

T.R.
GEBZE TECHNICAL UNIVERSITY
GRADUATE SCHOOL

FAN SHAPED FILM COOLING TECHNOLOGIES
FOR GAS TURBINES

SEMİH AVCUN

A THESIS OF MASTER OF SCIENCE
DEPARTMENT OF ENERGY TECHNOLOGIES
IN APPLIED PROPULSION SYSTEM DESIGN &
ENGINEERING FOR AEROSPACE TECHNOLOGIES

ADVISOR: DR. ERİNÇ ERDEM

MAY 2024

T.R.
GEBZE TECHNICAL UNIVERSITY
GRADUATE SCHOOL

FAN SHAPED FILM COOLING TECHNOLOGIES
FOR GAS TURBINES

SEMİH AVCUN

A THESIS OF MASTER OF SCIENCE
DEPARTMENT OF ENERGY TECHNOLOGIES
IN APPLIED PROPULSION SYSTEM DESIGN &
ENGINEERING FOR AEROSPACE TECHNOLOGIES

ADVISOR: DR. ERİNÇ ERDEM

MAY 2024

T.C.
GEBZE TEKNİK ÜNİVERSİTESİ
LİSANSÜSTÜ EĞİTİM ENSTİTÜSÜ

GAZ TÜRBİNLERİ İÇİN
FAN ŞEKİLLİ FİLM SOĞUTMA TEKNOLOJİLERİ

SEMİH AVCUN

YÜKSEK LİSANS TEZİ
ENERJİ TEKNOLOJİLERİ ANABİLİM DALI
HAVACILIK VE UZAY TEKNOLOJİLERİNDE
UYGULAMALI İTKİ SİSTEMİ TASARIM
MÜHENDİSLİĞİ

DANIŞMAN: DR. ERİNÇ ERDEM

MAYIS 2024



MASTER of SCIENCE JURY APPROVAL FORM

A thesis submitted by Semih AVCUN, defended on 30/05/2024 before the jury formed with the 12/02/2024 date and 2024/13 numbered decision of the GTU Graduate Administrative Board, has been accepted as a MASTER of SCIENCE thesis in the Department of Energy Technologies, Applied Propulsion System Design & Engineering for Aerospace Technologies Program.

JURY

MEMBERS

(THESIS ADVISOR) : Dr. Erinç ERDEM

MEMBER : Dr. Tolga YASA

MEMBER : Assoc. Prof. Dr. Ali Murat SOYDAN

APPROVAL

Gebze Technical University Graduate Administrative Board
...../...../..... date and/..... numbered decision.

SIGNATURE/SEAL

ABSTRACT

Gas turbines are utilized in aviation propulsion and power generation systems. They are complex and high technology machines that provide high performance and efficiency.

In recent years, advanced technology aircraft engines have requested in the aviation industry. These engines are consist of low fuel consumption and high turbine inlet temperature level in order to become high-tech machine.

For high levels of gas turbine performance, one of the most important key technologies is the increasing turbine inlet temperature. This increasing is achieved by means of advanced material development and complex cooling system design features.

Cooling system design which has greater impact than material development is improved day by day. High-energy burnt gases coming out of the combustion chamber first pass through the turbine blades and the most advanced cooling technologies are used on these parts. Impingement cooling, internal channels with turbulence enhancing structures, film cooling are examples of some cooling technologies.

Within the scope of this thesis, film cooling is selected in terms of validation and investigation studies. As the kind of this application, fan shaped film cooling which has been frequently used in recent years, will be examined. Lateral film effectiveness that is the most important design parameter of shaped film cooling, will be investigated with numerical models with different geometric features and flow properties.

Firstly, validation models will be created with test rig conditions. Then, different models will be set up with different feeding conditions and geometric properties. In addition, effects of coolant velocity in feeding channel which causes different flow physics behavior will be researched. The flow physics inhole and dowsntream of the hole will be investigated for all cases. The discharge coefficient and recovery factor for cylindrical and fan shaped holes are examined with the same boundary conditions.

Key words: Gas Turbine, Film Cooling, Fan Shaped Film Hole

ÖZET

Havacılık itki ve güç üretim sistemlerinde gaz türbinlerinden yararlanılır. Gaz türbinleri, yüksek performans ve verim sağlayan, kompleks ve yüksek teknoloji seviyesine sahip makinelerdir.

Son yıllarda havacılık sektöründe, ileri teknoloji uçak motorları isterleri ortaya çıkmıştır. Bu motorlar, yüksek teknoloji makineleri olabilmeleri için, düşük yakıt tüketiminden ve yüksek türbin giriş sıcaklık seviyesinden oluşur.

Yüksek seviye gaz türbini performansı için, en önemli anahtar teknolojilerden biri artan türbin giriş sıcaklıklarıdır. Bu artış, ileri seviye malzeme gelişimi ve kompleks soğutma sistem tasarım yapıları yardımı ile elde edilir.

Malzeme gelişiminden daha büyük etkiye sahip soğutma sistemi tasarımları günden güne gelişmektedir. Yanma odasından çıkan yüksek enerjili yanmış gazlar ilk olarak türbin palelerine uğramaktadır ve en ileri soğutma teknolojileri bu parçalarda kullanılır. Çarpma yöntemi, türbülans arttırıcı yapıların kullanıldığı iç kanallar, film soğutma soğutma teknolojilerine örnektir.

Bu çalışma kapsamında, validasyon ve araştırma çalışmaları açısından film soğutma seçilmiştir. Bu uygulamanın bir türü olarak, son yıllarda sıklıkla kullanılmaya başlanan fan şekilli film soğutma incelenecektir. Şekilli film soğutmanın en önemli tasarım parametresi olan yanal film etkinliği, farklı geometrik özelliklere ve akış özelliklerine sahip sayısal modeller ile incelenecektir.

İlk olarak test donanımı koşullarıyla doğrulama modelleri oluşturulacaktır. Daha sonra farklı geometrik özelliklere ve besleme koşullarına sahip farklı modeller kurulacaktır. Ek olarak, farklı akış fiziği davranışına neden olan soğutma sıvısı hızının besleme kanalındaki etkileri de araştırılacaktır. Delik içi ve delik çıkışındaki akış fiziği tüm durumlar için incelenecektir. Silindirik ve fan şekilli delik için tahliye katsayısı ve iyileşme faktörü aynı sınır koşulları ile birlikte incelenecektir.

Anahtar Kelimeler: Gaz Türbini, Film Soğutma, Fan Şekilli Film Deliği

ACKNOWLEDGEMENTS

Firstly, I would like to sincerely thank my Advisor, Dr. Erinc ERDEM for his respectable guidance and valuable support.

I am greatly appreciated to attend to master program Applied Propulsion System Design Engineering in Aerospace Technologies, in Gebze Technical University Turkey. Fan shaped film cooling technologies for gas turbines have detailly investigated with respect to topics of my master thesis.

I would like to thank my colleagues from TEI for their help and understanding.

Finally, my special thanks go to my family who always have patience and good feelings for me.

TABLE of CONTENTS

	<u>Page</u>
ABSTRACT	v
ÖZET	vi
ACKNOWLEDGEMENTS	vii
TABLE of CONTENTS	viii
LIST of SYMBOLS and ABBREVIATIONS	ix
LIST of FIGURES	xi
LIST of TABLES	xiii
1. INTRODUCTION	1
1.1. Definition of Gas Turbine Engine	1
1.2. Principles of Gas Turbine Engine	2
1.3. Brayton Cycle	3
1.4. Need for Turbine Cooling	6
2. THEORY and LITERATURE	9
2.1. Theory of Numerical Solver	9
2.1.1. Flow and Energy Equations	9
2.1.2. Turbulence Modeling, Realizable $k-\varepsilon$ (RKE) Model	11
2.2. Theory of Wall Turbulence Modeling	16
2.3. Theory of Film Cooling	19
2.4. Relevant Literature	21
3. BASELINE SHAPED HOLE	33
3.1. Why Called as Baseline Shaped Hole	33
3.2. Design of the Baseline Shaped Hole	34
4. TEST FACILITY	37
5. COMPUTATIONAL MODEL	40
6. RESULTS	53
6.1. Results of Mesh Sensitivity Cases	53
6.2. Results of the Different Turbulence Models	55
6.3. Results of the CFD Validation Cases	58
6.4. Results of the Geometry Cases	66
6.4.1 Results of the Different Feeding Condition Cases	66
6.4.2 Results of the Different Expansion Angle Cases	75
6.5. Results of the Coolant Velocity Cases	91
7. CONCLUSION	104
REFERENCES	106
BIOGRAPHY	109

LIST of SYMBOLS and ABBREVIATIONS

A	: Actual
aw	: Adiabatic wall
B	: Base mass flow rate
M	: Blowing ratio
k	: Conductivity
h	: Convection
c	: Coolant
ρ	: Density
D	: Diameter of film cooling hole
Cd	: Discharge coefficient
Φ	: Dissipation
γ	: Effective injection angle
s	: Entropy
ε	: Epsilon
β	: Expansion angle
∇	: Gradient
f	: Gravitational or Coriolis Forces
H	: Height
i	: Ideal
α	: Injection angle
ν	: Kinematic viscosity
$\bar{\eta}$	: Lateral Averaged Film Effectiveness
L	: Length of film cooling hole
η	: Local film effectiveness
Ma	: Mach number
∞	: Mainstream
m	: Mainstream
inf	: Mainstream
S	: Mean rate of shear strain
m	: Metering section of cooling hole
I	: Momentum flux ratio
X/D	: Normalized downstream distance
ω	: Omega
$\partial/\partial t$	: Partial time derivative
P	: Pitchwise distance
P/D	: Pitchwise spacing
p	: Pressure
Re	: Reynolds
τ	: Shear Stress
T	: Temperature
t	: Turbulence
Tu	: Turbulent intensity
k	: Turbulent kinetic energy
U	: Velocity
V	: Velocity
μ	: Viscosity

w	: Wall
x	: X direction
y	: Y direction
z	: Z direction
AR	: Area ratio, throat area to greatest expansion area
boi	: Body of influence
BC	: Boundary condition
CRVP	: Counter-rotating vortex pair
DR	: Density ratio
dyn	: Dynamic
fl	: First layer
inh	: Inside of cooling hole
fwd	: Laidback
lat	: Lateral
MWA	: Mass weighted average
max	: Maximum
PIV	: Particle image velocimetry
PR	: Pressure ratio
RKE	: Realizable k Epsilon
Rec	: Recovery
ref	: Reference value
RANS	: Reynolds Averaged Navier Stokes
RIT	: Rotor Inlet Temperature
sep	: Separation bubble
SX	: Single crystal
S-A	: Spalart-Allmaras
TBC	: Thermal barrier coating
VR	: Velocity ratio
vc	: Vena contracta

LIST of FIGURES

	<u>Page</u>
Figure 1.1: Typical Core Gas Turbine.	1
Figure 1.2: An Open Cycle of Gas Turbine Engine.	2
Figure 1.3: Brayton Cycle for Gas Turbine.	3
Figure 1.4: Gas Properties along Turbojet Engine.	4
Figure 1.5: Thermal Efficiency of the Brayton Cycle.	5
Figure 1.6: Turbine Entry Temperature Over the Years.	7
Figure 2.1: Turbulence Models Hierarchy.	12
Figure 2.2: Velocity Profile in Turbulent Flow Near Wall.	17
Figure 2.3: Shear Behavior in the Near Wall Region.	18
Figure 2.4: Schematic View of Film Cooling.	19
Figure 2.5: Local Effectiveness Distributions for Different DR Cases.	22
Figure 2.6: $\bar{\eta}$ Comparison for Different M Cases.	22
Figure 2.7: Dimensionless Temperature Contours at Mid Plane.	23
Figure 2.8: $\bar{\eta}$ Comparison for Two Hole Spacing.	24
Figure 2.9: $\bar{\eta}$ Comparison for Different Hole Angles.	24
Figure 2.10: Flow Characteristics for $VR < 0.5$ Case in the Cooling Hole.	25
Figure 2.11: Flow Characteristics for $VR > 0.5$ Case in the Cooling Hole.	25
Figure 2.12: Schematic Presenting Four Types of Vortices Structure.	26
Figure 2.13: Comparison of CFD and Test Velocity Distributions.	27
Figure 2.14: η_{film} Contours for Different M, DR = 1.5, $Tu_{\infty} = 0.5\%$.	29
Figure 2.15: η_{film} Contours for Different M, DR = 1.5, $Tu_{\infty} = 5.6\%$.	29
Figure 2.16: $\bar{\eta}$ of Shaped Hole for low stream turbulence at DR = 1.5.	30
Figure 2.17: Comparison of $\bar{\eta}$ at DR = 1.5 and DR = 1.2, $Tu_{\infty} = 0.5\%$.	31
Figure 3.1: Baseline of Fan Shaped Hole Design.	34
Figure 3.2: Representation of X and Z Direction in Effectiveness Formula.	35
Figure 4.1: Schematic View of Selected Test Rig.	37
Figure 5.1: View and Dimensions of Computational Models.	40
Figure 5.2: View of BOI Domains.	41
Figure 5.3: View of Minimum Local Mesh Sizing Surfaces.	42
Figure 5.4: Computational Models with Polyhedral Mesh Distribution.	44
Figure 5.5: Detailed Mesh Image of Hole Region.	44
Figure 5.6: Detailed Mesh Image of BOI Regions.	45
Figure 5.7: Main BCs of Validation Cases.	47
Figure 5.8: BCs of Different Feeding Condition Cases.	48
Figure 5.9: Upside View of Hole Exits.	50
Figure 5.10: Main BCs of Coolant Velocity Cases.	50
Figure 6.1: Comparison of $\bar{\eta}$ for Different Mesh Cases with Test Data.	53
Figure 6.2: Comparison of $\bar{\eta}$ for Different Solver with Test Data.	56
Figure 6.3: Computed Velocity Vectors of Turbulence Solvers.	57
Figure 6.4: Plotted Residual Behaviors for Flow and Turbulence Equations.	58
Figure 6.5: Monitored Critical Analysis Parameters.	59
Figure 6.6: Computed Velocity Vectors at DR = 1.5, M = 1.	60
Figure 6.7: η_{film} Contour for Shaped and Cylindrical Hole at DR = 1.5, M = 1.	61
Figure 6.8: Comparison of $\bar{\eta}$ for Different M, DR = 1.5 and $Tu_{\infty} = 0.5\%$.	63

Figure 6.9: Comparison of $\bar{\eta}$ for Different M, DR = 1.2 and $Tu_{\infty} = 0.5\%$.	64
Figure 6.10: Comparison of $\bar{\eta}$ for Different M, DR = 1.5 and $Tu_{\infty} = 5.6\%$.	65
Figure 6.11: Computed Velocity Vector for Different Feeding Condition Cases.	67
Figure 6.12: The Differences of Velocity Magnitude Contour at Throat Section.	69
Figure 6.13: Three Dimensional Streamlines with Plenum Condition.	70
Figure 6.14: Three Dimensional Streamlines without Plenum Condition.	71
Figure 6.15: Velocity Vectors after Ejection for X/D = 2.5 location.	72
Figure 6.16: Plotted Velocity Vectors after Ejection by Saumweber.	73
Figure 6.17: The Vortex Core Regions for Cylindrical and Shaped Holes.	74
Figure 6.18: Computed Velocity Vectors at Mid Plane for Geometry Cases.	76
Figure 6.19: Computed Velocity Vectors at Mid Plane for Geometry Cases - II.	77
Figure 6.20: Dimensionless Length and Height of Separation Bubbles	78
Figure 6.21: Velocity Magnitude Contours at Throat Sections.	80
Figure 6.22: Representative Scheme of Shaped Hole Flow Behavior.	81
Figure 6.23: Schematic Top View of the Flow Structure.	82
Figure 6.24: Dimensionless Velocity Contours at the Hole Exit.	83
Figure 6.25: Local Axial Velocity Distributions at X/D = 5.	84
Figure 6.26: Detailed Three Dimensional Streamlines in Cooling Hole.	85
Figure 6.27: Comparison of Local Film Effectiveness for X/D Locations.	88
Figure 6.28: Velocity Vectors after Ejection at Mid Plane.	89
Figure 6.29: Comparison of $\bar{\eta}$ for Geometry Cases.	90
Figure 6.30: Computed V Vectors at Midsection.	93
Figure 6.31: Computed Ma Number at Throat Section.	96
Figure 6.32: Comparison of η_{film} for Different Coolant Velocity Cases.	97
Figure 6.33: Comparison of $\bar{\eta}$ for Different Coolant Velocity Cases.	98
Figure 6.34: Representative Cylindrical and Shaped Hole Geometry with BCs.	99
Figure 6.35: The Ideal Mass Flow Rate Parameters for Global Cd Calculation.	101
Figure 6.36: Comparison of Cd for Cylindrical and Shaped Hole.	101
Figure 6.37: Dynamic Pressure at the Hole Inlet for Different Mach Number.	102
Figure 6.38: Pressure Ratio, $P_{s,inlet} / P_{s,throat}$ for Different Mach Number.	103

LIST of TABLES

	<u>Page</u>
Table 3.1: Geometric Parameters of Baseline Shaped Hole.	33
Table 4.1: Test Plate Geometry.	38
Table 5.1: Mesh Properties of Sensitivity Studies.	41
Table 5.2: The Effect of Deviation Assumption on Part Metal Temperature.	43
Table 5.3: Validation Matrix for Baseline Shaped Hole.	45
Table 5.4: Solution Parameters of Numerical Models.	46
Table 5.5: Matrix for Different Feeding Condition Cases.	48
Table 5.6: Matrix for Different Expansion Angle Cases.	49
Table 5.7: Summary of Created Coolant Velocity Cases.	51
Table 6.1: Error Summary of Mesh Cases.	54
Table 6.2: Total Pressure Loss of Different Expansion Angle Cases.	81
Table 6.3: Separation Bubble Size after Ejection for Different Geometry Cases	87
Table 6.4: Result Summary of Created Coolant Velocity Cases.	92
Table 6.5: Dimensionless Height of Vena Contracta Bubble.	94
Table 6.6: Bubble Deminsionless Height of Diffuser Section.	95
Table 6.7: Result Comparison for Cylindrical and Shaped Hole.	100

1. INTRODUCTION

1.1. Definition of Gas Turbine Engine

A gas turbine engine is designed to convert the thermal energy of a fuel into other form of useful power, such as mechanical power or high-speed thrust of a jet. The engine has mainly a power converter section and a gas generator, as shown in **Figure 1.1**.

All engine has core gas generator which is consist of compressor, combustor chamber and turbine sections. There are different types of gas turbine engines which are occurred by adding various helper components at inlet and exit of the gas generator. For example, the higher amount of work or thrust will be obtained from ducted fan in turbofan engine or a propeller in turboprop engine. Main purpose of turbine is that extracting shaft power for driving the compressor. The shaft work may be used to supply direct shaft work or an auxiliary power unit or produce electricity of power plant.

Airplanes have massive machine because of their power to weight ratio. Therefore, gas turbine engines are exclusively used to help them fly. They have also been utilized to generate electric power. Gas turbine engines have wide range for propulsion applications because of the great power to weight ratio as mentioned above.

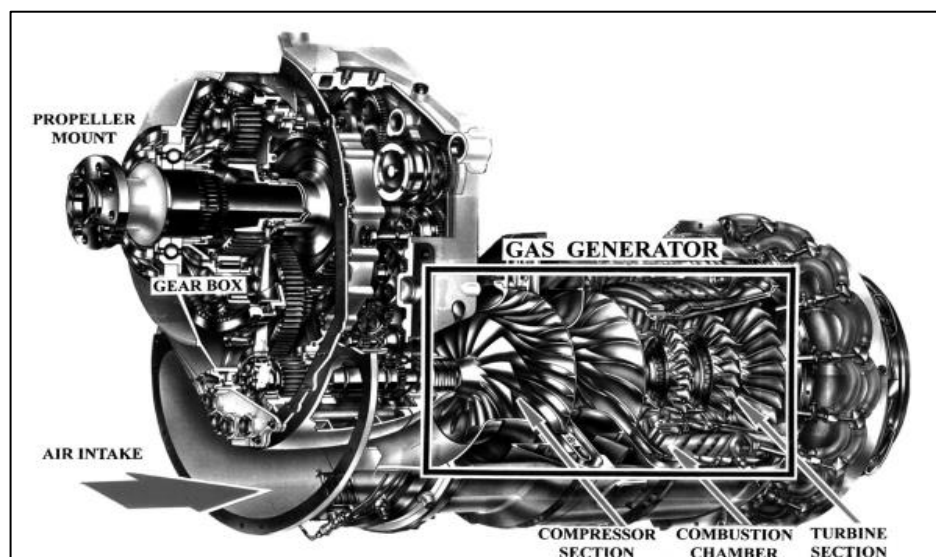


Figure 1.1: Typical Core Gas Turbine [1].

1.2. Principles of Gas Turbine Engine

The gas turbine engine has the compressor, the combustor and the turbine section respectively. Firstly, a fresh air from atmosphere go through inlet and its total properties are partially transformed to kinetic energy. Then, the fluid comes to compressor where there is a work to fluid resulting in an increase for its pressure. The compressor has progressively smaller areas stage by stage in order to maintain air axial velocity. With increasing of air density, continuity equation and constant mass flow rate along compressor are provided. This might be called as compressor design intent.

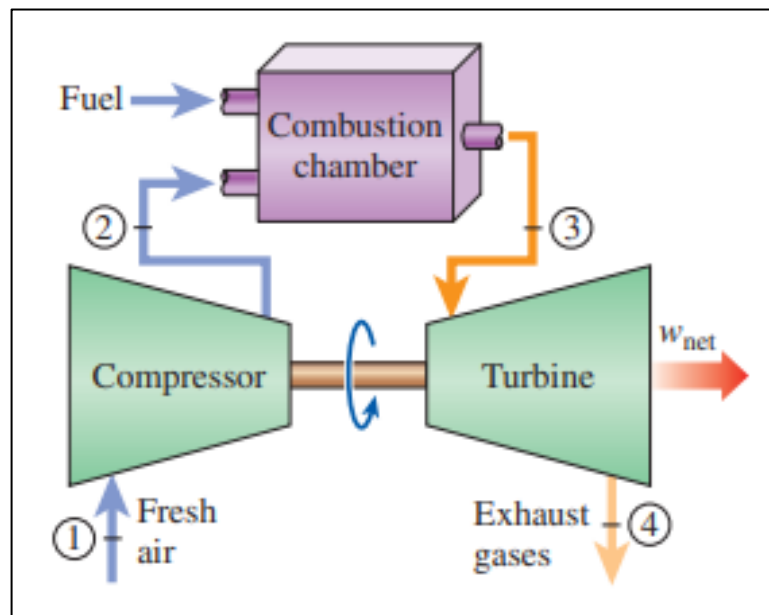


Figure 1.2: An Open Cycle of Gas Turbine Engine [2].

As shown in **Figure 1.2**, the squashed air comes into combustion chamber. There is a mixture between compressed air and fuel. High energy flow and temperature are provided with ignition. The fuel gets fire with oxygen in the compressed air and produce hot gases expanding.

After the combustion chamber, the high energy flow goes into the turbine. The turbine is assembled by a shaft to rotate the compressor blades. With respect to gas turbine type, this rotation gets some part of energy from the high energy flow. This energy is used to drive compressor. The gases which are produced in the combustion chamber go along the turbine and turn its blades. Turbine and compressor are fixed on shafts and have several sets of ball-bearing.

The thrust for the plane is produced from exhaust gases with the helping of nozzle component. The energy consumed working flow which passed the turbine blades and the colder air that bypassed the fan duct produce a force when leaving the nozzle. It behaves in order to propel the engine, and so the airplane ahead.

Air breathing jet engines are typically consist of rotating compressor component powered by a turbine. Also, thrust is provided with leftover power from propelling nozzle. This process is called as Brayton thermodynamic cycle.

1.3. Brayton Cycle

Brayton cycle is first suggested by George Brayton. It is proposed in order to use in the reciprocating oil-burning engine. Today, it is utilized for gas turbine engine to present the compression and the expansion steps take place in rotating machinery.

There are two types of cycle in figure. They are called as ideal and actual process in terms of entropy change. As it can be seen in below figure, entropy remains constant for ideal cycle and it is known as isentropic compression and expansion.

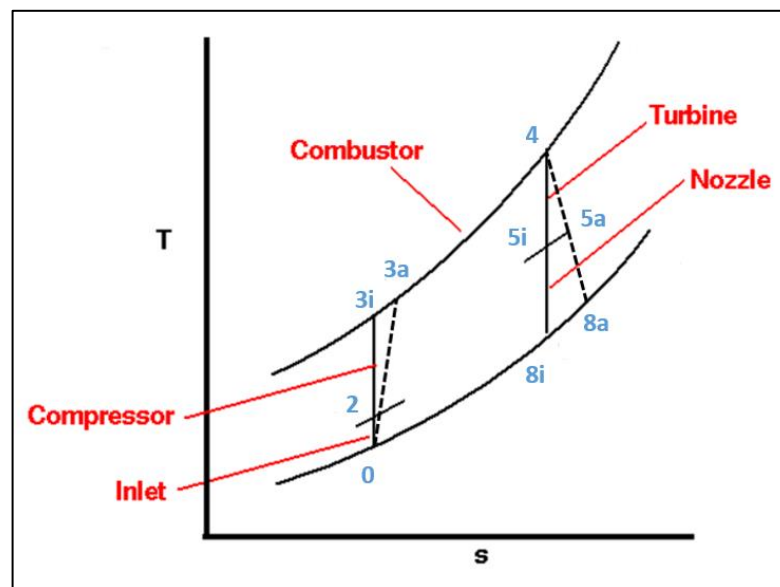


Figure 1.3: Brayton Cycle for Gas Turbine.

In Brayton cycle, a heat addition process occurs as combustion between point 3 and point 4 as shown in **Figure 1.3** and it is called as temperature and entropy (T-s) diagram. The power output and thermal efficiencies from cycle or engine are proportional to the below area under point 3 and point 4 in the diagram. The higher

location of point 3 and 4 on temperature axis causes the higher power output and efficiency.

The higher location of point 3 depends on pressurization of compressor. It means that compressor has the higher compression capability, cycle has higher compressed and hotter air.

The higher location of point 4 is connected with compressor work and fuel consumption. If the compressor has higher work, the location of point 4 will automatically be higher. The other factor is fuel consumption rate. With the compressor which has the same capability of compression, the higher location of point 4 occurs due to the higher fuel consumption rate.

The working Brayton cycle of the gas turbine engine has the similar processes with the Otto cycle of piston engine. Nevertheless, combustion happens at a constant pressure, whereas it occurs at a constant volume for the piston engine.

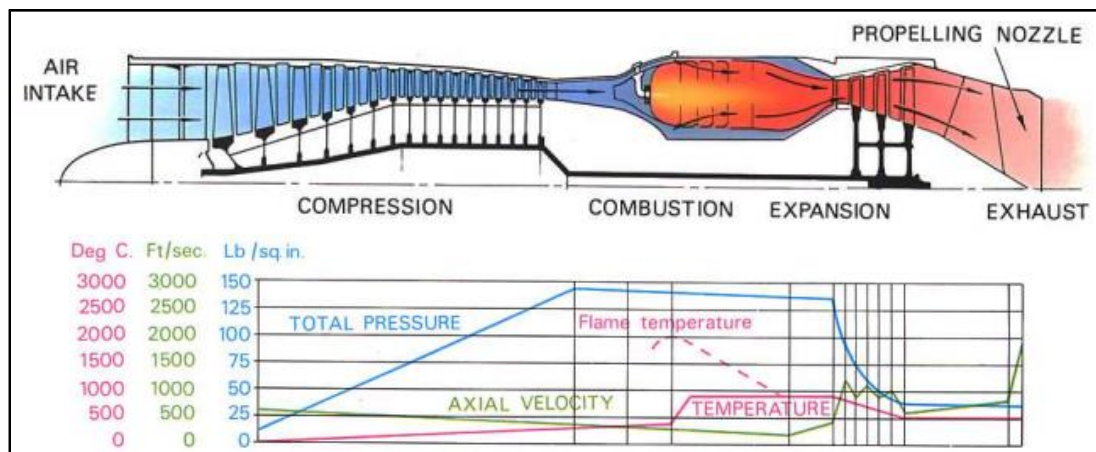


Figure 1.4: Gas Properties along Turbojet Engine [3].

Total pressure, axial velocity and temperature changes of jet engine components might be seen in **Figure 1.4**. During the passing of the gas through the compressor, a increase in the pressure of the air is demanded. Axial velocity is requested to be remain constant as it is explained before. Therefore, compressor generally has divergence passage with the increasing density in order to keep the velocity constant. In this way, it is aimed to have constant axial velocity because of nature of the mass conservation.

After the air is heated and the internal energy is processed through combustion, the turbine must be forced to rotate by increasing the gas velocity. When the thrust on the

aircraft is generated by the change in air momentum, a high exit velocity is required at the propelling nozzle. Additionally, localized air distribution needs to slow down as combustion occurs to create a low-velocity zone in which the flame can burn.

Change state of the air occurs under three basic conditions in engine operation. The temperature decreases the volume of air during compression and increases with the process of increasing pressure. When fuel is fueled and burned to increase temperature, there is an increase in volume while the amount of pressure remains constant during combustion. During the expansion of the turbine arrangement, when work is released from the gas analysis, the temperature and pressure decrease and the volume increases together.

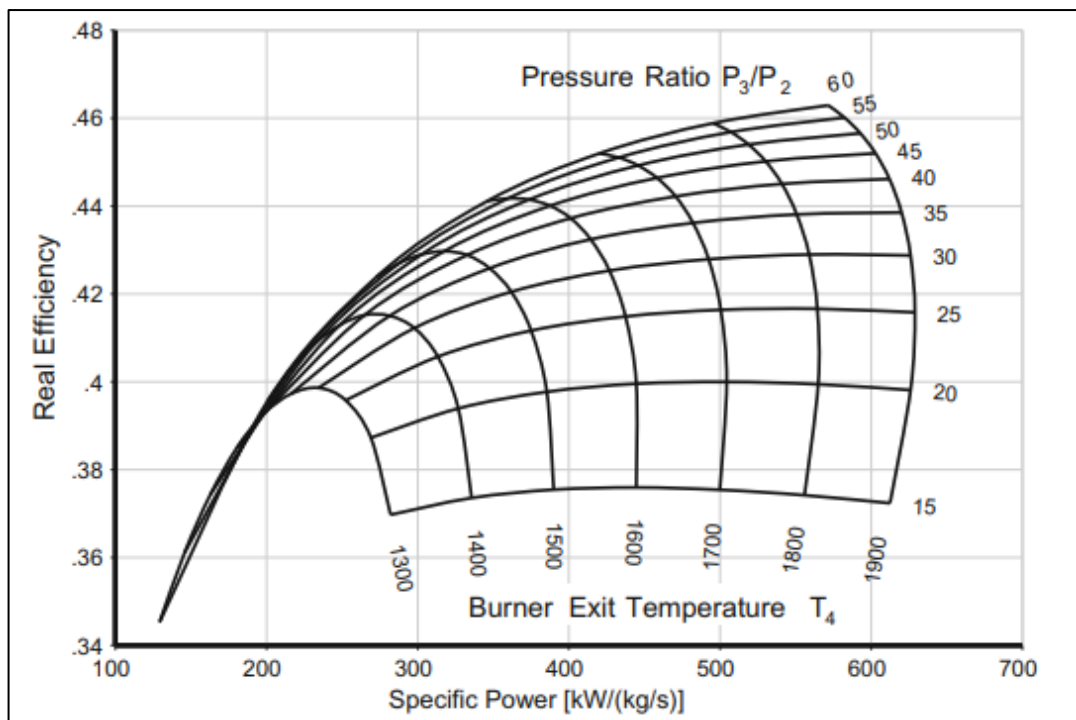


Figure 1.5: Thermal Efficiency of the Brayton Cycle [32].

There are two most important factors which affect thermal efficiency or power output of engine as it is mentioned above. One way to get higher power or efficiency is to achieve higher pressure ratio from engine. And, it is proportional to the mass flow rate through the engine. But, engine becomes heavier due to the higher radius with great pressure ratio. The other factor is the component efficiency of compressor and turbine. This kind of efficiency depends on temperatures of compressor and turbine. Higher

temperatures and pressure ratios are requested in order to increase engine real efficiency, as it can be realized in **Figure 1.5**.

1.4. Need for Turbine Cooling

Gas turbine engines are utilized for propulsion system of aircraft and generation of power on land or industrial applications. Aircraft engines are complex and high level working machine to meet high performance and efficiency.

There are two important components which work under extremely high temperatures at engines. Turbine blades are one of the most crucial components of these machines. Blades rotating at high speeds power turbines, which are subjected to extremely high temperatures in the process. However, these extreme temperatures can threaten the design of turbine blades. The other one is nozzle guide vanes. Vanes are used in order to guide main flow to blades and are exposed to combustion air which has very high temperature.

Turbine cooling systems are used to prevent damage to turbine blades and vanes from excessive operations. These systems generally provide cooling air into or around them. While the cooling air protects the blades and vanes from hot gases, the integrity and durability of the turbine increase at the same time. This cooling procedure let the engine to run at a higher RPM with generation of more power, which helps airplanes fly farther and faster.

Turbine cooling systems are developed and optimized to increase the reliability of aircraft engines. Low fuel consumption and high temperature level are critical factors in the design of modern aircraft engines, therefore advances in turbine cooling technology are constantly needed. These developments contribute to more efficiency and reduce environmental impacts in the aviation industry. Power output of gas turbine engines and its thermal efficiency increase by means of high turbine rotor inlet temperature (RIT) as it mentioned above. Clearly, increasing RIT is one of the most important key technologies for higher gas turbine engine performance.

Figure 1.6 shows that the RIT in advanced gas turbines are far higher than the softening point of the blade material. Therefore, turbine blade has to be cooled [8]. High service temperature development of material such as thermal barrier coating (TBC) or highly advanced cooling techniques is an important topic that needs to be

pointed out to provide high power and performance for the future engines. Uncooled turbines allow to low turbine entry temperature as it can be seen in figure. RIT has positive trend with addition of cooling system and new material to the engines.

Until the generation of single crystal SX casting material, material development benefits are third times lower than cooling technologies benefits in terms of RIT. With the development of SX casting, the new trend which has higher benefits is obtained.

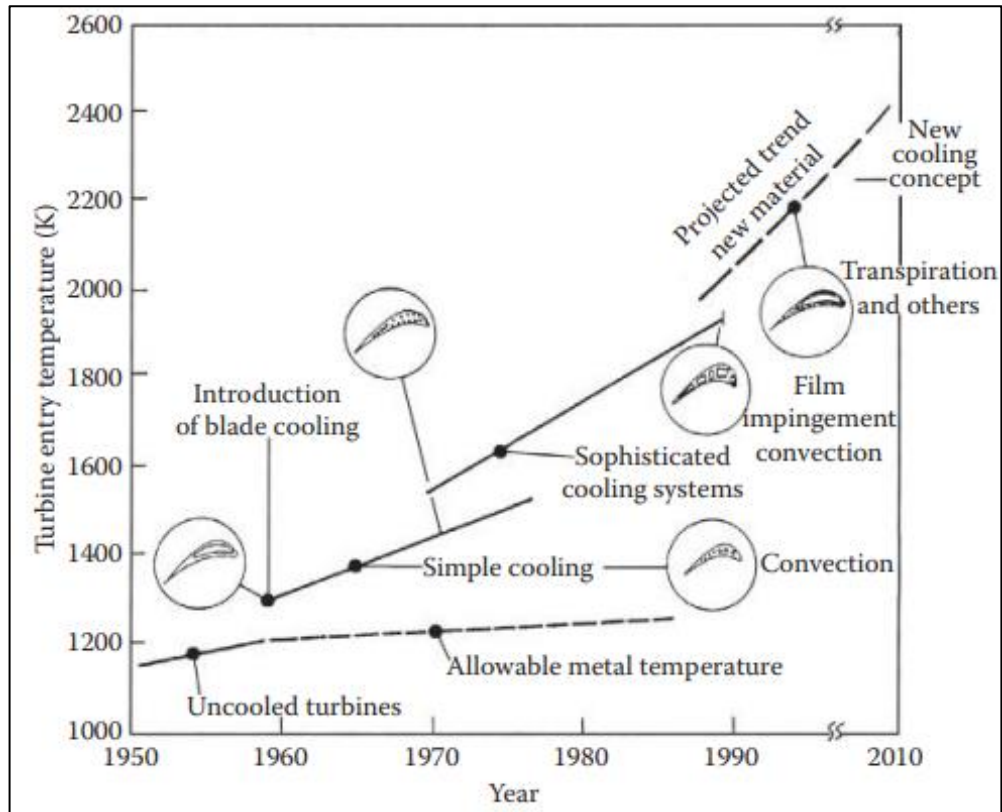


Figure 1.6: Turbine Entry Temperature over the Years [8].

Cooling needs to be used for safe operation. The air which is extracted from the compressor of the engine is utilized in order to cool the vanes and blades. Turbine cooling system is consist of film cooling, impingement cooling and enhanced convective cooling.

Film cooling technique which has coolant gas passes around vane or blade surface is used for a long time. This technique is the key of a coolant fluid at any points along surface which exposed to a high external temperature and heat transfer coefficient. This method helps to gain two benefits. First one is protection of surface in the hole location, the second one is the protection the downstream of the hole region.

Turbine and vane surfaces, platforms, shrouds are cooled by means of discrete film cooling holes. While the internal cooling techniques remove heat from the inside of metal surface, film cooling protects the blade surface from hot gases. Also, the cooling air passes through the film holes. In this way, it helps to remove heat load from surface by convection heat transfer.

As a result, film cooling technique which has coolant gas passes around coolant surface is used for a long time in turbine cooling with these intents.

2.THEORY AND LITERATURE

2.1. Theory of Numerical Solver

There are set of non-linear partial differential equations which are used due to solve the fluid movement and heat transfer mechanism of system. And, they are known as governing equations. The conservation of mass, momentum and energy flow through a system as solid, gas or liquid is fully described and solved by means of these differential equations. But, simulations which utilize RANS-based methods need to have additional turbulence equations.

The conservation of mass, momentum and energy equations help to govern the movement of any viscous flow and they are called the Navier-Stokes equations. Claude-Louis Navier and George Gabriel Stokes are the physicists who concluded the finding of equations. A finite control volume of fluid is focused in order to apply the basic physical principles. They are integrated form of governing equations. These equations can be written to solve physics as pure or complicated way.

2.1.1. Flow and Energy Equations

Fluid motion and heat transfer is governed by differential equations that are called continuity, momentum and energy equation. Budugur have completely explained all these equations and derivation or manipulation ways of them [25].

Continuity equation depends on the theory of mass balance. It means that mass is immutable in system. The three dimensional, unsteady and compressible flow continuity equation is given below formula as in its most common Cartesian co-ordinates form.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho V_i) + \frac{\partial}{\partial y}(\rho V_j) + \frac{\partial}{\partial z}(\rho V_k) = 0 \quad (2.1)$$

For steady flow, first term of formula which represents time dependent density change equals to zero. Also, ρ might be neglected for incompressible flow. After that, the

momentum equation for steady and incompressible flow can be derived by manipulating Formula 2.1 as below Formula 2.2.

$$\frac{\partial}{\partial x}(V_x) + \frac{\partial}{\partial y}(V_y) + \frac{\partial}{\partial z}(V_z) = 0 \quad (2.2)$$

Newton's second law is a phenomena which helps to derive momentum equations. They state that applied forces have to be remain constant on fluid volume.

$$\rho \frac{\partial V}{\partial t} = -\nabla P + \nabla \tau + f \quad (2.3)$$

Shear stress expression τ which is derived by George Stokes can be given for one direction by;

$$\tau_{xy} = 2\mu S_{xy} + \lambda \frac{\partial V_z}{\partial z} \delta_{xy} \quad (2.4)$$

Where shear strain rate S can be seen as;

$$S_{xy} = \frac{1}{2} \left(\frac{\partial V_y}{\partial x} + \frac{\partial V_x}{\partial y} \right) \quad (2.5)$$

General form of momentum equation is given in Formula 2.3. It is also known as Cauchy momentum equation. First term is about change in speed with direction. It is covered by the total, convective and local acceleration with respect to velocity change. Second term represents the pressure force that apply on surface. Third one is about viscous stresses. Gravitational or Coriolis forces are covered with last term.

Formulation which is consist of detailed explanation of the viscous terms was given in White [26]. But, this form is so useless due to so many independent terms.

X momentum equation:

$$\begin{aligned} \frac{\partial}{\partial t}(\rho V_x) + \frac{\partial}{\partial x}(\rho V_x^2) + \frac{\partial}{\partial y}(\rho V_x V_y) + \frac{\partial}{\partial z}(\rho V_x V_z) = \\ -\frac{\partial p}{\partial x} + \mu \left(\frac{\partial^2 V_x}{\partial x^2} + \frac{\partial^2 V_y}{\partial y^2} + \frac{\partial^2 V_z}{\partial z^2} \right) \end{aligned} \quad (2.6)$$

Y momentum equation:

$$\begin{aligned} \frac{\partial}{\partial t}(\rho V_y) + \frac{\partial}{\partial x}(\rho V_y V_x) + \frac{\partial}{\partial y}(\rho V_y^2) + \frac{\partial}{\partial z}(\rho V_y V_z) = \\ -\frac{\partial p}{\partial y} + \mu \left(\frac{\partial^2 V_y}{\partial x^2} + \frac{\partial^2 V_y}{\partial y^2} + \frac{\partial^2 V_y}{\partial z^2} \right) \end{aligned} \quad (2.7)$$

Z momentum equation:

$$\begin{aligned} \frac{\partial}{\partial t}(\rho V_z) + \frac{\partial}{\partial x}(\rho V_z V_x) + \frac{\partial}{\partial y}(\rho V_z V_y) + \frac{\partial}{\partial z}(\rho V_z^2) = \\ -\frac{\partial p}{\partial z} + \mu \left(\frac{\partial^2 V_z}{\partial x^2} + \frac{\partial^2 V_z}{\partial y^2} + \frac{\partial^2 V_z}{\partial z^2} \right) \end{aligned} \quad (2.8)$$

The velocity gradient and viscosity is substituted with shear stress terms in order to get pure form of momentum equation. Also, gravitational forces are neglected in pure form. X, Y and Z momentum equations are given as they can be seen above respectively.

$$\rho \left[\frac{\partial h}{\partial t} + \nabla h V \right] = -\nabla P + \nabla(k \nabla T) + \Phi \quad (2.9)$$

With respect to first law of thermodynamics, the sum of heat transfer and applied work of the system equals to the total energy. This is the statement of energy conservation law. First and second term of left side are about energy changes with time and convective heat transfer respectively. The first term of right side represents the energy transfer because of applied by compressive pressure work. The second one is about diffusion and equals to heat flux mentioned by Fourier's law. The last term represents dissipation and is connected with irreversible energy transfer and heat source.

2.1.2. Turbulence Modelling, Realizable $k-\varepsilon$ (RKE) Model

With a few exceptions, the governing equations that are utilized to solve fluid motion and heat transfer cannot be solved as analytical. So, the majority of flow cases have to be solved numerically by means of differential equations discretization. Various

numerical techniques have been developed to find a solution for random and irregular motions happening in extensive time and space.

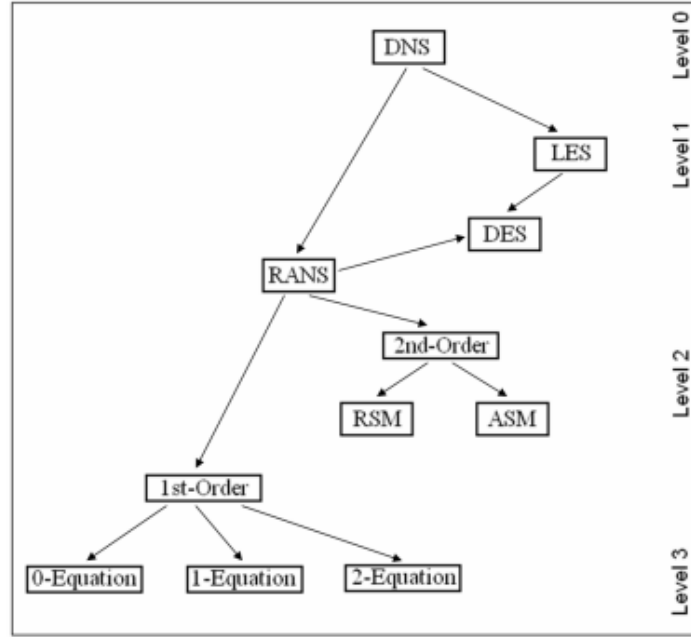


Figure 2.1: Turbulence Models Hierarchy [28].

While precise definition of turbulence requires extremely fine meshes and expensive computational time, less laborious techniques that obtain proper results have been created for industrial cases. The figure above provides an overview of the various turbulence models that is classified with respect to their decreasing complexity.

Turbulence modelling is the key for construction and using of a mathematical model to find the effects of turbulence. The Navier Stokes equations govern the velocity and pressure of fluid domain as it is mentioned before. Each of these parameters can be broken down into a mean part and a fluctuating part in a turbulent flow. The Reynolds-averaged Navier Stokes (RANS) equations, which control mean flow, are obtained by averaging the equations. However, these averaged equations have the nonlinearity. It means that velocity fluctuations still exist in the RANS equations as nonlinear term Reynolds stress. Its effect exists as stress term on the averaged flow similar to pressure and viscosity. Solution needs to close RANS equations by means of Reynolds stress term as a function of averaged flow. Also, fluctuating parts are removed in terms of velocity. It is called as closure problem. Joseph Valentin Boussinesq was the first person who has mentioned the concept of eddy viscosity in order to deal with closure problem.

Tennekes and Lumley [27] have provided introduction to turbulent flow. In addition, Budugur [25] and Blazek [28] have provided more advanced explanations about turbulence modeling and turbulent flow.

In modern engineering applications, there are commonly operated turbulence models in terms of RANS equations. The Boussinesq hypothesis which is connected with closure problem is utilized in the k - ε , k - ω and Spalart-Allmaras models. In this way, a relatively low cost computation is offered for the turbulence viscosity. Despite k - ω and k - ε models employ two additional equations to solve turbulence viscosity transport, S-A model has only one. Also, these three models are known as types of first order closure.

In this thesis, Realizable k - ε (RKE) model is preferred to solve validation and thesis studies. Because, there are separation bubble and recirculation in expansion section of fan shaped hole, separation at the exit of hole, jet spreading with injection angle as the common knowledge about fan shaped holes. Therefore, only RKE model is introduced but the effects of different turbulence solvers are considered at result section.

The most widely utilized two equation eddy viscosity model is the k - ε . The turbulent kinetic energy and turbulent dissipation rate are solved by means of this model [28].

The RKE model has been recently developed and has different two important addition considering standard k - ε model. First one is about turbulent viscosity and the other one is about dissipation rate. Also, it is a first order turbulence model as it is mentioned above.

The formed transport equations and parameters for RKE model are seen in below formulas.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial y}(\rho k V_y) = \frac{\partial}{\partial y} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial y} \right] + G_k + G_b - \sigma \varepsilon - Y_m + S_k \quad (2.10)$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial y}(\rho \varepsilon V_y) &= \frac{\partial}{\partial y} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial y} \right] + \rho C_1 S_\varepsilon \\ &\quad - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b \end{aligned} \quad (2.11)$$

$$C_1 = \max \left[0.43, \frac{\alpha}{\alpha + 5} \right] \quad (2.10)$$

$$\alpha = S \frac{k}{\varepsilon} \quad (2.11)$$

The generation of turbulence kinetic energy occurs because of the mean velocity gradients. It is given as G_k and can be calculated as written Formula 2.12. To evaluate consistent G_k with the Boussinesq hypothesis, two different formulas are utilized.

Also, the same kind of generation takes place due to buoyancy. It is seen with G_b and can be calculated for ideal gas as given Formula 2.14. But, the generation of k which occurs due to the effect of buoyancy when models include both a non-zero gravity field and a non-zero temperature gradient. In this thesis, gravity effect is not considered. Therefore, G_b can be removed from both two formulas.

Y_m represents the additive of the fluctuating expansion in turbulence of compressible flow to the entire dissipation rate. S is called as the modulus of the mean rate of strain tensor [29].

C_2 and $C_{1\varepsilon}$ are model constants. S_ε and S_k can be defined as source terms by user in Ansys Fluent.

$$G_k = -\rho \overline{V_x' V_y'} \frac{\partial V_y}{\partial x} = \mu_t S^2 \quad (2.12)$$

$$S = \sqrt{2S_{xy}S_{xy}} \quad (2.13)$$

$$G_b = -g_x \frac{\mu_t}{\rho Pr_t} \frac{\partial \rho}{\partial x} \quad (2.14)$$

$$Y_m = 2\rho\varepsilon M_t^2 \quad (2.15)$$

Where M_t is used for turbulent Mach number and defined as;

$$M_t = \sqrt{\frac{k}{c}} \quad (2.16)$$

Where c represents the speed of sound. The results are affected by this compressibility modification when fluid is used as compressible form of the ideal gas.

The term realizable helps to model satisfies accurate mathematical constraints on calculation of the Re stress. In this way, model is consistent with the turbulent flow behaviors [29].

RKE solves the spreading of the round jets more accurately as direct benefit. Also, it can provide excellent performance for flows that are consist of separation, recirculation and boundary layer with tough adverse pressure gradient.

With this model, a new formulation is contained for the turbulent eddy viscosity. A new eddy viscosity formulation which depends on a few constraints for the turbulent Re stress. As it is used in other k - ε models, turbulent eddy viscosity formulation can be seen in Formula 2.17.

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (2.17)$$

But, the C_μ is no longer a constant and it is computed from local flow parameters with constants together as they can be seen below formulas.

$$C_\mu = \frac{1}{A_0 + A_s \frac{kU^*}{\varepsilon}} \quad (2.18)$$

$$U^* = \sqrt{S_{xy}S_{xy} + \tilde{\Omega}_{xy}\tilde{\Omega}_{xy}} \quad (2.19)$$

$$\tilde{\Omega}_{xy} = \Omega_{xy} - 2\varepsilon_{xyz}\omega_k \quad (2.20)$$

$$\Omega_{xy} = \overline{\Omega_{xy}} - \varepsilon_{xyz}\omega_k \quad (2.21)$$

where $\overline{\Omega_{xy}}$ is the average rate of rotation tensor which seen in a rotation reference frame with angular velocity ω_k . The model constants are taken as;

$$A_0 = 4.04 \quad (2.22)$$

$$A_s = \sqrt{6}\cos\phi \quad (2.23)$$

where

$$\phi = \frac{1}{3} \cos^{-1}(\sqrt{6}W) \quad (2.24)$$

$$W = \frac{S_{xy}S_{yz}S_{xz}}{\tilde{S}^3} \quad (2.25)$$

$$\tilde{S} = \sqrt{S_{xy}S_{xy}} \quad (2.26)$$

$$S_{xy} = \frac{1}{2} \left(\frac{\partial V_y}{\partial x} + \frac{\partial V_x}{\partial y} \right) \quad (2.27)$$

2.2. Theory of Wall Turbulence Modeling

Computing turbulent flows which are affected by adjacent wall is the one of the most prevalent engineering issue. In turbomachinery industry, there are mainly two effects of a wall. First one is anisotropic turbulent flow which occurs due to changing of the wall normal components, the other one is increasing the generation of turbulence.

The wall creates thicker boundary layer where the velocity changes from zero condition on the wall to free stream velocity value. The highest friction difference and largest velocity gradients are usually found in the near wall region. At the same time, thermal boundary layer with largest gradients exists for heat transfer problem. It is crucial to accurately take these velocity, friction and temperature variations at the near wall. Because, these are computed by using gradients of the dependent variables.

In the turbulent flow, the region where very close to the wall shows a nearly linear velocity profile and viscous effects entirely dominate. It is called as the viscous sublayer and is shown in **Figure 2.2**. The velocity changes with distance from the wall. Although the viscous sublayer thickness is very thin, it has a dominant role on flow characteristics. Because, it is consist of large velocity gradients.

A very fine mesh which is close to the wall is applied in order to resolve the flow as the common way. It is very important to capture the near wall gradients as it is mentioned above. So, a large number of nodes is required.

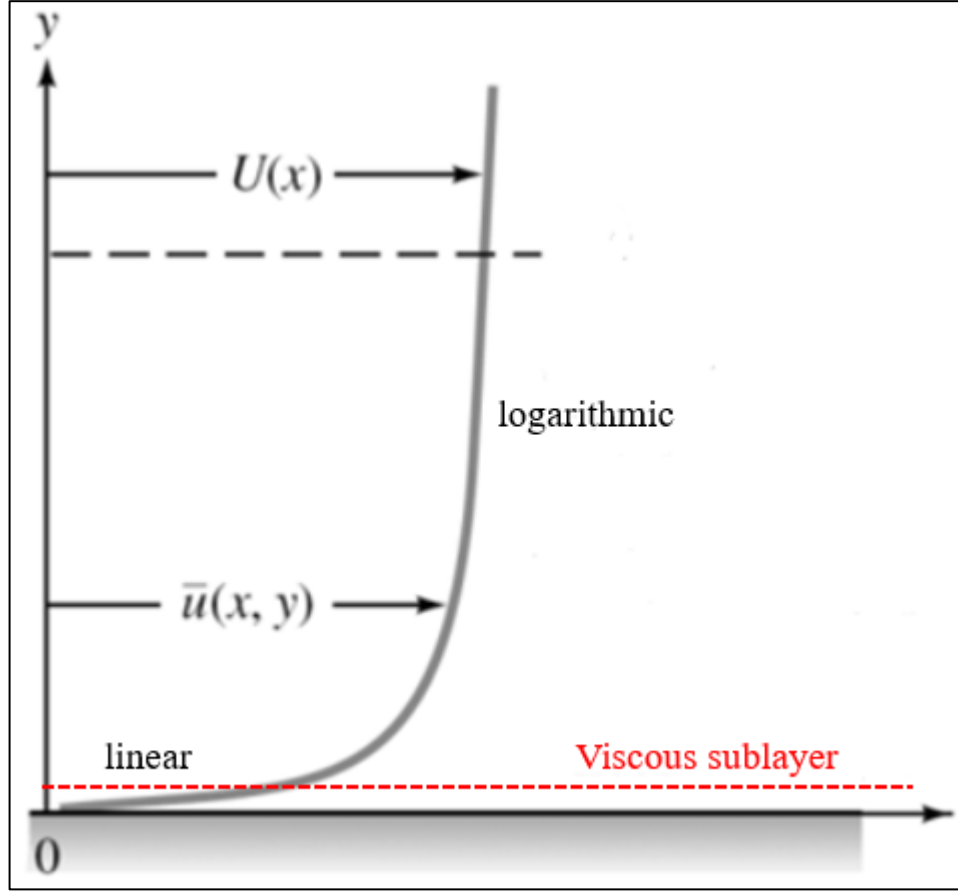


Figure 2.2: Velocity Profile in Turbulent Flow Near Wall [30].

The dimensionless parameter which is called as y plus in order to determine first cell distance from the wall. And, it is a function of friction velocity u^* , normal distance from the wall y and kinematic viscosity ν or time averaged velocity $\bar{u}$ and friction velocity u^* . Formulations are given in Formula 2.28.

$$y^+ = \frac{yu^*}{\nu} = \frac{\bar{u}}{u^*} \quad (2.28)$$

Where the friction velocity is a function of the wall shear stress and density. It can be seen as ;

$$u^* = \sqrt{\frac{\tau_w}{\rho}} \quad (2.29)$$

For the numerical studies, y^+ is aimed to be inside of the viscous sublayer in order to capture linear velocity profile and friction difference more accurately. With respect to this purpose, mesh y^+ of the thesis models is aimed to be 1 as highest value.

The viscous sublayer becomes thinner when the mean velocity increases. The velocity profile gets nearly flat and its distribution exists more uniform at very high Re number.

The near wall region can be divided into three sub-layers [27]. These are viscous sublayer, buffer layer and inertial sublayer.

- Viscous sublayer $0 < y^+ < 5$
- Buffer layer $5 < y^+ < 30$
- Inertial sublayer $30 < y^+ < 200$

While the viscous effects are very small in the inertial sublayer, the turbulence is negligible in the viscous sublayer. However, both viscous effects and turbulence are important in the buffer layer as it can be realized in **Figure 2.3**.

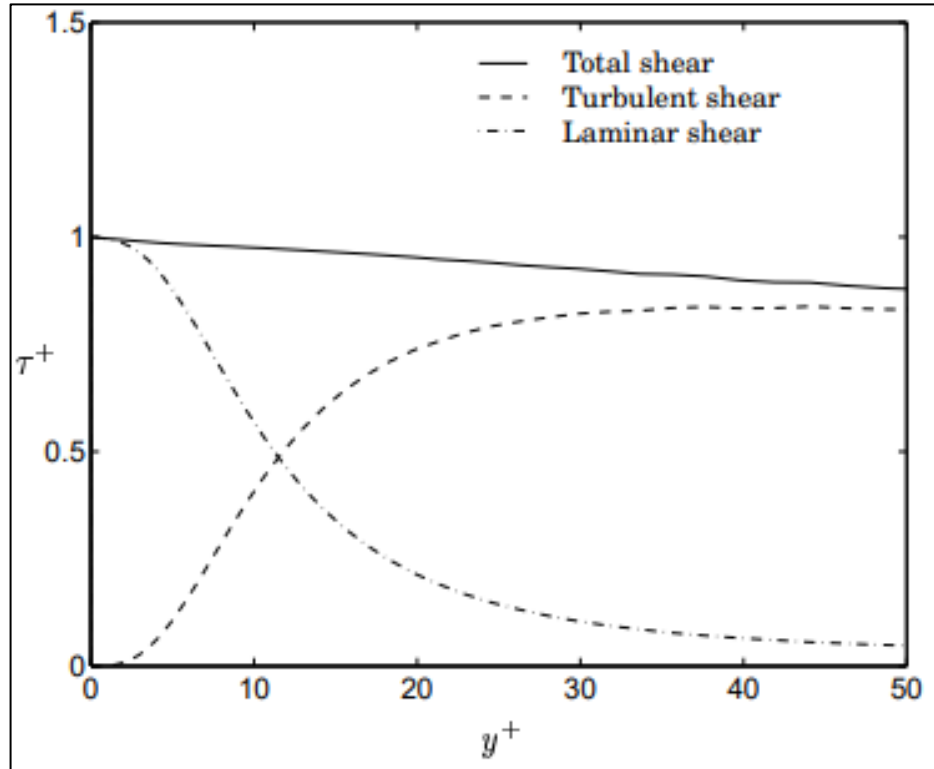


Figure 2.3: Shear Behavior in the Near Wall Region [31].

2.3. Theory of Film Cooling

Film cooling is known as convective heat transfer mechanism of turbine cooling design. Film cooling is the application which coolant air from compressor is fed into channel and passes through a series of holes onto surface of turbine components to cool and cover the external surface from hot combustion gases. The coolant flow ideally behaves as a preventive barrier on surface between the hot mainstream gases and turbine components. In this way, external convective heat transfer is reduced to increase gas turbine engine performance and RIT.

Success of film cooling in terms of heat transfer viewpoint depends on two performance variables. These are the adiabatic wall effectiveness and the heat transfer coefficient. The heat flux is described to component wall by external convection. This convection heat flux is given by Equation 2.30.

$$q'' = h_{inf}(T_{aw} - T_w) \quad (2.30)$$

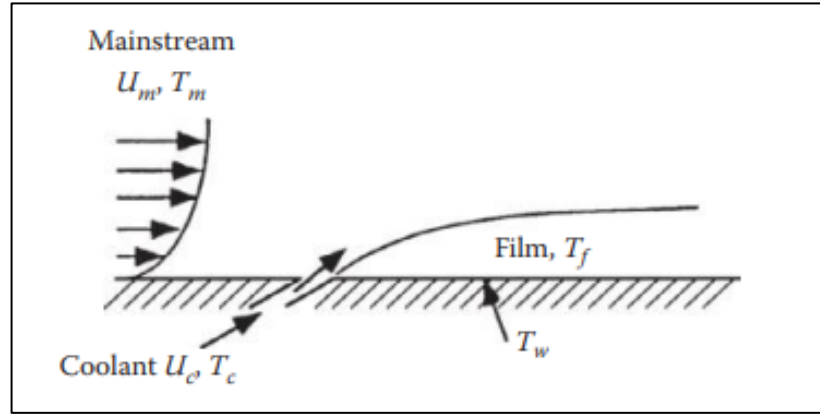


Figure 2.4: Schematic View of Film Cooling [8].

As it can be seen from equation and figure and formula, heat flux is proportional to heat transfer coefficient and temperature difference between adiabatic wall belongs to external gas and wall. Convection heat transfer is triggered by this phenomena. T_{aw} is the fluid temperature above the surface and called adiabatic wall temperature. Because, the surface gets this temperature if there is no heat transfer and heat flux equals to zero. Therefore, insulation material which has low conductivity is used to get adiabatic wall

temperature as much as possible. In this way, heat flux remains very low and T_{aw} can be taken from surface.

The aim of film cooling is to remain T_w under the material service limit. The adiabatic film cooling effectiveness is the non-dimensional parameter which is used to determine the performance of the coolant at system. This effectiveness is calculated with Equation 2.31 at below.

$$\eta_{film} = \frac{T_{\infty} - T_{aw}}{T_{\infty} - T_c} \quad (2.31)$$

T_c is the coolant temperature and T_{∞} is the hot mainstream temperature. If adiabatic wall temperature is assumed as coolant temperature, effectiveness is one and it means that system has 100% cooling effectiveness.

Also, film cooling performance depends on many non-dimensional parameters which describe the film cooling geometry and the flow behavior. These parameters are the coolant-to-mainstream ratios of density, velocity, mass flux and momentum flux as shown below in Equation 2.32.

$$DR = \frac{\rho_c}{\rho_{\infty}}, \quad VR = \frac{U_c}{U_{\infty}}, \quad M = \frac{\rho_c U_c}{\rho_{\infty} U_{\infty}} = \frac{\dot{m}_c / A_c}{\rho_{\infty} U_{\infty}}, \quad I = \frac{\rho_c U_c^2}{\rho_{\infty} U_{\infty}^2} \quad (2.32)$$

For cooling and test applications in gas turbine, general density ratios are between 1.2 and 2.0. It means proportion of temperature with respect to ideal gas formula and directly effects the adiabatic film effectiveness. Velocity ratio varies with respect to location and application of hole. It influences flow behavior and separation region. The mass flux ratio M is called as the blowing ratio and it is a function of multiplication DR and VR . With the higher coolant mass flux, jets have more cooling capacity but they have higher momentum, which can lead to separation from the surface for cylindrical hole. Momentum flux is a function of mass flux. So, it has the same trend with mass flux on effectiveness and flow behavior. The higher the momentum flux ratio the higher the jet penetration and spreading especially for fan shaped hole. These parameters are main influencer on the behavior of coolant flow and on the film effectiveness.

Generally, there is a separation or low velocity field in the hole due to vena contracta and jetting effect especially for plenum case. Therefore, these formations effect the

mass flow rate which passes through the hole. Film cooling hole is not able to provide the maximum flow rate which can pass through the throat of hole. The flow rate that is fed into main stream for plenum case is assumed as baseline flow rate.

2.4. Relevant Literature

At the beginning of the film cooling techniques, just cylindrical holes were utilized on surface of parts. Also, test facilities were developed with cylindrical shaped holes as first finding in order to investigate behaviors of film cooling. So, literature were firstly given with respect to cylindrical holes and then literature of fan shaped was considered.

Film cooling performance parameters which are momentum, density, velocity and blowing ratios have been studied in order to determine their effects. The film cooling jets might be attached to the surface, locally detached and after that attached or fully detached from cooling surface. An attached jets are preferred, because they provide the best cooling coverage from hot mainstream gases.

A study worked by Sinha et al. [9] find that the effect of different density ratios on lateral and centerline effectiveness cannot be scaled with VR, M and I across the range studied. For the attached jets film cooling effectiveness scaled with momentum flux ratio, it provides lower effectiveness at low blowing ratios.

Also, the high density ratio jet has better local lateral or laterally averaged cooling effectiveness as it can be seen in **Figure 2.5**. Lower centerline effectiveness than the low density ratio jet for an attached jet with matched momentum ratio.

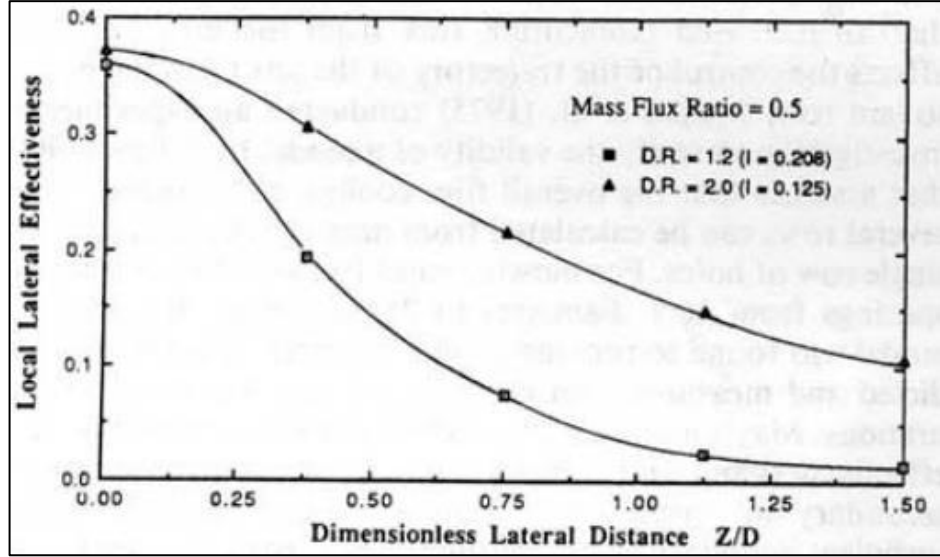


Figure 2.5: Local Effectiveness Distributions for Different DR Cases [9].

In Sinha studies, the most effective blowing ratio is found approximately as $M = 0.5$. **Figure 2.6** shows that maximum laterally averaged effectiveness is obtained at $M = 0.57$ case for $DR = 1.6$.

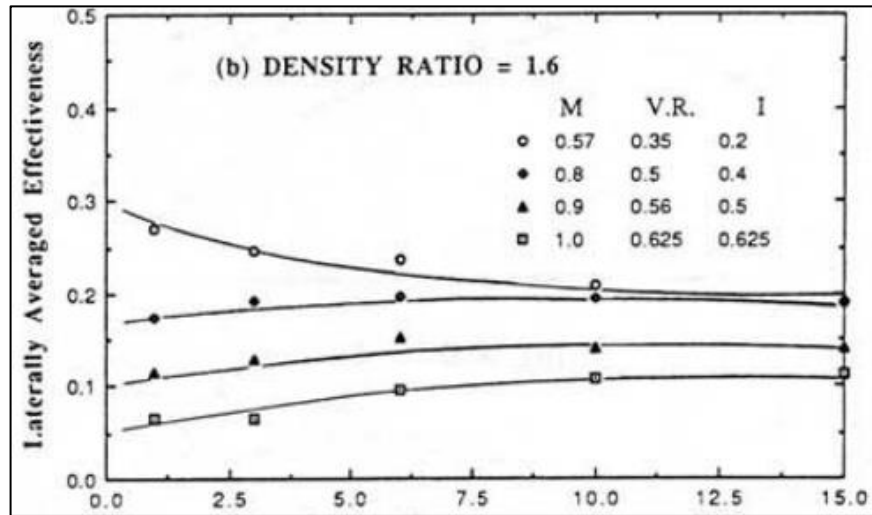


Figure 2.6: $\bar{\eta}$ Comparison for Different M Cases [9].

Thole et al. [10] had temperature measurements from downstream of film cooling holes for different range of conditions. They reported detachment conditions of film holes for scaled with momentum flux ratio. For a cylindrical hole, the jet started to detach from surface when momentum flux ratio was higher than 0.4. Between 0.4 and 0.8, the jet was partially detached and attached. For $I > 0.8$, the jet was completely

detached. Fully detached jet injection along the midline was presented in **Figure 2.7** with dimensionless temperature contours and detachment regions can be easily seen.

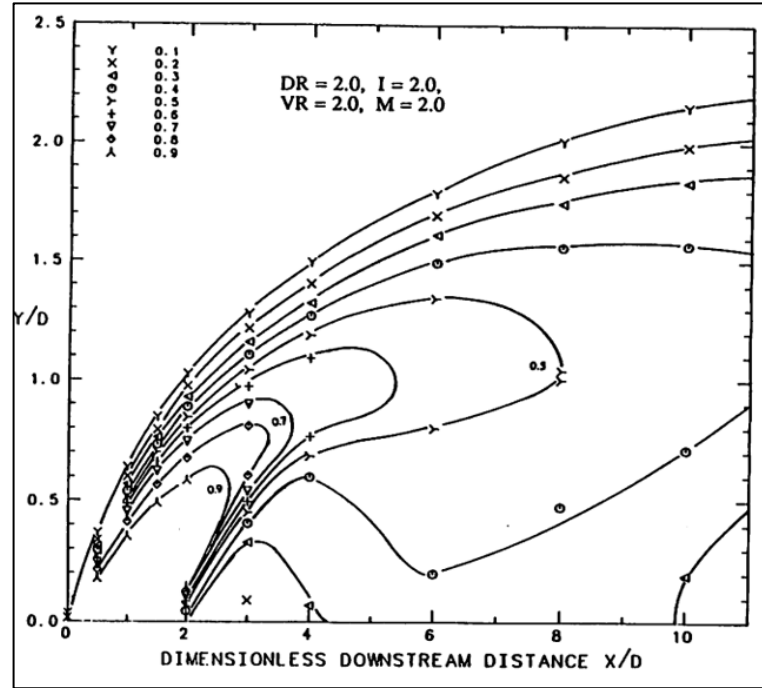


Figure 2.7: Dimensionless Temperature Contours at Mid Plane [10].

The effect of density ratio was studied by Pedersen et al. as first [7]. The proper density ratios by injecting as the coolant are obtained with approaching DR as high as 4.17. Also, the effect of density ratio on film cooling performance was investigated by Schmidt et al. [8] and Wayne and Bogard [9]. All density ratio studies found that a higher DR is contributor for a higher film cooling effectiveness due to the decreasing momentum flux ratio and increasing lateral expansion. The effect of density ratio was found as apparent for detached jets while relatively smaller for attached jets.

Numerous studies were conducted to investigate the effect of film cooling hole geometry on effectiveness. The data get by Wayne and Bogard [9] supported the idea which laterally averaged effectiveness scaled linearly with lateral hole spacing. In that study, 2.78 hole spacing had twice the laterally averaged effectiveness as 5.55 hole spacing. In **Figure 2.8**, laterally averaged effectiveness comparison between these two hole spacing is given. They also studied with compound angle and found that the peak effectiveness decreases after M equals to 0.6 for axial configuration and after M equals to 0.4 for compound angle configuration.

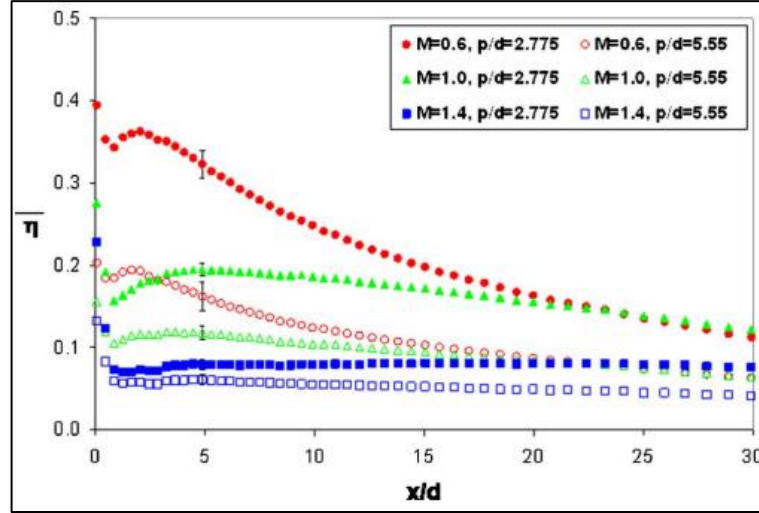


Figure 2.8: $\bar{\eta}$ Comparison for Two Hole Spacing [9].

Compound angle was also studied by Schmidt et al. [8]. In addition to improved lateral expansion of film cooling jets, it significantly improved effectiveness at high blowing ratios for the hole spacing at $P/D=6$. This contribution for $I = 0.98$ and $M = 1.25$ might be seen in **Figure 2.9**.

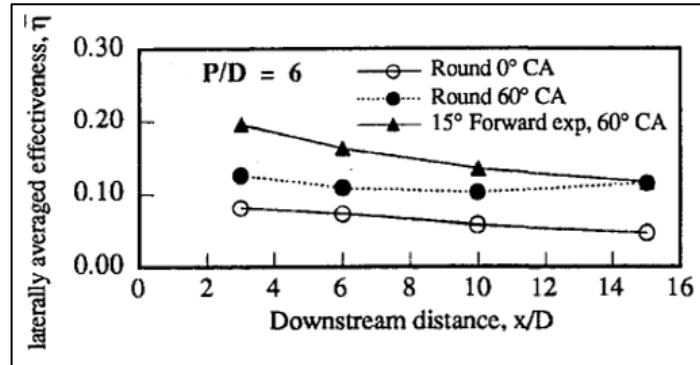


Figure 2.9: $\bar{\eta}$ Comparison for Different Hole Angles [8].

Fewer studies have been conducted to determine the flow behavior for test setup. Detailed investigations were completed on film cooling jets in crossflow by Pietrzyk [10] and Pietrzyk et al. [11,12]. At unit and high density ratio, Laser doppler velocimetry (LDV) was used to study the flowfield in a cylindrical hole by them. Pietrzyk postulated an in-hole separation region that governed many of the turbulence and shear characteristics of the film cooling flow [10]. Separation region was small in the film cooling hole for $VR < 0.5$. In the near wall region of the exit, most of coolant flow momentum was contained. There was a high turbulence region just exit

downstream and the cooling jet was suppressed on the surface. The separation region was larger in hole, when $VR > 0.5$. These small and larger separation region are given in **Figure 2.10** and **Figure 2.11**. The majority of the momentum was contained at the upstream edge location of hole exit for $VR > 0.5$ cases, shown in **Figure 2.11**. High turbulence region and interaction between mainstream and cooling jet occurred at the windward side. For $VR = 0.5$, the poorest shear layers existed and turbulence was minimum. In this study, turbulence is scaled with velocity ratio.

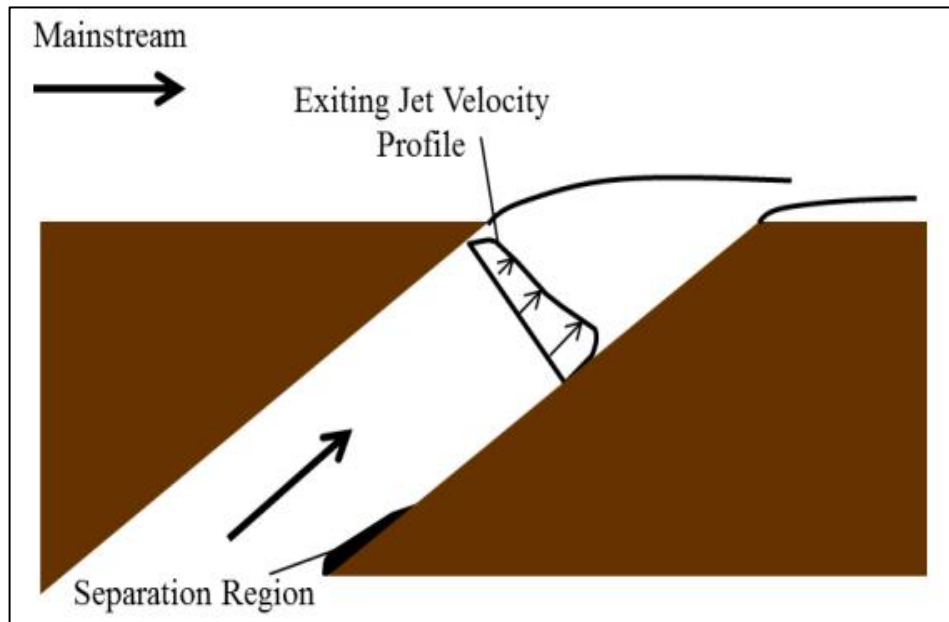


Figure 2.10: Flow Characteristics for $VR < 0.5$ Case in the Cooling Hole [12].

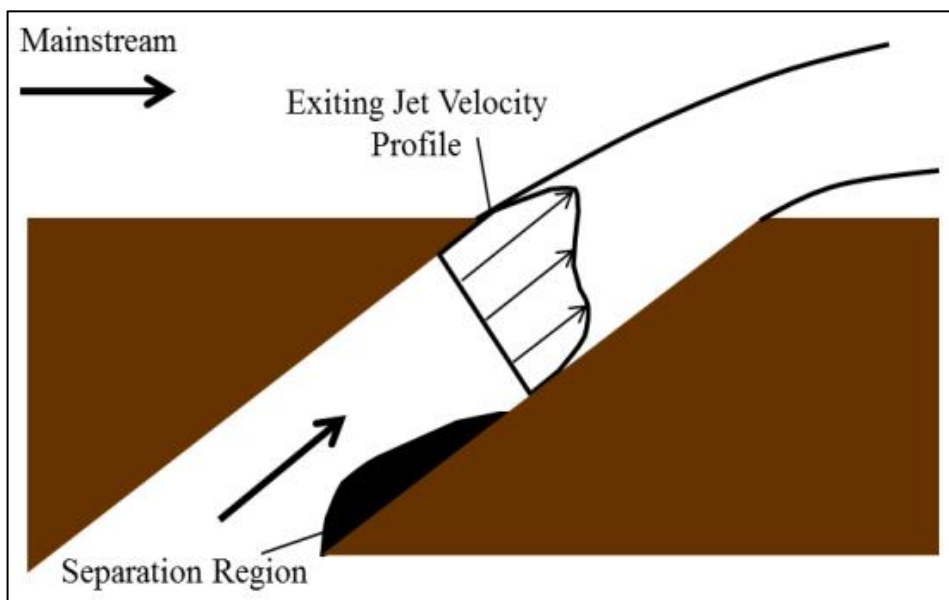


Figure 2.11: Flow Characteristics for $VR > 0.5$ Case in the Cooling Hole [12].

Pietrzyk et al. [11] found that higher jet penetration into the mainstream, higher blowing ratios. Also, the velocity profiles of both high and low density ratios were the same.

Time-resolved unsteady studies have been performed on jets in crossflow. They indicated that unsteady flow characteristics which become exist in turbulent jet flows are important. Moussa et al. [13] found characterization of jet injection at 90 degree into crossflow. They used temporal hot wire anemometry. They discovered that a jet injected into crossflow existed with wake vortices as the vortex shedding off the back side of jet injection.

Fric and Roshko [14] defined four dominant fluid structures. These are shear-layer vortices, horseshoe vortices, a counter-rotating vortex pair (CRVP) and wake vortices, as shown in **Figure 2.12**. The wake vortices have been mentioned by Moussa et al. But, Fric and Roshko studied by means of time-resolved flow visualization and they reported that the wake vortices occurred in the boundary layer and connected to the jet with periodic separation events.

Haven and Kurosaka [15] studied the measurement of both ‘kidney vortices’ (CRVP) and ‘anti-kidney vortices’ in a shear layer. The Laser-induced florescence technique was used for unsteady measurements. They indicated that the anti-kidney pairs rotate in the opposite direction to the counter rotating kidney pairs. They existed more abundantly in the high blowing ratio cases than in the low blowing ratio cases and had transient behavior.

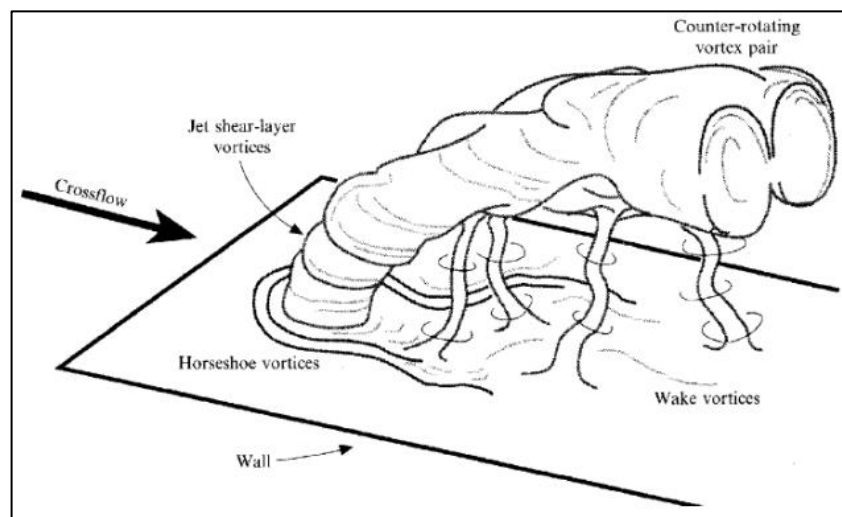


Figure 2.12: Schematic Presenting Four Types of Vortices Structure [14].

Particle image velocimetry (PIV) is one of the recently used common techniques to measure flowfields. It is preferred due to its chance to get two dimensional spatial map unlike point measurement techniques such as LDV and hot wire anemometry. Kampe et al. [16] studied the effects of diffused shape hole by means of PIV. The diffused shape was compared to a cylindrical hole. CRVP was the dominant structures at downstream of cylindrical hole. But, the familiar CRVP was not dominating for the coolant from diffused hole. Instead of this, a lateral expansion of the jet was dominant. Also, they observed asymmetric flow behavior for both test and numerical results. Although a reason for the asymmetric distribution has not yet been discovered until this study, it was thought to be related to the unsteady form of flow behavior. This asymmetric flow observation can be seen for $DR = 1.1$, $Tu = 7\%$ and $M = 2$ conditions in **Figure 2.13**.

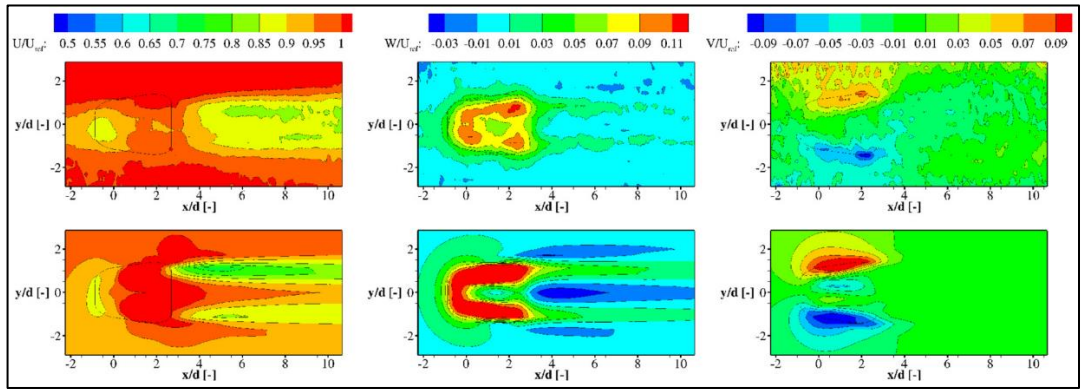


Figure 2.13: Comparison of Test (top) and CFD (below) Velocity Distributions [16].

X, Y and Z components represent streamwise, wall normal and lateral velocities respectively. Higher velocities for the computational results were taken in comparison to the test results. The streamwise velocity showed a light deceleration just upstream location of the hole exit. As the common literature knowledge, the wall normal component had a pronounced jet bifurcation with a recirculation region at the middle region of the diffuser.

Time-averaged stereoscopic PIV method was used by Meyer et al. [17] for different blowing and density ratios. This method consist of two high speed cameras which looking at the same flow section from two separate angles. Adding the combination of two different angles and calibrated images, three dimensional measurement was

obtained. Bernsdorf et al. [18] also utilized this PIV method to get measurement of jet in crossflow. They observed that when blowing ratio, density ratio and hole angle increased, the new boundary layer was thinned. They also had measurement for behavior of CRVP at varying DR and M. The higher M, the higher axial velocity of CRVP. The centerline of the CRVP was closer to the surface at higher DR and this finding makes the results logical with the CRVP data showed in Pietrzyk [10].

Fawcett et al. [19] specified unsteady flow structures in an engine nozzle guide vane geometry. This identification has been studied with time resolved flow visualization and time averaged PIV at DR = 1. Shear layer vortices and CRVP was focused. A streamwise and spanwise flow plane measurement were taken. In this way, different flow phenomena on each measurement plane have been captured. At high blowing ratio, they observed periodic shear layer vortices in the streamwise plane. Also, they could capture characteristics of the CRVP which more height with higher blowing ratio. An anti-kidney vortices at high blowing ratio existed transiently, as mentioned by Haven et al. [15].

Time-averaged and time-resolved particle image velocimetry technique has been used by Thole and Eberly [24, 27]. They provided datas which belong to low and high density ratios. Flat plate adiabatic effectiveness and flowfield measurement have been conducted at high DR. The facility utilized a heated mainstream and cryogenic cooling in order to get high DR jet injection. Time-resolved PIV data have recently been gathered, but only for low DR measurements. They created experimental setup to provide steady and unsteady data on realistic shaped film cooling geometries for the gas turbine community. By means of test facility, time-resolved studies are conducted at high density ratios.

Baseline shaped hole is studied by Thole and Schroeder [21]. This baseline film cooling hole geometry is known as 7-7-7 fan shaped hole. It has higher and improved cooling performance than cylindrical hole. They aimed to offer the community a shaped hole design which is representative of proprietary. They have presented comparison with open literature data.

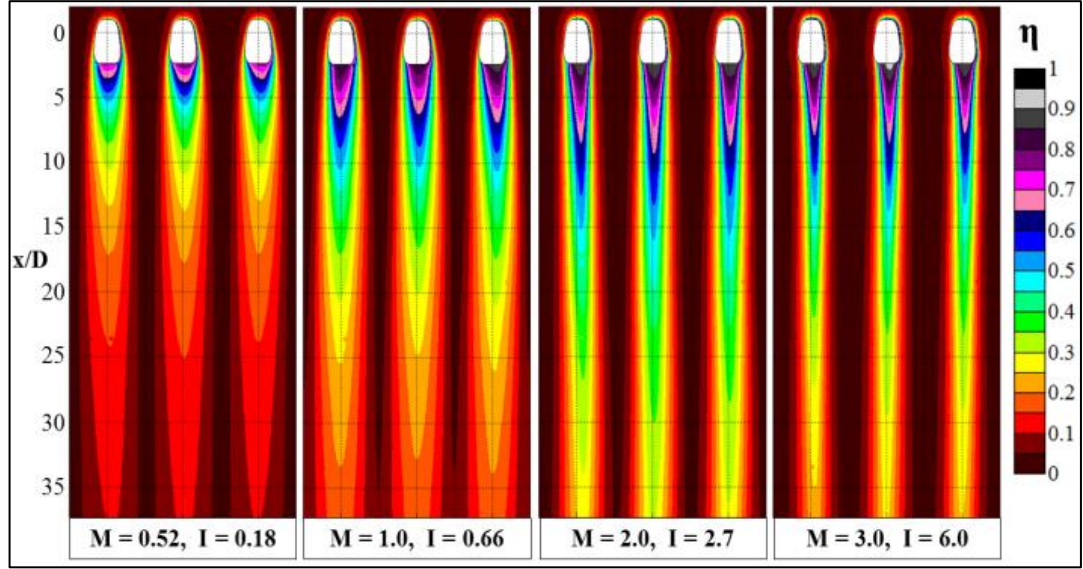


Figure 2.14: η_{film} Contours for Different M, DR = 1.5, $Tu_{\infty} = 0.5\%$ [22].

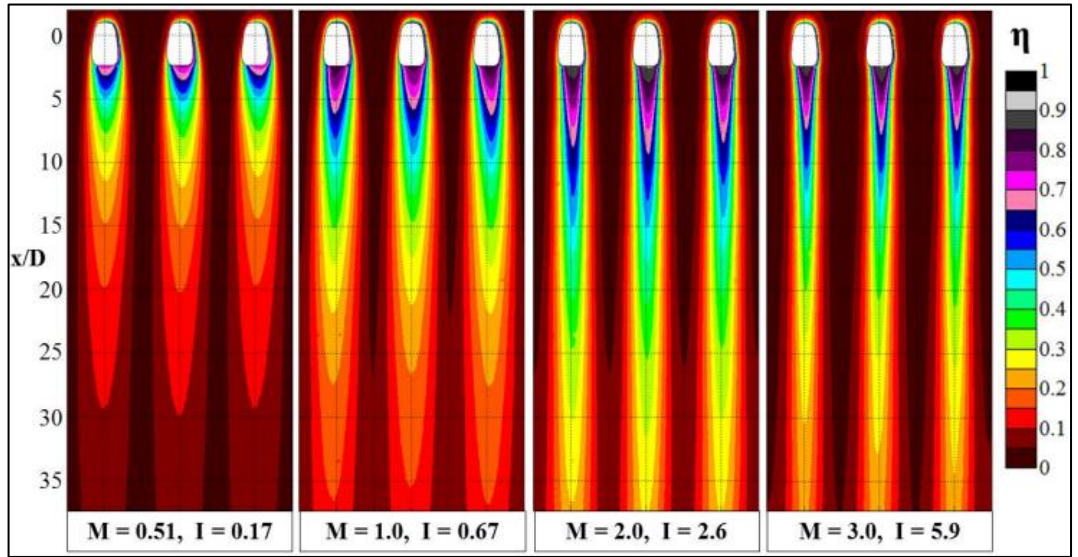


Figure 2.15: η_{film} Contours for Different M, DR = 1.5, $Tu_{\infty} = 5.6\%$ [22].

Figure 2.14 and **Figure 2.15** shows measured effectiveness for low and high freestream turbulence at high density ratio as the same DR = 1.5. Both two figures showed the same adiabatic effectiveness behavior. As it is realized that in figures, effectiveness firstly increased. And then it starts to decrease with increasing blowing ratio. This decreasing was also observed with the narrowing jet at the middle of the hole.

The main difference between low and high freestream turbulence cases was mixing level between coolant and mainstream. The higher mainflow turbulence has caused

higher mixing between the mainflow and the coolant jet. It helped to decrease film effectiveness earlier axial location. As a result, the effect of cooling which means adiabatic wall effectiveness has been lost earlier for higher freestream turbulence.

Also, they have had laterally averaged effectiveness distributions along downstream distance of the shaped hole for different cases. They have aimed to offer the gas turbine industry a shaped hole design for comparison goal. Two of these distributions can be seen in **Figure 2.16** and in **Figure 2.17**. At the immediate vicinity of the hole exit, data was not gathered. For this point at $x/D = 2.5$ location, data were extrapolated. Therefore, there is no any comparison between selected validation cases and thesis studies for this point.

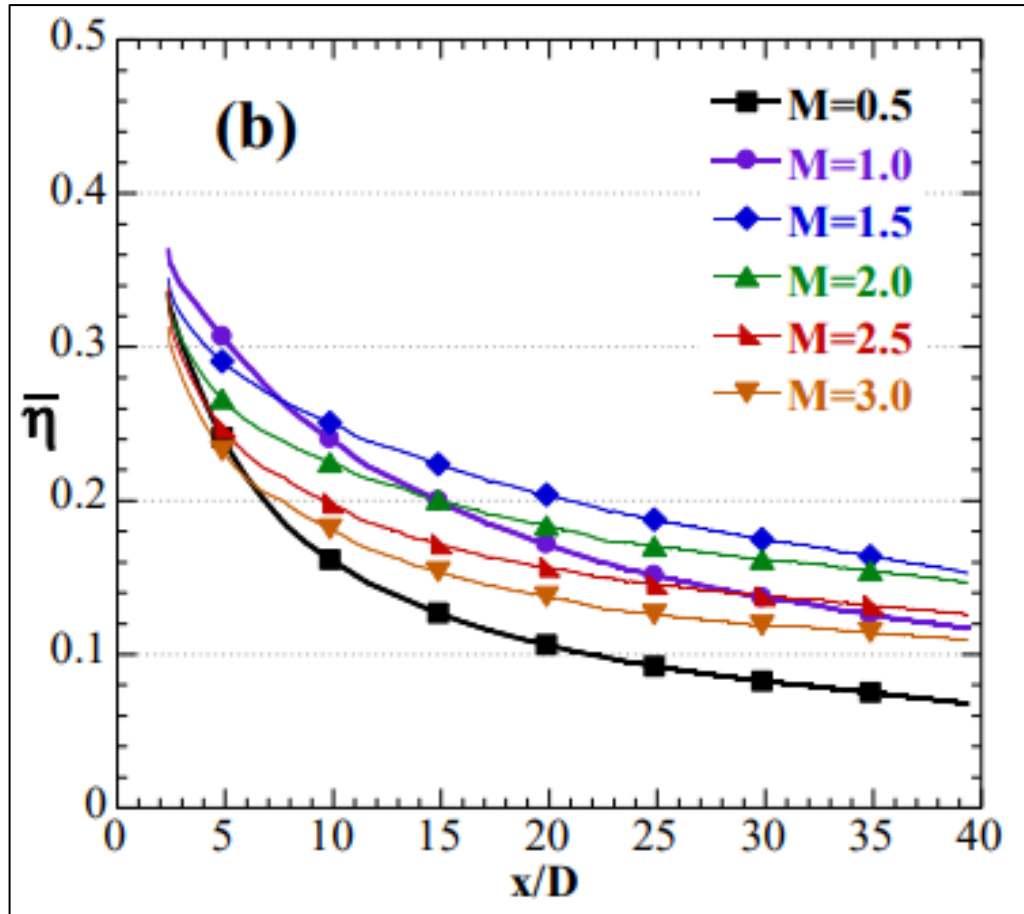


Figure 2.16: $\bar{\eta}$ of Shaped Hole for low stream turbulence at DR = 1.5 [22].

It belongs to low stream turbulence and DR = 1.5 case. They have proved that lateral spreading of coolant jet was highly depend on the blowing ratio. Due to the narrowing

jet spreading as it can be seen in effectiveness contours, the downstream lateral effectiveness decayed at blowing ratios above 1.5.

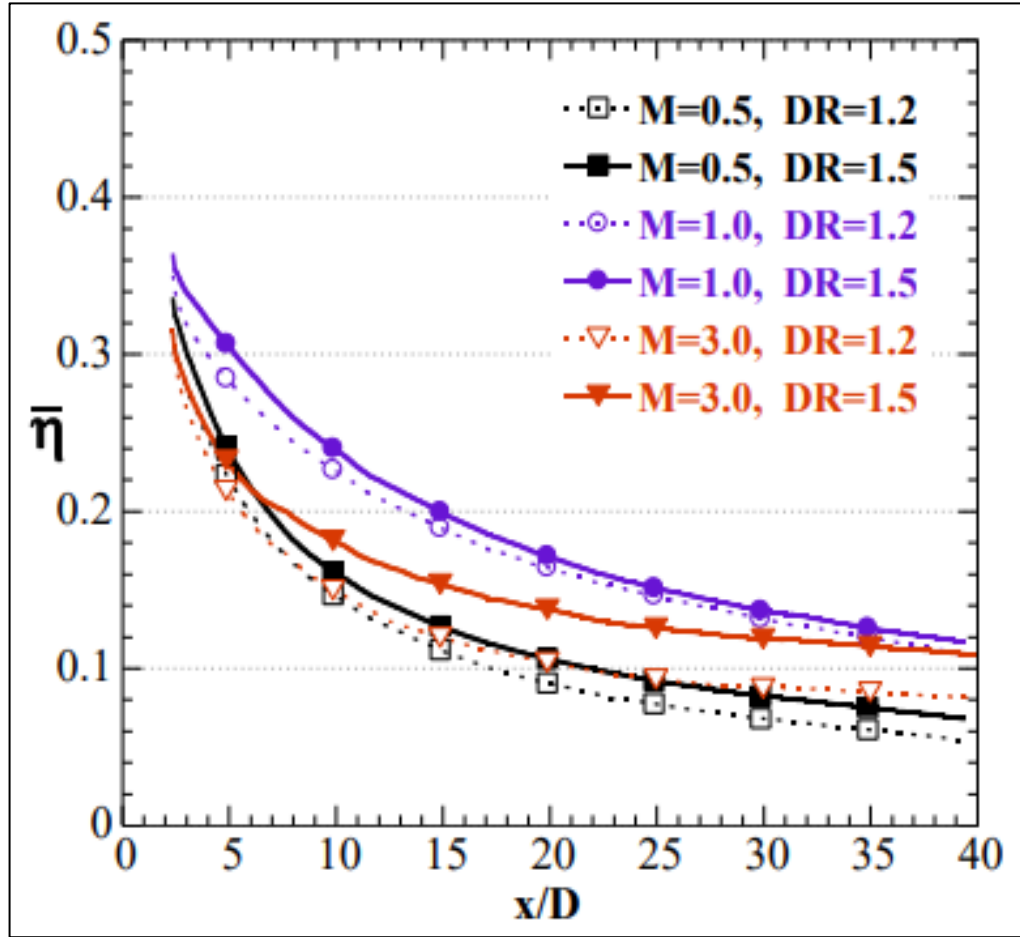


Figure 2.17: Comparison of $\bar{\eta}$ at DR = 1.5 and DR = 1.2, $Tu_\infty = 0.5\%$ [22].

The main difference between low and high freestream turbulence cases was mixing level between coolant and mainstream. The higher mainflow turbulence has caused higher mixing between the mainflow and the coolant jet. It helped to decrease film effectiveness earlier axial location. As a result, the effect of cooling which means adiabatic wall effectiveness has been lost for higher freestream turbulence.

Thole and Schroeder also studied effect of roughness in hole on adiabatic effectiveness [22]. Many studies consist of different factor as influencer on shaped hole performance. But, no studies have investigated how roughness along interior walls of the shaped hole affects cooling performance.

Three different cases which were studied by Thole and Schroeder are selected to utilize in this thesis scope. They are given with references as 25, 26 and 27. They are belong to same test facility and similar test conditions. Also, they utilized the baseline fan shaped geometry in terms of film cooling. But, they have diverse goals separately as explained above. Test facility details will be described below in Chapter 4. But, extensive ranges, unique applications, proper geometries for gas turbine and common benchmarking are involved to the facility. This is the reason why test facility given below is selected for this thesis.

3. BASELINE SHAPED HOLE

3.1. Why Called as Baseline Shaped Hole

In the film cooling literature, there is no standard shaped cooling hole as general. Moreover, there are a large variety in hole shapes and conflicting trends in the results.

The author selects the laidback fan shaped hole as a baseline geometry. Because, it is dominant in literature studies and has common usage in gas turbine industry. The ranges for geometric parameters commonly existed in the literature for laidback fan shaped hole are presented at **Table 3.1**.

Table 3.1: Geometric Parameters of Baseline Shaped Hole [22].

	Range Common in Literature	7-7-7 Shaped Hole
Injection Angle, α	30 to 55°	30°
L_m/D	1 to 4	2.5
L_{lat}/D , L_{fwd}/D	1.6 to 9.5	3.5
L/D	2.8 to 11.5	6
Laidback Angle, β_{fwd}	2 to 25°	7°
Lateral Angle, β_{lat}	2 to 18°	7°
P/D	2.8 to 8	6
Coverage Ratio, t/P	0.3 to 0.8	0.35
Area Ratio, AR	2.5 to 4.7	2.5
Sharpness of Inlet and Breakout Edges	Usually Sharp	Sharp
Rounding of Four Edges Inside Diffuser, R/D	0 to 0.5	0.5

In terms of expansion angle, 7 degree is accepted as baseline in each direction. Because, 7-7-7 shaped hole helps get the baseline shaped hole effectiveness performance which has comparison to the common literature test data. This investigation is detailly mentioned by Wittig et al. [24]. Also, the forward expansion of 7-7-7 hole starts well towards the inlet of the hole which provides forward diffusion prior to injection with the mainflow.

3.2. Design of the Baseline Shaped Hole

In terms of expansion angle, 7 degree is accepted as baseline in each direction. Because, higher expansion angle causes bigger jet separation in hole. Baseline design allows numerical model comparison with test data due to easier flow behavior.

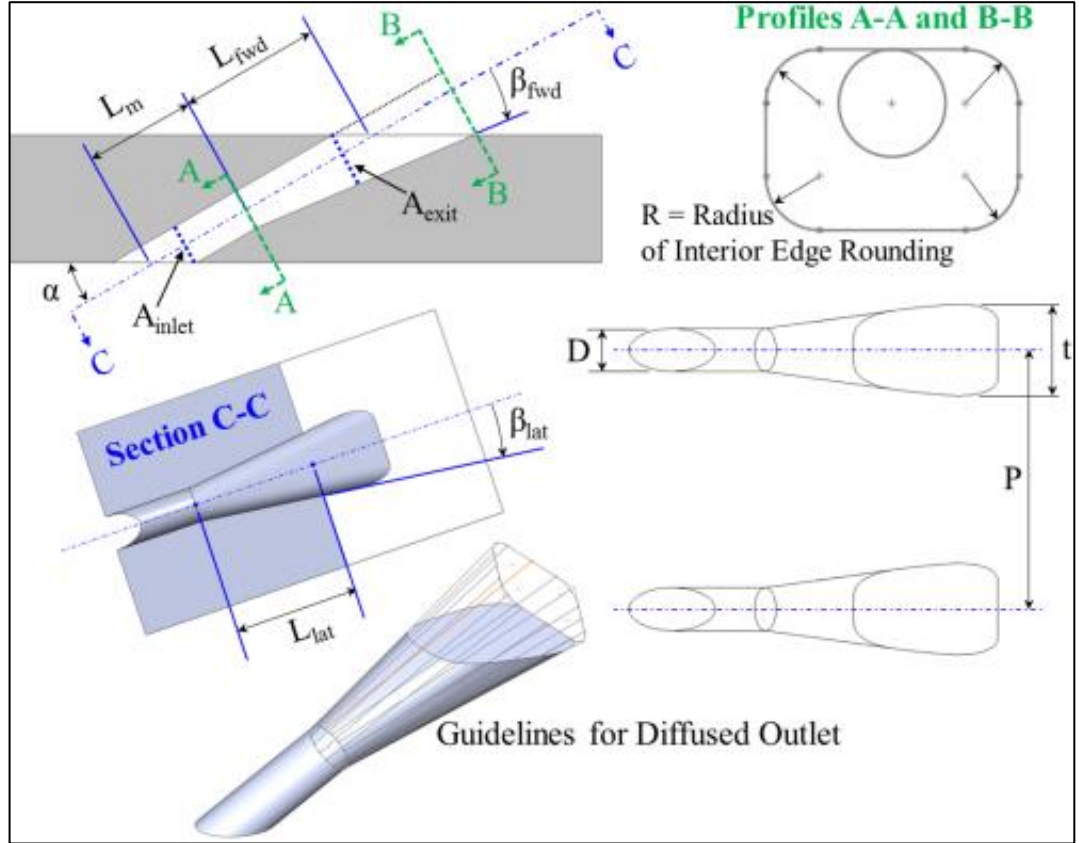


Figure 3.1: Baseline of Fan Shaped Hole Design [22].

As a simple design, cylindrical holes consist of constant hole diameter D , injection angle α to the surface, hole length, pitchwise distance P and pitchwise spacing P/D and orientation with respect to the mainflow direction as compound angle. These parameters are used to explain design of shaped hole with a few additional parameters. The design parameters and exact geometry are illustrated in **Figure 3.1**. Shaped hole has diffused outlet, expansion angle β in the forward or/and lateral directions, specified length after metering area L_m . These design parameters help to describe common specific values of the exit-to-inlet area ratio AR , the coverage ratio t/P and the effective injection angle which is given by the angle between the external surface and the diffuser floor of the hole γ ($\alpha - \beta_{fwd}$).

As illustrated at the bottom of **Figure 3.1**, the diffused outlet shape is guided by lines from the end of the throat section at plane A-A. And this guidelines are filleted until the rectangle of plane B-B. Lateral expansion is started from ending metering section as shown in **Figure 3.1**. The starting of laidback expansion angle may change for different purposes. But, the starting of lateral and laidback expansion angle is started from the same point as the end of metering section in given figure for baseline shaped hole.

The shaped hole diffused outlet helps to get higher adiabatic effectiveness than cylindrical holes at high blowing ratios. As mentioned above, cylindrical holes have detachment from cooling surface at high blowing ratios. The shaped holes have an exit to inlet ratio bigger than one. So, jet velocity at the hole exit is lower than at the hole inlet. Lower momentum of coolant jets reduces jet penetration into the mainflow and it helps to prevent jet detachment. Also, diffused outlets present a beneficial spreading of coolant over the surface with lateral expansion.

$$\bar{\eta}(X/D) = \frac{1}{Z} \int_{-Z}^Z \eta(X/D, Z/D) d(Z/D) \quad (3.1)$$

A new type adiabatic effectiveness term shows up itself as laterally averaged effectiveness. It is given by Equation 3.1.

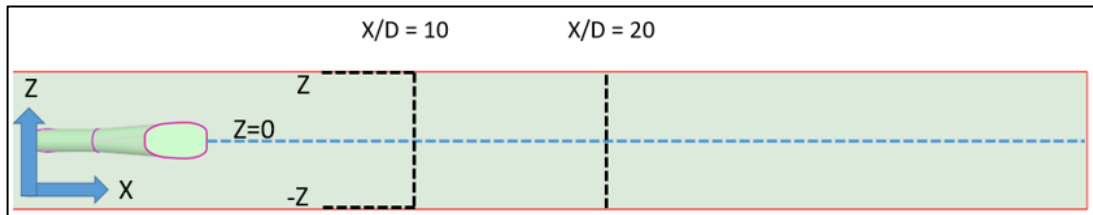


Figure 3.2: Representation of X and Z Direction for Effectiveness Formula.

Laterally averaged film effectiveness term depends on centerline and lateral effectiveness distribution at the same time. As it is shown in **Figure 3.2**, the symbols X and Z represent the specific downstream and lateral location of film cooling hole. For shaped holes, all literature studies are compared with the others by means of this equation.

For a CFD validation of shaped hole with test results, test facility and geometric parameters are selected the same with Thole and Schroeder [22]. Briefly, injection angle is 30°. The lateral and laidback expansion angle in each direction is 7° for shaped

hole validation cases in this thesis. The length of the metering section L_m is selected as $2.5D$ and the length of the expansion section equals to $3.5D$. The selected hole length and 7° expansion angles concluded with an area ratio $AR = 2.5$.

After the CFD validation, effects of expansion angles and coolant velocity in feeding channel will be examined with created different numerical models.

4. TEST FACILITY

Facility aims to collect comparison data with open literature for shaped hole, time-resolved and time-averaged data at high and low density ratios. Cylindrical and fan shaped holes can be tested in this facility because of removable test plate. Also, it is focused that effect of roughness in shaped hole, as explained above.

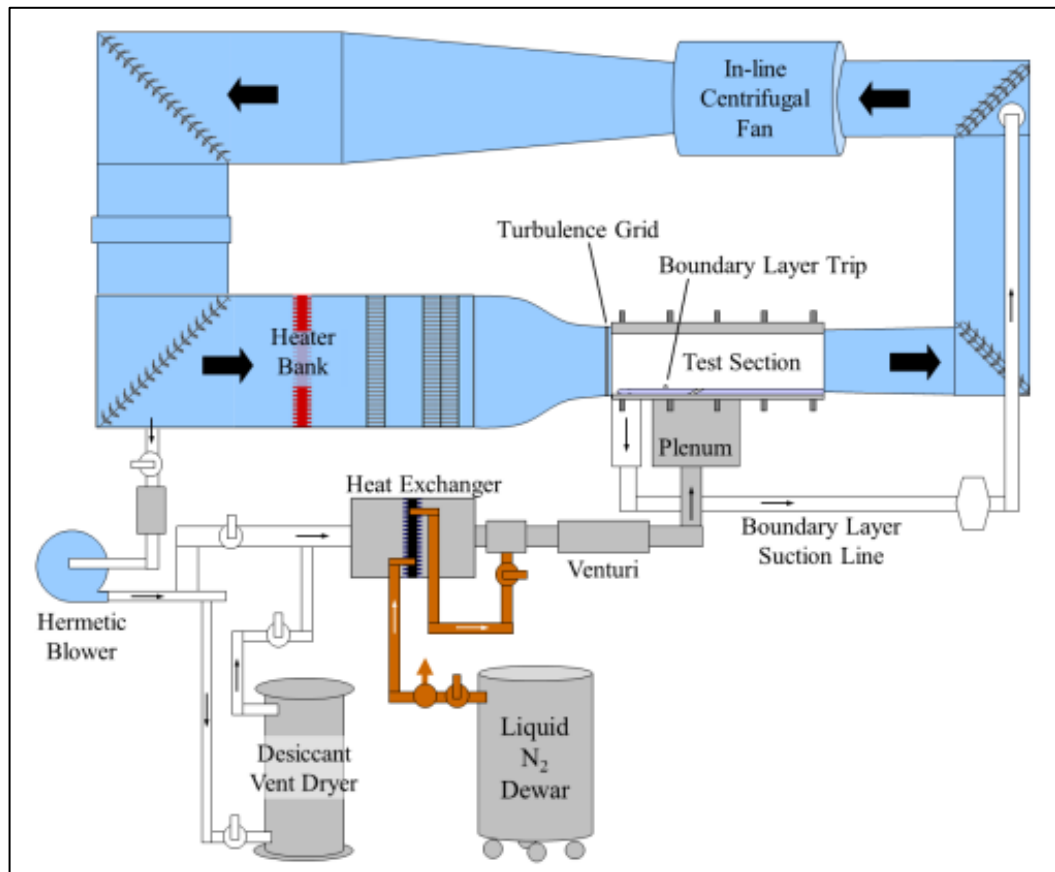


Figure 4.1: Schematic View of Selected Test Rig [21].

The facility was previously detailed by Eberly and Thole [23]. Schematic of heat transfer with the nitrogen cooling system and the entire flowfield is shown in **Figure 4.1**.

The test facility has two separate flow loops. First one is a mainstream loop which owns the mainflow of the tunnel and the other one is a coolant loop that housed the injection flow. Both of these loops can be set up with different facility conditions in

order to cover various purposes. The facility is developed with respect to outlined in Pietrzyk et al. [11].

Closed loop and low velocity wind tunnel is used in order to get flowfield behavior and adiabatic effectiveness measurements. For the present figure, in-line centrifugal fan is used to provide flow which is maintained at 295 K. A trip wire is located at $x/D = -33$ and initiates transition of the boundary layer to the turbulent phase.

A hermetically sealed variable frequency blower is used to remove coolant air for the film cooling injection from the mainstream. The coolant is directed by means of a vent drier that included solid desiccant in order to prevent frost formation that may occur from cryogenically chilling the coolant. A venturi flowmeter is used to measure the coolant flowrate at the downstream of the heat exchanger. This measurement is made before the coolant enters the plenum. Three flow conditioning screens are used to ensure uniform flow behavior to the cooling holes inside the plenum.

Film cooling geometry is made of Styrofoam which has 0.029 W/mK conductivity. In this way, a nearly adiabatic temperature for effectiveness calculation is measured on surface. The film cooling array is a row of five shaped holes and holes are used with 7.75 mm metering diameter. Also, parameters of baseline shaped hole validation geometry are summarized at **Table 4.1**.

Table 4.1: Test Plate Geometry.

Geometry	P/D	α	β_{fwd}, β_{lat}	AR	L_m/D	$L_{lat}/D, L_{fwd}/D$
Baseline Fan Shaped	6	30°	7°	2.5	2.5	3.5

FLIR SC620 infrared camera is utilized to measure the surface temperature distribution and effectiveness is determined with these measurements. Also, coolant flow and mainstream temperatures are separately measured by means of multiple thermocouples. The camera output is calibrated to precisely detect the whole range of temperatures. In this facility, adiabatic effectiveness measurements reported by Eberly and Thole for cylindrical and fan shaped holes presented good agreement with the literature [20].

Freestream turbulence intensity is reported nearly upstream of the hole. Blowing ratio and injection Reynolds number are determined at the throat section of the hole. The mainstream velocity is maintained 10 m/s for all test conditions.

5. COMPUTATIONAL MODEL

The computational model is handled with a single fan shaped cooling hole instead of an array due to limited computational resources. The geometry is created using the common rendering tool SpaceClaim.

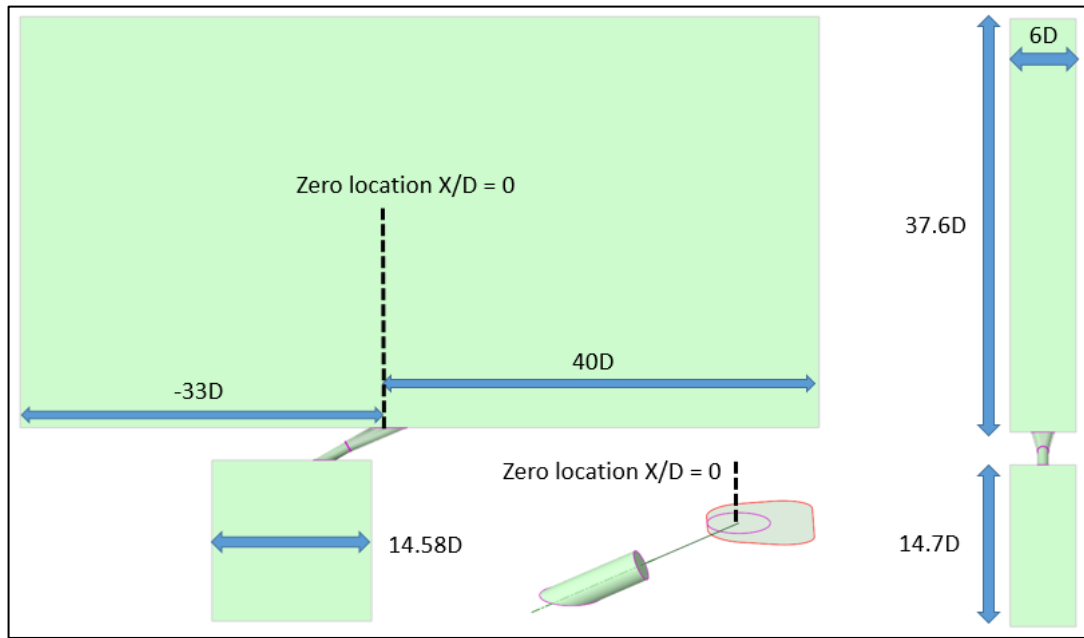


Figure 5.1: View and Dimensions of Computational Model.

The geometry dimensions are given in **Figure 5.1**. The hole is located at the center of the plenum, the width of inlet and plenum is defined as $6D$ because of single row approach. Height of main fluid domain is created as $37.6D$ and it is about actual test bench. The mainflow inlet is at $-33D$ with respect to hole origin location in order to get proper boundary layer behavior and thickness which are given at selected validation case for low turbulent intensity. The exit of the main fluid domain is extended to $40D$ downstream of origin. Coolant plenum is located at the bottom side of the hole and has the same thickness with the mainstream channel.

Domain is discretized to four subdomains and topologies are created. There are two different purposes with this step. First one is the easier meshing process at the inlet and exit of the sharpened cooling hole. The other one is the reporting non-dimensional

design parameters at the metering section. For this purpose, cooling hole is divided into two different domains at the end of the cylindrical section as illustrated in **Figure 5.2**.

Using ANSYS Fluent Mesher, mesh is created as an unstructured with polyhedral fillet option. Body of influence and local face mesh sizing options as size control type are used to capture the flow physics well. Body of influence BOI is known as volumetric refinement region. These volumetric refinement regions given at **Figure 5.2** are optimally determined while sensitivity studies and post-processing. Uniform boundary layer method is used and height of first cell layer is assigned as 0.01 mm which results Y^+ on a wall as highest 1.

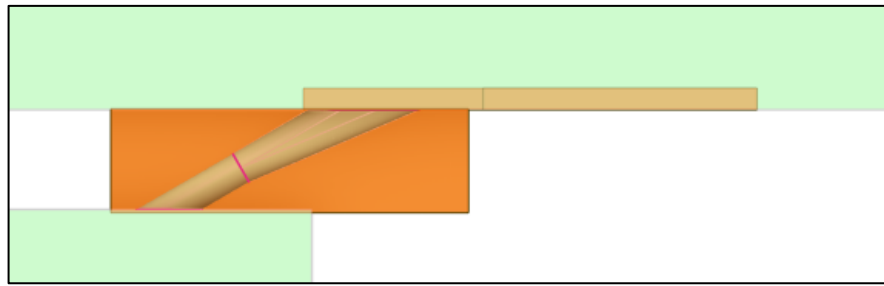


Figure 5.2: View of Body of Influence Domains.

The mixing between mainflow and coolant injection is aimed to accurately capture with defining first refinement region downstream of the hole. The second refinement region is used for hole itself in order to get more accurate flow behavior and separation region in the hole if there is.

Table 5.1: Mesh Properties of Sensitivity Studies.

Geometry	Name	BOI	y^+	Cell Count
7 7 7	MESH1	No	1	0.6 million
7 7 7	MESH2	No	1	1.2 million
7 7 7	MESH3	No	1	4.3 million
7 7 7	MESH4	Yes (1mm)	1	4.4 million
7 7 7	MESH5	Yes (0.4mm)	1	5 million
7 7 7	MESH6	Yes (0.2mm)	1	10 million

Six mesh options are created and investigated for mesh sensitivity studies and these options can be seen at **Table 5.1**. 7-7-7 baseline geometry is used for all sensitivity

cases. Three of them MESH1, MESH2 and MESH3 are about different local mesh sizing options. In this way, it is aimed to have effects of coarse and medium mesh distributions. Especially, coolant surface of mainstream and hole surfaces which are highlighted in **Figure 5.3** are determined in terms of local minimum mesh sizing. Because, these surfaces are important in terms of flow behavior in the hole and film effectiveness distribution. The other cases MESH4, MESH5 and MESH6 have different target boi size with respect to **Table 5.1** in order to investigate differences due to size of boi. As it is mentioned above, boi regions are determined in order to capture the coolant mainstream mixing and flow structure in cooling hole. As a result, 0.6 and 10 million mesh is created for coarse and fine situation respectively.

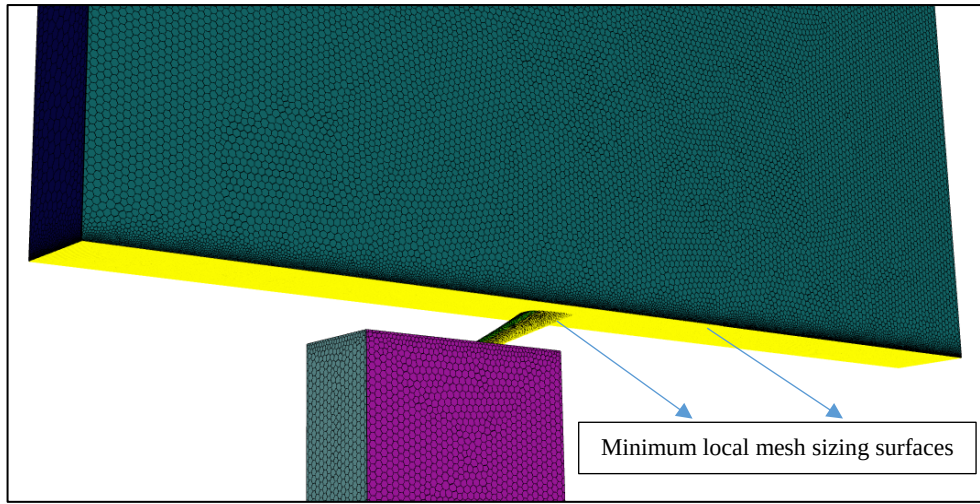


Figure 5.3: View of Minimum Local Mesh Sizing Surfaces.

20 % error between test data and numerical results of validation cases are determined in terms of maximum deviation. Because, selected literature test rig [27] reports that the maximum uncertainty is found as $\eta = \pm 0.025$ at high density ratio case. With respect to **Figure 2.16**, the percentage of maximum deviation is calculated as 10 % deviation for 0.31 value at $X/D = 5$ location. The computational solver might have its own deviation due to the numerical restriction of unsteady flow structure and it is assumed as 10 %. Therefore acceptable deviation between test and numerical results is assumed as maximum 20 % considering test uncertainty is 10 % for itself.

Table 5.2: The Effect of Deviation Assumption on Part Metal Temperature.

Modern Gas Turbine			
$\bar{\eta}$	0.1	0.2	0.3
T4 _{ref} [K]	1.44	1.44	1.44
T3 _{ref} [K]	0.63	0.63	0.63
% Error	20	20	20
T _{aw} [K]	1.36	1.28	1.20
Lower $\bar{\eta}$	0.08	0.16	0.24
Higher T _{aw} [K]	1.37	1.31	1.24
Higher $\bar{\eta}$	0.12	0.24	0.36
Lower T _{aw} [K]	1.34	1.24	1.15
$\pm T_w$ [K]	7	14	22

Table 5.2 summarizes that the effect of assumed thesis maximum deviation on wall metal temperature for different lateral film effectiveness with respect to modern gas turbine conditions. One dimensional in house code is utilized in order to calculate the wall temperatures. The minimum and maximum metal temperature deviations are found as 7 K and 22 K for lowest and highest film effectiveness respectively. Considering the heat balance solution of in house code, the chordwise and spanwise conduction is neglected. Therefore, these wall temperature deviations will decrease by means of the chordwise and spanwise conductions. In addition, the earlier X/D distance which has the highest effectiveness value and the highest metal temperature error as 22 K is not a stress critical location in terms of component life. Because, the internal impingement insert which is located to backward side of film cooling hole helps to cool metal temperature down. With respect to these knowledge, the wall temperature deviations given in **Table 5.2** can be assumed as acceptable. As a result, the maximum deviation between test data and numerical results are determined as 20 %.

At the end of the sensitivity studies, MESH5 option is found as proper and utilized for all validation and thesis studies. MESH5 mesh case has target mesh size in terms of face and BOI regions. Results of sensitivity studies will be given in Section 6. In **Figure 5.4** and **Figure 5.5**, general and detailed polyhedral mesh views which are belong to MESH5 case can be seen.

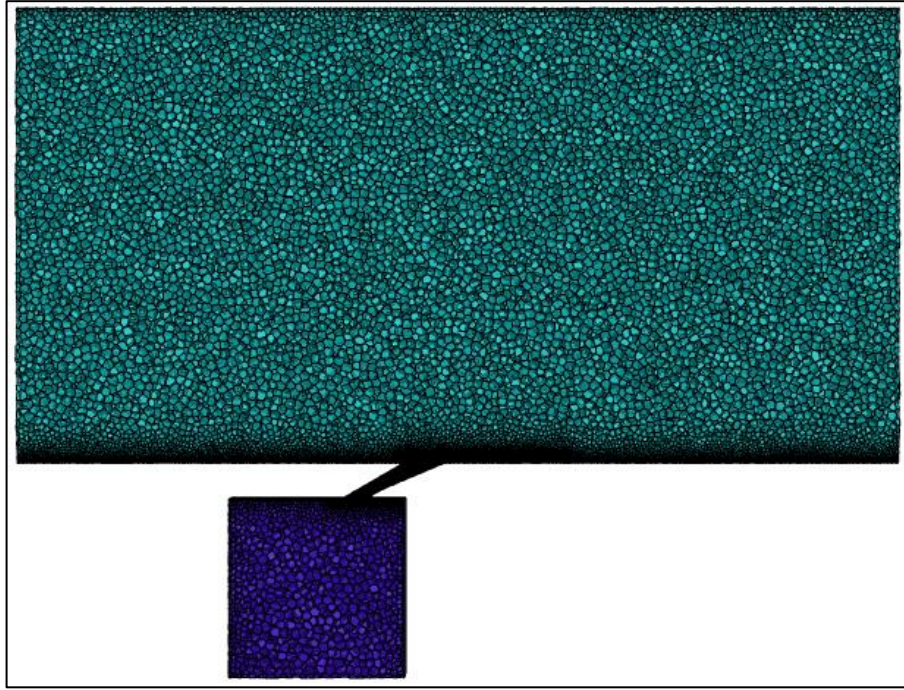


Figure 5.4: Computational Models with Polyhedral Mesh Distribution.

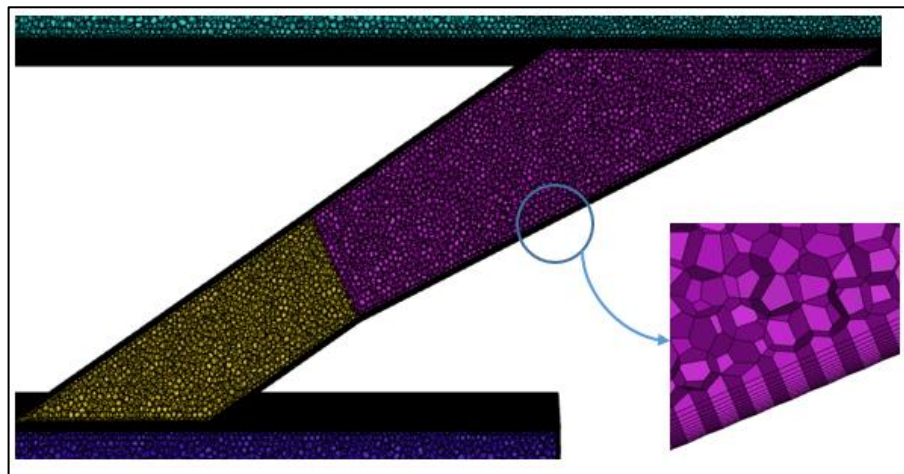


Figure 5.5: Detailed Mesh Image of Hole Region.

In **Figure 5.6**, the detailed mesh image of boi regions are shown below. Refined regions which are selected to capture flow behavior in hole and mixing between mainstream and coolant can be easily noticed in figure. As it can be realized in below figure, earlier X/D location is selected for the end of boi domain in terms of mixing region. Because, the close results are observed between earlier and longer X/D location domains. Considering consumption of source and time, earlier X/D location is determined with detailed mesh studies.

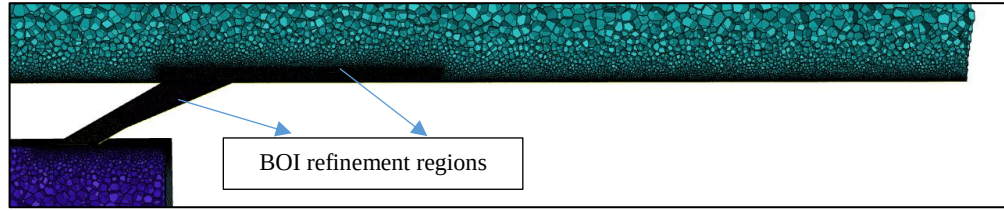


Figure 5.6: Detailed Mesh Image of BOI Regions.

After the determination of proper mesh and turbulence model options, validation cases are set up to validate CFD results with selected test rig data. The test rig has 7-7-7 fan shaped hole and the measurements for different conditions has been provided by Thole and Schroeder [22]. The Fluent is used as a solver for computing of validation cases. They have different density ratio, mainstream turbulent intensity and blowing ratio. Also, these variables are changed with respect to test data as given at **Table 5.3**. In this way, blowing and density ratios as main design parameters are covered for the validation purposes. Baseline fan shaped hole details have already been presented as it can be seen at **Table 4.1**.

Table 5.3: Validation Matrix for Baseline Shaped Hole.

Geometry	$Tu_{\infty,m}$	DR	M	Re_D
7 7 7	0.5%	1.2	0.5, 1	2800-7000
7 7 7	0.5%	1.5	0.5, 1, 1.5	3400-12500
7 7 7	5.6%	1.5	0.5, 1	3400-8200

Boundary conditions and solution options are set to solve for steady RANS model. The turbulence model is chosen as Realizable K-Epsilon and enhanced wall treatment is switched on with low Y^+ mesh grid. The term realizable means that the turbulence model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. A main benefit of the realizable $k-\epsilon$ model is that the spreading rate of round jets is predicted more accurately. Also, it helps to provide excellent performance for boundary layer under strong negative pressure gradients, separation and recirculation.

Solution parameters which are selected with respect to the best solution results and convergence residuals are given at **Table 5.4**. While RKE is suggested for this type of

study because of jets and recirculation region, the other turbulence models are used to investigate their effects on lateral adiabatic effectiveness. Results of the other turbulence models will be given in Section 6.

Air properties specific heat, conductivity and viscosity are defined as a function of static temperature, only molecular weight of air which will not be dramatically effect the results remains constant. Functions of air properties are exported from NUMECA software and utilized in Fluent with respect to temperature.

Table 5.4: Solution Parameters of Numerical Models.

Parameter	Type or Value
Solution scheme	Steady
Solution type	RANS
Model	RKE
Wall function	Enhanced wall treatment
Equation of state	Ideal gas
Max Y^+	1
Scheme	Coupled
Fluid Properties	Polynomial

3D numerical solver geometry of validation cases can be seen at **Figure 5.7**. Red and green domains represent coolant and mainflow respectively. Also, black arrows can be observed for inlet and exit boundary conditions with respect to given conditions. Different mass flow rates and temperatures for inlet coolant are adjusted in order to evaluate proper blowing ratios which belong to selected validation cases. Mainflow conditions are kept constant as uniform velocity inlet and pressure outlet conditions, and side walls of fluid domains are defined as periodic. Mainflow exit is defined as opening to the atmosphere. Also, the given geometry in **Figure 5.7** which have 7-7-7 fan shaped cooling hole is used for mesh sensitivity and effects of different turbulence solver cases.

For fluid type, the ideal gas is used with the variable thermophysical properties respect to temperature. For the test and numerical comparison, experimental velocity 10 m/s

and experimental temperature 295 K with average low turbulent intensity are defined as velocity inlet boundary condition. Mass flow inlet for coolant feeding as the plenum condition is adjusted with equivalent M and DR of test condition in order to get correct comparison between numerical and test cases. As a result, coolant flow will have very low velocity at the inlet as plenum because of huge normal area to flow direction.

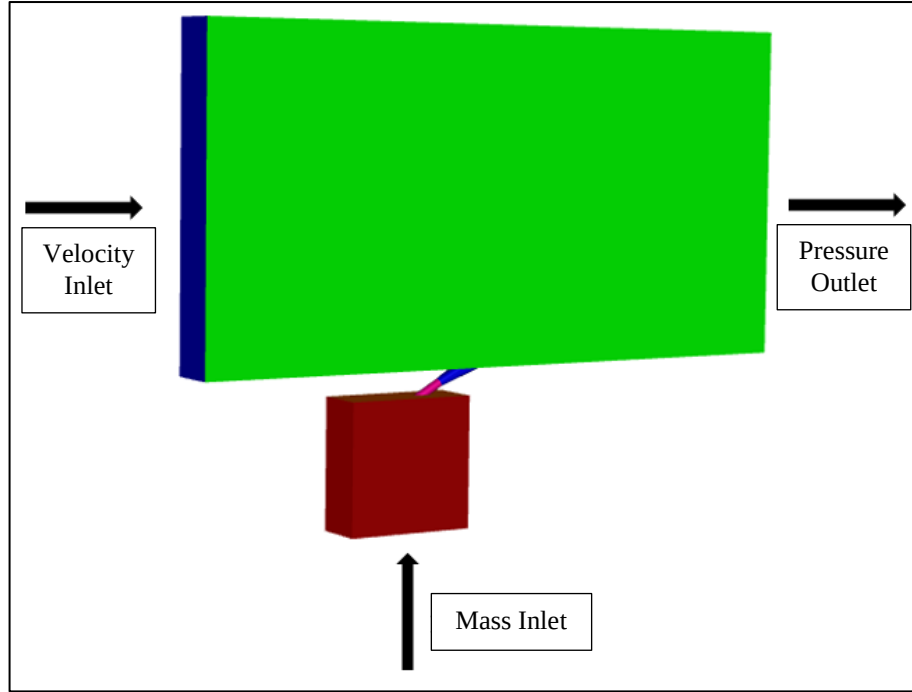


Figure 5.7: Main BCs of Validation Cases.

Then, the geometry cases of this thesis which consist of different feeding situations and different expansion angles of fan shaped hole are separately created. After the validation of lateral film effectiveness with given test data for baseline fan shaped hole by Thole et al. [22], the effects of different feeding situations on coolant flow physics are observed. Four different cases are determined with 7-7-7 fan shaped and cylindrical holes for investigation of coolant flow behavior considering the feeding case as with and without plenum condition. It is also investigated the effects of geometry parameters on lateral adiabatic effectiveness and flow behavior are investigated. These parameters are laidback and lateral expansion angles. The other geometric parameters are kept constant.

At **Table 5.5**, the matrix which belongs to different feeding conditions are given with different case properties. While the two of them have 7-7-7 fan shaped cooling hole,

the other two cases are created with cylindrical hole. These shaped and cylindrical holes are performed with the same throat diameter as it has been used for selected literature test rig. In terms of boundary conditions, experimental velocity 10 m/s and experimental temperature 295 K with average low turbulent intensity are defined as velocity inlet mainstream conditions for these cases. Temperature and mass flow rate of coolant flow is adjusted in order to provide the proper M and DR condition given below. The other boundary conditions are kept as the same in validation cases.

Table 5.5: Matrix for Different Feeding Condition Cases.

Case	Geometry	Plenum	M	DR	Flow Direction
1	Cylindrical	Yes	1	1.5	$X = 0, Y = 1$
2	Cylindrical	No	1	1.5	$X = 0.86, Y = 0.5$
3	7 7 7	Yes	1	1.5	$X = 0, Y = 1$
4	7 7 7	No	1	1.5	$X = 0.86, Y = 0.5$

Two different numerical models can be seen in **Figure 5.8**. On the left, domains are used to reflect the common selected test rig conditions. On the right, numerical models which is created to investigate the effects of different feeding condition is given without plenum.

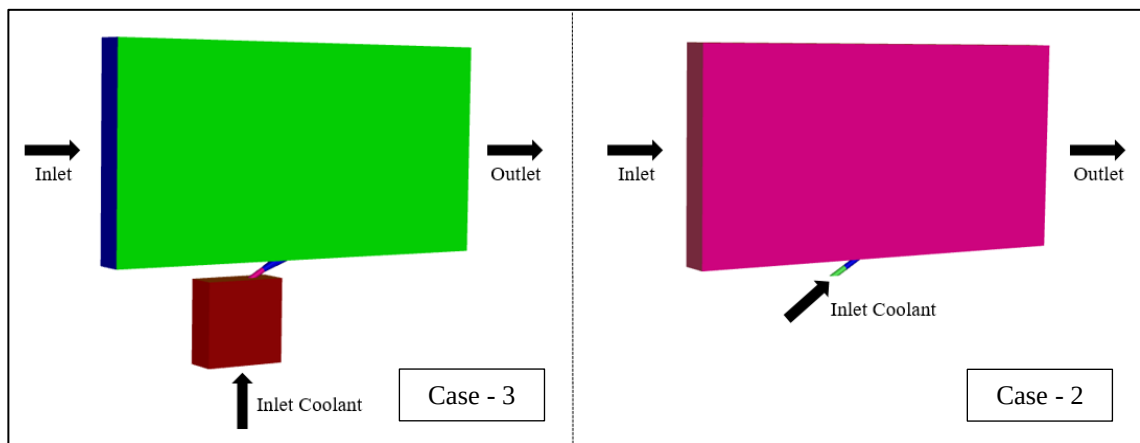


Figure 5.8: BCs of Different Feeding Condition Cases.

Both 7-7-7 fan shaped and cylindrical holes are utilized for each feeding situation model at the same time. BCs are defined as mentioned above and mass flow inlet condition with appropriate flow direction is defined at the inlet with respect to feeding situation. For the without plenum cases, flow directions are considered with respect to 30° injection angle.

Table 5.6: Matrix for Different Expansion Angle Cases.

Geometry	$Tu_{\infty,m}$	α	M	DR	P/D	β_{fwd}, β_{lat}	L_m/D	$L_{lat}/D, L_{fwd}/D$	AR
4 4 0	0.5%	30°	1	1.5	6	0°,4°	2.5	3.5	1.47
4 4 4	0.5%	30°	1	1.5	6	4°,4°	2.5	3.5	1.79
7 7 0	0.5%	30°	1	1.5	6	0°,7°	2.5	3.5	1.82
7 7 7	0.5%	30°	1	1.5	6	7°,7°	2.5	3.5	2.5
10 10 0	0.5%	30°	1	1.5	6	0°,10°	2.5	3.5	2.18
10 10 10	0.5%	30°	1	1.5	6	10°,10°	2.5	3.5	3.32

At **Table 5.6**, the matrix which belongs to different expansion angle cases are given with different geometric information and flow properties. Laidback and lateral expansion angles are selected 4, 7 and 10 with respect to the common literature knowledge. Six different cases are formed with selected angle values. Numerical models are performed at $M = 1$, $DR = 1.5$ and $Tu_{\infty,m} = 0.5\%$ conditions as the BCs of different feeding condition cases.

Three of them have only lateral expansion angles and the others have both lateral and laidback angles. The higher degree of lateral and laidback angles causes the higher space as laterally and axially respectively.

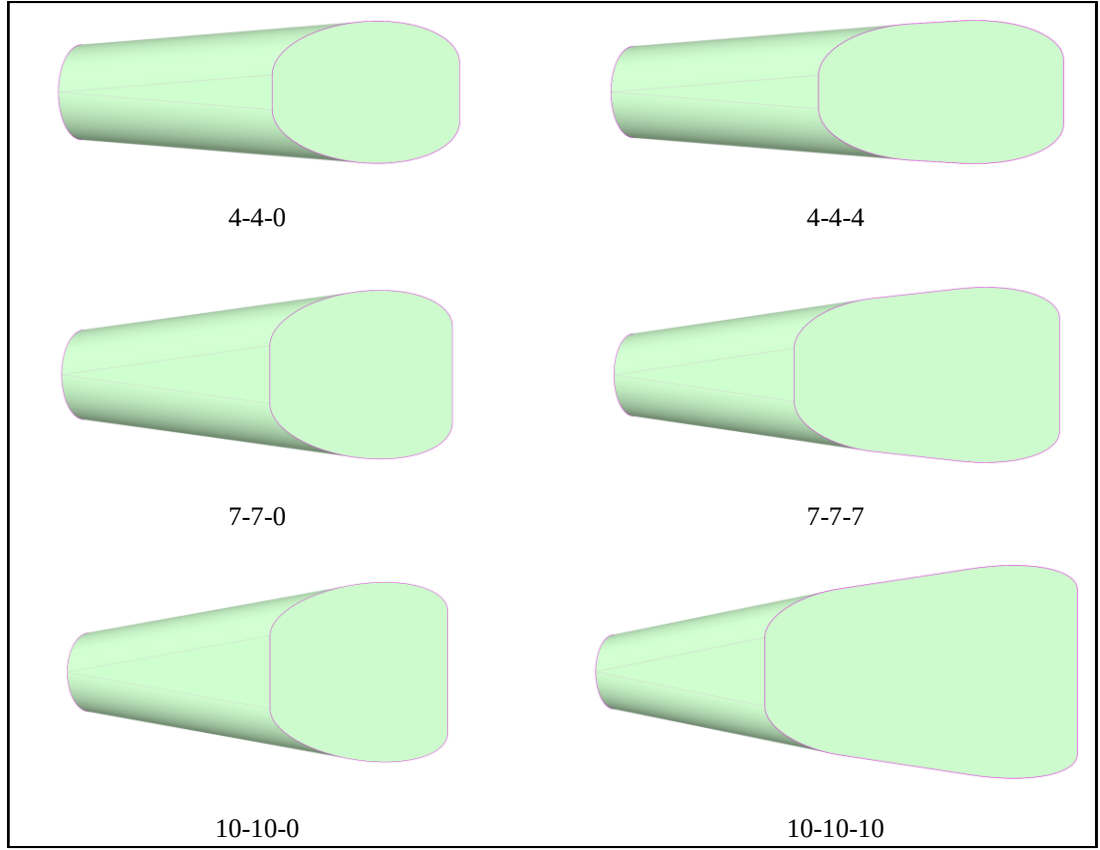


Figure 5.9: Upside View of Hole Exits.

Only shaped section of cooling hole is changed with respect to determined expansion angle and the other domains remain constant. Main channel, coolant channel, cylindrical section of film hole and diameter of throat are kept the same as CFD validation case. Only expansion sections of fan shaped hole are rendered. The effects on hole exit of this rendering process are represented in **Figure 5.9**.

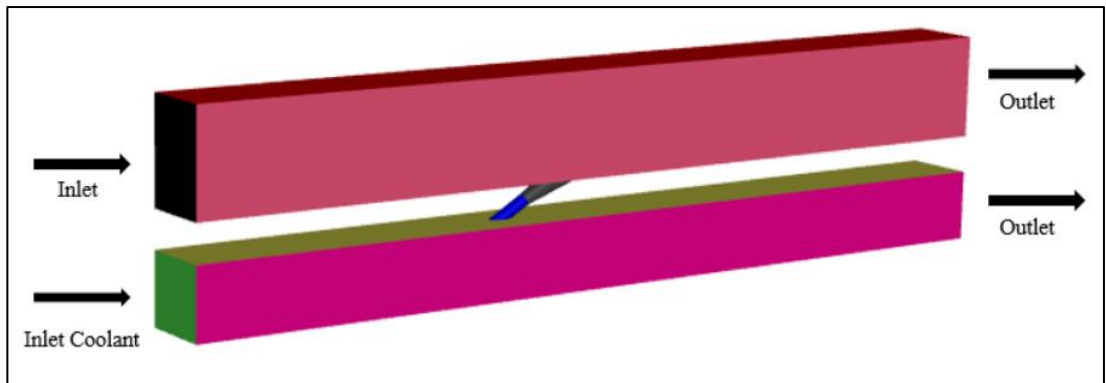


Figure 5.10: Main BCs of Coolant Velocity Cases.

Plenum inlet of coolant is used to reflect test rig condition until now. The different feeding situations and expansion angle cases are studied after the validation step between test and numerical results. But, the effect of coolant velocity in feeding channel will also be investigated in this thesis scope. Different numerical geometry is created for this purpose as it is given in **Figure 5.10**. Height of main flow domain is decreased. Also, the new coolant channel which has smaller section area than validation case is set up to obtain coolant velocity in supply channel.

The plenum condition is not common situation for internal supply passage of turbine cooling. In modern gas turbine, the coolant flow might have different Mach number instead of plenum behavior for the upstream condition after the impingement process. Therefore, the different coolant velocity cases are created with respect to **Table 5.7**. But, the mainstream conditions are kept as constant in order to maintain stability of this thesis topic. In this way, only the effect of coolant velocity in feeding channel which represents case between impingement insert and component inner wall will be investigated without any mainstream difference.

Velocity inlet BC of main flow is kept the same but its value is updated as 80 m/s. The temperature of main flow equals to 400 K. For the coolant flow, pressure inlet BC is defined instead of mass flow. At the exit of the channels, pressure outlet conditions are adjusted in order to keep coolant total pressure in channel feeding to mainstream static pressure ratio as constant. This pressure ratio is controlled just before the inlet and just before the exit of the cooling hole. In this way, PR is considered as the same without any pressure or mixing losses.

Table 5.7: Summary of Created Coolant Velocity Cases.

Name	Geometry	$Tu_{\infty,m}$	$P_{t,c} / P_{s,m}$	Ma_c	Ma_m	$P_{s,m \text{ exit}}$	DR
Case0	7 7 7	0.5%	1.05	0	0.2	76.4 kPa	2
Case1	7 7 7	0.5%	1.05	0.1	0.2	76.4 kPa	2
Case2	7 7 7	0.5%	1.05	0.2	0.2	76.4 kPa	2
Case3	7 7 7	0.5%	1.05	0.25	0.2	76.4 kPa	2

Pressures are determined below the atmospheric condition representatively in order to provide proper flow system. Static pressure value of the main stream is fixed and coolant static pressure is updated at the exit for each coolant Mach number case.

In terms of geometry, 7-7-7 baseline fan shaped geometry with 7.75 mm throat diameter is utilized as test rig configuration. Numerical geometries have a ratio pitchwise distance to diameter as 6. Effects of the coolant velocity on mass flow rate, blowing ratio, lateral effectiveness distribution are investigated with created coolant velocity cases. Also, the geometry effect on discharge coefficient is investigated for cylindrical and shaped hole at the same condition.

6. RESULTS

6.1. Results of Mesh Sensitivity Cases

Mesh sensitivity study is known as running the simulation with different grid. The main purpose is how the converged solution changes with each different mesh structure. This study is also called as mesh independence and convergence study.

In order to determine proper mesh options, six different cases are considered as it is mentioned above. There are two main objectives in sensitivity scope. The first one is about local mesh sizing and global mesh count, the other one is about box size. The mesh sensitivity cases are performed with respect to $M = 1$, $DR = 1.5$ and low turbulent intensity conditions.

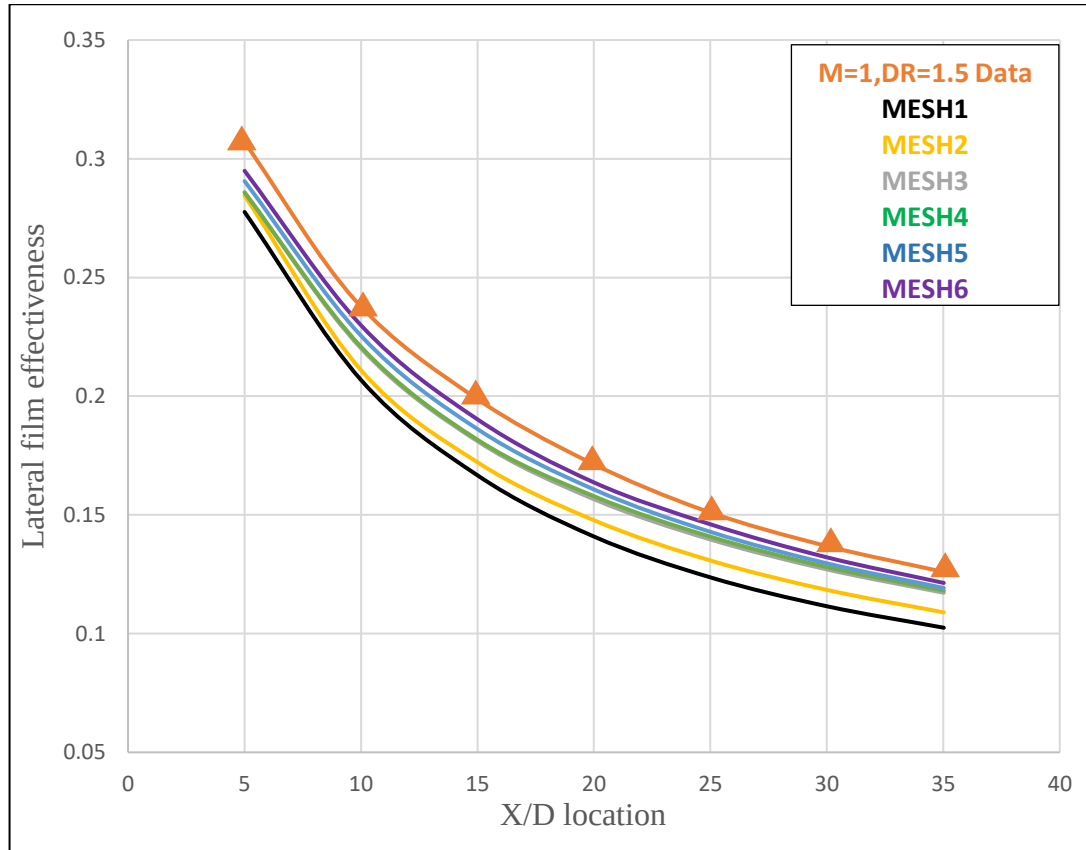


Figure 6.1: Comparison of $\bar{\eta}$ for Different Mesh Cases with Test Data.

Lateral adiabatic effectiveness distribution of mesh cases are given in **Figure 6.1**. MESH5 and MESH6 have the closest match. MESH1 and MESH2 cases are coarse mesh without boi size. As it is expected, effectiveness error is far away the target 20 % for both cases. MESH3 and MESH4 cases have the same surface mesh sizing. But, the difference is existing of boi. While MESH3 has no any boi sizing, MESH4 has the boi with mesh 1 mm size. This 1mm boi size is relatively called as great when it is compared to the other boi values. The boi with great target size as 1 mm has no conspicuous effect on lateral effectiveness distribution. Therefore, they are about at the same position and have close lateral effectiveness each other as it can be realized in **Figure 6.1**.

MESH5 and MESH6 cases have the boi applications where are utilized for the same purpose as MESH4. Whereas MESH4 has no critical improvements in terms of effectiveness, MESH5 and MESH6 have major influencer with the less boi size than 1mm. They have 0.4 mm and 0.2 mm boi target respectively. The each lateral film effectiveness for any location is very close to the sensitivity target for these two mesh cases.

Table 6.1: Error Summary of Mesh Cases.

M = 1, DR = 1.5, $Tu_{\infty,m} = 0.5$		% Error & % Deviation for					
x/D	Test Data	MESH1	MESH2	MESH3	MESH4	MESH5	MESH6
5	0.30	10.6	7.9	7.4	7.4	5.7	4.1
10	0.24	16.2	13.9	9.2	8.9	6.6	4.6
15	0.20	20.2	16.3	10.6	10.2	7.6	5.3
20	0.17	22.3	16.7	10.1	9.2	7.2	5.3
25	0.15	22.7	16.0	8.7	7.8	6.2	3.9
30	0.13	23.0	15.8	7.9	7.1	5.8	3.8
35	0.12	23.2	15.8	7.7	6.9	5.9	4.0

The summary of errors for different X/D locations is given at **Table 6.1**. Red color represent the position where errors between test and mesh cases are calculated as higher than the sensitivity target. Dark green color states that calculated errors are

under the limit. For MESH5 and MESH6, all errors for any location seem acceptable. But, MESH6 has 10 million mesh count and it causes the waste of time and source. Also, it can be proved that there is no major enhancement difference between MESH5 and MESH6 cases.

At the end of the post processing, the global error for each location is under the sensitivity target and they have very close effectiveness to test results for MESH5 and MESH6. But, MESH5 case is selected in terms of insensitive residuals and lateral adiabatic effectiveness. And, it is consist of surface mesh sizing and boi domains. Size of surface and boi are determined to cause proper insensitive CFD results and appropriate source time.

6.2. Results of the Different Turbulence Models

After the determination of proper mesh options, the different turbulence models are considered in terms of lateral film effectiveness and flow structure inside the cooling hole. Turbulence modelling is the key to find the effects of flow turbulence behavior. The Navier Stokes equations are capable of solving the velocity and pressure of fluid domain. Each of these parameters can be broken down into a mean part and a fluctuating part in a turbulent flow. RANS equations which are used by turbulence solver and obtained by averaging the equations. But, velocity fluctuations are still involved to RANS equations with specific solution terms. Each turbulence solver has the own different additional terms. Therefore, studies which have different turbulence solver need to be performed in order to be sure how the results will change.

As it has been mentioned earlier, RKE model is found as turbulence solver for this kind of flow and hole structures. Because, it is suggested as common literature model due to capability of solving separation bubble regions, recirculation and jet spreading for this kind of studies. Firstly, this situation has to be proved with comparison to different turbulence models.

The standard $k-\varepsilon$ and $k-\omega$ SST models are also considered for comparison. As it is suggested in literature, the best results have been gathered with RKE model. The detailed results are given and shown below.

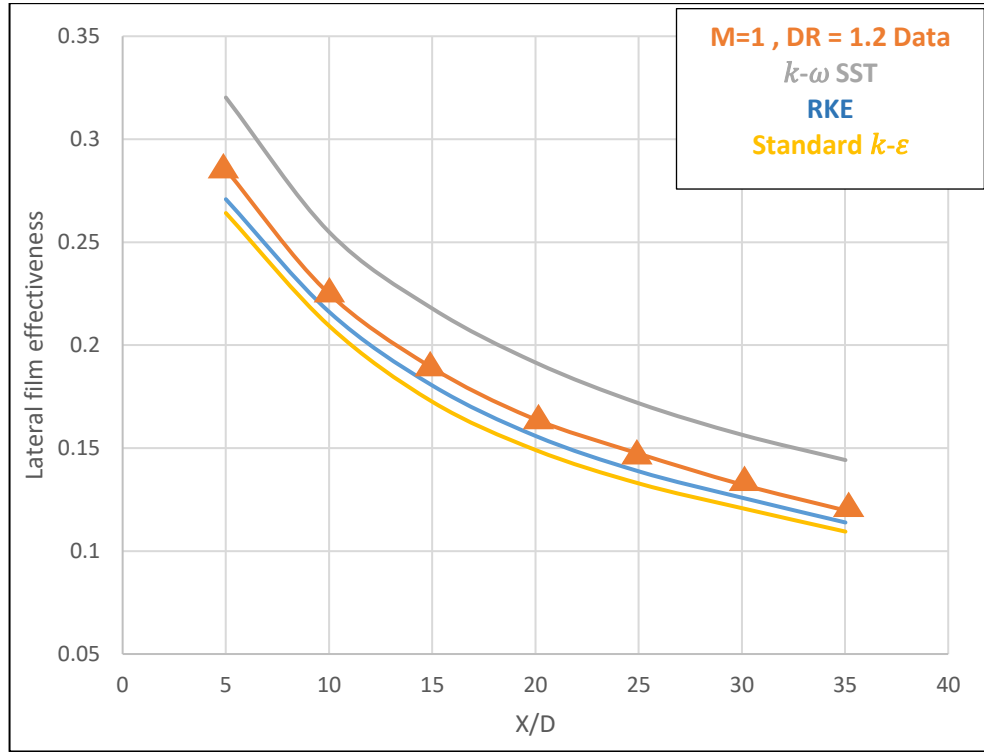


Figure 6.2: Comparison of $\bar{\eta}$ for Different Solver with Test Data.

Lateral film effectiveness comparison of turbulence models are presented in **Figure 6.2**. As it can be realized in the figure, lateral effectiveness distribution is overpredicted with $k-\omega$ SST model which is used with grey color. As a main reason, computed velocity magnitudes for $k-\omega$ SST model at midsection shows that this turbulence model with selected mesh grid and solution parameters is not be able to capture the separation bubbles or recirculation regions accurately. While $k-\omega$ SST model has no any separation bubbles, they exist for both $k-\epsilon$ models. Finally, RKE model presented with blue color which gives the closest numerical results in terms of laterally averaged film effectiveness is determined as the main turbulence model in these thesis studies. Also, Saumweber and Schulz have reported that RKE turbulence model and the same boundary layer approach in terms of Y^+ with this thesis provide the good approach for resolution of the near wall area [34].

Figure 6.3 proves that the main difference between three models occurs on the leeward side of the jet hole. For $k-\epsilon$ models, coolant flow seems attached condition along cylinder section of shaped hole and detachment situation takes place when coolant flow comes to the expansion section of shaped hole. But, coolant flow is able to pass through cylinder and expansion sections of hole as contiguous for $k-\omega$ SST model. In

this reason, overpredicted lateral effectiveness distribution with this model seems reasonable. Because, the separation bubbles found in $k-\varepsilon$ models are not observed with $k-\omega$ SST model.

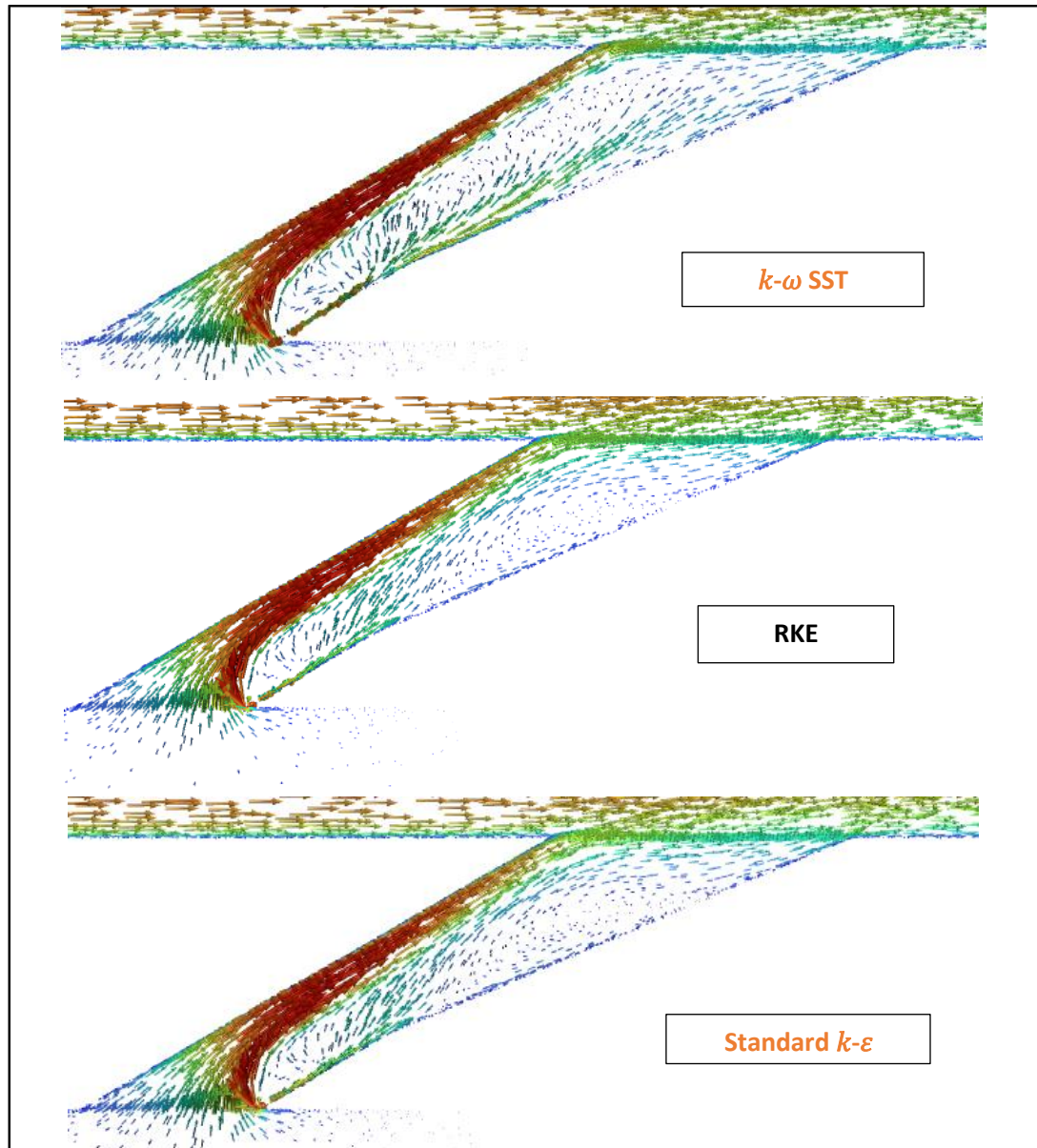


Figure 6.3: Computed Velocity Vectors of Turbulence Solvers.

Figure 6.3 also shows that RKE and Standard $k-\varepsilon$ models have the same velocity vector distributions at midsection. But, RKE model has the closer lateral effectiveness to test data than Standard $k-\varepsilon$ model. As it has been mentioned earlier, realizable means model is able to satisfy more accurate mathematical constraints in calculation formula of the

Re stress and model is coherent with the turbulent flow physic. In this way, RKE solves the spreading of the jets more accurately as common literature knowledge.

After this point, RKE turbulence model is utilized in the future works. The other solution parameters are properly determined in order to get the closest lateral effectiveness to test results and flow behaviors. Solution parameters which have the best results has been given at **Table 5.4**.

6.3. Results of the CFD Validation Cases

The validation and thesis cases are performed with respect to ‘none’ convergence criteria in Fluent. In this way, residuals are not limited in terms of criteria but all of them are monitored for all cases. The maximum residual is determined as $1e-4$ for continuity and residuals of velocity components, turbulence parameters and energy are followed to be acceptable as lower than this value.

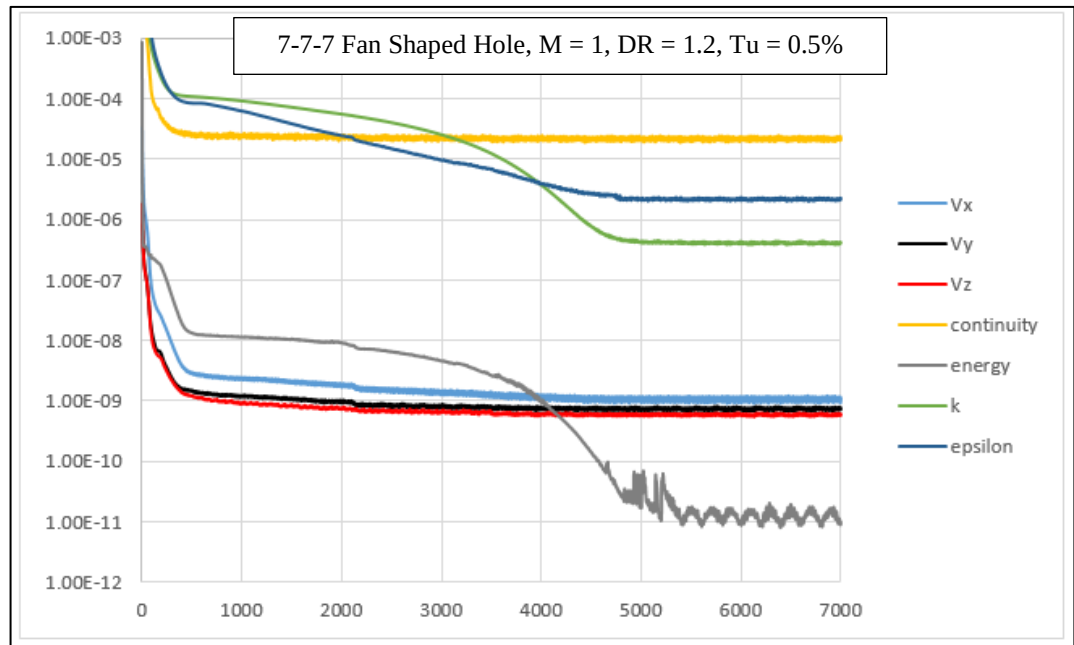


Figure 6.4: Plotted Residual Behaviors for Flow and Turbulence Equations

Figure 6.4 belongs to 7-7-7 fan shaped hole at $M = 1$, $DR = 1.2$ and low turbulent intensity case. And, it shows that turbulence and energy equations are converged well after 5000 iterations. Continuity and velocities are converged earlier and the same behavior are observed for all validation cases in terms of convergence parameters.

After 5000 iteration, residuals are observed less than determined convergence criteria and have horizontal behavior. Even if energy equation has oscillation, it seems acceptable due to very low difference between two time steps as $1e-11$. As a result, all thesis cases are solved until 7000 iterations in terms of convergence.

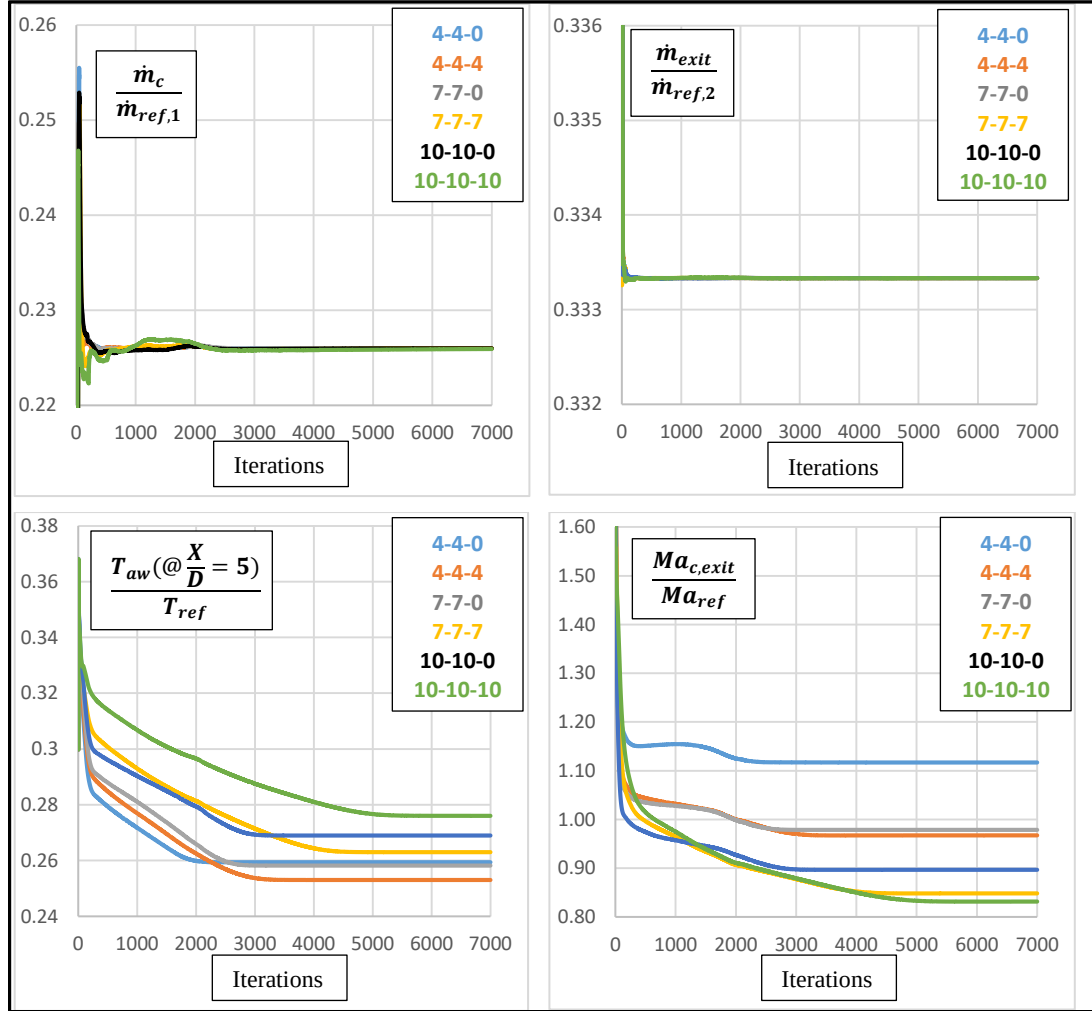


Figure 6.5: Monitored Critical Analysis Parameters

Only behavior of residuals monitoring is not enough to be sure that thesis numerical models converged well. In addition, analysis parameters are also investigated in terms of convergence. Therefore, some analysis parameters are given in order to control convergence situation in **Figure 6.5**. Computed coolant mass flow rate at throat of film cooling hole, mass flow rate at the exit of mainstream domain, local wall temperature downstream of the coolant jet for midspan $X/D = 5$ location and MWA coolant Mach number at the hole exit which belong to geometry cases are shown as dimensionless in **Figure 6.5**. The same parameters are also controlled for the validation cases.

Parameters are used as dimensionless with respect to reference values. Figure prove that monitored dimensionless parameters are converged well for selected 7000 iterations without any oscillation. While throat and exit mass flow rate are the same, computed local wall temperature and exit Mach number of coolant jet seems different because of the different hole geometry as it is expected. Findings about different hole geometry are particularly explained in the section of geometry cases.

The velocity vector contours on the center Z axis midplane of the geometry is shown in **Figure 6.6** for one of the validation cases which belongs to 7-7-7 baseline fan shaped.

Below domain acts like a plenum and coolant air has low velocity magnitude until inlet of the hole. So, velocity vectors of coolant in plenum starts to appear just before coolant hole inlet. Mainflow velocity equals to 10 m/s in main channel as it is remembered from boundary conditions and maximum velocity is monitored about 11 m/s in cooling hole.

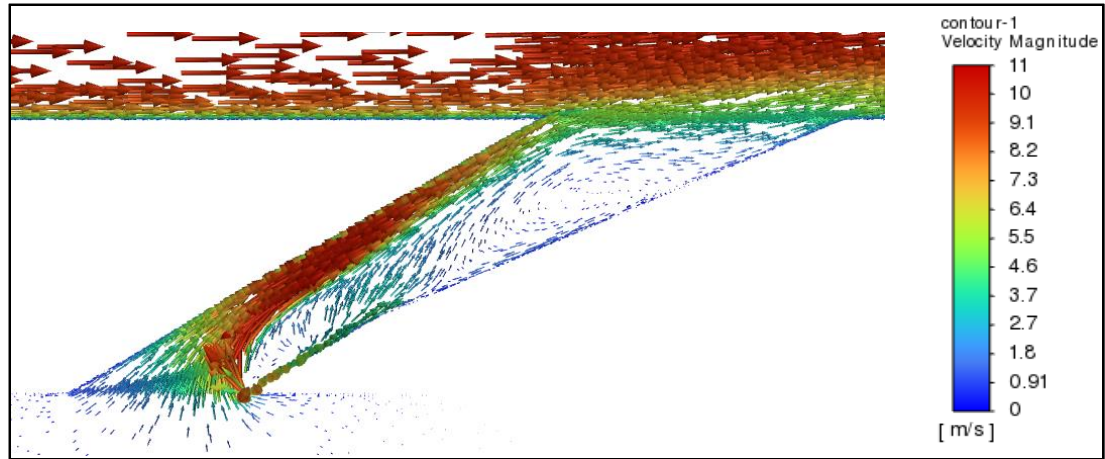


Figure 6.6: Computed Velocity Vectors at DR = 1.5, M = 1.

There are different low and negative velocity regions in hole as common literature knowledge. First one occurs due to vena contracta at the entrance of the film cooling hole as it has been mentioned earlier by Pietrzyk [10].

The other one is separation bubble region in expansion section of fan shaped hole as it can be seen in the **Figure 6.6**. This issue has been already covered by Wittig [24]. These two low or negative velocity regions are affected not only by blowing, density and pressure ratio but also expansion angles. In addition, these regions really depend

on the feeding situation of coolant flow. They are not found as the same for different cases.

The adiabatic film effectiveness contour on the downstream surface is shown at $DR = 1.5$, $M = 1$ and low Tu mainstream condition in **Figure 6.7**. Local adiabatic film effectiveness equation which is given in Formula 2.31 is defined as custom field function with respect to temperature of main stream, coolant flow and adiabatic wall temperature. Then, effectiveness distributions which are belong to one of the validation case of 7-7-7 shaped hole and cylindrical hole are obtained by means of this function as below. For the same M and DR , the found main goal is local film effectiveness distribution of 7-7-7 shaped hole seems more effective then cylindrical as the common literature knowledge already mentioned in literature section.

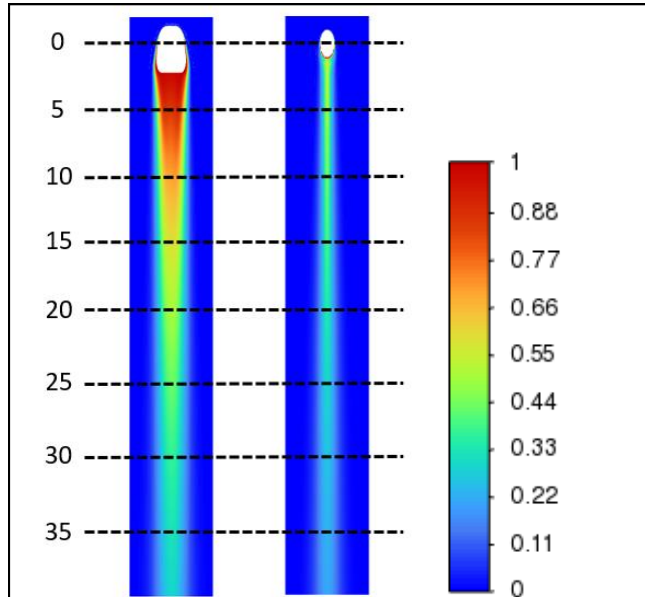


Figure 6.7: η_{film} Contour for Shaped and Cylindrical Hole at $DR = 1.5$, $M = 1$.

Then, laterally averaged film effectiveness is calculated with respect to Formula 3.1 for each X/D location given in **Figure 6.7** with respect to location of 7-7-7 shaped hole test rig results. Literature results are given for these specific X/D locations which represent with black line except $X/D = 0$ and this location represents the coolant flow itself.

The lateral film effectiveness comparisons between literature and thesis cases for different density ratio and turbulent intensity are given in below figures. Results are

not compared from $X/D = 0$ to $X/D = 5$ distance due to nonexistent test rig results for these area and inefficient CFD region which is just exit of the cooling hole. This is the common literature way for validation because of RANS CFD modelling restriction. $X/D = 0$ location represents the middle of the cooling hole as it is shown in **Figure 6.7** and test results of wall temperature are inherently not exist for this region. Therefore, thesis studies are also computed for $X/D = 5$ location as earliest comparison location. The compared X/D locations in terms of test and numerical film effectiveness might be acceptable, considering the suction and pressure side distance of gas turbine cooled components.

In terms of gas turbine cooling, critical X/D location depends on design. As it has been mentioned above, the earlier X/D distance of engine component generally has internal impingement cooling. In this way, metal temperature can be decreased for these earlier distance. Therefore, life and stress are not critical due to lower metal temperature of this location. But, impingement insert is not able to reach to forward location. It can be summarized that the forward location is more important than just downstream of cooling hole in terms of cooling and part life.

In general, literature and thesis numerical CFD results are very close to each other for all blowing ratio cases and literature ones are slightly higher than CFD results in this thesis. Comparisons for all film cooling parameters which belong to the test rig X/D specific location are separately given in below figures. In terms of turbine cooling design, the critical X/D location is not an only one specific point. The cooling design should be separately considered for all X/D locations.

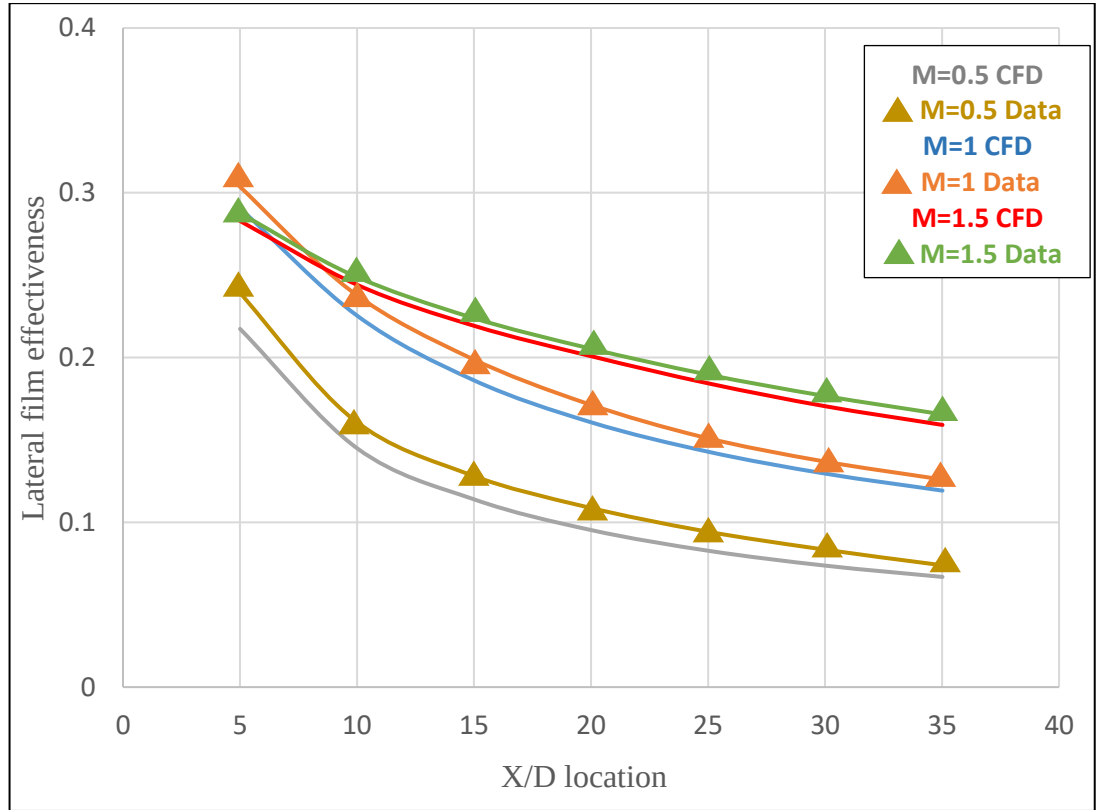


Figure 6.8: Comparison of $\bar{\eta}$ for Different M, DR = 1.5 and $Tu_{\infty} = 0.5\%$.

For $Tu = 0.5\%$ and $DR = 1.5$ cases, maximum deviation or error is get at $X/D = 20$ location for $M=0.5$ case and it equals to 13%. And, this deviation is acceptable in terms of validation standard. As it can be realized in **Figure 6.8**, less error between test and numerical results might be seen for higher blowing ratios.

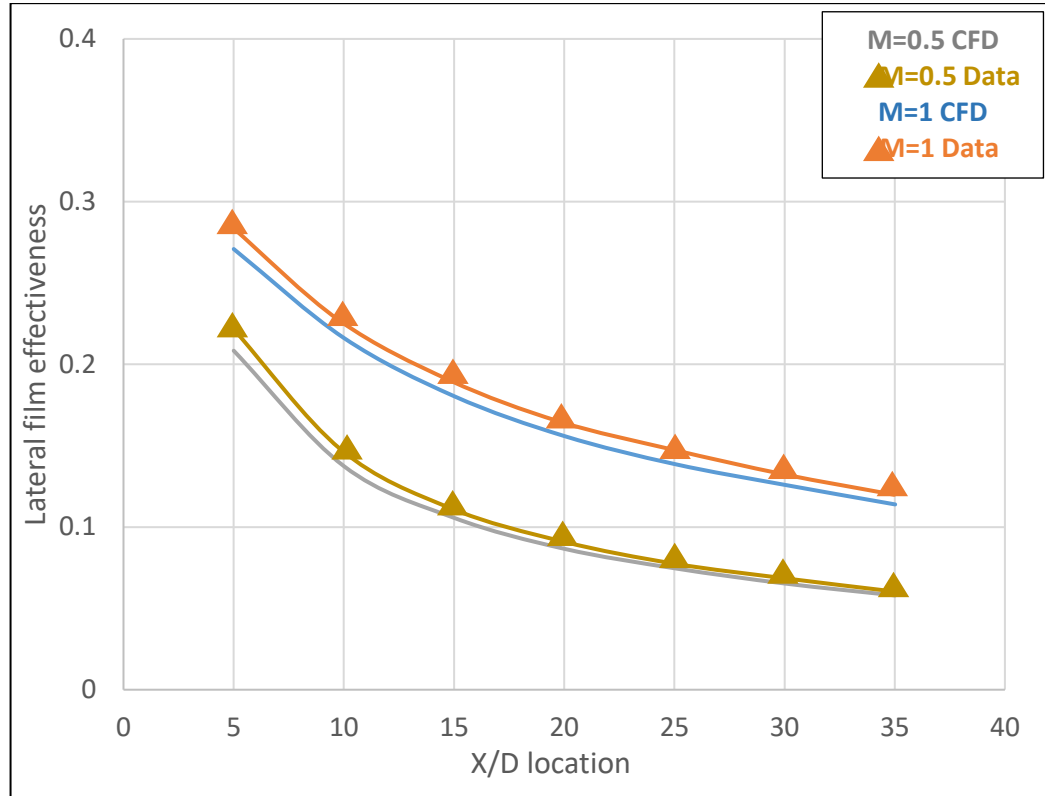


Figure 6.9: Comparison of $\bar{\eta}$ for Different M, DR = 1.2 and $Tu_{\infty} = 0.5\%$.

For $Tu = 0.5\%$ and $DR = 1.2$ cases, maximum deviation or error is get at $X/D = 25$ location for $M=1$ case and it equals to 6%. And, this deviation is acceptable in terms of validation standard.

As it can be realized in **Figure 6.9**, less error between test and numerical results might be seen for lower blowing ratios in comparison to higher density ratio case.

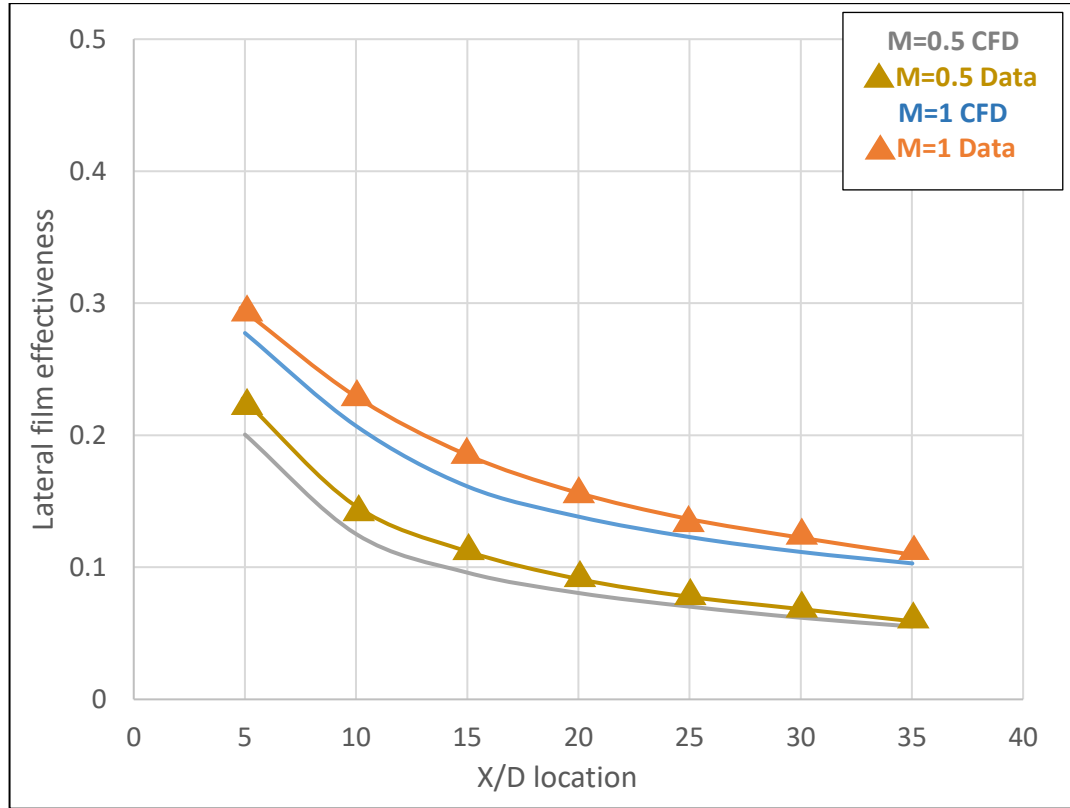


Figure 6.10: Comparison of $\bar{\eta}$ for Different M, DR = 1.5 and $Tu_{\infty} = 5.6\%$.

For $Tu = 5.6\%$ and DR = 1.5 cases, maximum deviation or error is get at X/D = 15 location for M=1 case and it equals to 16%. And, this deviation is acceptable in terms of validation standard.

For all validation cases, maximum deviation is obtained at high turbulent intensity, density and blowing ratio condition. The main reason is that the highest uncertainty formation is observed with highest design parameters. For example, it was reported that maximum test uncertainty for blowing ratio is found for the highest blowing ratio M = 1.5 case [22].

Also, there might be the other influencer. The same mainflow domain is used for all validation cases in terms of axial length in order to get lower computational time and source. But, boundary layer thickness and velocity fluctuations in this region are different for high turbulent intensity case. Using the same inlet domain with lower turbulent intensity case causes thinner boundary layer thickness than actual one. Because of the reasons which are given above, maximum deviation might be found for higher turbulent intensity and with highest blowing ratio.

6.4. Results of Geometry Cases

After validation progress, specific thesis cases are created for different feeding situation and hole geometry. There are two main subsections in this section and these are determined in order to investigate effects of different feeding situation and expansion degree of shaped hole.

First subsection is about flow behavior of coolant with plenum condition. Different feeding cases are created with 7-7-7 and cylinder holes. The coolant is defined with velocity direction with respect to applications as it is mentioned in **Figure 5.8**. For the second subsection, six geometry cases which have different expansion angle of fan shaped hole are created. The effects of expansion angles on the coolant flow structure and film cooling effectiveness will be investigated with the created 6 different fan shaped hole geometries which are given in **Figure 5.9**.

The same boundary conditions are followed for all created cases of Section 6.4. Main flow has 10 m/s velocity inlet and mass flow rate of coolant is fed from plenum as validation cases. M , DR and turbulent intensity are constant as 1, 1.5 and 0.5 % respectively. The low turbulent intensity is utilized for all new cases in this thesis scope. It effects the mixing after ejection between hot gas and coolant flow. In this way, mixing effect is aimed to decrease and the scope of thesis is narrowed in terms of flow and effectiveness parameters.

6.4.1. Results of Different Feeding Condition Cases

As it is mentioned above, this section aims to investigate flow behavior of coolant flow with different hole geometry and feeding situation. With respect to this aim, four different cases are separately created for 7-7-7 fan shaped and cylindrical holes. Created models are performed with and without plenum condition in terms of coolant feeding, and the changing of coolant behavior is observed. Models are performed at $M = 1$, $DR=1.5$ case.

The first post processing of different feeding condition cases is about separation bubbles. There are two main separation bubble in cooling hole with plenum feeding condition as it is mentioned in Section 6.3.

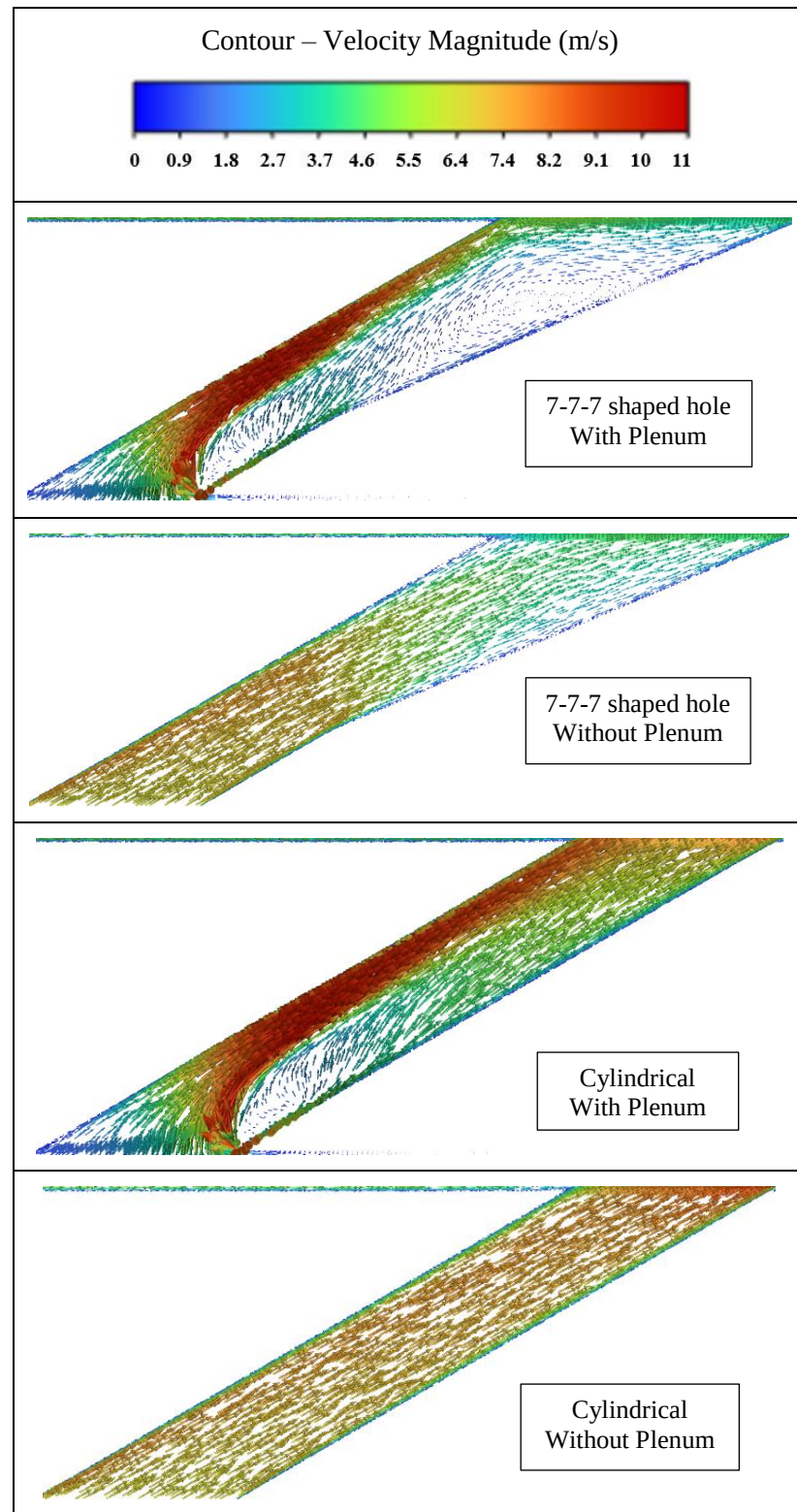


Figure 6.11: Computed Velocity Vectors for Different Feeding Condition Cases.

The first one occurs due to vena contracta effect at the hole inlet and the other is observed due to strong adverse pressure gradient which depends on hole cross section area and expansion angle of fan shaped hole. The findings about these separation

bubbles with and without plenum feeding for two types of cooling hole are represented in **Figure 6.11**. **Figure 6.11** proves that plenum condition and expansion angle are separately influencer for formation of separation bubbles. Considering first and third images with plenum cases, vena contracta is observed for both cases and it means that the plenum feeding creates the vena contracta at the inlet of cooling hole for shaped and cylindrical holes. Considering second and fourth images without plenum cases, the vena contracta is not exist. Because, the coolant is uniformly defined with flow direction at the inlet as it is explained earlier and it causes the uniform velocity profile without separation at the inlet. In addition, the other separation bubble which occurs in expansion section of fan shaped hole is not found for without plenum feeding due to the uniform velocity profile. The coolant in hole is able to deal with moderate adverse pressure gradient at each location. Even if there is the vena contracta effect for cylindrical hole at third images in **Figure 6.11**, the other separation bubble is not observed. For the formation of the separation bubble in expansion section, plenum feeding and expansion angle must be exist at the same time. The coolant flows close to the windward or upper side of cooling hole because of plenum feeding and it causes the velocity deficit region below coolant flow for fan shaped cooling hole. Therefore, the flow tends to detach because of strong adverse pressure gradient. But, this adverse pressure gradient seems as moderate for cylindrical hole which has lower cross sectional area than fan shaped hole. As a result, the formation of these main two separation bubbles is really depends on feeding situation and type of hole.

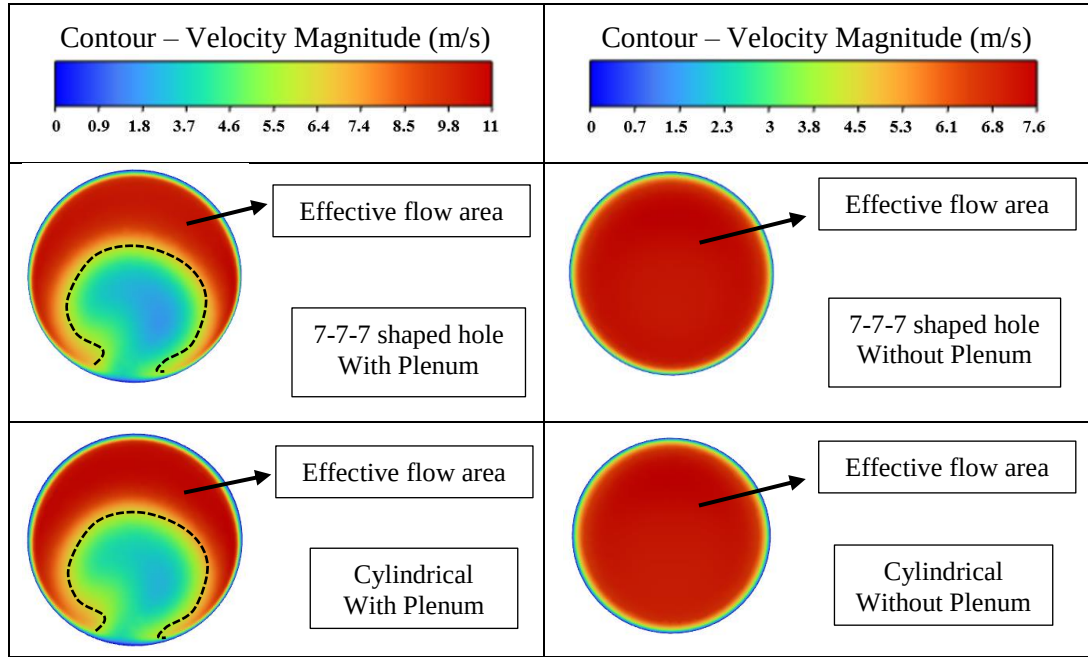


Figure 6.12: The Differences of Velocity Magnitude Contours at Throat Section

The second finding of different feeding condition cases is about computed velocity magnitudes at throat. Velocity magnitude contours of four cases are given in **Figure 6.12**. On the right in above figure, velocity magnitude contours can be seen for without plenum condition. Throats have uniform velocity distribution because of direction definition with respect to 30° injection angle at the hole inlet. On the left, velocity magnitude contours are given for plenum condition and non-uniform velocity distributions are observed for both 7-7-7 shaped and cylindrical hole. Considering the same amount of cooling air exists for all cases, maximum velocity is about 11 m/s for the plenum condition due to the lower effective flow area than without plenum condition. The coolant flow passes along the more effective area at throat section and uniform velocity is found as 7.6 m/s with respect to mass balance equation which means the higher area causes the lower velocity for the same mass flow rate. As a result, the plenum feeding causes irregularity at throat and the coolant flows close to the windward side of cooling hole as it can be realized in **Figure 6.12**.

The other finding of different feeding condition cases is about plotted three dimensional velocity streamlines of coolant flow from hole inlet to mainstream exit. Firstly, velocity streamlines which are belong to the with plenum condition cases for cylindrical and shaped hole are given in **Figure 6.13**.

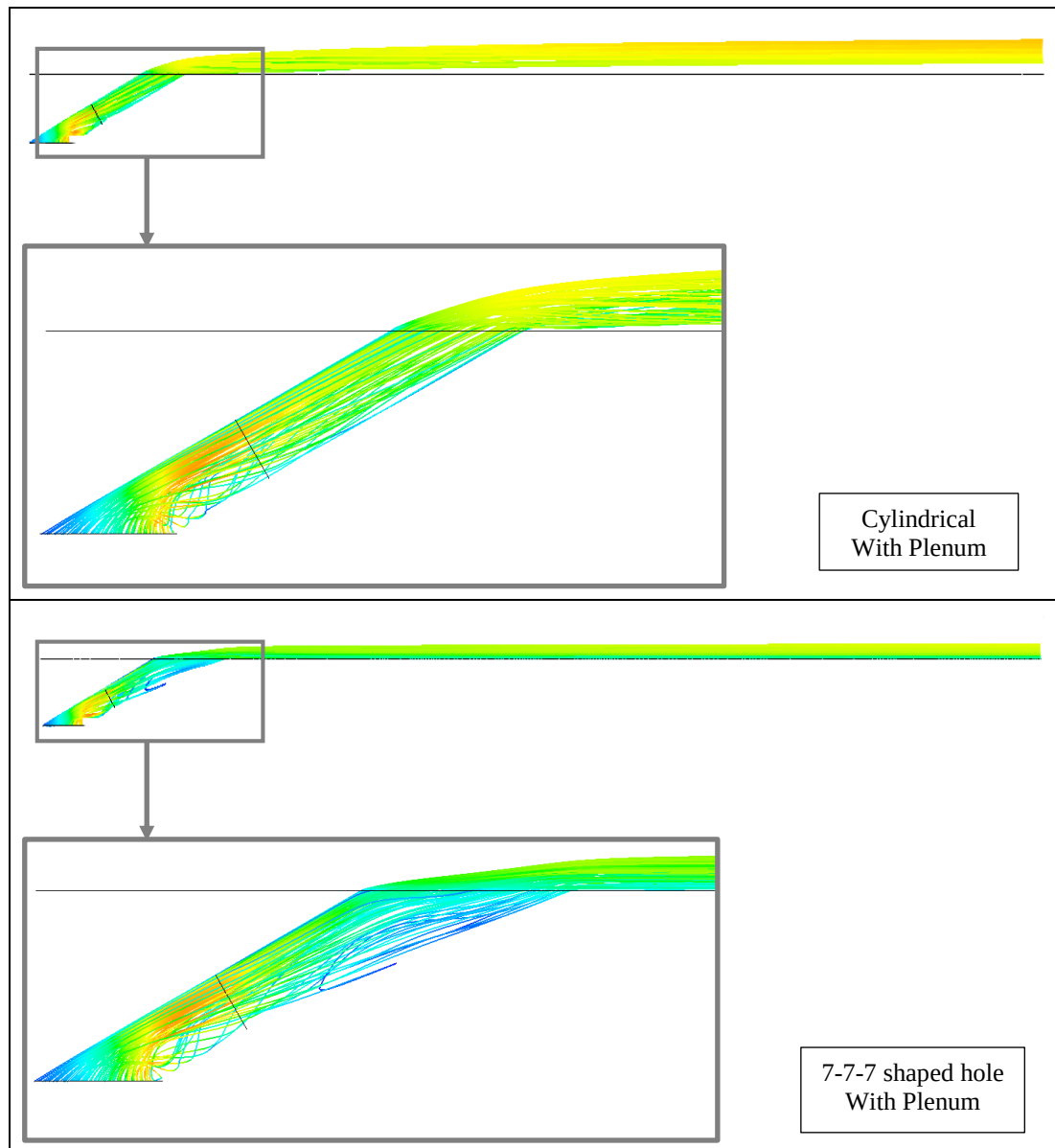


Figure 6.13: Three Dimensional Streamlines with Plenum Condition.

Figure 6.13 proves that type of film cooling hole effects the downstream behavior of coolant flow from cooling hole exit to mainstream exit. While the cylindrical hole causes detachment situation, it is not exist for the shaped hole. Above images in figure show that the streamlines are subjected to the early detachment issue just exit location of cooling hole and this issue always exist along downstream surface for cylindrical hole. But, the fan shaped hole prevents the detachment situation and the coolant tends to stick to the downstream surface for any location as it is shown in above figure.

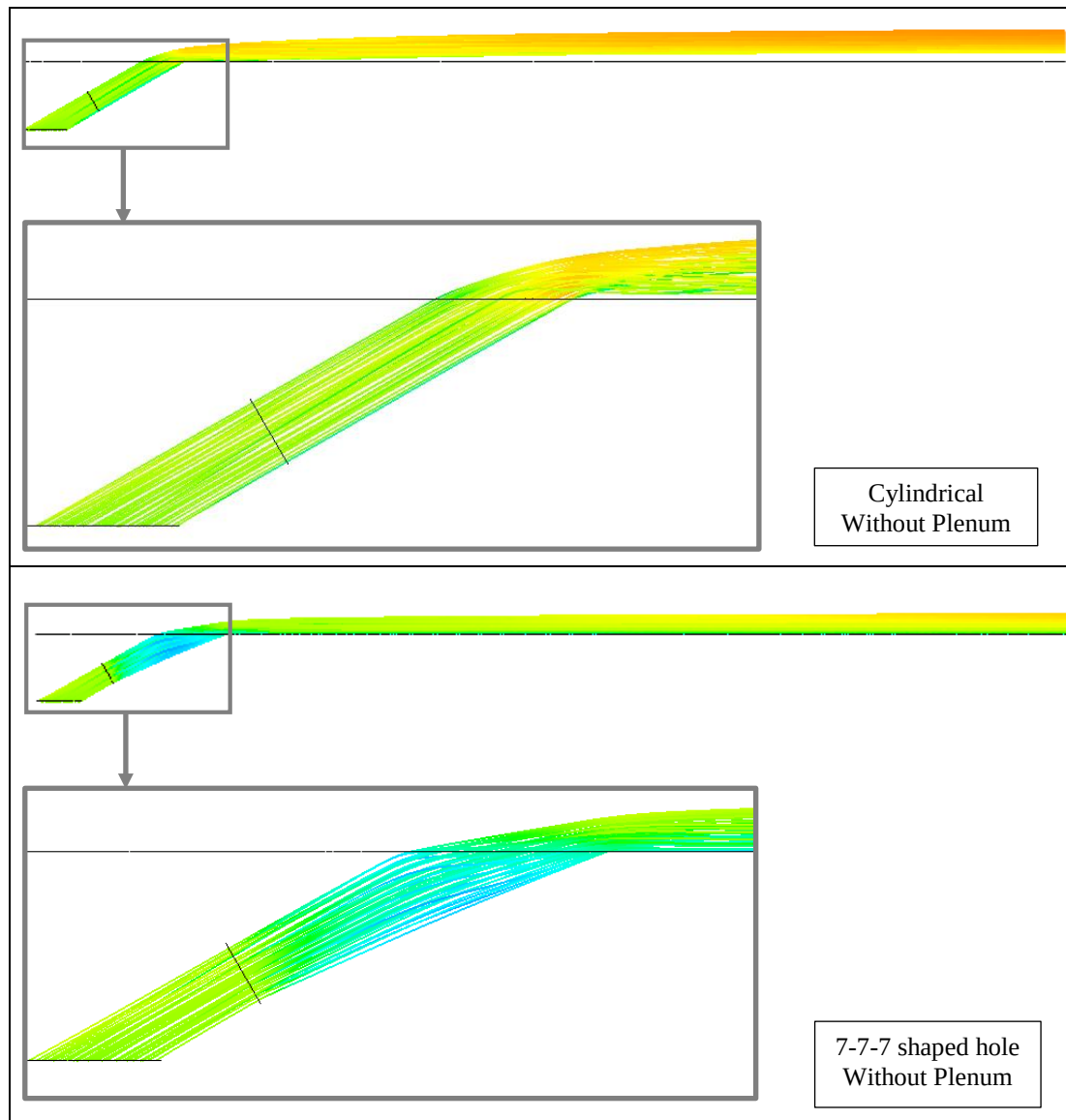


Figure 6.14: Three Dimensional Streamlines without Plenum Condition.

Then, velocity streamlines which are belong to the without plenum condition cases for cylindrical and shaped hole are given in **Figure 6.14**. The same flow behaviors are observed for the cylindrical and the shaped hole as the with plenum cases. It means that the coolant feeding situation is not an influencer for the coolant downstream behavior. The detachment issue only depends on the shape of cooling hole and the cylindrical hole leads to the detachment situation for both cases with and without plenum condition. As a result, the fan shaped hole prevents the detachment for any coolant feeding conditions.

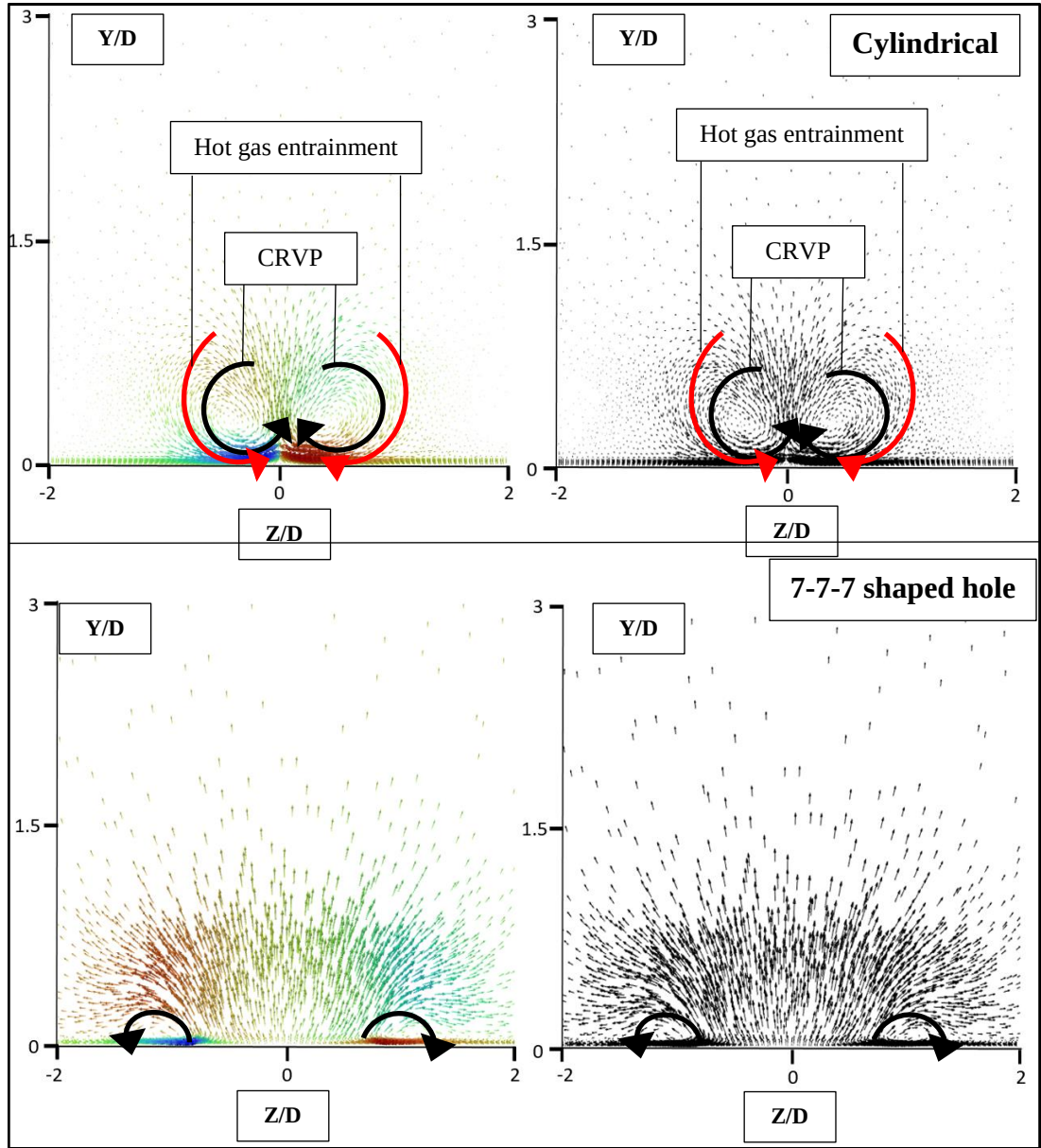


Figure 6.15: Velocity Vectors after Ejection for $X/D = 2.5$ location

The other observation about velocity vectors after mainstream and coolant ejection. Velocity vectors after ejection are given in **Figure 6.15** for the cylindrical and 7-7-7 shaped hole for the plenum feeding condition. The velocity vectors and the secondary flows are about the same for with and without plenum feeding conditions. Therefore, the vector distributions in figure are given for only plenum condition. With this figure, it is aimed that understanding of three dimensional flow structure which effect the film cooling performance.

Eric and Roshko [14] have already defined the counter-rotating vortex pair (CRVP) for the cylindrical hole. Above figure proves that the CRVP is formed due to the

interaction between the coolant jet and the mainstream for cylindrical shape. The rotational motion of the vortex pair causes the hot gas entrainment toward the wall surface and the coolant is subjected to the lift off condition by hot mainstream entrainment. As a result, both entrainment and coolant lift off situation degrade the film cooling effectiveness of cylindrical hole.

Saumweber et al. have already mentioned the velocity vector after ejection and below **Figure 6.16** [35] shows almost the same behavior as above **Figure 6.15** which belongs to thesis shaped hole studies. As it can be realized in both figures, the secondary flows are observed quite small and weak for fan shaped hole when considered with the cylindrical hole geometry.

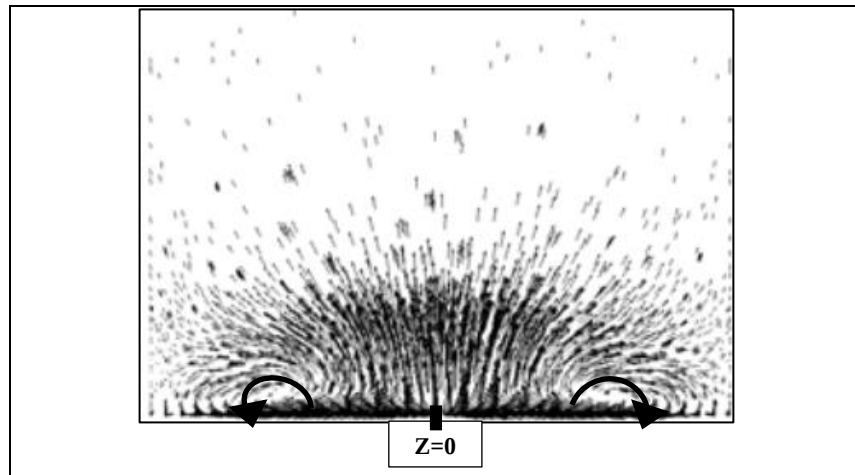


Figure 6.16: Plotted Velocity Vectors after Ejection by Saumweber [35].

The kidney vortices seem close to the lower solid wall and far away from the $Z=0$ center location. In this way, the hot gas entrainment is prevented because of using fan shaped hole and the higher laterally averaged film effectiveness is obtained for this shaped hole geometry. The lift off condition which decreases the film cooling performance is not exist for the shaped hole geometry, while it is negative influencer with the hot gas entrainment for the cylindrical hole. These findings prove the difference between the cylindrical and shaped hole about the detachment situation along downstream surface which has been shown with three dimensional streamlines in above **Figure 6.13** and **Figure 6.14**.

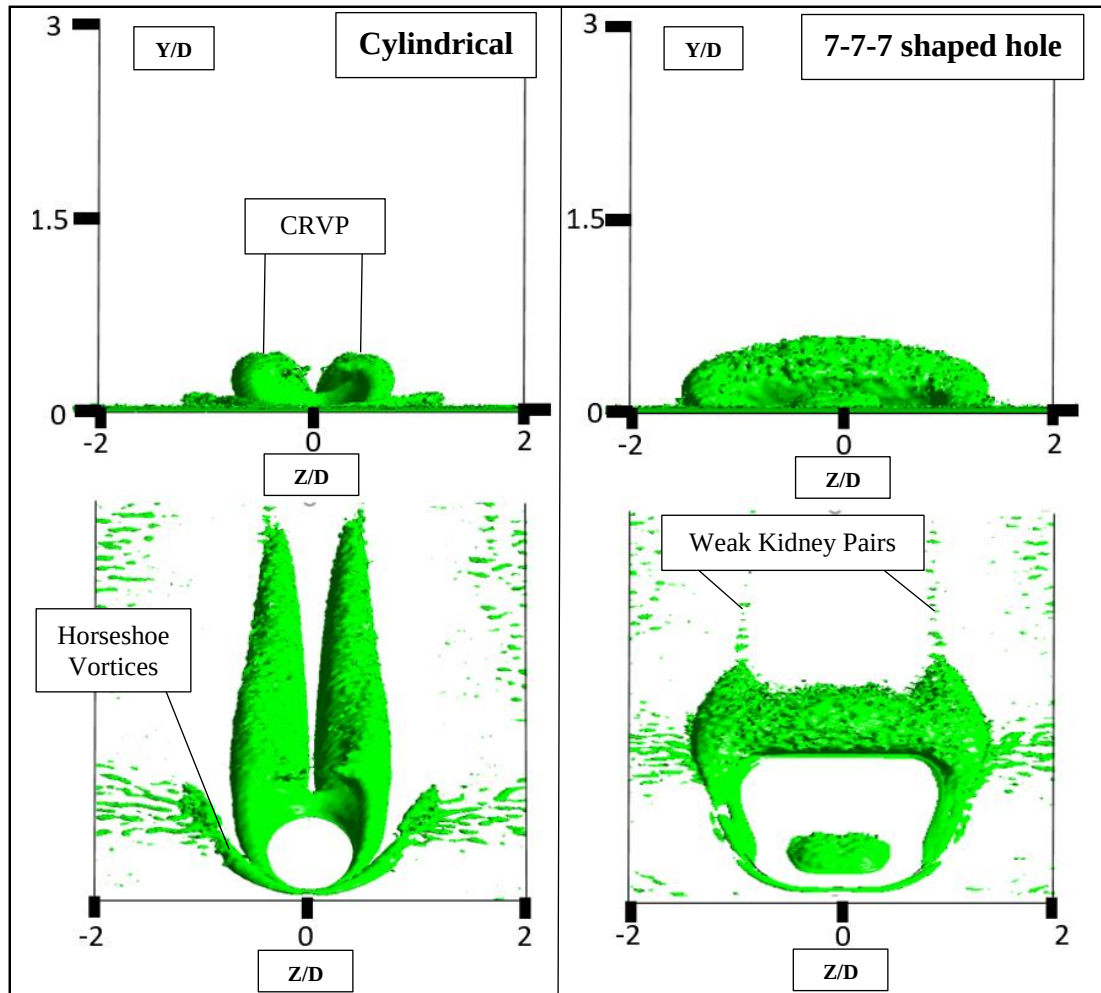


Figure 6.17: The Vortex Core Regions for Cylindrical and Shaped Holes.

In **Figure 6.17**, the vortex core regions for the cylindrical and shaped hole are plotted by means of Lambda 2-Criterion method to investigate the behavior of secondary flows inside the mainstream. As Fric and Roshko [14] has mentioned, the horseshoe vortices and CRVP are observed for the cylindrical hole. And, these vortices are stronger when consider with the shaped hole.

The CRVP of cylindrical hole leads to the hot gas entrainment below itself and degrades the film effectiveness especially for lateral position with respect to $Z=0$ center location as it has been explained above. But, the fan shaped hole reduces the secondary flow effects and weak kidney pair appears as it is shown with right image in figure. These observations prove the different film cooling effectiveness distributions for shaped and cylindrical holes in **Figure 6.7**.

6.4.2. Results of Different Expansion Angle Cases

The geometry differences are about laidback and lateral expansion angles in this section. Angles are determined as four, seven and ten with respect to common literature knowledge. Dimensions of mainstream and coolant domain are the same with test rig.

Generally, the laidback and lateral angles should not be greater than overexpansion angle of diffuser. So, angle is limited as ten degree in order to cover this situation. 7-7 fan shaped hole means that hole has forward laidback and lateral expansion angles at the same time. But, 7-7-0 fan shaped hole means hole has only lateral expansion angle.

Firstly, the coolant flow behavior inside the hole is computed at midsection of domains as it can be seen in **Figure 6.18** and in **Figure 6.19**. Velocity vectors for different geometry cases can be seen below in these figures. For all geometry cases with plenum condition of coolant flow except 4-4-0 case, two separation bubble regions which have been explained above are observed as common literature knowledge. The separation at the inlet that happens due to vena contracta effect exists for all cases. In addition, the flow in the diffuser tends to separate because of adverse pressure gradient. And, this separation situation really depends on the geometry angle. The separation occurs with different size due to the different pressure gradient with respect to boundary layer theory. This pressure gradient seems as strong or weak influencer with respect to the cross section area which is changed with expansion angle.

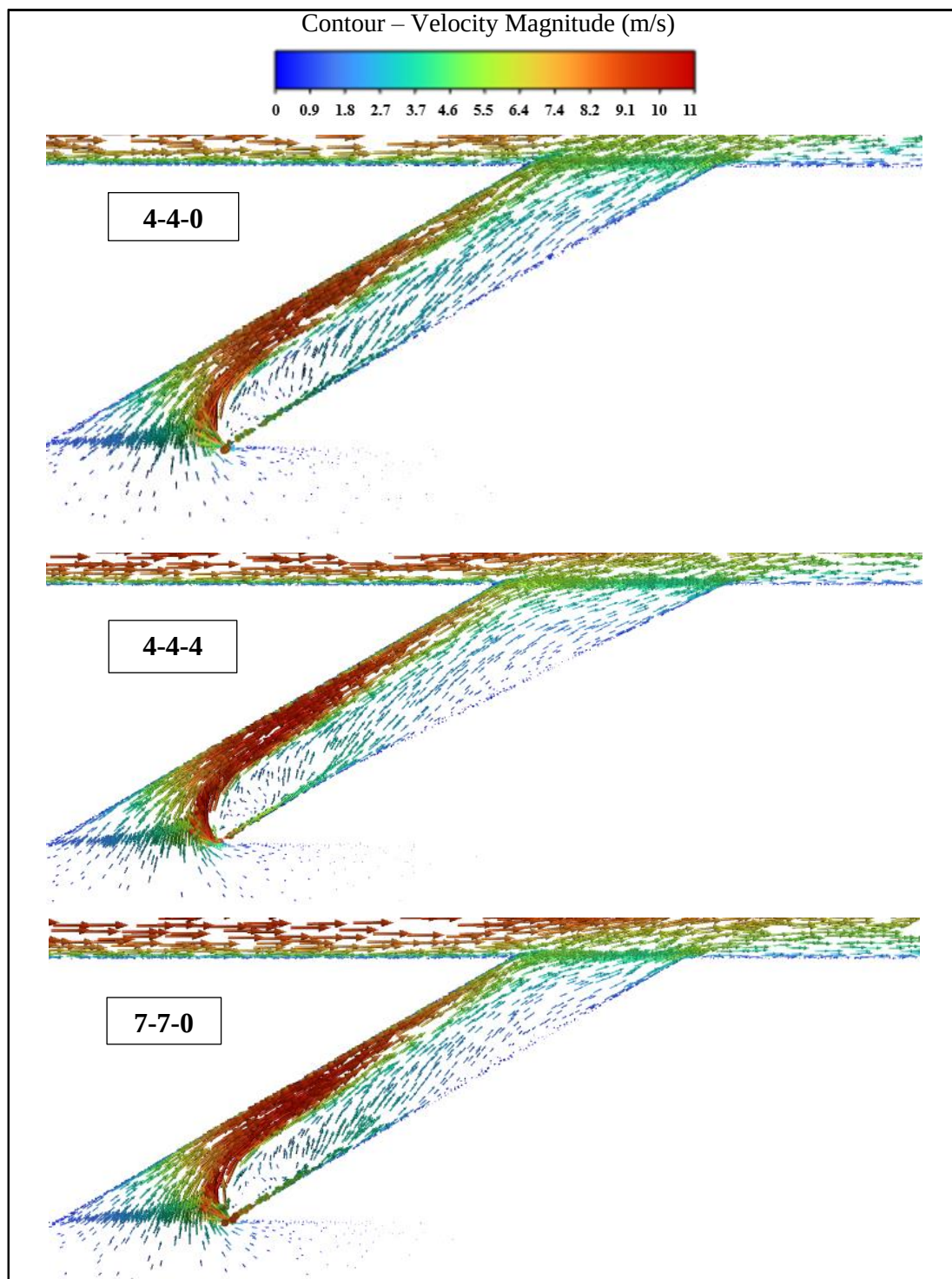


Figure 6.18: Computed Velocity Vectors at Mid Plane for Geometry Cases.

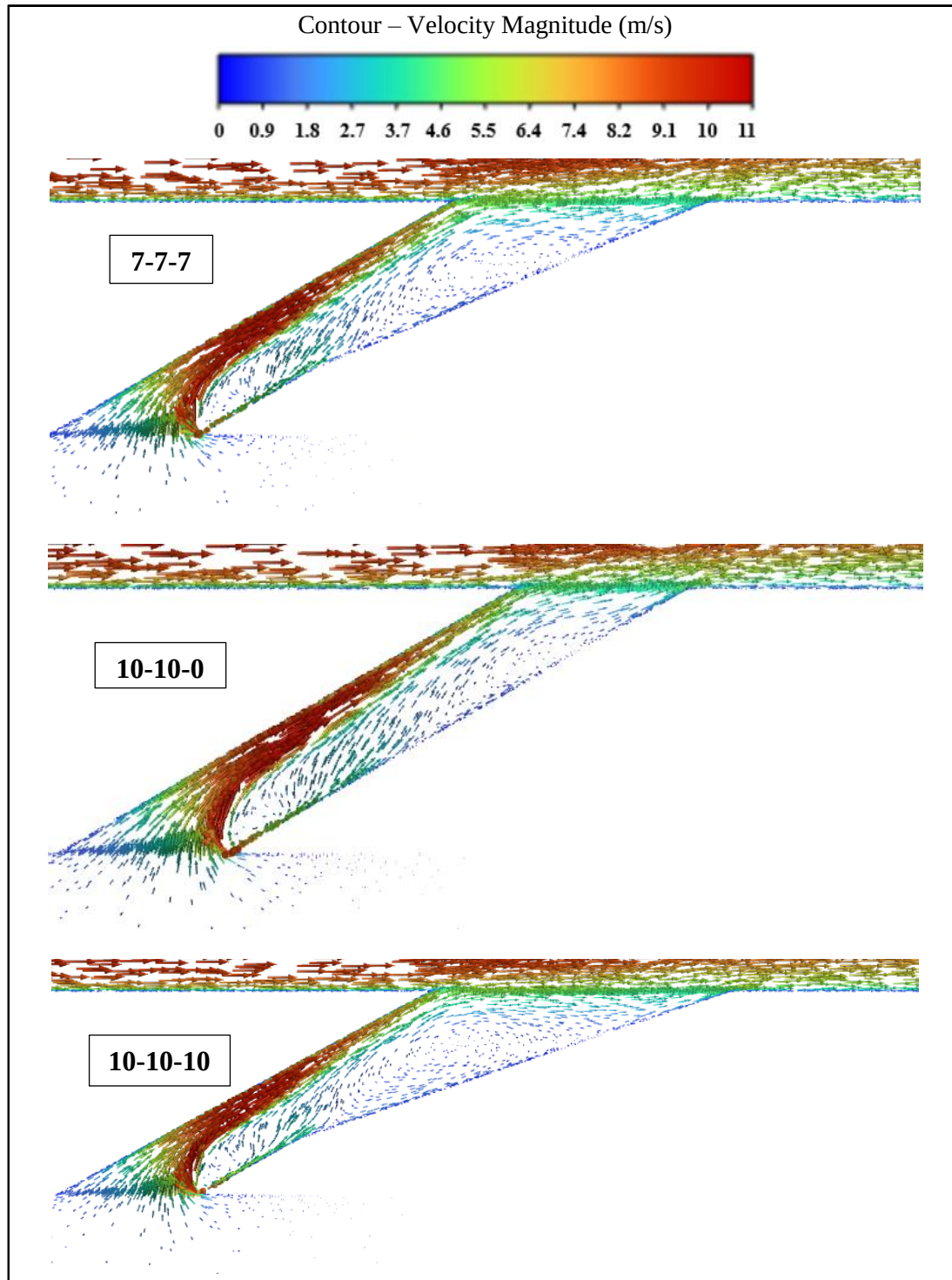


Figure 6.19: Computed Velocity Field at Mid Plane for Geometry Cases - II.

Figure 6.18 and **Figure 6.19** prove that the expansion angle for fan shaped hole effects the flow behavior especially in diffuser section. But, the velocity vectors at the inlet of cooling hole are observed with the common behavior. It means that the expansion

angle is not a influencer for vena contracta flow structure at the hole inlet. The vena contracta always seems due to below feeding plenum condition.

Length and height of separation bubbles in diffuser section which belong to different geometry cases are given in **Figure 6.20**. Values of length and height are normalized with respect to throat diameter of cooling hole.

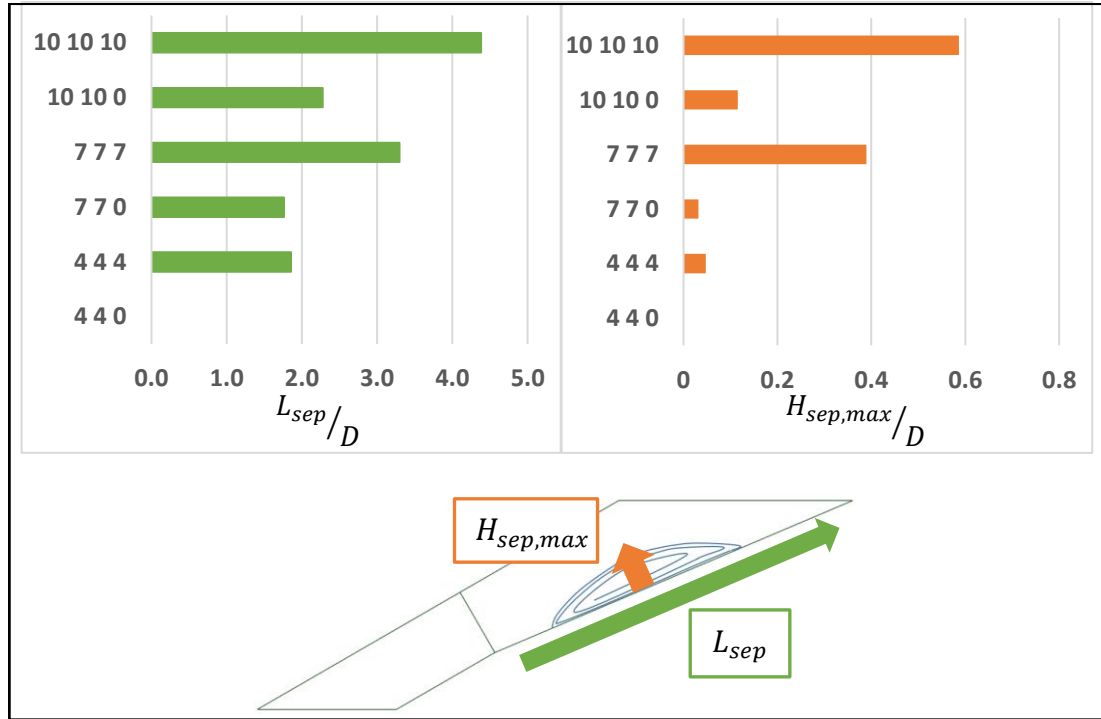


Figure 6.20: Dimensionless Length and Height of Separation Bubbles

Figure proves that the separation bubble region which normally occurs in expansion section of fan shaped hole is not found for only 4-4-0 case. The holes with laidback angle always have separation bubble in expansion section. And, separation bubble exists with different size for all geometry cases. Because, all cases have the different cross sectional area in diffuser section. The different cross sectional area leads to the different reduction of momentum and moderate or strong adverse pressure gradient in this section. Velocity gradient along the same streamline is calculated as negative and static pressure term in momentum equation will become positive ($dP/d > 0$) due to front minus sign. The higher static pressure, the lower kinetic energy which can lead to zero velocity or reverse flow. But, this configuration has the lowest cross sectional area and the highest coolant momentum in diffuser section causes the lower static

pressure gradient. As a result, only 4-4-0 case has the moderate adverse pressure gradient along expansion section.

Considering the length of separation with green bars, the longest separation bubble is observed as 4.4D for 10-10-10 case. The greater expansion angle causes the longer separation bubble. Considering the height of separation with orange bars the thinnest separation region is observed for 7-7-0 cases as 0.03D. In flow direction, the adverse pressure gradient is observed as moderate because of the small section area at the lower expansion angles. The velocity deficit at the leeward side of the hole caused by the jetting effect observes more discernible with higher expansion angles. The thickest one is found as 0.58D for 10-10-10 case.

Briefly, the greatest separation bubble is found for 10-10-10 case in terms of length and height as it is shown in **Figure 6.20**. As a result, the greater expansion angles which have the higher cross sectional area causes the greater separation bubble because of strong pressure gradient in diffuser section.

Velocity magnitude contours at throat section of cooling hole is seen in **Figure 6.21**. Maximum velocity magnitude is about constant for all degree cases and it is approximately found as 11 m/s. As it can be mentioned earlier, blowing ratio is constant for all cases and it means that the same amount of coolant flow is fed into shaped hole section. The effective mass flow area with respect to the equal amount of coolant air is nearly obtained as the same and coolant air behaves almost the same character at throat section in terms of maximum velocity magnitude regions with red color. The coolant flows close to the windward side of cooling hole for all cases because of plenum feeding condition as it can be seen in **Figure 6.21**. As a result, displacement of coolant flow to the windward side is exists for all expansion angles due to the vena contracta of plenum inlet and the same behavior at throat section seems for the different lateral and laidback angles.

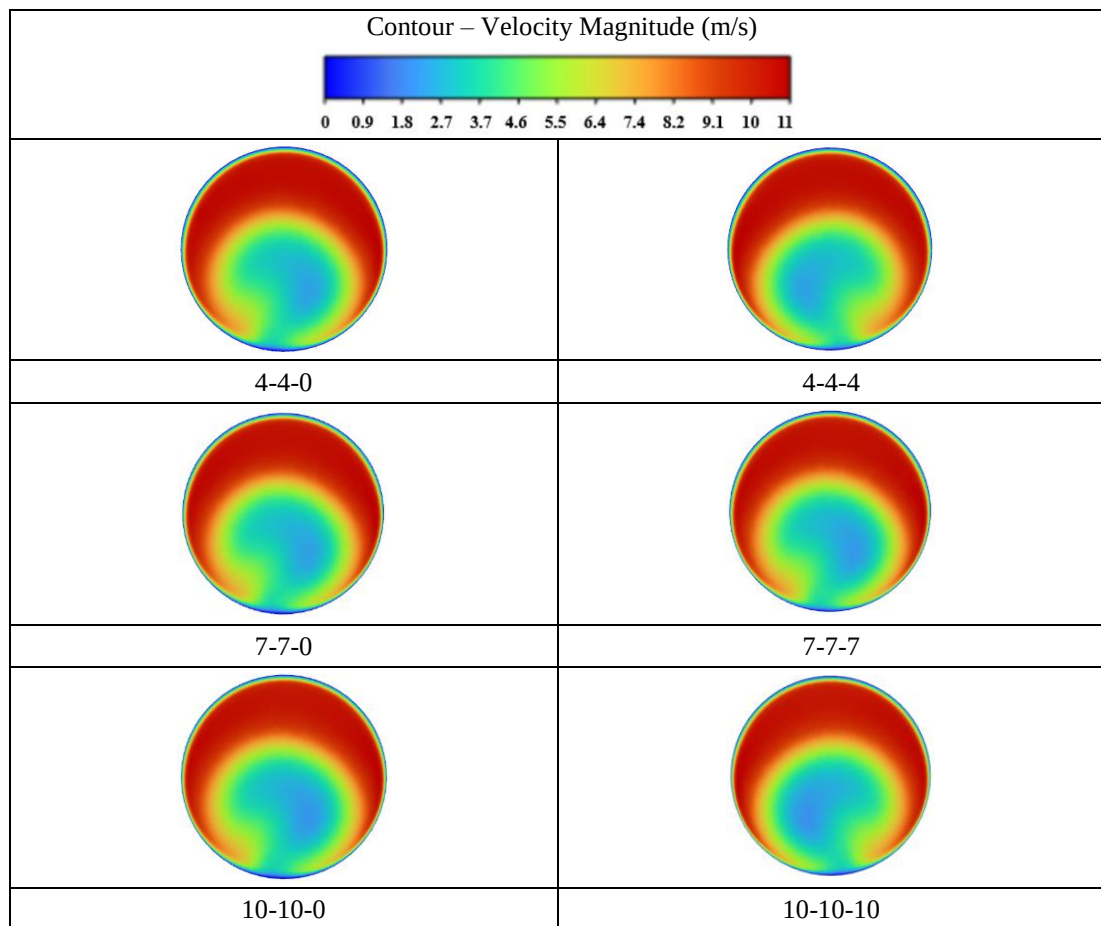


Figure 6.21: Velocity Magnitude Contours at Throat Sections.

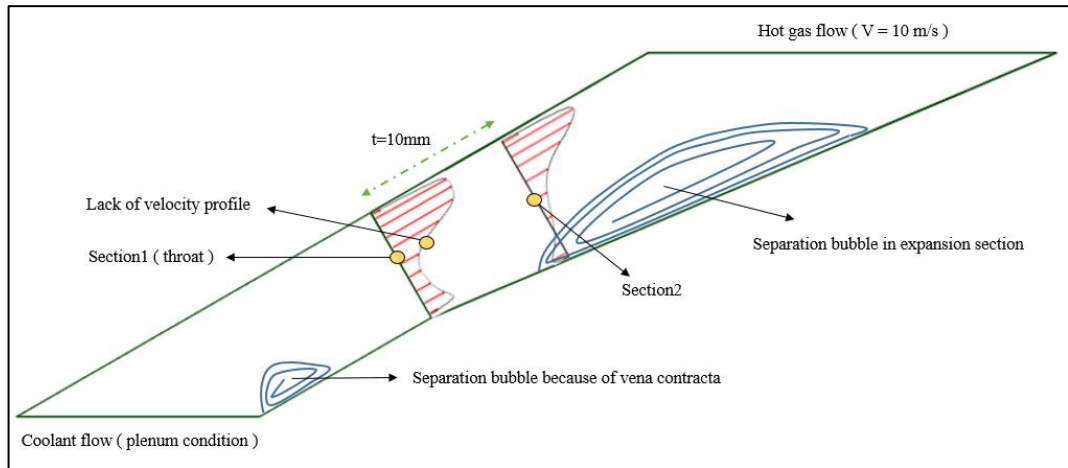
The one of the important film cooling design phenomena is the total pressure loss inside the hole. The redundant pressure loss might lead to mainstream injection into the component. Therefore, the designer must be aware of the hole design in terms of total pressure loss. **Table 6.2** shows the total pressure loss of created geometry cases for cylindrical and diffuser sections.

The total pressure loss at cylindrical section is almost the same for all geometry cases. As it is explained above, the coolant has the same flow behavior until the throat section. The windward side displacement with the same maximum velocity as 11 m/s is observed for all cases. Therefore, the total pressure loss at cylindrical section is about constant with the increased expansion angle. But, the total pressure loss at diffuser section is really depend on the form and size of separation bubble. The 10-10-10 case which has the biggest separation bubble is subjected to the highest loss. As a result, the greater separation bubble causes the higher total pressure loss in diffuser section.

Table 6.2: Total Pressure Loss of Different Expansion Angle Cases.

Case	$P_{t,loss}$ @cylindrical section	$P_{t,loss}$ @diffuser section
4-4-0	28.3 Pa	14.5 Pa
4-4-4	28.5 Pa	14.6 Pa
7-7-0	28.6 Pa	15.2 Pa
7-7-7	28.8 Pa	16.8 Pa
10-10-0	28.9 Pa	16.5 Pa
10-10-10	29 Pa	19.6 Pa

Scheme of 7-7-7 shaped hole flow behavior in the center plane is representatively given in **Figure 6.22**. Separation bubbles and velocity profiles for sections are shown. Axial velocity profiles of coolant flow are given at Section1 and Section2. There is 10 mm perpendicular distance between Section1 and Section2, these two sections represent throat and expansion parts of selected hole geometry respectively. And, separation bubble regions that are mentioned above are schematically drawn with blue lines.

**Figure 6.22:** Representative Scheme of Shaped Hole Flow Behavior.

The first observation is about passing way of coolant flow along hole. As it is mentioned above, coolant flow moves windward side of the cooling channel. Also, **Figure 6.22** shows this windward side movement with red arrow velocity profiles. Especially in symmetry plane, it moves close to the windward side of cooling. The highest velocity magnitudes which is known as jetting effect are observed as 11 m/s close to the windward side of cooling hole. The jetting effect creates velocity deficit

region at the hole leeward side [33] and the separation region inherently forms at this region.

The other observation is about non-uniform velocity profile in smooth channel. As it can be mentioned in section of different feeding condition cases, the uniform velocity profile along cross section can be achieved without plenum inlet. If the coolant has proper velocity directions as X and Y component with respect to injection angle, the uniform flow structure can be occur. But, plenum condition creates the secondary flow region which is called as vena contracta at the hole inlet. Velocities are disturbed by these secondary flows and the jetting effect zone takes place. Velocity profiles are given with red lines in **Figure 6.22** and the maximum velocity seems for the jetting effect zone and it equals to 11 m/s.

It can be summarized that non-uniform velocity area at throat is exists with the plenum condition for all expansion angles. But, the coolant might have velocity in feeding channel and this velocity in feeding channel will affect the amount of coolant flow rate and velocity distribution at throat section. In this thesis scope, the different coolant velocity cases in feeding channel will also be investigated to determine the changing of mass flow rate and velocity distribution at throat section.

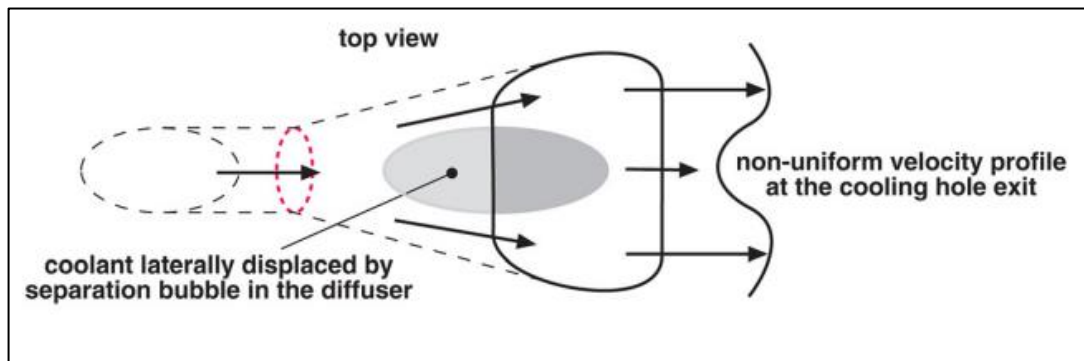


Figure 6.23: Schematic Top View of the Flow Structure [33].

Flow structure of the lateral expansion fan shaped hole geometry has been investigated by Saumweber et al. They have comparison for cylindrical and 14 degree lateral expansion. In addition, the effect of laidback angle has not been included to this investigation scope. They have reported that the coolant flow is displaced by separation bubble in the diffuser and it goes through wall creating the bimodal patterns as it can be seen in **Figure 6.23** [33].

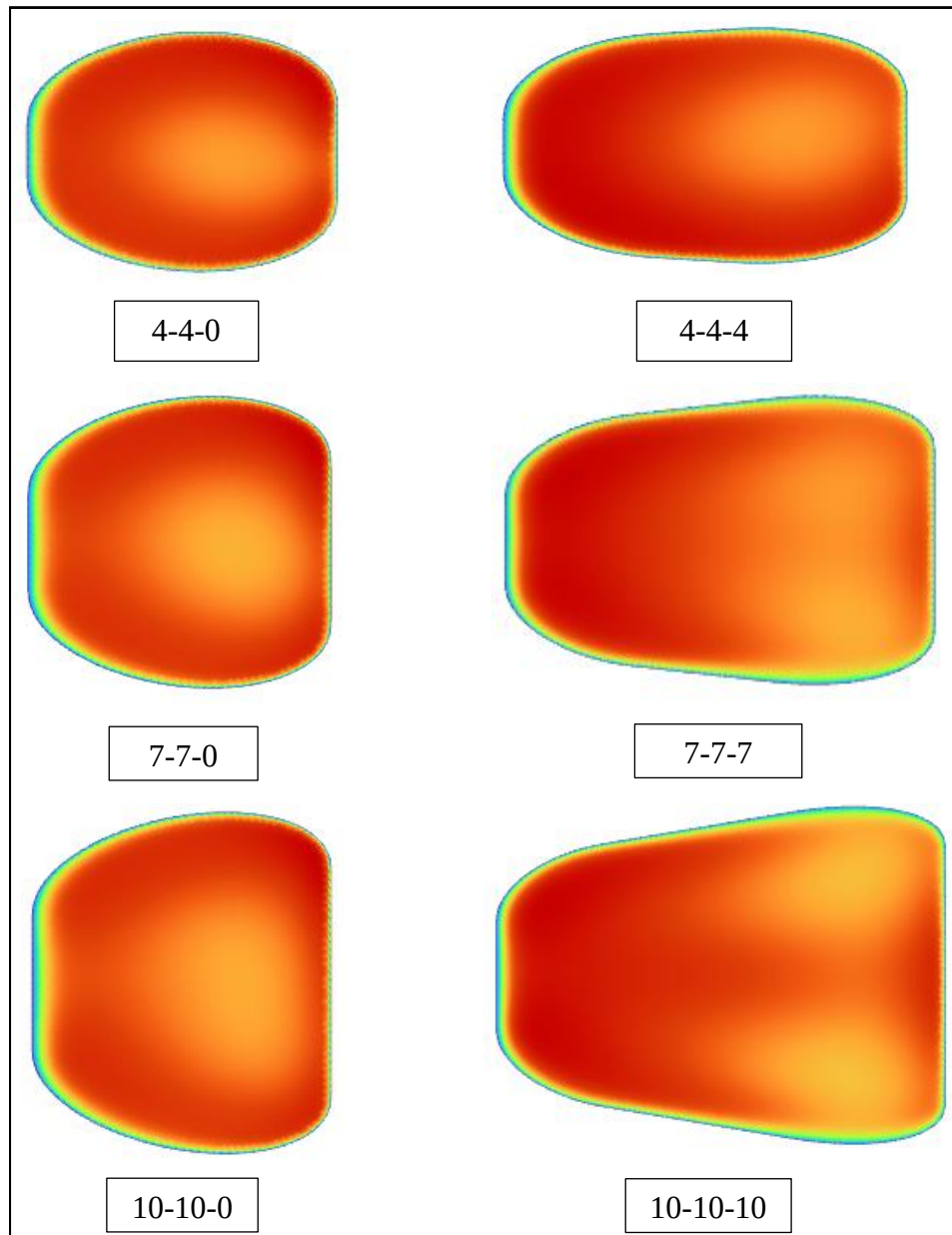


Figure 6.24: Dimensionless Velocity Contours at the Hole Exit.

In this thesis scope, this flow phenomena is considered for selected lateral and laidback angles. Flow behaviors and lateral film effectiveness of different shaped holes are discussed below. Exit velocity magnitude contours of fan shaped holes are shown in **Figure 6.24**. Coolant flow is laterally displaced by the separation bubble and expansion angle for the fan shaped hole as it has been shown in **Figure 6.23**.

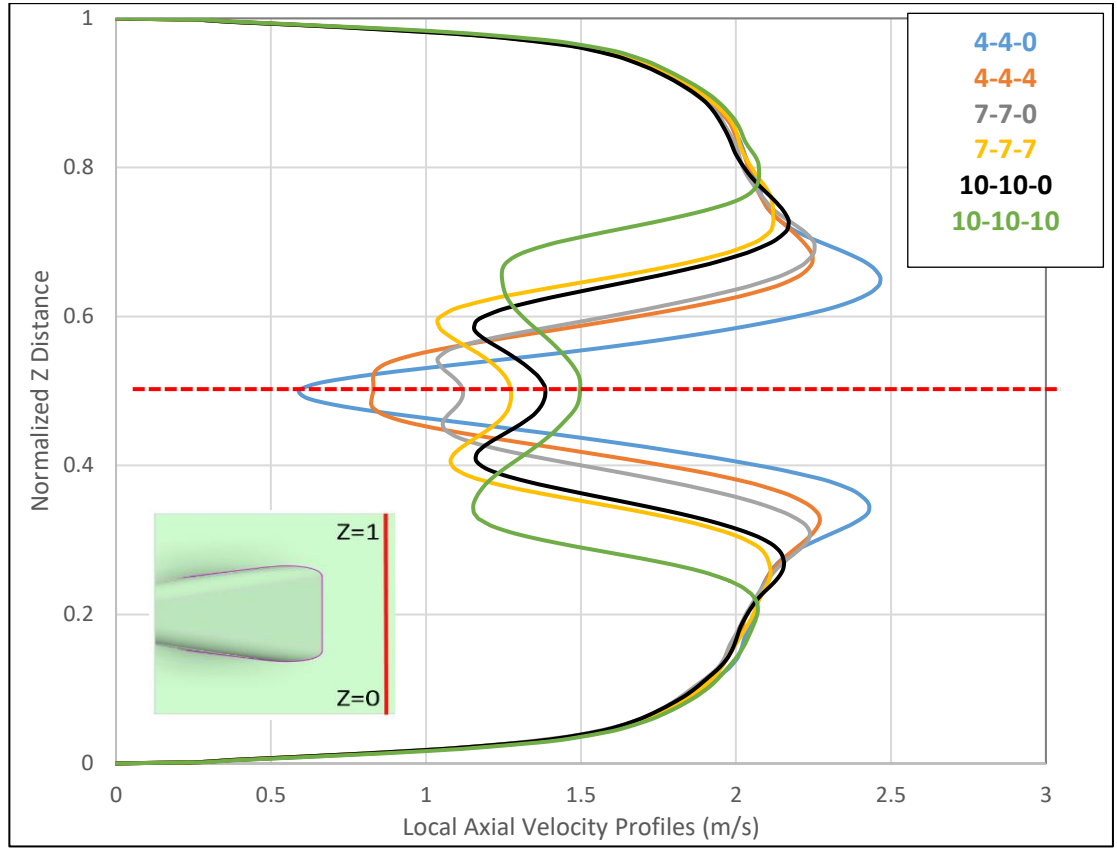


Figure 6.25: Local Axial Velocity Distributions at $X/D = 5$.

In this thesis, local axial velocity distribution just above the wall is investigated after mixing between coolant and mainstream. **Figure 6.25** shows axial velocity distribution of different geometry cases. Velocity distribution is taken 0.1 mm away from wall surface at $X/D = 5$ location. In this way, results are not effected by no slip condition and velocity is not equal to zero. And, the literature test rig has the measurements and findings at $X/D = 5$ location as earliest location.

As it is mentioned before, separation region in expansion section of the fan shaped hole is not exist just for 4-4-0. Therefore, the axial velocity profile is observed more symmetrical especially for 4-4-0 geometry if the symmetry plane is created for $Z_{\text{norm}} = 0.5$ as it can be seen in **Figure 6.25**. But, lateral displacement of coolant flow occurs when separation region starts to become exist. The bigger lateral displacement is observed for the greater angles and it causes the greater nonsymmetric flow behavior and local film effectiveness distribution. In literature, it is known as the bimodal effectiveness pattern.

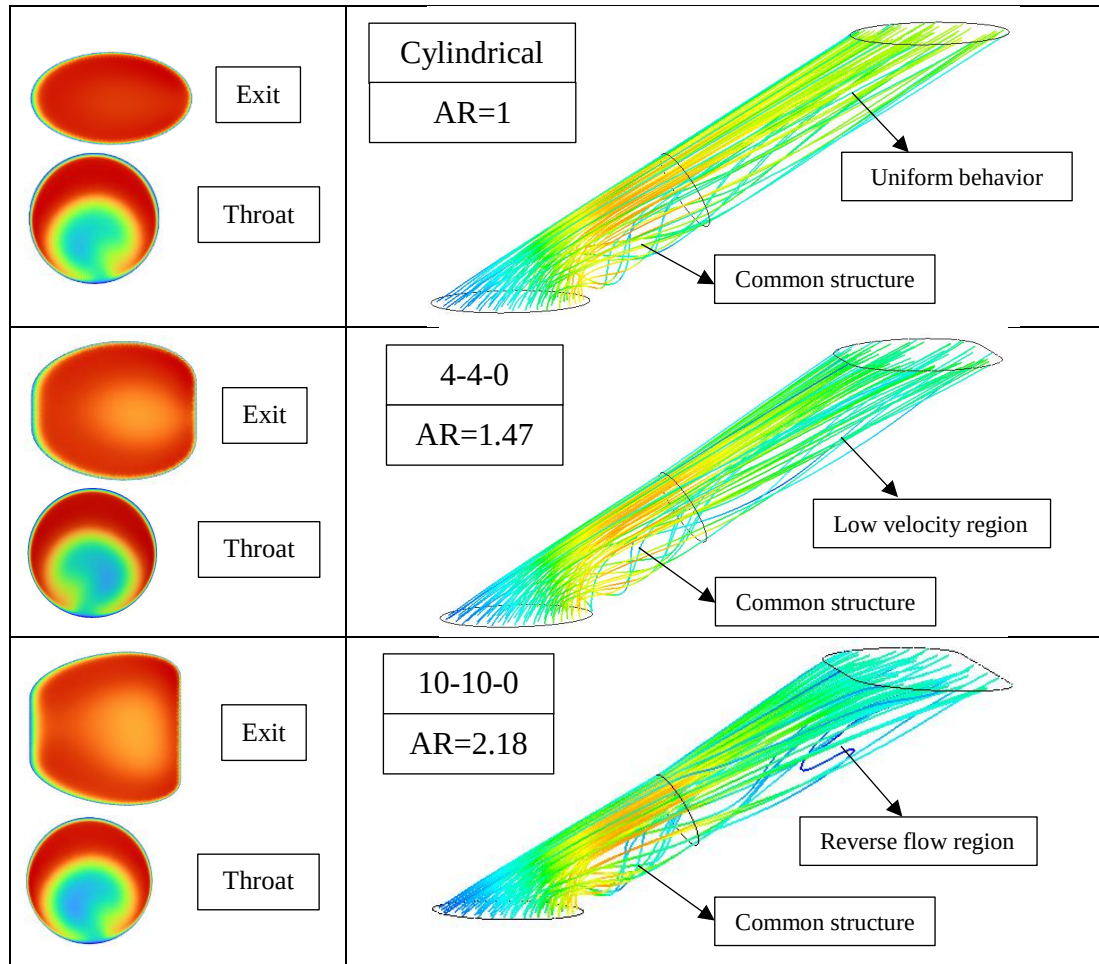


Figure 6.26: Detailed Three Dimensional Streamlines in Cooling Hole

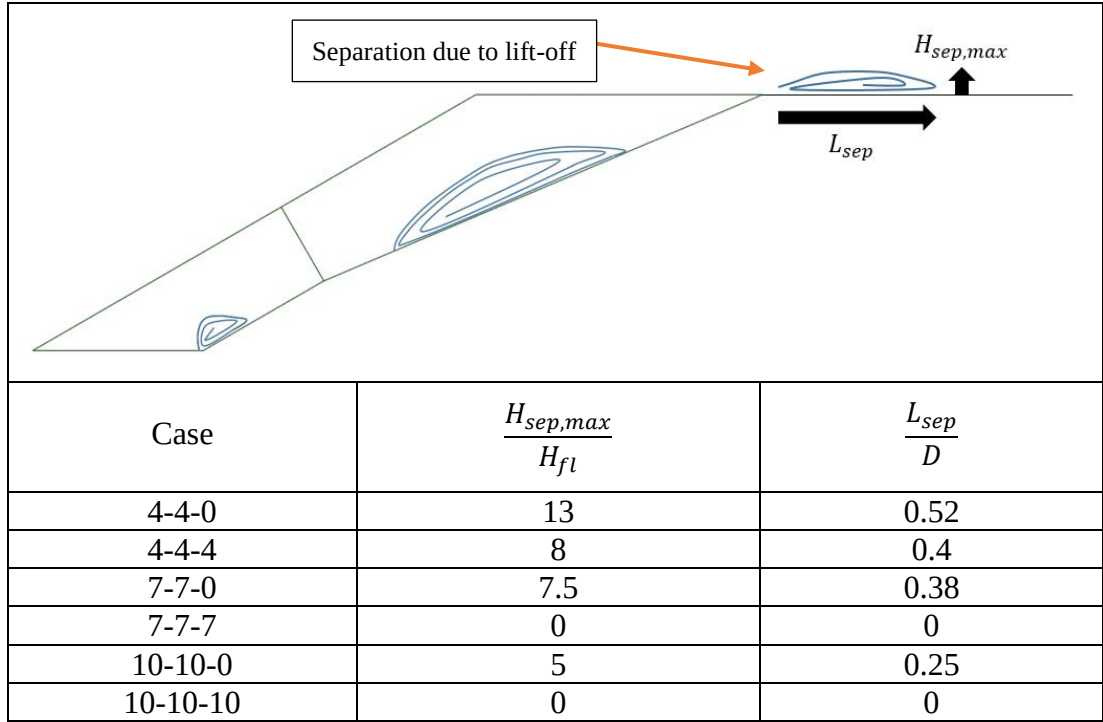
Figure 6.26 shows that three dimensional flow structure for different cooling hole geometries with velocity magnitude contours at the throat and exit. Three dimensional flow structure inside the cooling hole for different geometries is plotted to investigate low velocity region in the middle of hole exit. The common finding is about three dimensional flow behavior in cylinder section due to the plenum inlet. As it is mentioned earlier, the plenum inlet condition causes the vena contracta at the hole inlet and it leads to three dimensional flow especially close to the leeward side of cylindrical section. Considering the velocity magnitude contours at the hole exit, uniform velocity distribution without low velocity region in the middle is observed for only cylindrical hole. Because, the coolant flow recovers itself in terms of three dimensional effect due to the narrowest cross section area close to the cooling hole exit. But, low velocity region in the middle of hole exit is observed for both 4-4-0 and 10-10-0 cases. The lateral expansion angles as 4 and 10 degree lead to sidewall displacement of coolant flow and it creates low velocity region and separation bubble situations which depend

on expansion angle. Even if the separation bubble is not exist for 4-4-0 case as it is mentioned earlier, the low velocity region especially at the midsection is observed due to the sidewall displacement and the jetting effect inside the hole. But, the moderate adverse pressure gradient which means no any separation is still found due to the lowest static pressure gradient along streamline for the 4-4-0 case. For 10-10-0 case, the separation bubble can be realized in **Figure 6.26** and the greater AR causes the strong adverse pressure gradient. Then, the separation bubble takes place in expansion section of cooling hole at it is quantified before. As a result, the greater exit area effects the three dimensional flow behavior inside the hole and the velocity distribution at the hole exit.

The maximum height and length of separation bubbles which take place because of lift off condition just the beginning of cooling hole downstream surface, are given as dimensionless at **Table 6.3**. The height is normalized with respect to mesh first layer thickness and the length is normalized with respect to film cooling hole diameter. Considering only the lateral expansion angle cases, table proves that lift off condition is always observed for lateral expansion angle. But, lift off issue which effects the centerline effectiveness as negative is reduced with the higher lateral expansion angle from 4 to 10 degree. Considering the lateral and laidback angle cases, the separation due to lift off occurs for only 4-4-4 case. The higher expansion angles as 7 and 10 degree, the lift off is prevented by means of the greater exit area.

The maximum height and length of separation bubbles which take place because of lift off condition just the beginning of cooling hole downstream surface, are given as dimensionless at **Table 6.3**. The height is normalized with respect to mesh first layer thickness and the length is normalized with respect to film cooling hole diameter. Considering only the lateral expansion angle cases, table proves that lift off condition is always observed for lateral expansion angle. But, lift off issue which effects the centerline effectiveness as negative is reduced with the higher lateral expansion angle from 4 to 10 degree. Considering the lateral and laidback angle cases, the separation due to lift off occurs for only 4-4-4 case. The higher expansion angles as 7 and 10 degree, the lift off is prevented by means of the greater exit area.

Table 6.3: Separation Bubble Size after Ejection for Different Geometry Cases.



Comparison of local film effectiveness for different X/D locations can be seen in **Figure 6.27**. Results are given for the same analysis conditions as $M = 1$, $DR = 1.5$. When considering only the lateral expansion angle, it is found that jet effect triggers the centerline effectiveness at mid plane is reduced with increased lateral angle. While centerline effectiveness is expected lower for 7 degree than 4 degree expansion, almost the same centerline effectiveness is found as 0.9 for 4-4-0 and 7-7-0 cases. Because, the 4-4-0 case has jet effect which causes higher lift off condition just downstream of the hole than 7-7-0 case. This finding is proved at **Table 6.3**.

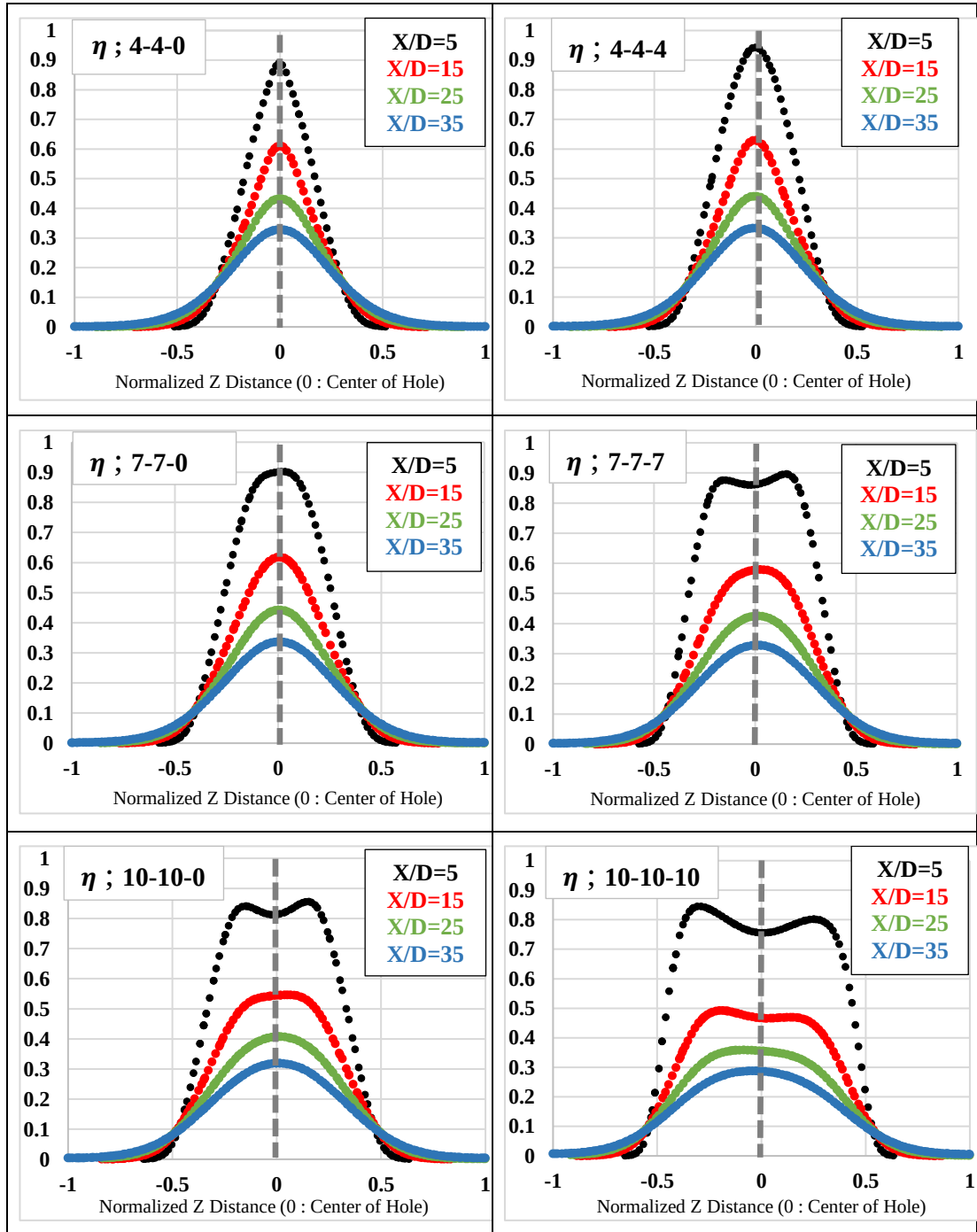


Figure 6.27: Comparison of Local Film Effectiveness for X/D Locations.

Lateral spreading starts to exist for 7 degree lateral case. But, it seems still symmetric as it can be seen in **Figure 6.27**. This situation takes place due to the very thin separation region as 0.03D of 7-7-0 case. In addition, the greater 0.11D 10-10-0 case separation bubble dislocates the coolant flow side wards leads to the bimodal effectiveness patterns. Therefore, the greatest lateral displacement and the lowest centerline effectiveness is get at the biggest lateral angle 10-10-0 case as it is expected.

When considering the lateral and the laidback expansion angles together, lateral spreading of lateral and laidback expansion cases starts earlier than only lateral cases. But, it seems symmetric for 4-4-4 case due to very thin 0.04D separation bubble in comparison with the higher laidback angle cases. In this reason, the bimodal effectiveness pattern is not observed for 4-4-4 case. Also, the centerline effectiveness of 4 degree laidback case has the highest value. Because, the lift off condition which decrease the centerline effectiveness is reduced with 4 degree laidback angle compared to only lateral 4 degree expansion angle. This finding is proved with velocity magnitude vectors at midsection in **Figure 6.28**. Figure also shows that the velocity magnitudes of separation region are reduced with the higher AR.

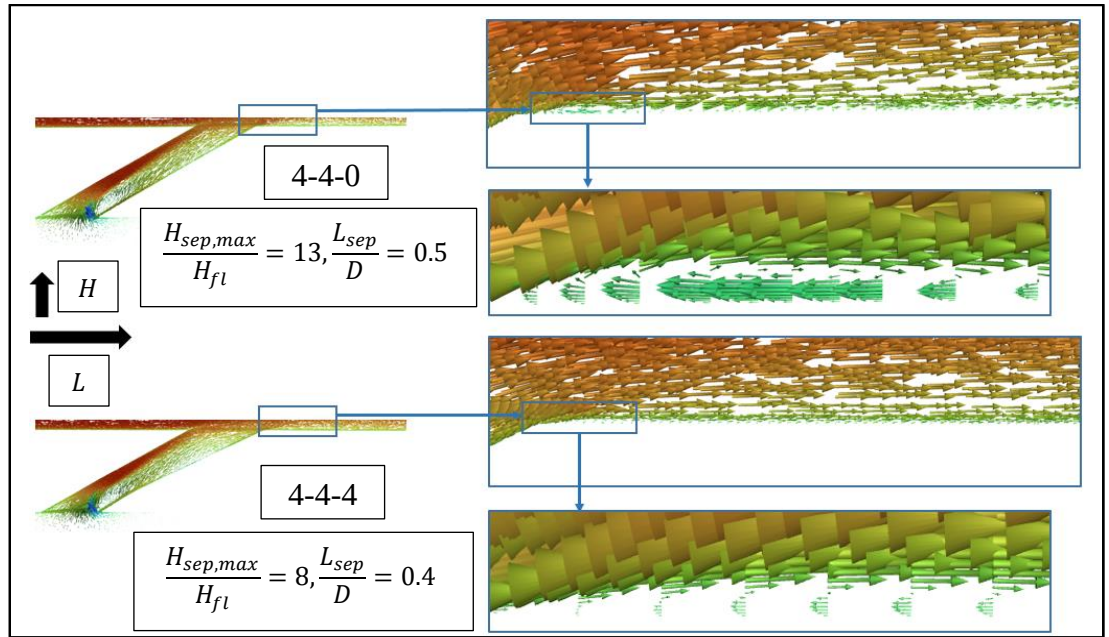


Figure 6.28: Velocity Vectors after Ejection at Mid Plane.

It is found that jet effect triggers the centerline effectiveness at mid plane is reduced with increased lateral angle. So, the higher laidback angle which contains the greater coolant displacement to side wards leads to decrease the centerline effectiveness, increase the lateral spreading.

When considering the effect of axial distance, the greatest spreading for any location is observed for 10-10-10 case. The centerline effectiveness decrease along axial path from $X/D = 5$ to $X/D = 35$ in each separate case. When looking at the black and the

blue lines for the same case in **Figure 6.27**, the lateral spreading of coolant increases in terms of just distance or length not magnitude.

As a result, 10-10-10 the lateral and the laidback angles case have the longest lateral spreading, the lowest centerline effectiveness due to the reasons given above. **Figure 6.27** proves that the bimodal effectiveness pattern of coolant flow may exist for different expansion angle. For 4 and 7 degree lateral and 4 degree laidback angles, the bimodal pattern of effectiveness is not noticeable because of unexisting or lower separation region.

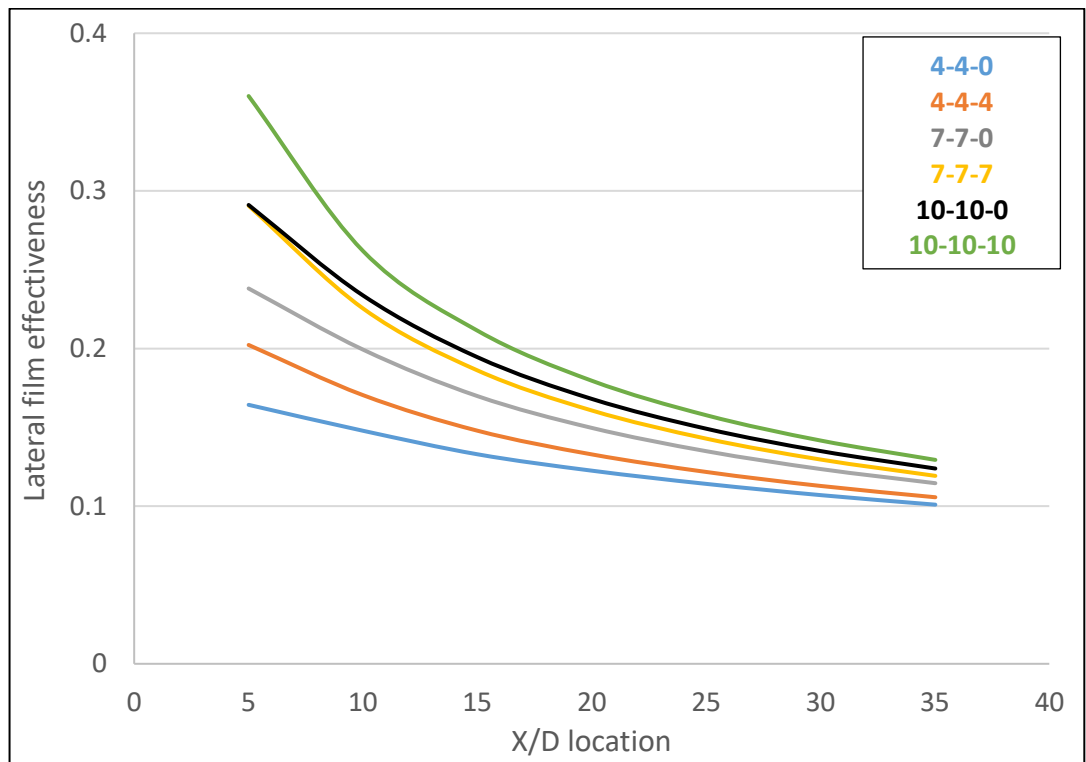


Figure 6.29: Comparison of $\bar{\eta}$ for Geometry Cases.

Figure 6.29 shows the lateral film effectiveness distributions for different geometry cases. Results are belong to $M = 1$, $DR = 1.5$ and $Tu_{\infty} = 0.5\%$ conditions. For the same analysis conditions, the different angle of fan shaped hole causes significant shifts in effectiveness except 7-7-7 and 10-10-0 geometries.

The generated geometry with 4, 7 and 10 degree lateral and expansion angles proves that the bigger separation bubble is an intended nature for a fan shaped hole. Because, higher expansion angle and cross section area enhance the lateral effectiveness and

consequent spreading of the coolant flow. But, similar lateral effectiveness distributions are attained only for the 7-7-7 and 10-10-0 geometries.

The effectiveness distributions show the decreasing character with downstream distance as it can be seen in **Figure 6.29**. This decreasing behavior proves that the expansion angle dramatically effects the close distance of downstream region especially before the $X/D = 20$ location. For the far away distance from the cooling hole exit ($X/D > 20$), the expansion angle is not an important influencer on the lateral film effectiveness. As Thole at al. has mentioned before [22], the blowing ratio is an important film cooling parameter on lateral film effectiveness for the far distance of cooled surface. The greater blowing ratio leads to the higher lateral film effectiveness for any location of cooled surface unlike the geometry expansion angle.

The lateral averaged maximum effectiveness seems at $X/D = 5$ location as 0.36. And, it belongs to the highest lateral and laidback expansion angles 10-10-10 case. The higher expansion angle has the increased exit area for the diffuser. In this way, an intense contact can be acquired for earlier distance of hole downstream to gain higher lateral film effectiveness.

For the comparison between 4-4-0 and 10-10-10 cases, the maximum enhancement is up to 0.21 or about 120% and the minimum enhancement is up to 0.025 or about 23% in effectiveness. For far away from the injection location ($X/D > 20$) enhancement factor dramatically decreases. The other gains are observed between these two percentages.

In summary of geometry cases with plenum condition, the higher expansion angle effects the flow structure in cooling hole. The separation bubble region in expansion section of fan shaped hole starts to form with increased expansion angle. It causes nonsymmetric velocity profile and local film effectiveness distribution downstream of the cooling hole. The coolant flow is replaced the wall side wards by separation bubbles and the greater lateral film effectiveness can be achieved.

6.5. Results of Coolant Velocity Cases

Until now, the plenum condition is utilized to feed the film cooling holes in this thesis. While the plenum condition is widely used in terms of film cooling studies, it is not

necessary to represent the internal supply passage of a blade. Internal supply coolant passage can be observed with velocity.

Four representative crossflow conditions are created in order to investigate the effect of coolant supply passage velocity. These velocity cases have been already given at **Table 5.7**. Briefly, first case as a plenum is the main condition that is used in validation and geometry cases. Different Mach numbers in the feeding channel are used to reflect crossflow effect. Coolant velocities are defined as 0, 0.1, 0.2 and 0.25 Ma respectively. Main stream velocity is defined as 80 m/s, 0.2 Ma for different four cases. In addition main stream temperature is assigned as 400 K and pressure ratio is kept the same. The effects of different mainstream velocity is not included into this thesis scope in order to investigate only the effect of coolant velocity between impingement insert and inner wall for stability of comparison as it is mentioned above.

Table 6.4: Result Summary of Created Coolant Velocity Cases.

Name	$P_{t,c} / P_{s,m}$	Ma_c	DR	Ma_m	$\dot{m}_c$	M
Case0	1.05	0	2	0.2	B	1.5
Case1	1.05	0.1	2	0.2	1.2B	1.9
Case2	1.05	0.2	2	0.2	1.42B	2.25
Case3	1.05	0.25	2	0.2	1.43B	2.28

Table 6.4 shows the result summary of created coolant velocity cases. Coolant mass flow rate is computed at throat section of fan shaped hole for each case. Mass flow rate is computed for Case0 plenum condition and is called the base mass flow rate as ‘B’. For other cases, computed coolant mass flows are normalized with this base mass flow with respect to ratio.

While the pressure ratio $P_{t,c} / P_{s,m}$ is kept as the same 1.05, increased coolant mass flow rate is observed with the higher coolant velocity in feeding channel. Also, blowing ratio is directly proportional to the coolant mass flow with respect to formula as it is given in theory section with 2.32 reference number. The higher coolant mass flow rate causes the higher blowing ratio and the changing of blowing ratio inherently occurs with the coolant velocity in feeding channel.

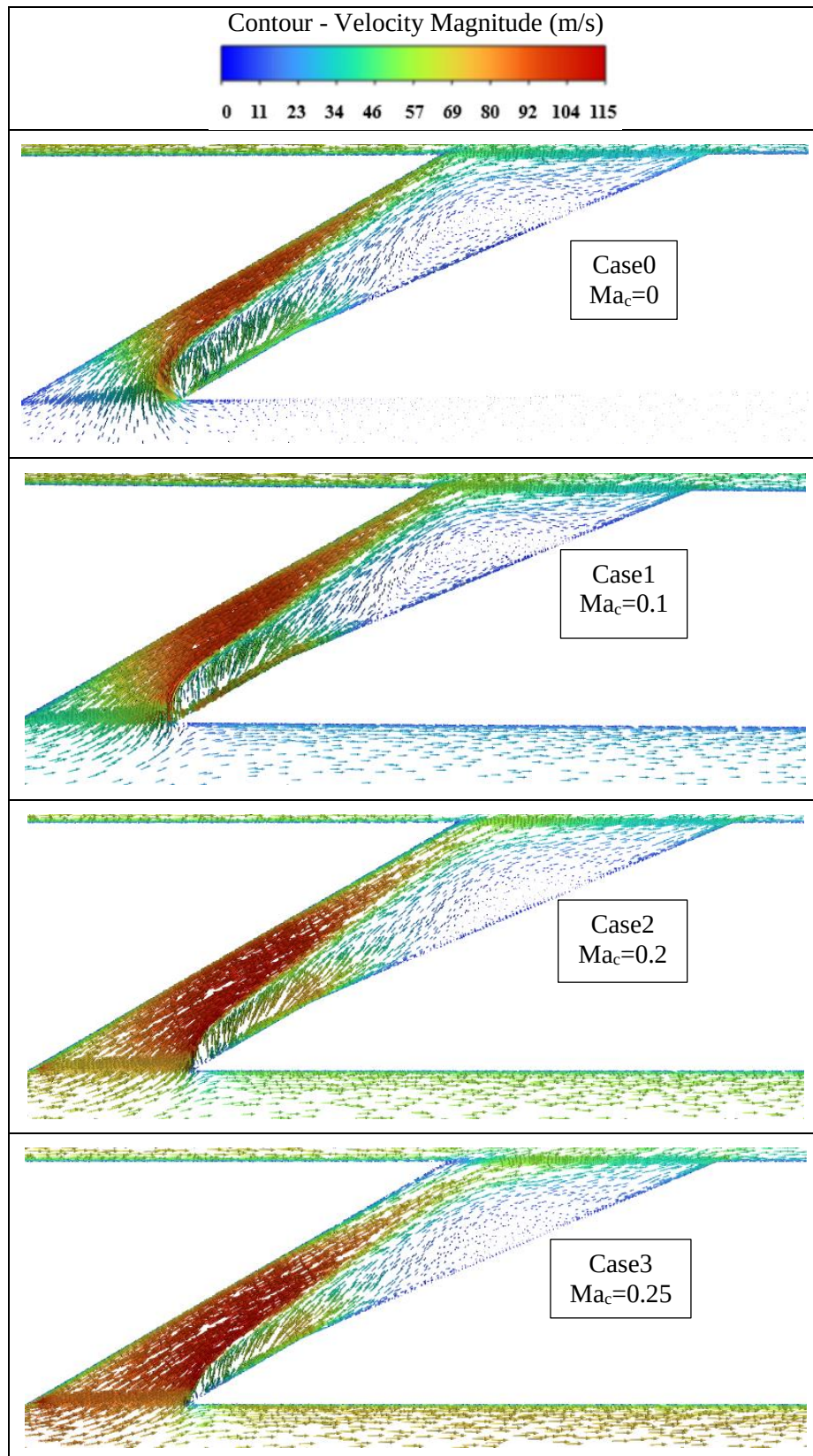
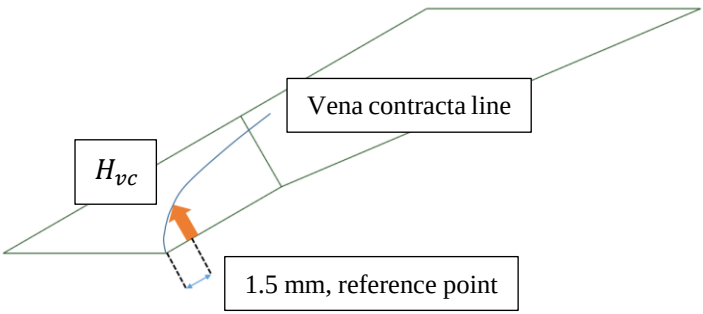


Figure 6.30: Computed V Vectors at Midsection.

Contours of velocity magnitude at mid plane and Mach number at throat section can be seen in **Figure 6.30**. Figure shows that jetting effect is observed for all cases. But, they are computed as different size with respect to feeding velocity. The higher coolant velocity in feeding channel causes the greater jetting effect area and magnitude. The maximum coolant velocity and mass flow area are found at $Ma_c=0.25$ case.

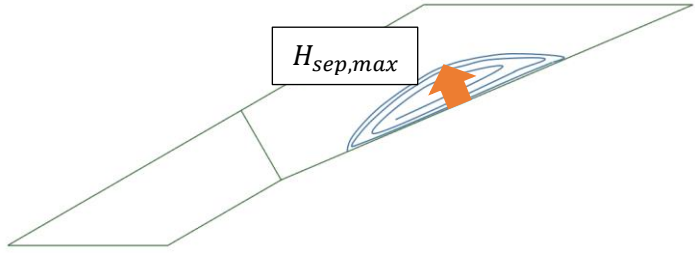
Computed velocity vectors show that the region at the entrance takes place due to vena contracta is suppressed and reduced with the higher coolant velocity. This situation is proved by means of **Table 6.5** findings. For determined reference point which is 1.5 mm away from hole inlet location as it is shown below, the height of vena contracta is investigated. The height of vena contracta decreases from 0.5D to 0.16D for the increased coolant velocity with respect to below table.

Table 6.5: Dimensionless Height of Vena Contracta Bubble.

Name	$\frac{H_{vc}}{D}$	
Case0, $Ma_c=0$	0.5	
Case1, $Ma_c=0.1$	0.37	
Case2, $Ma_c=0.2$	0.25	
Case3, $Ma_c=0.25$	0.16	

Computed velocity vectors in **Figure 6.30** also show that the different size of separation bubble is observed with the increased coolant velocity. The normalized maximum height of separation bubble at mid plane in expansion section of fan shaped hole is given for the different coolant velocity cases as it can be seen in **Table 6.6**.

Table 6.6: Bubble Dimensionless Height for Diffuser Section.

Name	$\frac{H_{sep,max}}{D}$	
Case0, Ma _c =0	0.45	
Case1, Ma _c =0.1	0.4	
Case2, Ma _c =0.2	0.27	
Case3, Ma _c =0.25	0.04	

The faster coolant is able to flow moderate along cross section and adverse pressure gradient at the expansion section of fan shaped hole almost vanishes. Therefore, very thin velocity deficit region as 0.04D is observed for 0.25 Ma coolant cases. It is found that the size of separation region is reduced with the higher coolant velocity with respect to **Table 6.6**. As a result, the faster coolant flow which have high dynamic pressure may easily deal with the adverse pressure gradient and the formation of separation bubble is reduced.

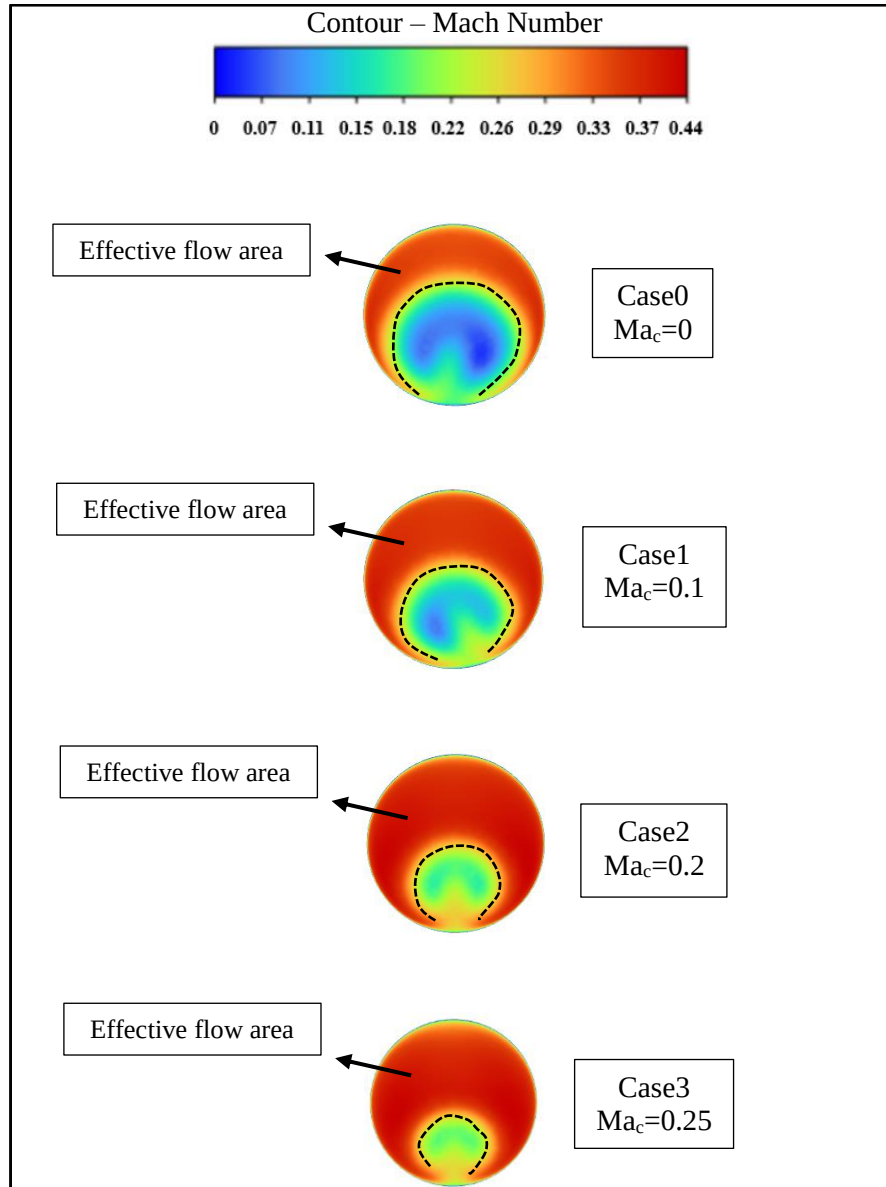


Figure 6.31: Computed Ma Number at Throat Section.

Contours of Mach number at throat section can be seen in **Figure 6.31**. Mach number contours at throat section prove that the coolant pass through this section more effective. More effective means that maximum velocity region which is shown with red color at throat section or effective mass flow area increases. As it has been mentioned above, nonsymmetric flow behavior occurs due to coolant displacement and vena contracta effect. This flow behavior disappears and flow pass through the throat section more symmetric. Because, vena contracta and separation region are suppressed by the faster coolant flow. Additionally, recovery effect which comes from higher dynamic pressure at the entrance of cooling hole enhances the amount of film cooling flow.

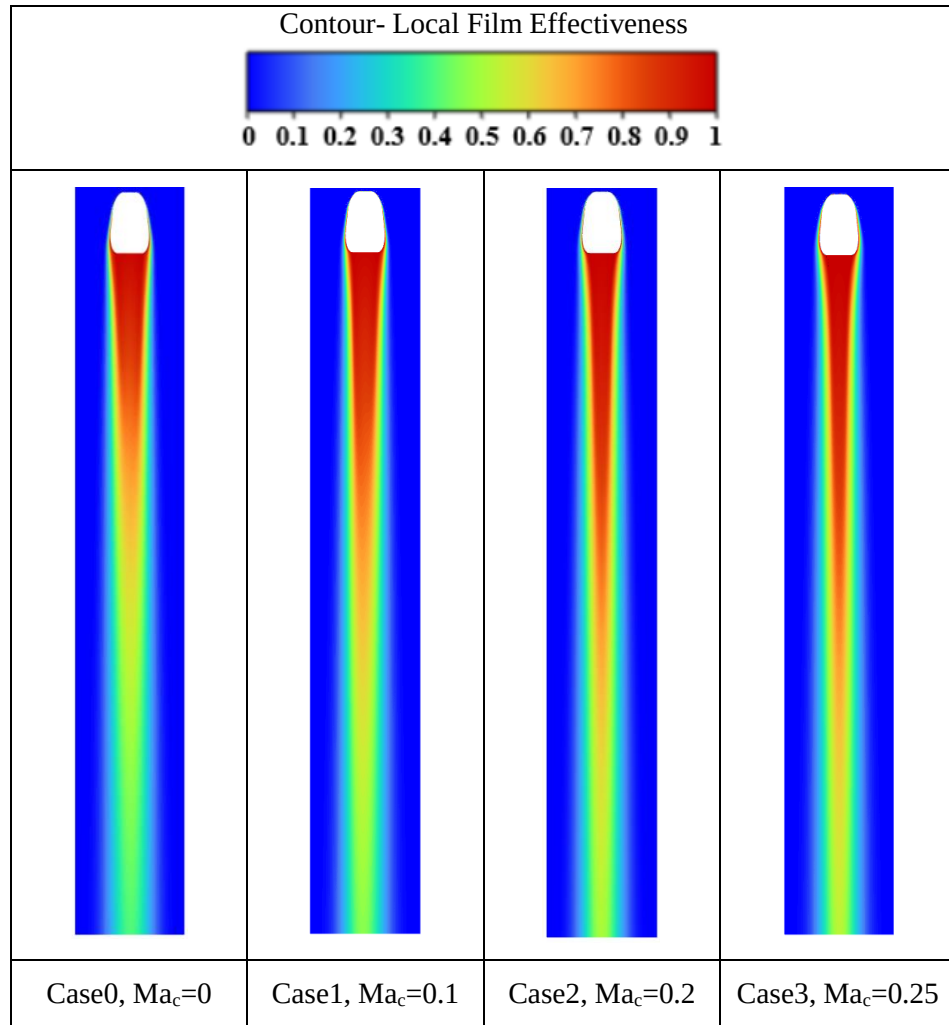


Figure 6.32: Comparison of η_{film} for Different Coolant Velocity Cases.

Figure 6.32 shows contours which mean comparison of local film effectiveness for different coolant velocity cases. As it can be seen in figure, the more coolant velocity means the far reached downstream surface of the cooling hole. It means that the greater coolant velocity causes the red color which has the relatively higher local film effectiveness at the end of cooled surface, only green color which has lower film effectiveness than red one is observed for plenum zero coolant velocity case.

Especially at the beginning of the cooled surface, lateral spreading reduces with the increased coolant velocity or blowing ratio. Because, separation bubble region which displaces the coolant flow lateral way decreases as it has been mentioned above. In this way, coolant tends to pass through straighter without lateral side wall displacement.

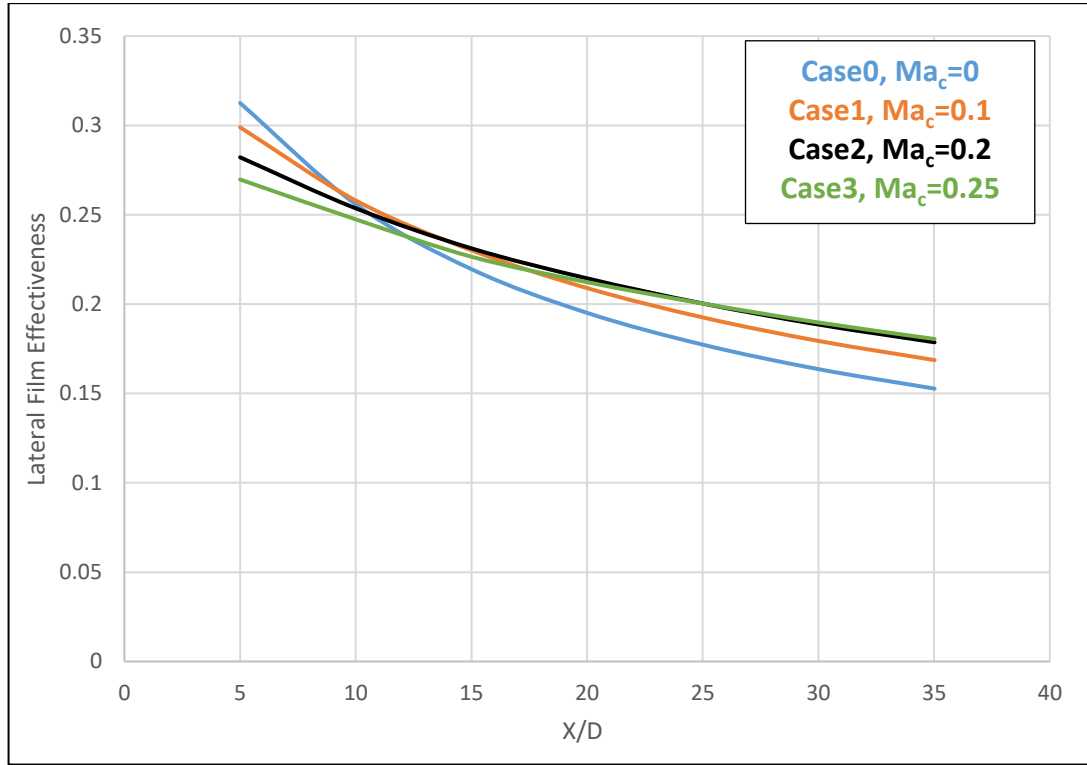


Figure 6.33: Comparison of $\bar{\eta}$ for Different Coolant Velocity Cases.

Figure 6.33 shows distributions which represent comparison of lateral film effectiveness for different coolant velocity cases. There are two different effected regions in terms of positive and negative influence. Lateral effectiveness is gained and lost at the same time for each case with the greater coolant velocity. The location of these gain and loss variation is about $X/D = 10$ point as it can be seen in figure.

Before the location of $X/D = 10$, the lateral effectiveness decreases with the greater coolant velocity in feeding channel. The flow gained more jet penetration effect causes the less lateral spreading. Mainstream which has the higher velocity in hole can easily disturb the lateral mixing effectiveness especially earlier X/D location. The decreased lateral effectiveness takes place because of the greater coolant velocity is the common observation for all numerical conditions.

On the other hand, the faster coolant flow is able to reach the farther location of cooled surface with the greater local film effectiveness. The lateral effectiveness increases with the higher coolant velocity after the location of $X/D = 10$ if the coolant velocity is from 0 to 0.2. For $Ma_c=0.25$ case, different lateral effectiveness behavior from the other cases is observed except $X/D < 10$ region.

Considering any cooled surface location, optimum lateral effectiveness distribution exists in $Ma_c=0.2$ case. $Ma_c = 0.2$ case has the highest lateral effectiveness distribution except small axial distance ($X/D < 10$). This is the special topic when its velocity compares with the mainflow velocity. They both are equal to the each other. If the coolant Mach number is bigger than the mainstream Mach number ($Ma_\infty = 0.2$), the observations lose their stability and trend. And, it is hard to understand physics of system.

In summary, taking into account the gained experiences from literature and this thesis in considering the effects of coolant velocity, it is difficult to guess how the results are affected by increased coolant velocity. But, there might be a problem in terms of lateral film effectiveness or effective mass flow area at throat when the coolant Mach number is higher than the mainstream Mach number.

Four crossflow conditions are also performed for cylindrical cooling hole configuration in order to observe the differences at the same upstream and downstream conditions. In this way, the geometry effect on discharge coefficient is evaluated for cylindrical and fan shaped holes.

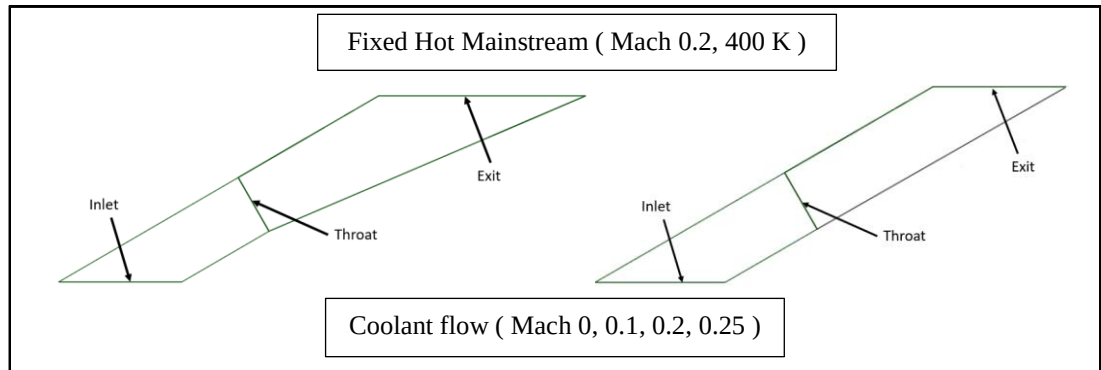


Figure 6.34: Representative Cylindrical and Shaped Hole with BCs.

Mainstream and coolant flow conditions together with hole shapes are shown in **Figure 6.34**. The pressure ratio, which is defined according to coolant total pressure to mainstream static pressure is kept constant as 1.05 for all cases. In addition, the mainstream conditions are kept as 400 K, 0.2 Ma and density ratio is taken as 2 for each case. Also, locations of computed analysis parameters are highlighted as ‘Inlet’, ‘Throat’ and ‘Exit’ in **Figure 6.34**.

Table 6.7: Result Comparison for Cylindrical and Shaped Hole.

Name	Ma_c	$P_{dyn,inlet}$ [Pa]	Cd	$P_{s,inlet} / P_{s,throat}$	M
Case0	0	2133	0.82	1.053	1.5
Case0-cyl	0	1385	0.66	1.032	1.28
Case1	0.1	2614	0.99	1.052	1.9
Case1-cyl	0.1	1583	0.76	1.024	1.48
Case2	0.2	4238	1.16	1.049	2.25
Case2-cyl	0.2	2619	0.78	1.001	1.51
Case3	0.25	5104	1.18	1.032	2.28
Case3-cyl	0.25	3395	0.72	0.991	1.4

Table 6.7 shows the comparison between cylindrical and shaped hole for created coolant velocity cases. The ideal mass flow rate is calculated for cylindrical hole properties which are $P_{t,c} / P_{s,m} = 1.05$, 7.75 mm diameter and coolant temperature.

Figure 6.35 shows the calculation parameters (total pressure of coolant in feeding channel and the static pressure of mainstream) to calculate the ideal mass flow rate, which is used to find the global discharge coefficient. Hence, values above one for fan shaped hole are reported in **Table 6.7**, also as reported by Taslim [36] and Schulz [37]. From a practical point of view these are the parameters that can be set/controlled in the design phase of turbine cooling flow network. As the coolant flow is accelerated inside the feeding channel, its static pressure drops and dynamic pressure rises for the set constant total pressure, and the dynamic pressure recovery of cooling hole is dependent upon the geometry and the ability of the hole to take in the flow efficiently. Essentially, when the streamlines are aligned towards the hole with reduced separation regions occurring at the entrance of the hole, dynamic pressure recovery increases. The effectiveness of the hole admitting coolant flow rises due to this effect, which is demonstrated in discharge coefficient values.

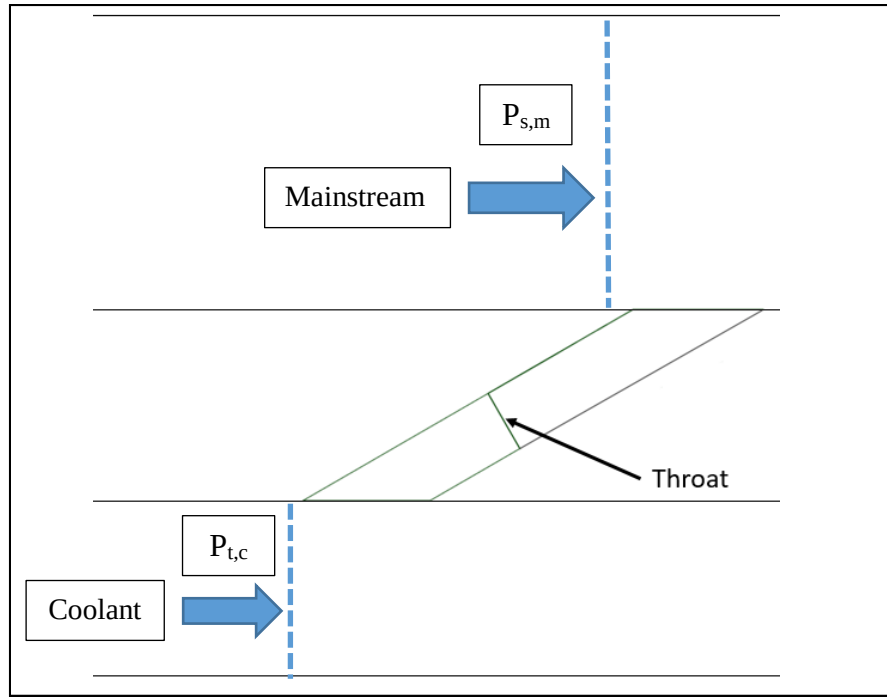


Figure 6.35: The Ideal Mass Flow Rate Parameters for Global Cd Calculation.

The discharge coefficient for each case is shown in **Figure 6.36**. Blue and red color represent cylindrical and shaped hole respectively. Due to the calculation way of global discharge coefficient, fan shaped hole has Cd value bigger than one for 0.2 and 0.25 coolant Mach number case. But, it seems reasonable to remain comparison stability with respect to the same pressure ratio ($P_{t,c} / P_{s,m}$) value which belongs to each case.

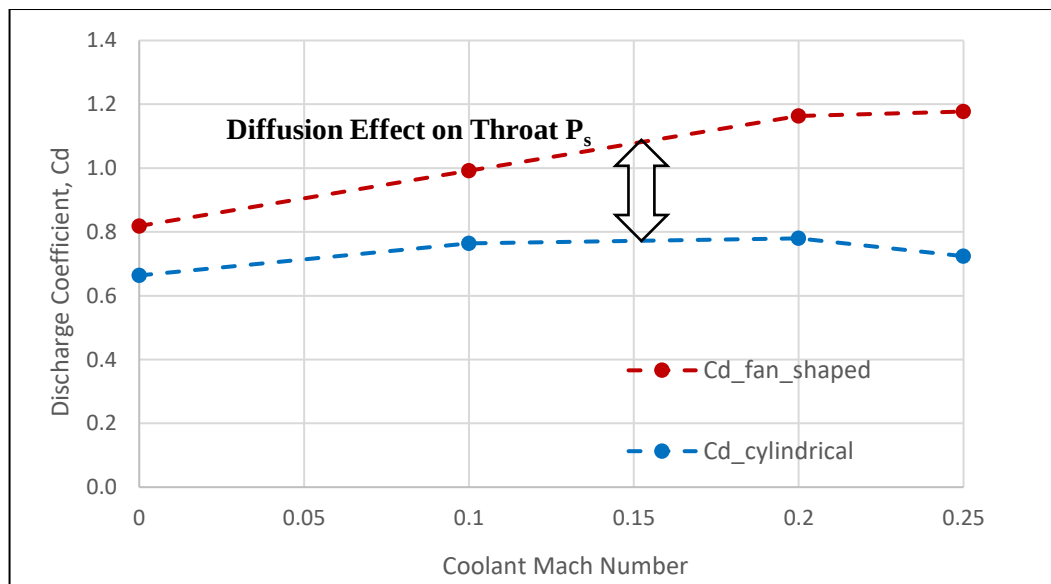


Figure 6.36: Comparison of Cd for Cylindrical and Shaped Hole.

Considering the same coolant Mach number case, shaped film cooling hole leads to higher discharge coefficient than cylindrical film cooling hole. It means that the higher blowing ratio, which is a function of amount of coolant flow can be achieved with shaped hole at the same condition. The reason for that is the presence of lower static pressure obtained locally at throat region with fan shaped hole compared to cylindrical hole. This local low static pressure draws flow effectively through the entrance. These observations are demonstrated in **Figure 6.37** and **Figure 6.38**.

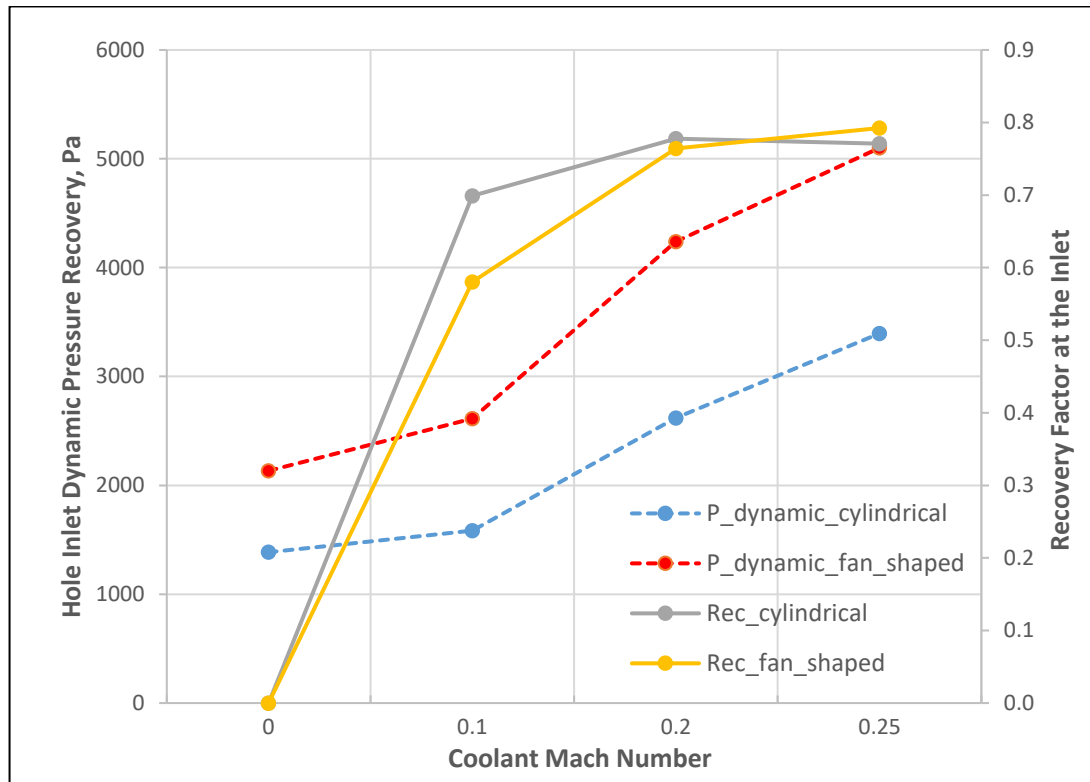


Figure 6.37: Dynamic Pressure at the Hole Inlet for Different Mach Number.

Figure 6.37 shows the dynamic pressure at the inlet of the cooling hole for different coolant Mach number cases. This figure shows that the greater coolant Mach number in feeding channel causes the greater dynamic pressure enhancement at the hole inlet. Therefore, the amount of coolant flow increases when the static pressure at the inlet is greater than the static pressure at the throat section.

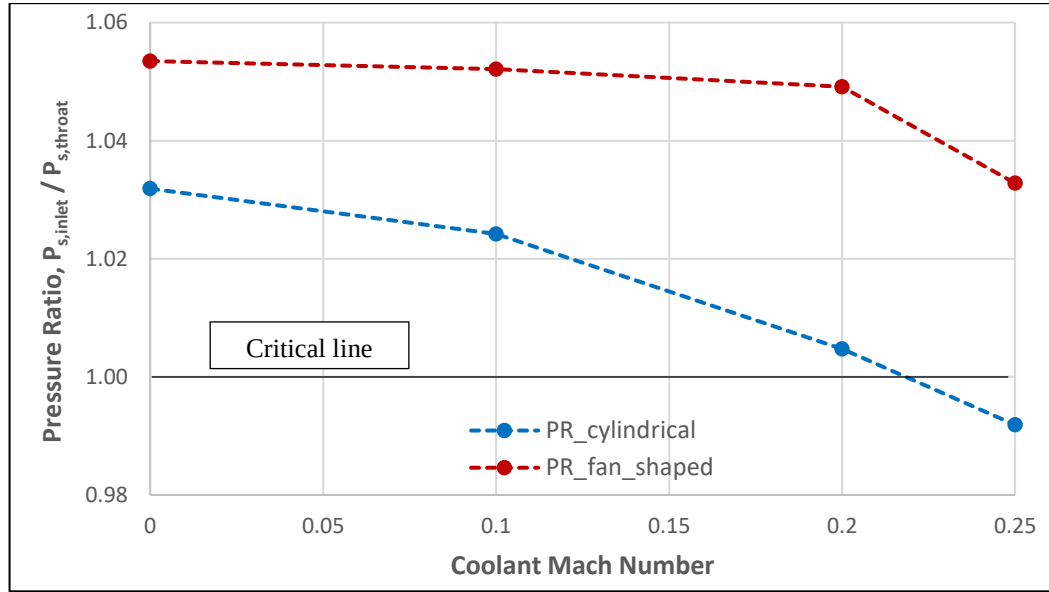


Figure 6.38: Pressure Ratio, $P_{s,inlet} / P_{s,throat}$ for Different Mach Number.

The static pressure ratio of inlet to throat section is given for each case in **Figure 6.38**. This figure proves that shaped hole has the higher pressure ratio than cylindrical hole. Because, fan shaped hole acts as a diffuser after the throat section, matching same mainstream condition at the exit. The lower static pressure at throat section is observed for shaped hole and this situation leads to the greater amount of coolant flow for shaped cooling hole at the same condition.

When the inlet static pressure starts to drop below the throat static pressure, the amount of coolant flow starts to decrease. As it is shown in **Figure 6.38**, there is a critical pressure ratio line for $P_{s,inlet} / P_{s,throat} = 1$ value and the coolant mass flow rate of cylindrical hole decreases for the 0.25 Mach number which is found below this line. Also, the pressure ratio of shaped hole dramatically decrease but it is still higher than 1 for 0.25 Mach number. The dynamic pressure recovery still helps to increase the amount of coolant flow.

The region up to the throat becomes a diffusing channel when the cooling hole consist of expansion section. When coolant feeding channel has a higher Mach number with the same upstream total pressure, the greater discharge coefficient is observed because of the higher dynamic pressure recovery at the entrance of the hole and diffusion process in expansion section. Therefore, entire hole becomes a diffusing channel composed of front cylindrical segment and rear diffuser segment which effect the actual mass flow rate for fan shaped hole.

7. CONCLUSION

The different geometry and coolant velocity cases are mainly investigated in this thesis. Firstly, numerical models are validated with selected literature test rig results. Solution parameters are determined in order to get proper results in terms of validation. The closest numerical results to selected literature test rig is obtained with RKE turbulence model. Also, boi mesh control domains are utilized to get the proper results.

There are two main geometry cases which about different feeding condition and expansion angle in this thesis scope. The cases have different coolant feeding condition are created with and without plenum condition for cylindrical and fan shaped holes. These are utilized in order to investigate the behavior of internal flow inside cooling hole, the formation of separation bubbles, the velocity distribution at throat section, the three dimensional streamlines, the vortex structures after coolant ejection. While the feeding condition effects the coolant behavior inside cooling hole, the detachment issue is not effected by the upstream feeding condition. The detachment situation only depends on the geometry type and the cylindrical hole leads to CRVP and the hot gas entrainment toward the wall surface. The rotational motion of CRVP causes the hot gas entrainment which degrade the film cooling effectiveness on surface. But, the fan shaped hole prevents the CRVP and weak secondary flows are observed with this configuration. In this way, the fan shaped hole is able to have high laterally averaged film effectiveness.

After that, the different fan shaped geometries are created to investigate effects of expansion angles. The different flow behavior in hole and lateral effectiveness distribution occur for each expansion angle. The increase of film hole cross section at expansion section leads to velocity deficit region. Therefore, the coolant flow is subjected to an adverse pressure gradient which triggers an inherent separation. For the greatest the expansion angle, the biggest the separation region as 4.4D and 0.58D for 10-10-10 case. In this way, the side wards displacement of coolant flow which increase lateral effectiveness exists. The highest lateral film effectiveness is found as 0.36 at $X/D = 5$ location of 10-10-10 case. So, the separation in the diffuser section of fan shaped hole is a desirable feature. In addition, the shaped holes lead to reduction

in coolant velocity by the increased exit area and it causes the benefit in terms of film cooling effectiveness even the higher blowing ratio is utilized. The highest the expansion angle, the biggest the three dimensional flow structure and separation bubble inside the cooling hole and this structure creates the greatest total pressure loss of coolant flow as 19.6 Pa for 10-10-10 case.

Then, coolant velocity cases are considered. The main observation is about effective mass flow area and blowing ratio. The higher coolant velocity in feeding channel causes the higher effective area and blowing ratio for the same boundary conditions except coolant Mach number. Because, the coolant with greater velocity decrease the separation size and increase recovery effect at the entrance. The maximum blowing ratio is found as 2.28 for the greatest 0.25 coolant Mach number case of 7-7-7 fan shaped hole. The other observation is about lateral effectiveness. There are positive and negative effected regions are obtained for increased coolant velocity. Also, the complex and unpredictable trends might be occur when the coolant Mach number is higher than the mainstream Mach number.

Considering the same Mach number case for cylindrical and shaped hole, shape hole has the higher amount of coolant flow than cylindrical hole due to dynamic pressure recovery at the entrance and diffusion in expansion section. Fan shaped hole can adjust itself with lower static pressure at throat section, matching same exit mainstream condition. In this way, it utilizes higher dynamic pressure gain which causes the greater amount of coolant flow. Maximum discharge coefficients are found as 1.18 and 0.78 for 7-7-7 fan shaped and cylindrical holes respectively. Also, the greater coolant Mach number leads to the greater amount of coolant flow until static pressure of inlet and throat equals to each other. When the inlet static pressure starts to drop below the throat static pressure, the amount of coolant flow starts to decrease. For the higher Mach number case which has lower pressure ratio than one, the hot gas ingestion from mainstream into cooling hole and feeding channel can be observed.

The findings of this thesis clearly show that the designer of fan shaped cooling hole must know the geometry details and operating conditions to determine cooling performance of an intended configuration.

REFERENCES

- [1] Baskharone E. A., (2006), “Principles of Turbomachinery in Air-Breathing Engines”, 2nd Edition, Cambridge University Press.
- [2] Çengel Y. A., Boles M. A., (2015), “Thermodynamics an Engineering Approach”, 8th Edition, McGraw-Hill.
- [3] Rolls Royce, (2005), “The Jet Engine”, 5th Edition, Wiley.
- [4] Han J. C., Dutta S., Ekkad S., (2013), “Gas Turbine Heat Transfer and Cooling Technology”, 2nd Edition, CRC Press Taylor & Francis Group
- [5] Sinha A. K., Bogard D. G., Crawford M. E., (1991), “Film Cooling Effectiveness Downstream of a Single Row of Holes with Variable Density Ratio”, *Journal of Turbomachinery*, 113(3), p. 442-449.
- [6] Thole K. A., Sinha A. K., Bogard D. G., Crawford M. E., (1992), “Mean Temperature Measurements of Jets with a Crossflow for Gas Turbine Film Cooling Application” *Rotating Machinery Transport Phenomena*, p. 69-85.
- [7] Pedersen D. R., Eckert E. R. G., Goldstein R. J., (1977), “Film Cooling With Large Density Differences Between the Mainstream and the Secondary Fluid Measured by the Heat-Mass Transfer Analogy”, *Journal of Heat Transfer*, 99(4), pp. 620-627.
- [8] Schmidt D. L., Sen B., Bogard D. G., (1996), “Film Cooling With Compound Angle Holes: Adiabatic Effectiveness”, *Journal of Turbomachinery*, 118(4), pp. 807-813.
- [9] Wayne S. K., Bogard D. G., (2007), “High Resolution Film Cooling Effectiveness Comparison of Axial and Compound Angle Holes on the Suction Side of a Turbine Vane” *ASME Journal of Turbomachinery*, 129(2), pp. 195-203.
- [10] Pietrzyk J. R., (1989), “Film-Cooling Effectiveness Downstream of a Single Row of Holes With Variable Density Ratio”, *ASME Journal of Turbomachinery*, 113(3), pp. 442-449.
- [11] Pietrzyk J. R., Bogard D. G., Crawford M. E., (1990), “Effects of Density Ratio on the Hydrodynamics of Film Cooling”, *ASME Journal of Turbomachinery*, 112(3), pp. 437-443.
- [12] Pietrzyk J. R., Bogard D. G., Crawford M. E., (1989), “Hydrodynamic Measurements of Jets in Crossflow for Gas Turbine Film Cooling Applications”, *ASME Journal of Turbomachinery*, 111(2), pp. 139.
- [13] Moussa Z. M., Trischka J. W., Eskinazi S., (1976), “The Near Field in the Mixing of a round Jet with a Cross-Stream”, *Journal of Fluid Mechanics*, 80(1), pp. 49-80.

- [14] Fric T. F., Roshko A., (1994), "Vortical Structure in the Wake of a Transverse Jet" *Journal of Fluid Mechanics*, 279, pp. 1-47.
- [15] Haven B. A., Kurosaka M., (1997), "Kidney and Anti-Kidney Vortices in Crossflow Jets", *Journal of Fluid Mechanics*, 352, pp. 27-64.
- [16] Kampe T. auf dem, Völker S., Sämel T., Heneka C., Ladisch H., Schulz A., Bauer H. J., (2011), "Experimental and Numerical Investigation of Flow Field and Downstream Surface Temperatures of Cylindrical and Diffuser Shaped Film Cooling Holes", *ASME Journal of Turbomachinery*, 135, pp. 21-34.
- [17] Meyer K. E., Özcan O., Westergaard C.H., (2002), "Flow Mapping of a Jet in Crossflow with Stereoscopic PIV", *Journal of Visualization*, 5(3), pp. 225-231.
- [18] Bernsdorf S., Rose M. G., Abhari R. S., (2006), "Experimental Validation of Quasi-Steady Assumption in Modelling of Unsteady Film-Cooling", *ASME Journal of Turbomachinery*, 130, pp. 127-137.
- [19] Fawcett R. J., Wheeler A. S., He L., Taylor R., (2012), "Experimental Investigation into Unsteady Effects on Film Cooling", *ASME Journal of Turbomachinery*, 134, pp. 1491-1502.
- [20] Eberly M. K., Thole K. A., (2014), "Time-Resolved Film-Cooling Flows at High and Low Density Ratios", *ASME Journal of Turbomachinery*, 136(6).
- [21] Schroeder R. P., Thole K. A., (2014), "Adiabatic Effectiveness Measurements for a Baseline Shaped Film Cooling Hole", *ASME Journal of Turbomachinery*, 144(12).
- [22] Schroeder R. P., (2015), "Influence of In-Hole Roughness and High Freestream Turbulence on Film Cooling from a Shaped Hole", Doctorate Thesis, The Pennsylvania State University The Graduate School College of Engineering.
- [23] Eberly M. K., (2012), "Time-Resolved Studies of High Density Ratio Film-Cooling Flows", Master of Science Thesis, The Pennsylvania State University The Graduate School Department of Mechanical and Nuclear Engineering.
- [24] Thole K. A., Gritsch M., Schulz A., Wittig S., (1998), "Flowfield Measurements for Film Cooling Holes with Expanded Exits", *Journal of Turbomachinery*, 120(2), pp. 327-336.
- [25] Lakshminarayana B., (1996), "Fluid Dynamics and Heat Transfer of Turbomachinery", Wiley & Sons, Inc.
- [26] White F. M., (2016), "Fluid Mechanics", 8th Edition, McGraw-Hill.
- [27] Tennekes H., Lumley J. L., (1972), "A first course in Turbulence", Massachusetts Institute of Technology, Cambridge.
- [28] Blazek J., (2001), "Computational Fluid Dynamics: Principles and Applications" Elsevier.

- [29] Web 1, (2009), <https://www.afs.enea.it/project/neptunius/docs/fluent/html/>, (Eriřim Tarihi: 16/12/2023)
- [30] Web 2, (2010), <http://www.itcmp.pwr.wroc.pl/~znmp/fundmech.htm#>, (Eriřim Tarihi: 17/12/2023)
- [31] Bredberg J., (2000), “On the Wall Boundary Condition for Turbulence Models”, Internal Report 00/4, Department of Thermo and Fluid Dynamics Chalmers University of Technology, Göteborg, Sweden.
- [32] Kurzke J., Halliwell I., (2018), “Propulsion and Power An Exploration of Gas Turbine Performance Modeling”, eBook, Springer.
- [33] Saumweber C., Schulz A., (2012), “Free-Stream Effects on the Cooling Performance of Cylindrical and Fan-Shaped Cooling Holes”, ASME Journal of Turbomachinery, 4, pp. 879-891.
- [34] Saumweber C., Schulz A., (2008), “Comparison of the Cooling Performance of Cylindrical and Fan-Shaped Cooling Holes with Special Emphasis on the Effect of Internal Coolant Cross-Flow”, ASME Journal of Turbomachinery, GT2008-51036, 893-903.
- [35] Adami P., Martelli F., Montomoli F., Saumweber C., (2002), “Numerical Investigation of Internal Crossflow Film Cooling”, ASME Journal of Turbomachinery, GT2002-30171, 51-63.
- [36] Taslim M. E., Ugarte S., (2003), “Discharge Coefficient Measurements for Flow Through Compound-Angle Conical Holes with Cross-Flow”, International Journal of Rotating Turbomachinery, 10, pp. 145-153.
- [37] Gritsch M., Schulz A., Wittig S., (1998), “Discharge Coefficient Measurements of Film-Cooling Holes with Expanded Exits”, ASME Journal of Turbomachinery, 120(3), pp. 557-563.

BIOGRAPHY

The author, Semih AVCUN, graduated from Eskişehir Osmangazi University Department of Mechanical Engineering in 2017. He has started his career in the gas turbine industry as a Thermal System Design Engineer and is still working in this field.