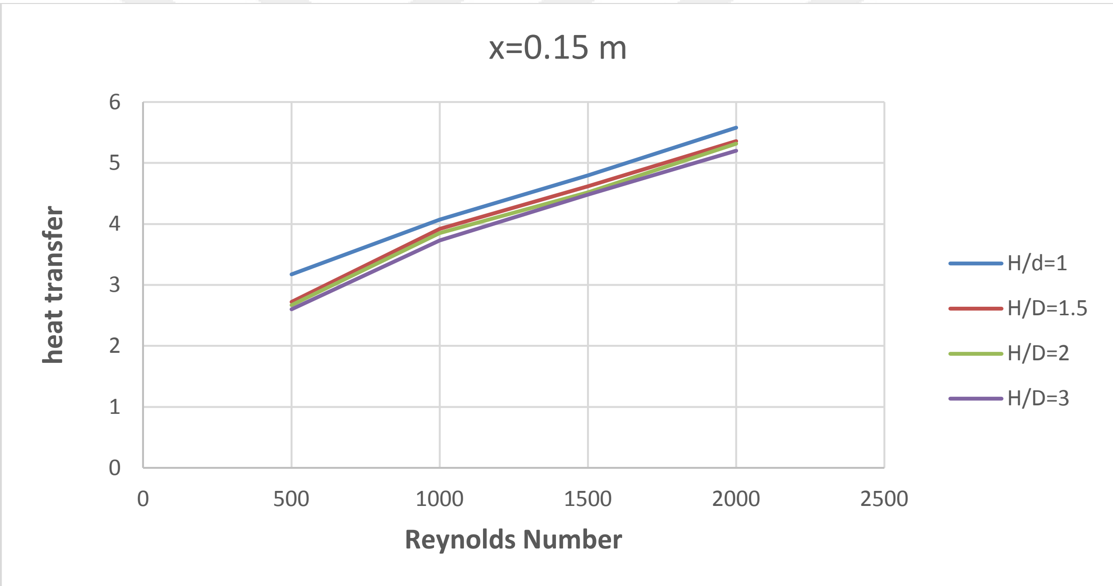


Figure 3.10 The heat transfer coefficient at different Reynolds numbers for different jet nozzle-plate distances (At horizinal distance from the stagnation point x =0,15 m).

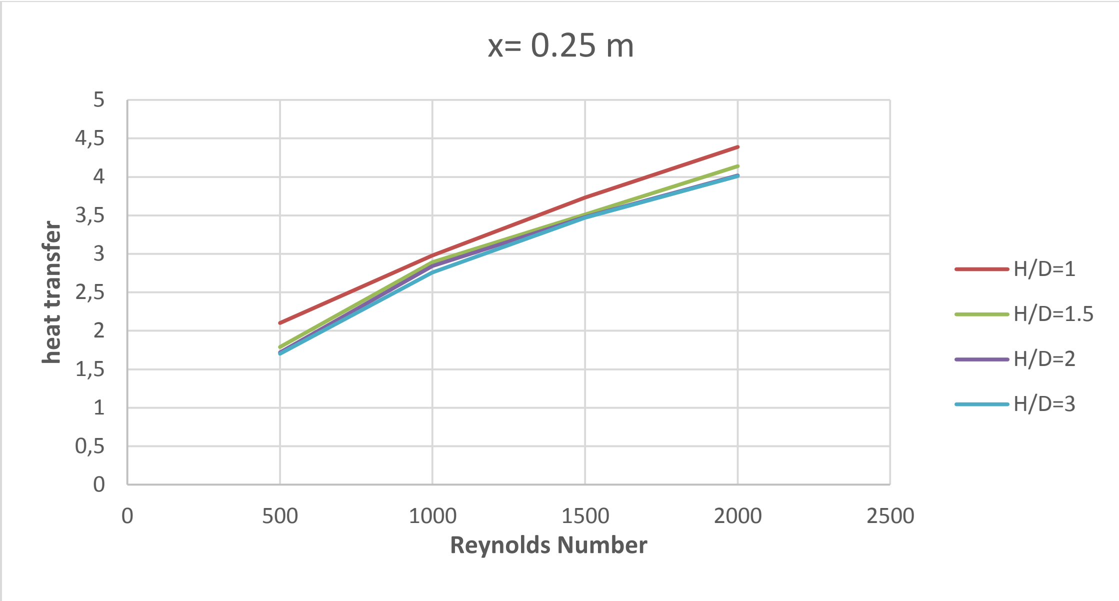


Figure 3.11 The heat transfer coefficient at different Reynolds numbers for different jet nozzle-plate distances (At horizinal distance from the stagnation point x =0,25 m)

This result suggests that the stagnation point heat transfer is a stronger function of Reynolds number than that within the wall jet region. The reason is because the flow velocity in the wall jet region decreases.

3.3.2 Effect of Nozzle to Plate Spacing (H/D, case study 2)

In Figure 3.12 is shown the jet impingement and the wall regions for non-dimensional nozzle to plate spacing of H/D =1, 1.5, 2 and 3 .and jet of diameter is 13 mm.

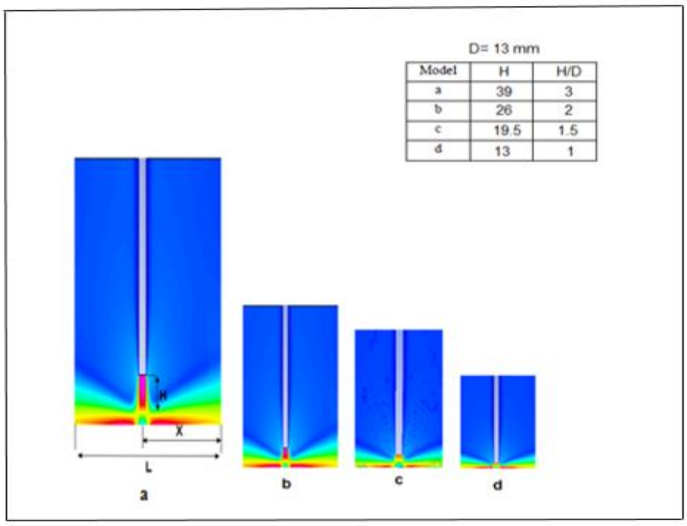


Figure 3.12 Geometry of impinging jet at different nozzle-plate spacing

3.3.2.1 Effect of Nozzle to Plate Spacing (H/D) on Nusselt Number (Nu)

The Nusselt number is a dimensionless heat transfer coefficient. The heat transfer coefficient is a relationship between the heat flux and the temperature difference between the target surface and the air. The heat transfer coefficient has complex dependence on many variables, such as the fluid properties and the flow velocity. Where for laminar flow, Incropera and DeWitt (1990) used the following expression for the local Nusselt number, Nu_x

$$Nu_x = 0.332\ Re_x^{-1/2}\ pr^{1/3} \qquad (3.3)$$

In Figure 3.13, the variation of the Nusselt number Nu is shown at Re = 500 .1000.1500, and 2000 with distance between the nozzle exit and impingement H/D = 1, 1.5, 2, and 3, respectively. As mentioned previously the maximum heat transfer is observed at the high Re, and heat transfer decreases in the horizontal direction for all nozzles to plate spacing values. The highest Nu is for H/D =1 and it decreases with increasing H/D.

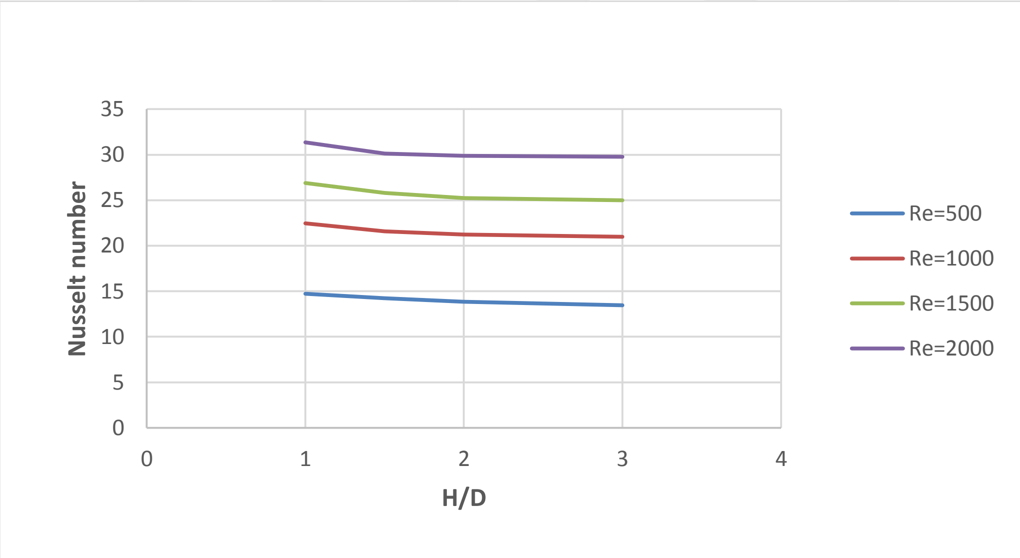


Figure 3.13 Nussle number for different jet nozzle-plate spacing at different Reynolds Number.

It can be seen that the highest Nu is for H/D = 1 at Re = 2000 and it decreases with increasing H/D. For example, at a Reynolds number of 2000, and H/D = 2 the Nusselt number Nu is 29.77, that is mean the Nusselt number increase when increase value rise as much as raise value of the Reynolds number, on other hand it is exposed to fall as soon as there is increase of H/D.

The maximum heat transfer is observed at the highest Reynolds number and the lowest nozzle to plate spacing the highest Nu is for Re = 2000 and it decreases with decreasing Reynolds number. As stated Nusselt, number is maximum at the lowest nozzle to plate spacing, because the impinging cooling area at a lower nozzle to plate spacing is larger than in higher nozzle to plate spacing, as shown in Figures 3.14 and 3.15.

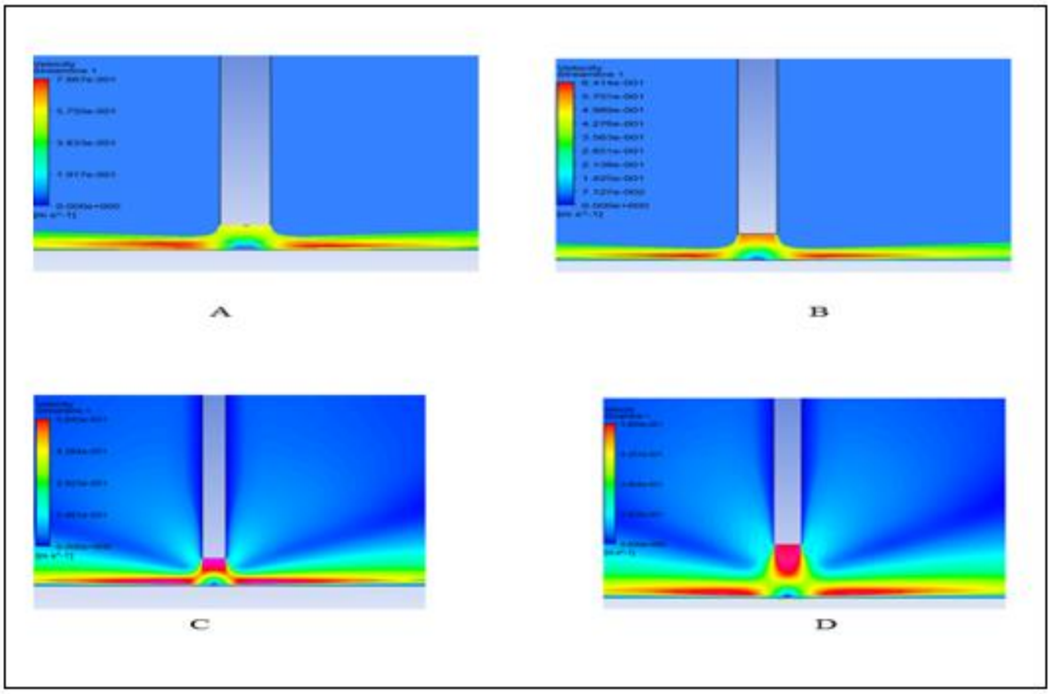


Figure 3.14 Effect of different nozzle-plate spacing on the stagnation cooling area at 2000 Reynolds number; spacing of a) H/D=1, b) H/D= 1.5, c) H/D= 2 and d) H/D= 3

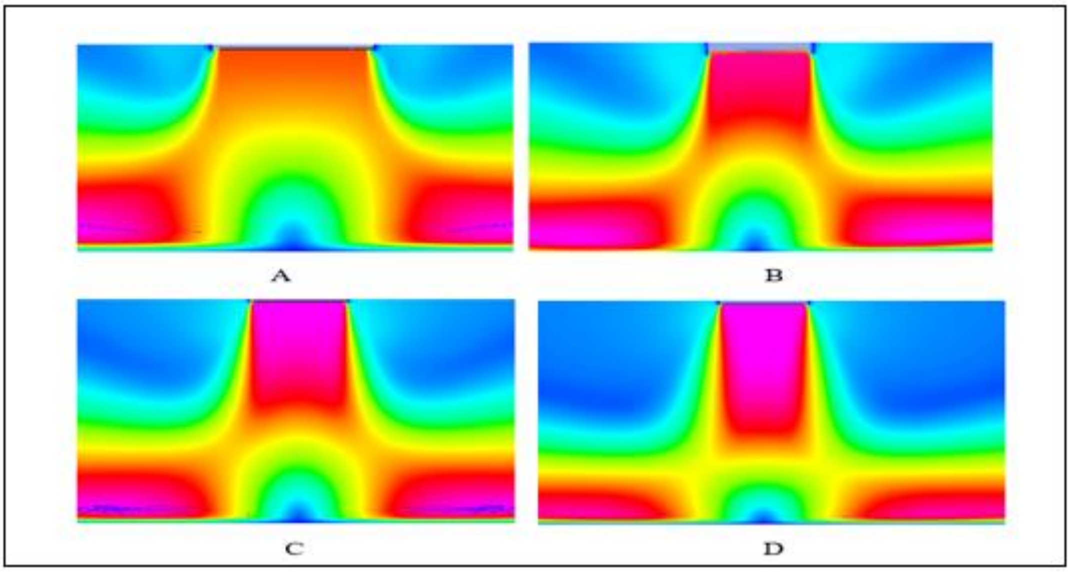


Figure 3.15 Zooming in on the stagnation cooling area at 2000 Reynolds number and different nozzle-plate spacing; for a) H/D= 1, b) H/D= 1.5, c) H/D= 2 and d) H/D= 3.

3.3.2.2 Effect of Nozzle to Plate Spacing (H/D) on heat transfer coefficient (h)

The numerical analysis of heat transfer is performed in order to study the influenceof various Reynolds numbers. Heat transfer (h) is determined from the local heat flux (q"), the local surface temperature (Ts) and the inlet jet air temperature (Ti):

$$h= \frac{q''}{(Ts-Ti)} \quad (3.4)$$

The local Nusselt number at a location x for laminar flow over a flat plate was determined by

$$Nu_x= \frac{h\ x}{k} = 0.332\ Re_x^{0.5}\ pr^{1/3}\ ,\ Pr > 0.60 \quad (3.5)$$

Where h is the local heat transfer coefficient, where x is the distance from the leading edge and edge of the plate k is the thermal conductivity of the air.

In the Figures 3.16, 3.17 show variations heat transfer cofficent with different nozzle to plate distance H= 13, 19.5, 26 and 39. mm correspoinding to H/D = 1,1.5,2 and 3 respectively.As mentioned previously, the the maximum heat transfer is observed at the stagnation point, and heat transfer decreases in the horizontal direction for all nozzles to plate spacing values.

In Figure 3.16, the variation of heat transfer cofficent with different nozzle to plate spacing H/D ,where the heat transfer cofficent has high value h= 5.581 w/ m^2 when Reynolsd Number Re =2000 and distance from stagination point x = 0.15 but when increase the value of H/D will noticed the value of the heat transfer cofficent decrease , for example when H/D= 3 will noticed h= 5.20 w/m^2 .

The reason for this difference with H/D is that the acceleration of the fluid through the small distance between the wall and nozzle exit is more active than with a longer distance.

These results indicate that the heat transfer coefficient decreases monotonically with increasing distance from the jet nozzle.

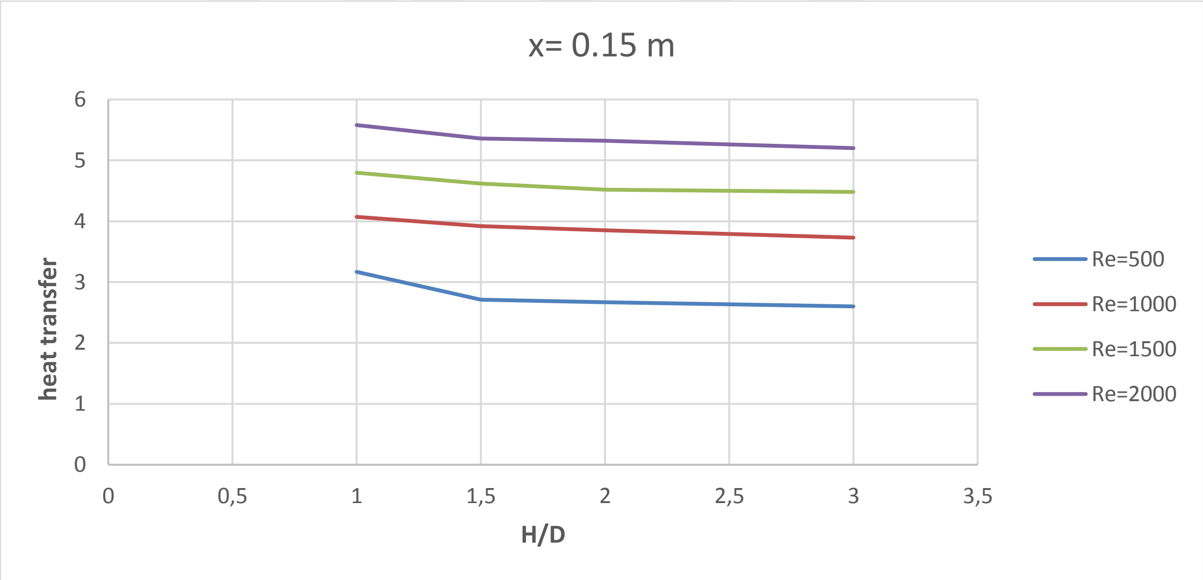


Figure 3.16 The heat transfer coefficient at several jet nozzle-plate distances foir different Reynolds numbers (At horizinal distance from the stagnation point x =0.15 m).

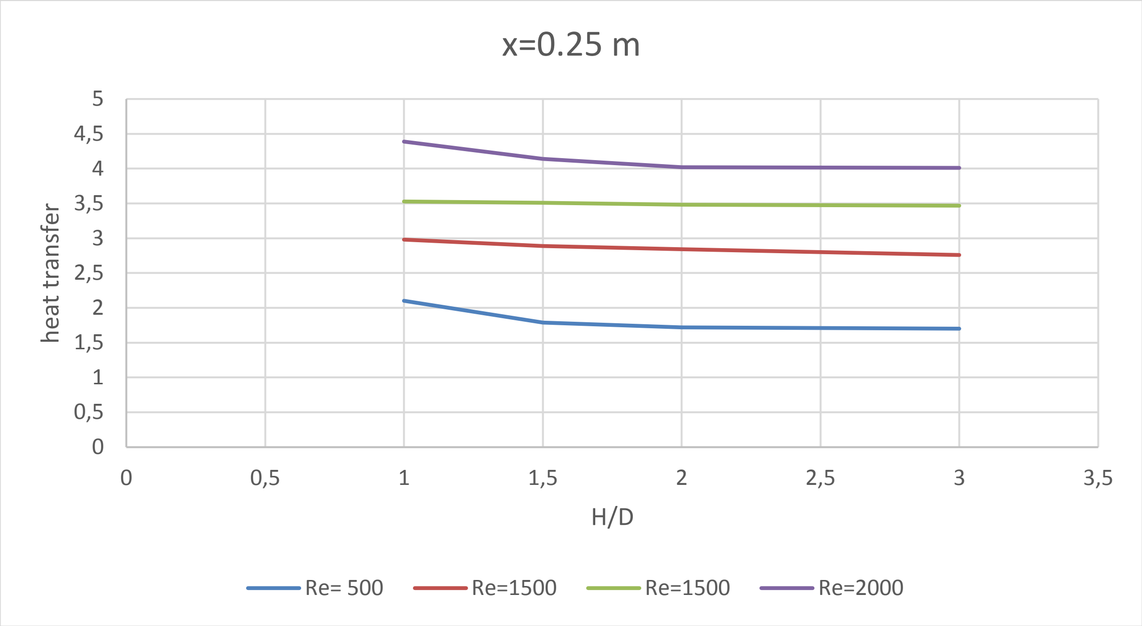
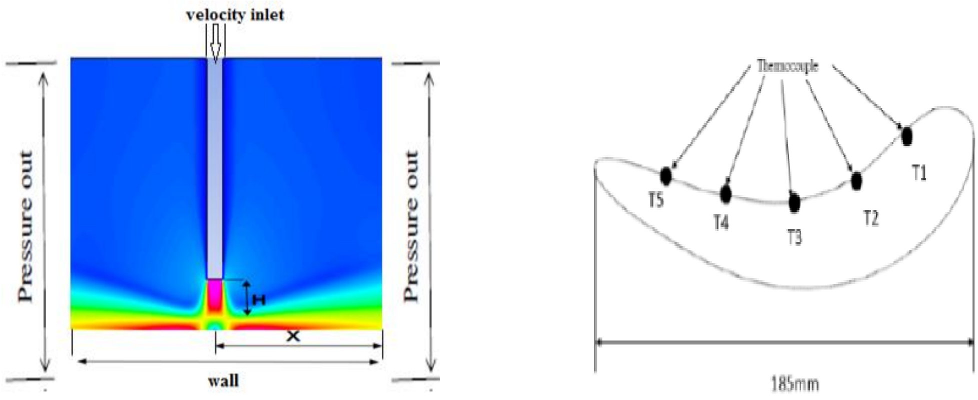


Figure 3.17 The heat transfer coefficient at several jet nozzle-plate distances for different Reynolds numbers (At horizinal distance from the stagnation potin x =0.25 m)

3.4 Research work based upon comparing with Previous Experimental Results

The comparing study is vital method used in this research since the model performed through strong investigation show the necessary to compare our study to expermintal work. Attibuted to Mohammed Mosa. [4]. In this case the impinging plate has been implemented by numerically and it has been compatered the results with previous experimental results. It can be seen that the present model is in agreement with the experimental results, especially in the valus of the heat transfer coffricent and Nusselt number. In this research, numerically study was concentrated on determination effectiveness of jet impingement cooling system which has different on a flat plate by using two variable parameters which are Reynolds number and nozzle exit to surface spacing, H/D .On other hand the comparing experimental has used to determine effectiveness of jet impingement cooling system on a turbine blade pressure side by using two variable parameters which are mass flow rate that refer to Reynolds number and nozzle exit to surface spacing, H/D as seen in Figure 2.1 , Where it notied that the highest Nusselt number is produced when Reynolds number = 2637, followed by 1970, 1322 and lastly 646, when Reynolds number increase, the Nusselt number will increase as well, As surface spacing increases H/D, the Nusselt number will decrease and the heat loss to the surrounding will decrease. To conclude the comparing between to expermintal performances shows the correct results appeared between two studies with resemble conclusion. This research can also help future engineers to get on high heat transfer over flat plate .



a b

Figure 3.18 The comparing experimental between (a) jet impingement cooling system on flat plate and (b) turbine blade .

Table 3.1 The parameters used in this study Re (500)

V_{jet}	Re_{jet}	$Nu_{0.15}$	$Nu_{0.25}$	H/D	$h_{0.15}$	$h_{0.25}$
0.58	500	14.74	14.61	1	3.18	2.10
0.58	500	14.22	13.46	1.5	2.72	1.79
0.58	500	13.85	12.93	2	2.67	1.72
0.58	500	13.47	12.13	3	2.60	1.70

Table 3.2 The parameters used in this study Re (1000)

V_{jet}	Re_{jet}	$Nu_{0.15}$	$Nu_{0.25}$	H/D	$h_{0.15}$	$h_{0.25}$
1.15	1000	22.46	21.12	1	4.07	2.98
1.15	1000	21.59	20.18	1.5	3.92	2.89
1.15	1000	21.23	20.15	2	3.85	2.84
1.15	1000	21.00	20.09	3	3.73	2.76

Table 3.3 The parameters used in this study Re (1500)

V_{jet}	Re_{jet}	$Nu_{0.15}$	$Nu_{0.25}$	H/D	$h_{0.15}$	$h_{0.25}$
1.73	1500	26.88	25.51	1	4.79	3.53
1.73	1500	25.79	23.20	1.5	3.62	3.51
1.73	1500	25.23	23.51	2	3.52	3.48
1.73	1500	25.00	22.50	3	3.48	3.47

Table 3.4 The parameters used in this study Re (2000)

V_{jet}	Re_{jet}	$Nu_{0.15}$	$Nu_{0.25}$	H/D	$h_{0.15}$	$h_{0.25}$
2.31	2000	31.36	30.15	1	5.58	4.39
2.31	2000	30.13	29.51	1.5	5.36	4.14
2.31	2000	29.87	29.35	2	5.32	4.02
2.31	2000	29.77	29.15	3	5.20	4.01

4. CONCLUSION

In this thises, the results of a numerical investigation are presented on the effect of the Reynolds number, nozzle to plate spacing, and configuration of the heated target surface on the flow field of jet impinging on a heated surface. The velocity have been studied to evaluate the heat transfer coefficient. The results indicate that Re has a significant effect on the flow field and heat transfer. The heat transfer rate in the stagnation and wall jet regions is enhanced with an increase in Re. When the Reynolds number increases, the consequence is an increase in the velocity of the flow and an increase in the convective heat transfer which results in better surface cooling. The following are the brief conclusions of this chapter:

- Two regions of jet impingement surface are identified. These regions are x = (0.15 ,0.25 m)
- The Nusselt numbers increase with a decrease in H/D. This result may be due to flow acceleration at lower H/D. Where the heat teansfer coefficient is high at x = 0.15, whereas at x= 0.25 will be decrease, The heat transfer coefficient depend on the distance from the stagnation point and it is decreases from the stagnation point till the edge of the stagnation region.
- It is noticed that with the increase in value of the Reynolds number (Re), the heat transfer rate (h) at the stagnation point increases as well at a given nozzle to plate spacing. This result is because of the cooling stagnation area’s being larger at higher Reynolds number.

REFERENCES

[1] Wolf D H; Incropera F P; Viskanta R. 1993: "Jet Impingement Boiling", Advances in Heat Transfer

[1] Gardon, R & Akfirat, JC (1965). The role of turbulence in determining the heat transfer characteristics of impinging jets, Int. j. heat Mass Transfer, vol. 8, pp. 101-8.

[2] Incropera, FP & DeWitt, DP (1990). Fundamentals of Heat and Mass Transfer, 3rdedn, John Wiley and Sons, New York

[3] Hofmann, HM, Kind, M & Martin, H (2007). Measurements on steady state heat transfer and flow structure and new correlations for heat and mass transfer in submerged impinging jets', Int. J. Heat Mass Transfer, vol. 50, pp. 3957-65

[4] Mohamad Nor Musa (2010). Jet Impingement Cooling System on the Pressure Side of Turbine Blade, Universiti Teknologi Malaysia, 81310 UTM Skudai.

[5] T.S. O'Donovan, (2005). Fluid flow and heat transfer of an impinging air jet, PhD Thesis, University of Dublin, Dublin

[6] Mark Michael Dobbertean October 18, 2011. Steady and Transient Heat Transfer for Jet Impingement on Patterned Surfaces University of south Florida

[7] Dulfharah Nizam Ali 2009. Effect of nozzle diameter on jet impingement cooling system Master Thesis, University Malaysia Pahang.

[8] Nakod, PM, Prabhu, SV & Vedula, RP 2008. Heat transfer augmentation between impinging circular air jet and flat plate using finned surfaces and vortex generators, Experimental Thermal and Fluid Science, vol. 32, pp. 1168 -87.

[9] Farial A. Jafar August 2011. Flow fields and heat transfer of liquid falling film on horizontal cylinders , PhD Thesis, University of Victoeia,

[10] Ail Mohammed Hayder December 2014. Numerical Studyo of jet impingement cooling on a smooth curve surface, Master Thesis, University Tun Hussein Onn Malaysia.

[11] Chung, YM, Luo, KH & Sandham, NDC 2002. Numerical study of momentum and heat transfer in unsteady impinging jets, International Journal of Heat and Fluid Flow, vol. 23, pp. 592-600.

[12] Chitranjan Agarwall August 2012. Effect exit to surface spacing on the cooling of electrically heated surface with jet impingement, International Journal of Advanced Research in Electrical, ISSN 2278-8875.

[13] Viskanta, R. (1993). Heat transfer to impinging isothermal gas and flame jets. Exp. Thermal Fluid Sci. 6, 111–134.

[14] Huang, L & El-Genk, MS 1994, 'Heat transfer of an impinging jet on a flat surface', International Journal of heat and mass transfer vol. 37, no. 13, pp.1915 – 23.

[15] Narayanan, V, Seyed-Yagoobi, J & Page, RH 2004, An experimental study of fluid mechanics and heat transfer in an impinging slot jet flow, International Journal of Heat and Mass Transfer vol. 47, no. 1827–1845.

[16] Han, J.-C., Dutta, S., and Ekkad, S. (2000). ‘‘Gas Turbine Heat Transfer and Cooling Technology’’. Taylor & Francis, New York.

[17] Chougule N.K., Parishwad G.V (2011), CFD Analysis of Multi-jet Air Impingement on Flat Plate Proceedings of the World Congress on Engineering 2011 Vol III WCE 2011, July 6 - 8, 2011, London, U.K.

[18] Jambunathan, K, Lai, E, Moss, MA & Button, BL 1992, 'A review of heat transfer data for single circular impingement, ' Internat.J. Heat Fluid Flow, vol. 13, no. 2, pp. 106-15.

[19] Siba, EA, Ganesa-Pillai, M, Harris, KT & Haji-Sheikh, A 2003, 'Heat transfer in a high turbulence air jet impinging over a flat circular disk,' J. Heat Transfer, ASME, vol. 125, pp.

[20] Lee, DH, Song, J & Jo, MC 2004, 'The effect of nozzle diameter on impinging jet heat transfer and fluid flow', J. Heat Transfer, vol. 126, pp. 554-7.

[21] K Marzec and A Kucaba-Pietal (2014), Heat transfer characteristic of an impingement cooling system with different nozzle geometry Journal of Physics: Conference Series 530 (2014) 012038.

[22] De Witt, D. P. (1990). Fundamentals of Heat and Mass Transfer. New York: Wiley.

[23] Taylor, R.A. (2012). "Socioeconomic impacts of heat transfer research". International Communications in heat transfer 39 (10)1467-1473 doi110.1016/j.icheatmastransfer.2012.09.

[24] Clancy, L.J. (1975), Aerodynamics, Pitman Publishing Limited, London.

[25] Lam, C. G. K. & Bremhorst, K. A., "Modified form of the k-εε_ model for predicting wall turbulence," Trans. ASME Journal of Fluid Engineering, Vol. 103, pp. 456-460, 1981.

[26] http://www.academi.edu /496980/part.B.Heat Transfer Principlas in Electronics Cooling 54 MPE 635.Electronics Cooling 8 Forced Convection Heat Transfer .pdf.

[27] D. Cooper, D. C Jackson, B. E. Launder, and G. X. Liao. Impinging jet studies for turbulence model assessment - i. ow-_eld experiments. International journal of heat and mass transfer, 36(10):2675{2684, 1993.

[28] S. Z. Shuja, B. S. Yilbas, and M. Rashid (2003), Confined swirling jet impingement onto an adiabatic wall. International journal of heat and mass transfer, 46:2947{2955, 2003.

[29] Y. M. Chung, K. H. Luo, and N. D. Sandham, Numerical study of momentum and heat transfer in unsteady impinging jets. International journal of heat and uid ow, 2002.

[30] T. J Craft, L. J. W. Graham, and B. E. Launder. Impinging jet studies for turbulence model assessment - ii. an examination of the performance of four turbulence models. International journal of heat and mass transfer, 36(10):2685{2697, 1993.

[31] Sahbaz Mehmet, Kentli Aykut, Koten Hasan "Thermal analysis and optimization of high power led armature", 2017.

[32] Asem Nabadavis and Dipti Prasad Mishra. Numerical Investigation of Jet Impingement Heat Transfer on a Flat plate. Applied Science Innovations Pvt. Ltd., India. ISSN 0974 – 0546. Carbon – Sci. Tech. 8/4(2016).

[33] K. Siva Satya Mohan1 and Dr. S. K. Bhatti. Analysis of Comparison of Fluid Jet Impingement Heat Transfer on Flat Plate Using CFD. International Journal of Advanced Science and Technology Vol.94 (2016), pp.67-74.

APPENDIX

AIR PROPERTIES – CHART

Temperature (K)	Specific Heat - c_p - (kJ/kgK)	- c_v - (kJ/kgK)	Ratio of Specific Heats - k - (c_p/c_v)	Dynamic Viscosity - μ - (10^{-5} kg/m s)	Thermal Conductivity (10^{-5} kW/m K)	Prandtl Number	Kinematic Viscosity[1)] - ν - (10^{-5} m^2/s)	Density[1)] - ρ - (kg/m^3)	Diffusivity - α - (10^{-6} m^2/s)
175	1.0023	0.7152	1.401	1.182	1.593	0.744	0.586	2.017	
200	1.0025	0.7154	1.401	1.329	1.809	0.736	0.753	1.765	10.17
225	1.0027	0.7156	1.401	1.467	2.020	0.728	0.935	1.569	
250	1.0031	0.7160	1.401	1.599	2.227	0.720	1.132	1.412	15.67
275	1.0038	0.7167	1.401	1.725	2.428	0.713	1.343	1.284	
300	1.0049	0.7178	1.400	1.846	2.624	0.707	1.568	1.177	22.07
325	1.0063	0.7192	1.400	1.962	2.816	0.701	1.807	1.086	
350	1.0082	0.7211	1.398	2.075	3.003	0.697	2.056	1.009	29.18
375	1.0106	0.7235	1.397	2.181	3.186	0.692	2.317	0.9413	
400	1.0135	0.7264	1.395	2.286	3.365	0.688	2.591	0.8824	36.94
450	1.0206	0.7335	1.391	2.485	3.710	0.684	3.168	0.7844	
500	1.0295	0.7424	1.387	2.670	4.041	0.680	3.782	0.7060	
550	1.0398	0.7527	1.381	2.849	4.357	0.680	4.439	0.6418	
600	1.0511	0.7640	1.376	3.017	4.661	0.680	5.128	0.5883	
650	1.0629	0.7758	1.370	3.178	4.954	0.682	5.853	0.5430	
700	1.0750	0.7879	1.364	3.332	5.236	0.684	6.607	0.5043	
750	1.0870	0.7999	1.359	3.482	5.509	0.687	7.399	0.4706	
800	1.0987	0.8116	1.354	3.624	5.774	0.690	8.214	0.4412	
850	1.1101	0.8230	1.349	3.763	6.030	0.693	9.061	0.4153	
900	1.1209	0.8338	1.344	3.897	6.276	0.696	9.936	0.3922	
950	1.1313	0.8442	1.340	4.026	6.520	0.699	10.83	0.3716	
1000	1.1411	0.8540	1.336	4.153	6.754	0.702	11.76	0.3530	
1050	1.1502	0.8631	1.333	4.276	6.985	0.704	12.72	0.3362	
1100	1.1589	0.8718	1.329	4.396	7.209	0.707	13.70	0.3209	
1150	1.1670	0.8799	1.326	4.511	7.427	0.709	14.70	0.3069	
1200	1.1746	0.8875	1.323	4.626	7.640	0.711	15.73	0.2941	
1250	1.1817	0.8946	1.321	4.736	7.849	0.713	16.77	0.2824	
1300	1.1884	0.9013	1.319	4.846	8.054	0.715	17.85	0.2715	
1350	1.1946	0.9075	1.316	4.952	8.253	0.717	18.94	0.2615	
1400	1.2005	0.9134	1.314	5.057	8.450	0.719	20.06	0.2521	
1500	1.2112	0.9241	1.311	5.264	8.831	0.722	22.36	0.2353	
1600	1.2207	0.9336	1.308	5.457	9.199	0.724	24.74	0.2206	
1700	1.2293	0.9422	1.305	5.646	9.554	0.726	27.20	0.2076	
1800	1.2370	0.9499	1.302	5.829	9.899	0.728	29.72	0.1961	
1900	1.2440	0.9569	1.300	6.008	10.233	0.730	32.34	0.1858	

CURRICULUM VITAE

I was born in Libya - Benghazi in 1983. I completed my high school education in Libya Salah din High School. I studied mechanical engineering in Garyounis University in Benghazi and graduated in 2007. Then I started Master of Science program in mechanical engineering in Marmara University in 2013.