Numerical Analysis of Film Cooling on Gas Turbine Blades

Submitted in partial fulfillment of the requirement for the degree of Bachelor of Science in Mechanical Engineering

Submitted By:

Rifat Bin Alam Rohit

Student Id # 141412

Zawad Sultan

Student Id # 141414

SK Tashowar Tanzim Rahul

Student Id # 141416

Ibrahim Hossain Hridoy

Student Id # 141452

Under the supervision of

Dr. Hamidur Rahman

Associate Professor

Department of Mechanical & Chemical Engineering

Islamic University of Technology (IUT)

Organization of Islamic Co-operation (OIC)

ACKNOWLEDGEMENT

First of all, we would like to thank Allah (SWT) for giving us the patience, potential and blessings. We would like to thank our supervisor, Associate Professor Dr. Hamidur Rahman for his guidance, enthusiasm and unwavering willingness to spend his time and wisdom for the care of his students that is truly inspiring. We are extremely grateful to have received his sincere advices and earnest support. On that note, we would also like to sincerely thank our respected teachers Assistant Professor Mr. Sayedus Salehin and Mr. Abdul Karim for their suggestions on the topic and their guidance thereafter. Given the time and various other constraints, we have tried our best in coming up with results and reaching our target, and we sincerely apologize for any mistakes or shortcomings that had occurred. Nevertheless, whatever we have learned or achieved, would not have been possible without the support of the above mentioned people.

Lastly, we would like to thank our parents for the pivotal part they played in allowing us to come this far in life. Without their constant support and encouragement we would have never managed achieve anything in life.

ABSTRACT

Various papers and research works regarding film cooling performance were studied. Parameters, conditions and geometry factors that affect the film cooling effectiveness were analyzed. The objective was to see how modifications such as dual row of anti-vortex hole configuration, and diffuser shaped anti-vortex configuration affected the overall cooling effectiveness, when compared to the performance for the conventional single row of cylindrical holes. The basis of comparison was film cooling applied over a flat plate, and the resulting cooling effectiveness produced. The numerical study was executed on CFX package. Validation of the simulation technique was attempted with reference to the experimental works of A.K Sinha in “Film Cooling Effectiveness Downstream of a Single Row of Holes with Variable Density Ratio”.

Contents

Chapter -1

INTRODUCTION

When designing gas turbines, a high combustion temperature is desirable to obtain a good thermal efficiency. At the same time, the thermal limitations of the gas turbines components must not be exceeded. High temperatures can lead to large thermal stresses that can reduce the life span of the components and increase the risk of fatigue and failure. The trade-off between efficiency on the one hand, and reliability, life span, service interval etc. on the other hand, must be handled early in the design process. At the same time, many other aspects such as aerodynamics, structural strength, manufacturing and assembly must be considered simultaneously.

Advanced and innovative cooling techniques are essential in order to improve the efficiency and power output of gas turbines. Turbine inlet temperatures of 1,600 K are typical for current gas turbines, and there is an interest in increasing the temperatures for the next generation of gas turbines. Over the past decades, significant effort has been devoted to developing effective cooling strategies to maintain the blade temperature below the melting point of alloys used to construct the airfoils. As a result, various cooling strategies have been developed such as film, impingement and multi-pass cooling. The following paper is concerned with the implementation of film cooling to improve gas turbine efficiency.

PROBLEM AT HAND

Extreme temperatures encountered at the inlet of gas turbines exert prolific thermal stresses on the turbine inlet material components, or more precisely, the first stage stator and rotor sections. In order to keep metallurgical limits intact and the alloys from melting under duress, cooling is incorporated. However, reducing turbine inlet temperatures, while it may preserve metal solidity, greatly compromises plant efficiency since the working temperature range of the cycle in operation is shrunk. Optimized cooling is therefore of paramount importance to ensure working blade material sustenance whilst not compromising thermal efficiency of the plant. This paper addresses the problem by applying a special cooling technique called film cooling to a simplified virtual model of a gas turbine inlet.

A BRIEF LOOK INTO GAS TURBINES

The use of gas turbines for generating electricity dates back to 1939. Today, gas turbines are one of the most widely-used power generating technologies. Gas turbines are a type of internal combustion (IC) engine in which burning of an air-fuel mixture produces hot gases that spin a turbine to produce power. It is the production of hot gas during fuel combustion, not the fuel itself that the gives gas turbines the name. Gas turbines can utilize a variety of fuels, including natural gas, fuel oils, and synthetic fuels.

Gas turbines are comprised of three primary sections mounted on the same shaft: the compressor, the combustion chamber (or combustor) and the turbine. The compressor can be either axial flow or centrifugal flow. Axial flow compressors are more common in power generation because they have higher flow rates and efficiencies. Axial flow compressors are comprised of multiple stages of rotating and stationary blades (or stators) through which air is drawn in parallel to the axis of rotation and incrementally compressed as it passes through each stage. The acceleration of the air through the rotating blades and diffusion by the stators increases the pressure and reduces the volume of the air. Although no heat is added, the compression of the air also causes the temperature to increase.

The compressed air is mixed with fuel injected through nozzles. The fuel and compressed air can be pre-mixed or the compressed air can be introduced directly into the combustor. The fuel-air mixture ignites under constant pressure conditions and the hot combustion products (gases) are directed through the turbine where it expands rapidly and imparts rotation to the shaft. The turbine is also comprised of stages, each with a row of stationary blades (or nozzles) to direct the expanding gases followed by a row of moving blades. The rotation of the shaft drives the compressor to draw in and compress more air to sustain continuous combustion. The remaining shaft power is used to drive a generator which produces electricity. Approximately 55 to 65 percent of the power produced by the turbine is used to drive the compressor. To optimize the transfer of kinetic energy from the combustion gases to shaft rotation, gas turbines can have multiple compressor and turbine stages.

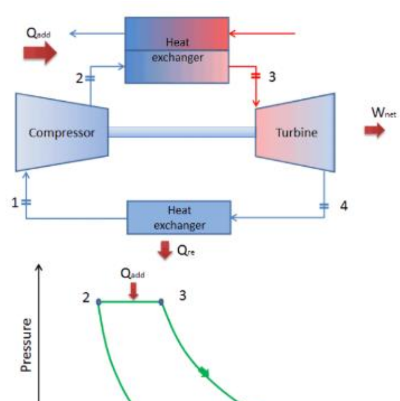
The thermodynamics centering on the operation of gas turbines is the **Brayton Cycle**. In a closed ideal Brayton cycle, the system executing the cycle undergoes a series of four processes: two isentropic (reversible adiabatic) processes alternated with two isobaric processes:

Isentropic compression (compression in a compressor) – The working gas (e.g. helium) is compressed adiabatically from state 1 to state 2 by the compressor (usually an axial-flow compressor). The surroundings do work on the gas, increasing its internal energy (temperature) and compressing it (increasing its pressure). On the other hand the entropy remains unchanged. The work required for the compressor is given by $W_c = H_2 – H_1$.

Isobaric heat addition (in a heat exchanger) – In this phase (between state 2 and state 3) there is a constant-pressure heat transfer to the gas from an external source, since the chamber is open to flow in and out. In an open ideal Brayton cycle, the compressed air then runs through a combustion chamber, where fuel is burned and air or another medium is heated ($2 \rightarrow 3$). It is a constant-pressure process, since the chamber is open to flow in and out. The net heat added is given by $Q_{add} = H_3 – H_2$.

Isentropic expansion (expansion in a turbine) – The compressed and heated gas expands adiabatically from state 3 to state 4 in a turbine. The gas does work on the surroundings (blades of the turbine) and loses an amount of internal energy equal to the work that leaves the system. The work done by turbine is given by $Wt = H_4 – H_3$. Again the entropy remains unchanged.

Isobaric heat rejection (in a heat exchanger) – In this phase the cycle completes by a constant-pressure process in which heat is rejected from the gas. The working gas temperature drops from point 4 to point 1. The net heat rejected is given by $Q_{re} = H_4 – H_1$.



FILM COOLING

Film cooling is an extensively studied method for cooling of gas turbine hot section airfoils. Increases in the maximum operating temperature of the turbine lead to increases in the power output and thermal efficiency of the turbine engine, but cause decreases in the durability for the components in the hot section of the engine. In order to allow for higher operating temperatures and improved component durability, film cooling is employed to protect the components from the hot mainstream gas, and to keep the temperatures of the components within acceptable ranges. Film cooling studies often demonstrate effectiveness of cooling in experimental and computational studies with contour plots of film-cooled component surface non-dimensional temperatures, and stream wise variation plots of centerline and span wise averaged adiabatic effectiveness.

The pressure side of the first stage blades consists of reservoirs known as plenums from where a coolant jet is issued at some speed. This jet flows along a narrow channel before being expelled through a cylindrical hole at high velocity into the hot mainstream gas domain in the duct. The two variable temperature fluids interact and emerge downstream with the plate wall of the pressure side adequately cooled to safe temperature limits. Ideally, the coolant jet exits tangentially across the surface, forming a film that is thick enough to prevent hot spots and thin enough to prevent mainstream flow choking.

PARAMETERS

Blowing ratio is a measure of how much, and with what velocity, cooling air is being used to create the cooling film. Blowing ratio is defined as:

$$M = \frac{\rho_c U_c}{\rho_\infty U_\infty}$$

The **downstream distance** is, like many parameters in film cooling, scaled with the hole diameter, D, to create a dimensionless distance. Downstream distance is an essential parameter in film cooling and in graphs the film effectiveness is most commonly plotted versus this parameter. Thus, these graphs show how the film effectiveness changes with downstream distance from the hole, and a number of plots like this will be presented later.

Turbulence: Downstream from the point of ejection, the increased mixing between coolant and free stream caused by the higher turbulence results in a larger amount of hot gas reaching the surface, and thus a reduction in effectiveness. This reduction is most pronounced at low blowing ratios, where the turbulence from the jet injection is low. The importance of free stream turbulence is also dependent on hole geometry.

The **momentum ratio** is defined as:

$$I = \frac{\rho_c U_c^2}{\rho_\infty U_\infty^2}$$

Where ρ_c and ρ_∞ is the coolant and free stream density while Uc and $U\infty$ is the coolant and free stream velocity. As can be seen this closely resembles the blowing ratio.

The **density ratio**, DR, is defined as the ratio of coolant and free stream density, ρ_c / ρ_∞, and in a gas turbine it is larger than or equal to one, DR $\geq$ 1.

LITERATURE REVIEW

The present study numerically investigates the effect of different number of holes and various turbulence models to best capture results of established and hence proven experimental study. A new methodology can be studied upon thereafter by performing different simulations on the benchmark case, the Sinha geometry.

Experimental Facility and Instrumentation

Experiments were conducted in the closed-loop wind tunnel facility shown schematically in the figure below. A comprehensive description of this facility was given by Pietrzyk et al. (1990). The secondary flow loop in this facility uses cryogenic cooling of the injectant to obtain different density ratios between the jets and the mainstream. The test plate was modified for this study to reduce conduction problems associated with adiabatic wall temperature measurements.

The figure below that shows the geometry of the new test plate. A new test plate was installed in the facility to reduce conduction errors. The test plate was constructed from extruded polystyrene foam (Styrofoam), which has a thermal conductivity (k = 0.027 W/m-K) significantly lower than the previous test plate. An analysis of conduction errors using a three-dimensional conduction heat transfer code indicated negligible conduction errors for the new test plate.

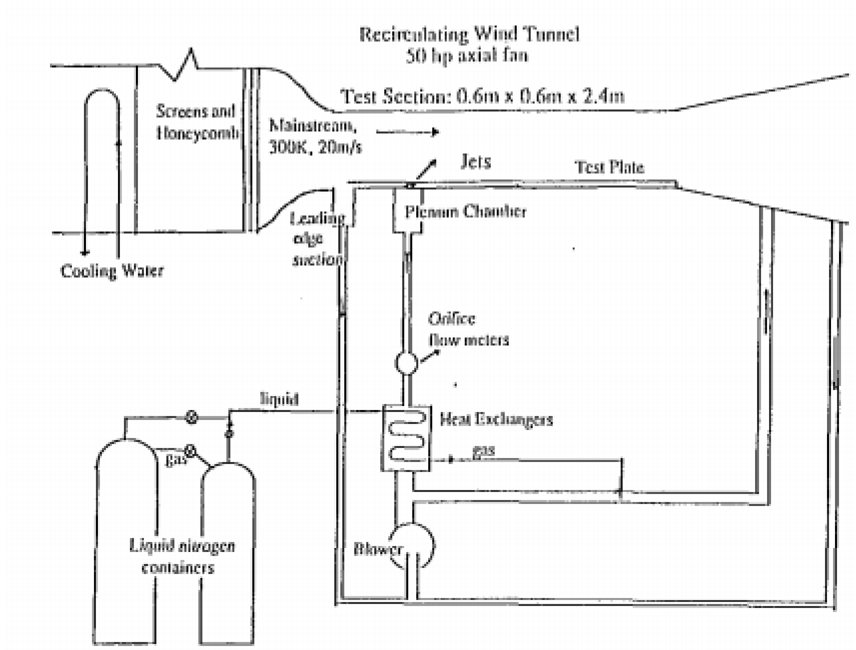
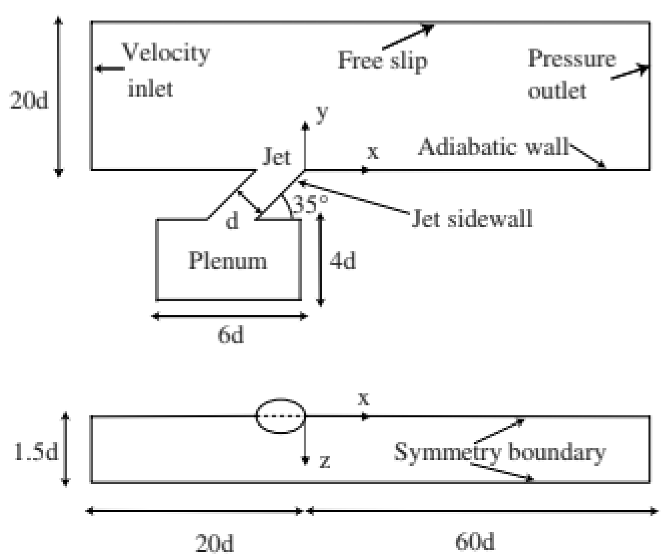


Fig. 1 Schematic of film-cooling test facility



The Sinha geometry serves as an apt standard case for its experimental results were the latest work done on film cooling studies, before the advent of numerical studies took over. Besides, it employs the simplest of working models, a flat plate to represent the surface of the pressure side of the first stage turbine blade that incurs the peak thermal stresses and also makes it more convenient to work with. Since recent film cooling studies have all selected the Sinha case as the model to replicate and a model against which new methodologies are compared and evaluated thereof, it seems only fitting to conduct studies in an attempt to best capture relevant results.

Chapter-2

Governing Equation

Navier-Stokes Equations

The Navier-Stokes equations govern the motion of fluids and can be seen as Newton's second law of motion for fluids. In the case of a compressible Newtonian fluid, this yields:

$$\underbrace{\rho\left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u}\cdot\nabla\mathbf{u}\right)}_{1} = \underbrace{-\nabla p}_{2} + \underbrace{\nabla\cdot\left(\mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \frac{2}{3}\mu(\nabla\cdot\mathbf{u})\mathbf{I}\right)}_{3} + \underbrace{\mathbf{F}}_{4}$$

Where **u** is the fluid velocity, ***p*** is the fluid pressure, ***ρ*** is the fluid density, and ***μ*** is the fluid dynamic viscosity. The different terms correspond to the inertial forces (1), pressure forces (2), viscous forces (3), and the external forces applied to the fluid (4). The Navier-Stokes equations were derived by Navier, Poisson, Saint-Venant, and Stokes between 1827 and 1845. These equations are always solved together with the continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla\cdot(\rho\mathbf{u}) = 0$$

These equations are at the heart of fluid flow modeling. Solving them, for a particular set of boundary conditions (such as inlets, outlets, and walls), predicts the fluid velocity and its pressure in a given geometry.

Film Cooling Effectiveness Equations

The aim of film cooling is to introduce the coolant into the boundary layer without significantly increasing turbulence and entraining additional hot free stream gas. There are three temperatures in this problem, the free stream temperature, the coolant temperature and the wall temperature. For incompressible flow with constant fluid properties, the heat transfer rate to the surface, $\dot{q}$, may be expressed in the following form:

$$\dot{q} = h(T_{aw} - T_w)$$

Where h is a heat transfer coefficient. T_{aw} and T_w are the adiabatic wall temperature, which is now different from the free stream temperature, and wall temperature, respectively. Alternatively, the relationship may be given in terms of the known temperature differences as:

$$\dot{q} = \alpha(T_g - T_w) + \beta(T_g - T_c)$$

α and β are solely functions of the flow field. The film cooling effectiveness, η, is thus given as:

$$\eta = \frac{T_g - T_{aw}}{T_g - T_c} = -\frac{\beta}{\alpha} \text{ also } h = \alpha$$

TURBULENCE MODELS IN CFX

Turbulence consists of fluctuations in the flow field in time and space. It is a complex process, mainly because it is three dimensional, unsteady and consists of many scales. It can have a significant effect on the characteristics of the flow. Turbulence occurs when the inertia forces in the fluid become significant compared to viscous forces, and is characterized by a high Reynolds Number.

In principle, the Navier-Stokes equations describe both laminar and turbulent flows without the need for additional information. However, turbulent flows at realistic Reynolds numbers span a large range of turbulent length and time scales, and would generally involve length scales much smaller than the smallest finite volume mesh, which can be practically used in a numerical analysis. The Direct Numerical Simulation (DNS) of these flows would require computing power that is many orders of magnitude higher than available in the foreseeable future.

To enable the effects of turbulence to be predicted, a large amount of CFD research has concentrated on methods that make use of *turbulence models*. Turbulence models have been specifically developed to account for the effects of turbulence without recourse to a prohibitively fine mesh and direct numerical simulation.

A number of models have been developed that can be used to approximate turbulence based on the Reynolds Averaged Navier-Stokes (RANS) equations:

- Eddy-viscosity models:
    - Zero equation model.
    - Standard k-ε model.
    - RNG k-ε model.
    - Standard k-ω model.
    - Baseline (BSL) zonal k-ω based model.
    - SST zonal k-ω based model.
- Reynolds stress models (RSM):
    - Launder, Reece and Rodi Isotropization of Production model (LRR Reynolds Stress).
    - Launder, Reece and Rodi Quasi-Isotropic model (QI Reynolds Stress).
    - Speziale, Sarkar and Gatski (SSG Reynolds Stress).
    - SMC-ω model (Omega Reynolds Stress).
    - Baseline (BSL) Reynolds stress model.
    - Explicit Algebraic Reynolds stress model (EARSM).

The turbulence models used in our various simulations are:

1. **The k-omega model**: One of the advantages of the k-ω formulation is the near wall treatment for low-Reynolds number computations. The model does not involve the complex nonlinear damping functions required for the k-ε model and is therefore more accurate and more robust. The k- ω models assume that the turbulence viscosity is linked to the turbulence kinetic energy and turbulent frequency via the relation:

$$\mu_t = \rho \frac{k}{\omega}$$

2. **SST model**: The k-ω based SST model accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients. The proper transport behavior can be obtained by a limited to the formulation of the eddy-viscosity:

$$v_t = \frac{a_1 k}{\max(a_1 \omega, SF_2)}$$

3. **k-ε model**: One of the most prominent turbulence models, this model has been implemented in most general purpose CFD codes and is considered the industry standard model. It has proven to be stable and numerically robust and has a well-established regime of predictive capability. For general purpose simulations, this model offers a good compromise in terms of accuracy and robustness.

4. **RNG k-ε model**: The RNG k-ε model is an alternative to the standard k-ε model. In general it offers little improvement compared to the standard k-ε model.

Chapter 3

Methodology

Tools Used

- SolidWorks – Creating the 3D Geometry
- Ansys Simulation Software – Running the Simulations
  - ICEM CFD – For generating the mesh.
  - Ansys CFX package – To setup the simulation and execute it.

Geometry Used

- Sinha Geometry (as Reference)

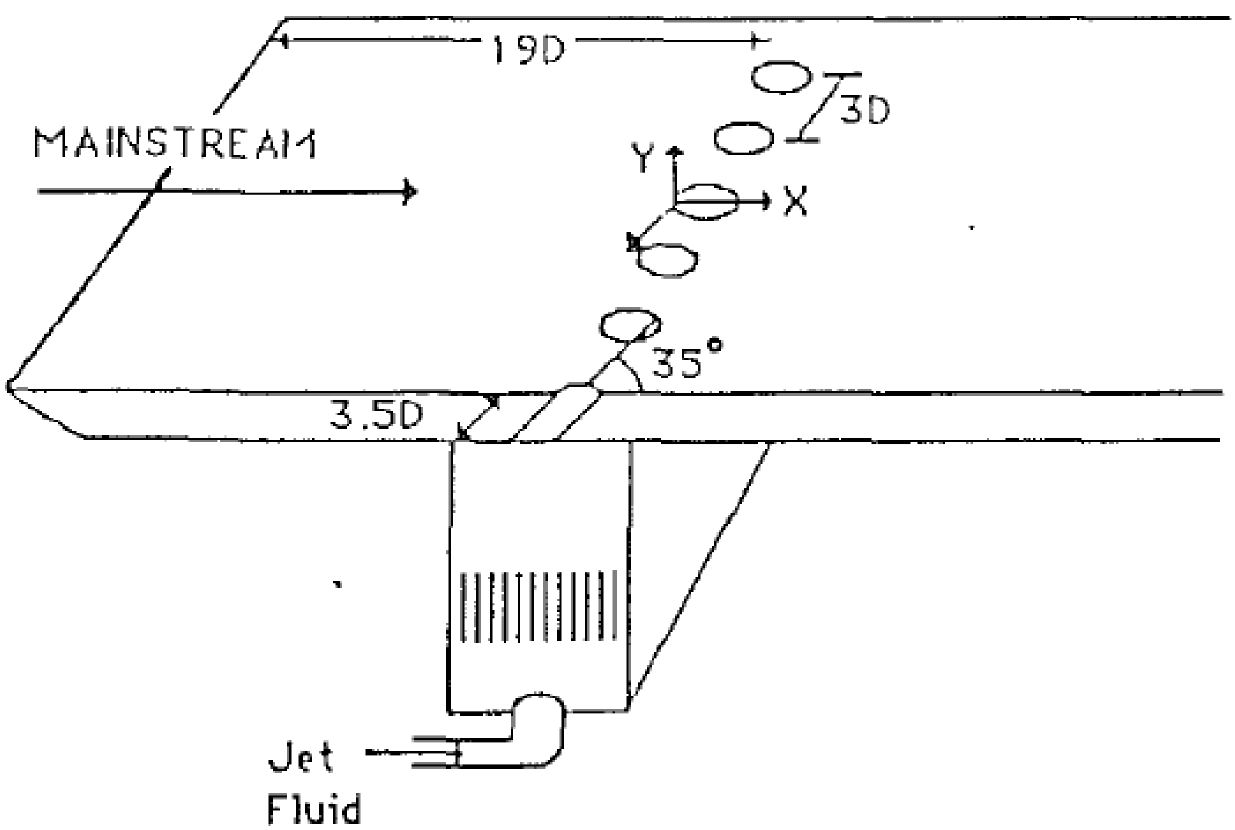
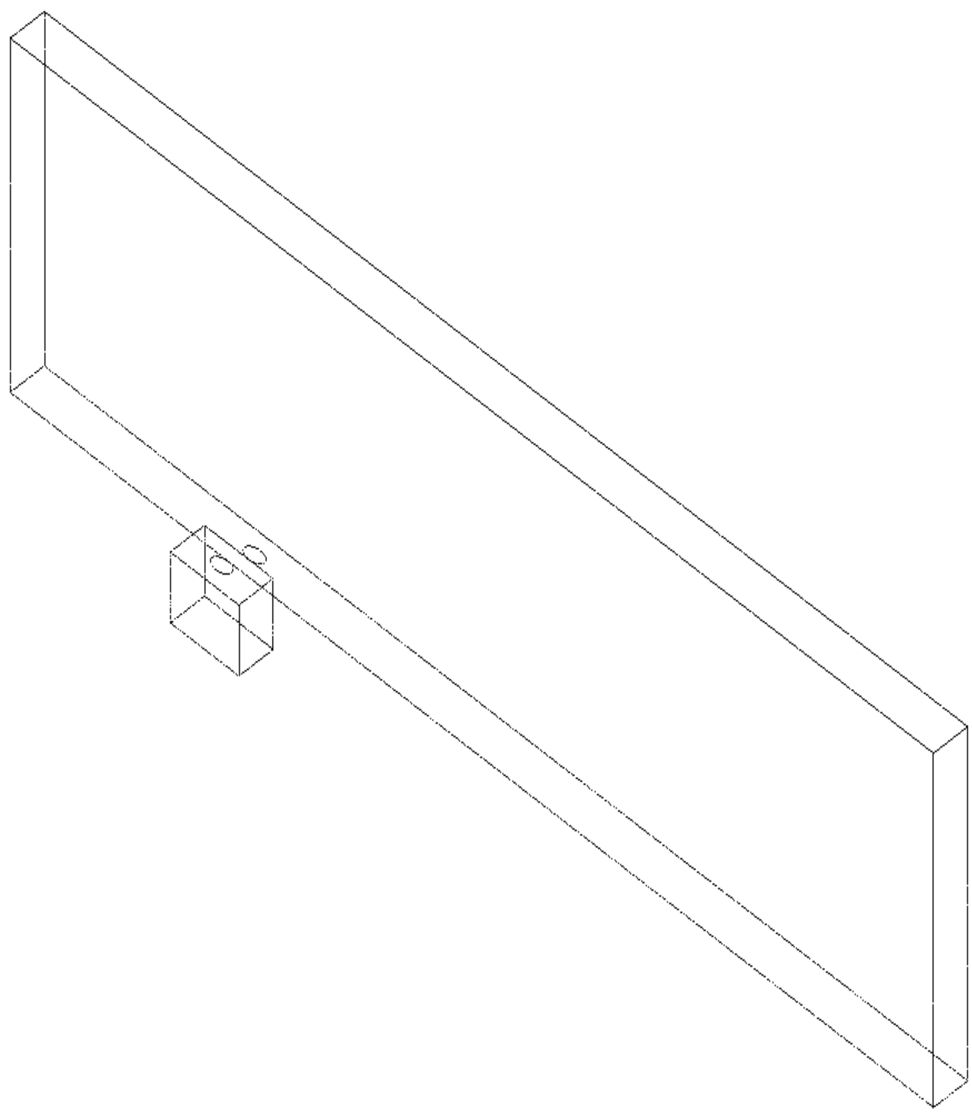
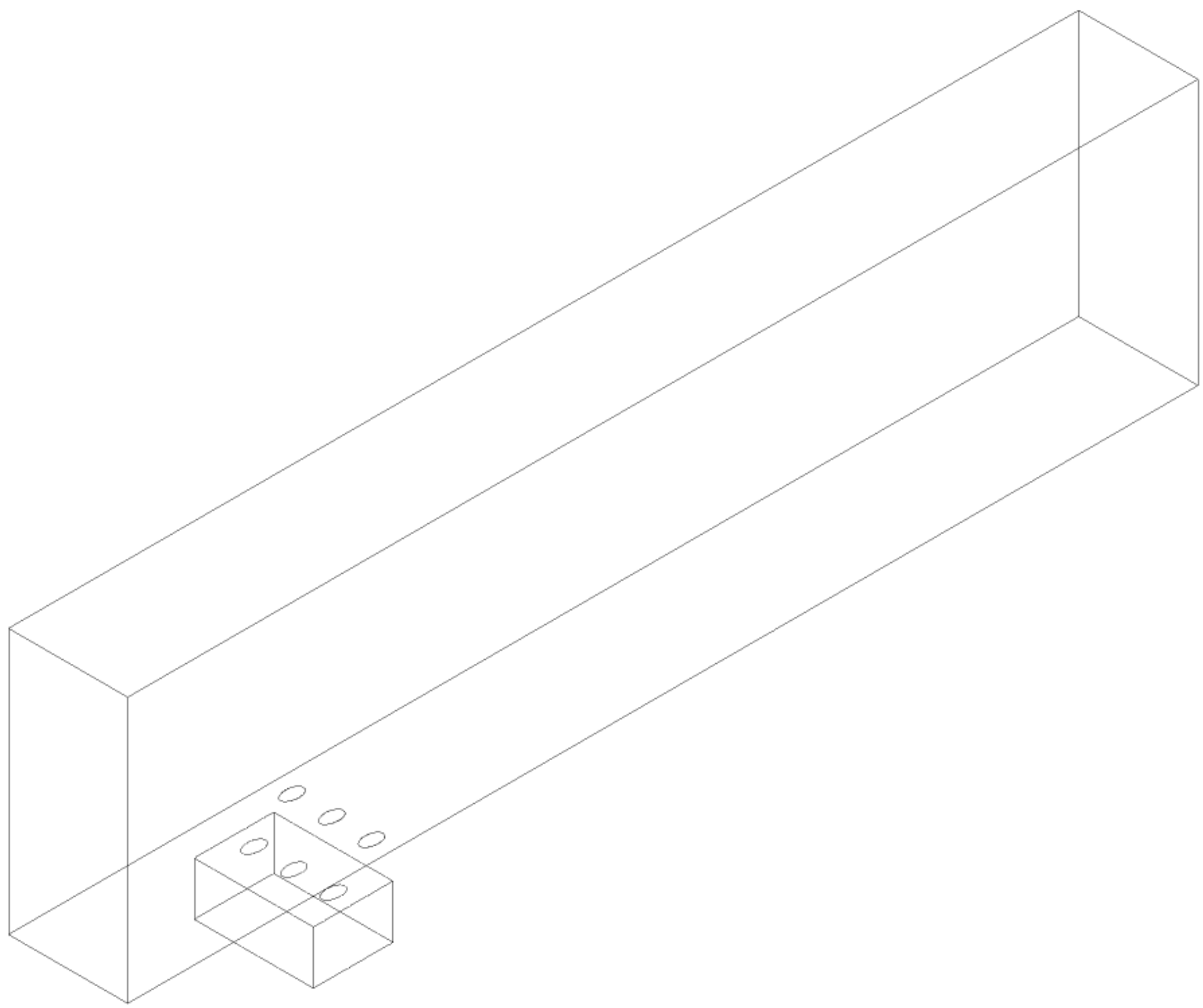


Fig. 2(*a*) Film-cooling test geometry and coordinate system

- The generated geometry (Single Hole)



- Three Hole



Boundary Conditions

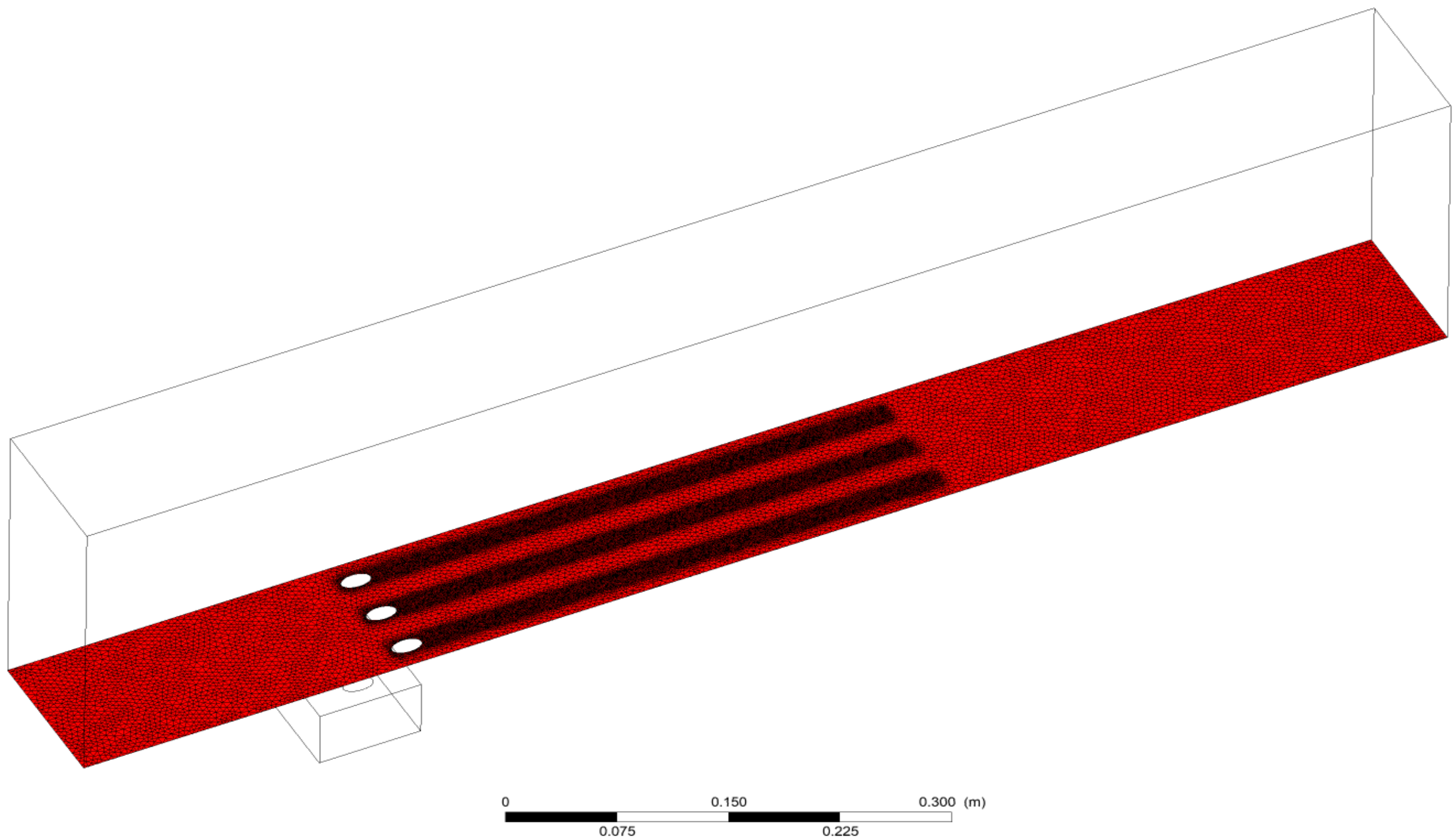
1. Main Inlet
    a. Mainstream Air Temperature = 300 K
    b. Mainstream Air Velocity = 20m/s
    c. Turbulence Model = Intensity and Auto compute (0.0002)
2. Main Outlet
    a. Open to Atmosphere
    b. Pressure at exit= Atmospheric Pressure
3. Top, Plate & Plenum Walls
    a. Modeled as walls with the following conditions
        i. Adiabatic
        ii. No-Slip Condition
4. Plenum Inlet
    a. Plenum Inlet mass flow rate = 0.00250kg/ms^2
    b. Turbulence Model= Intensity & Auto compute (0.02)
    c. Plenum Inlet Temperature = 150k

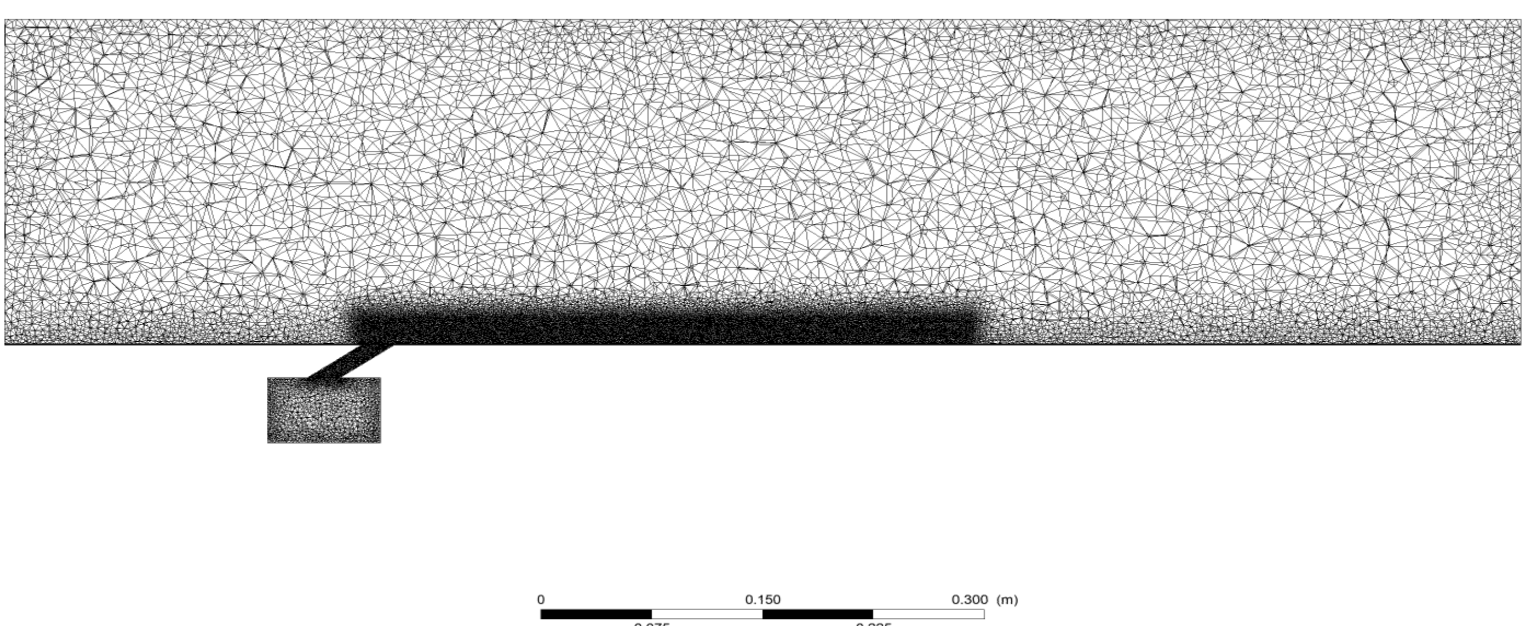
Overall Assumptions

1. Air behaves as an "IDEAL GAS"
2. Turbulence Model = RNG K EPSILON
3. Steady State Heat transfer

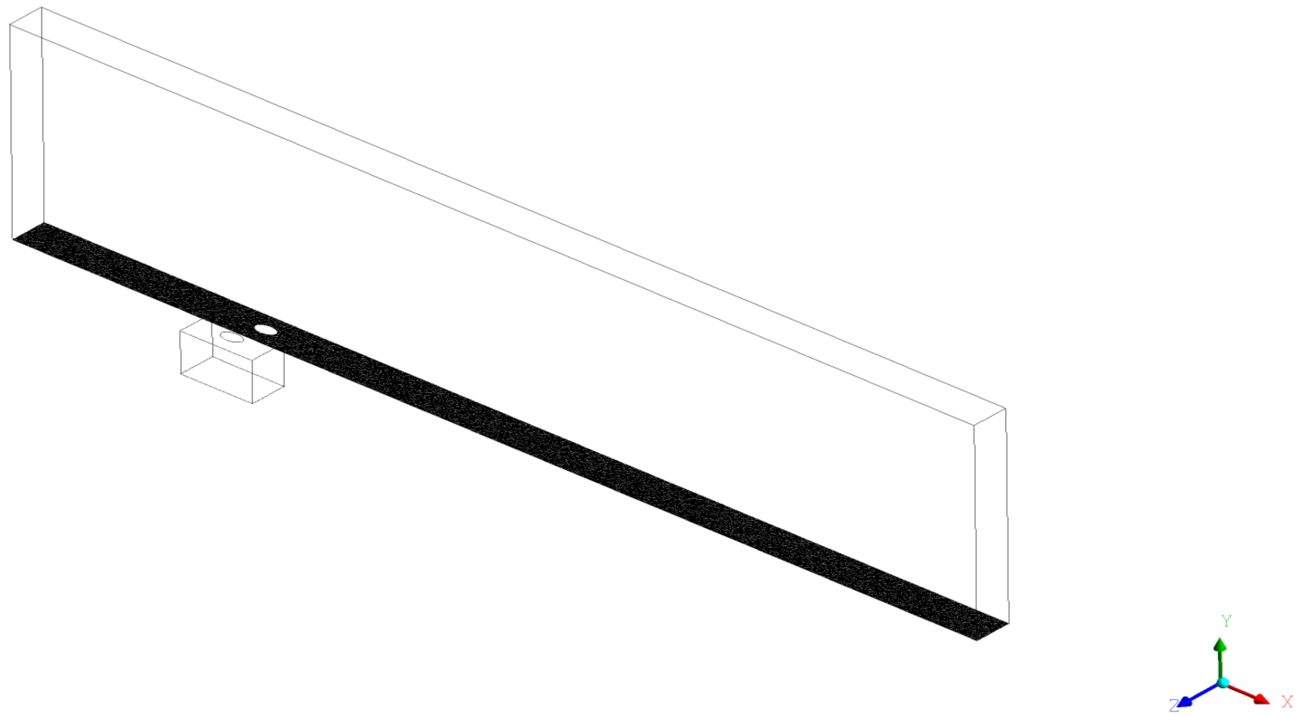
Mesh Generated

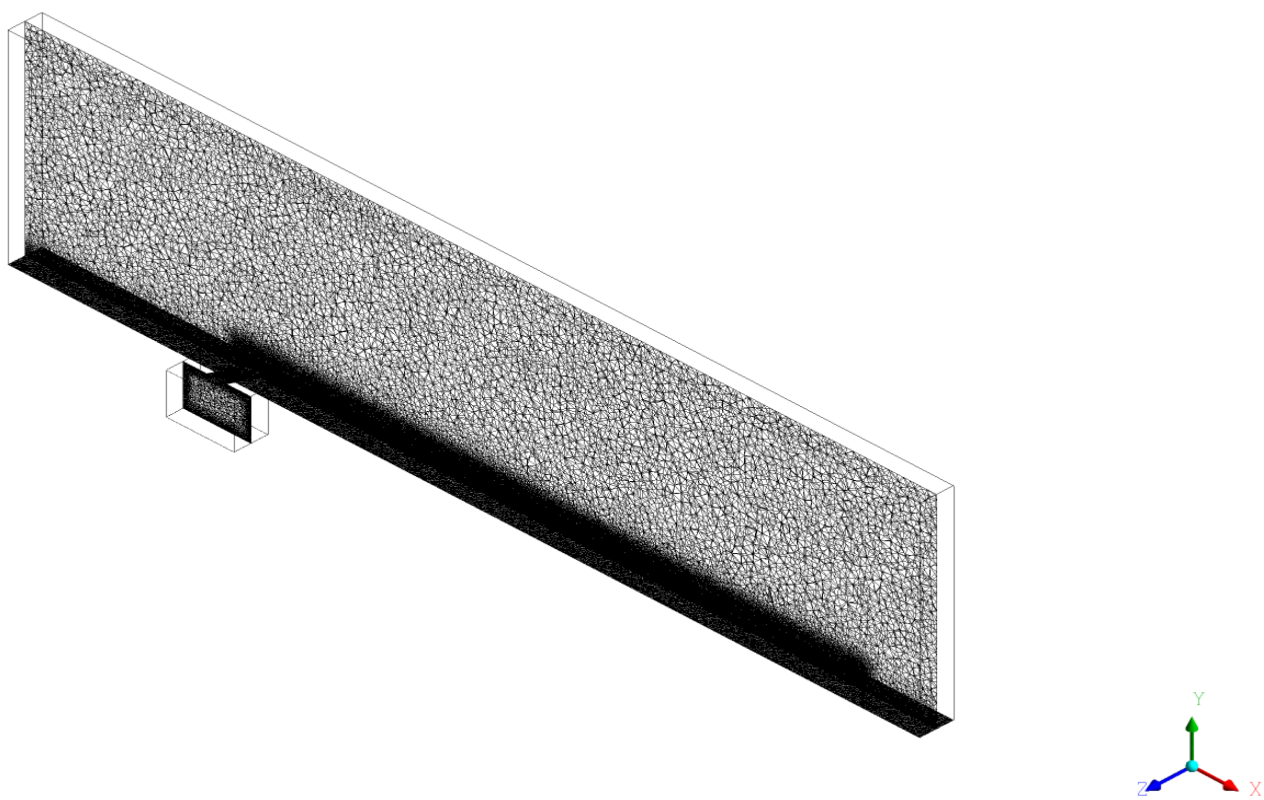
- Three Hole Geometry





- Single Hole

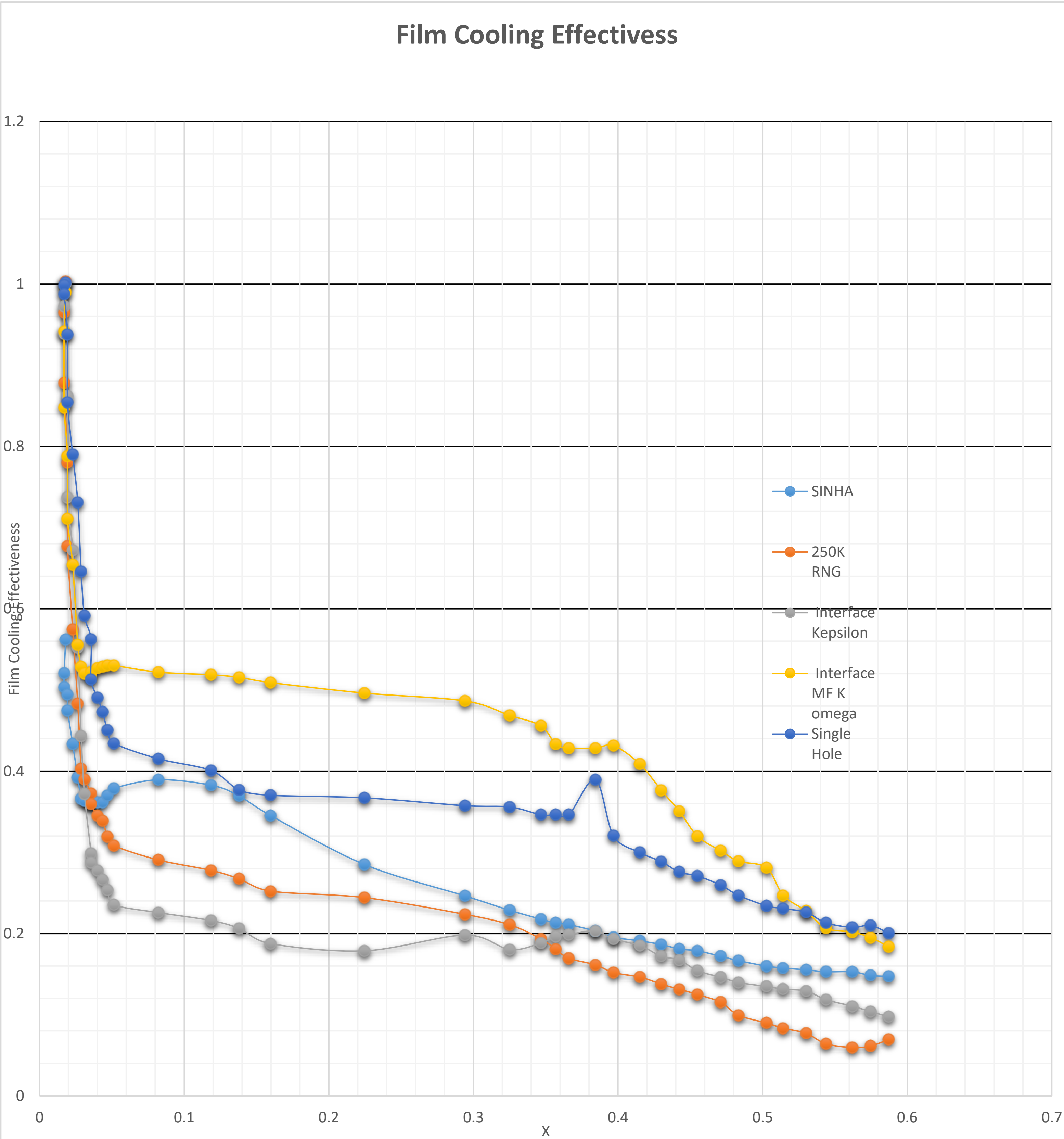




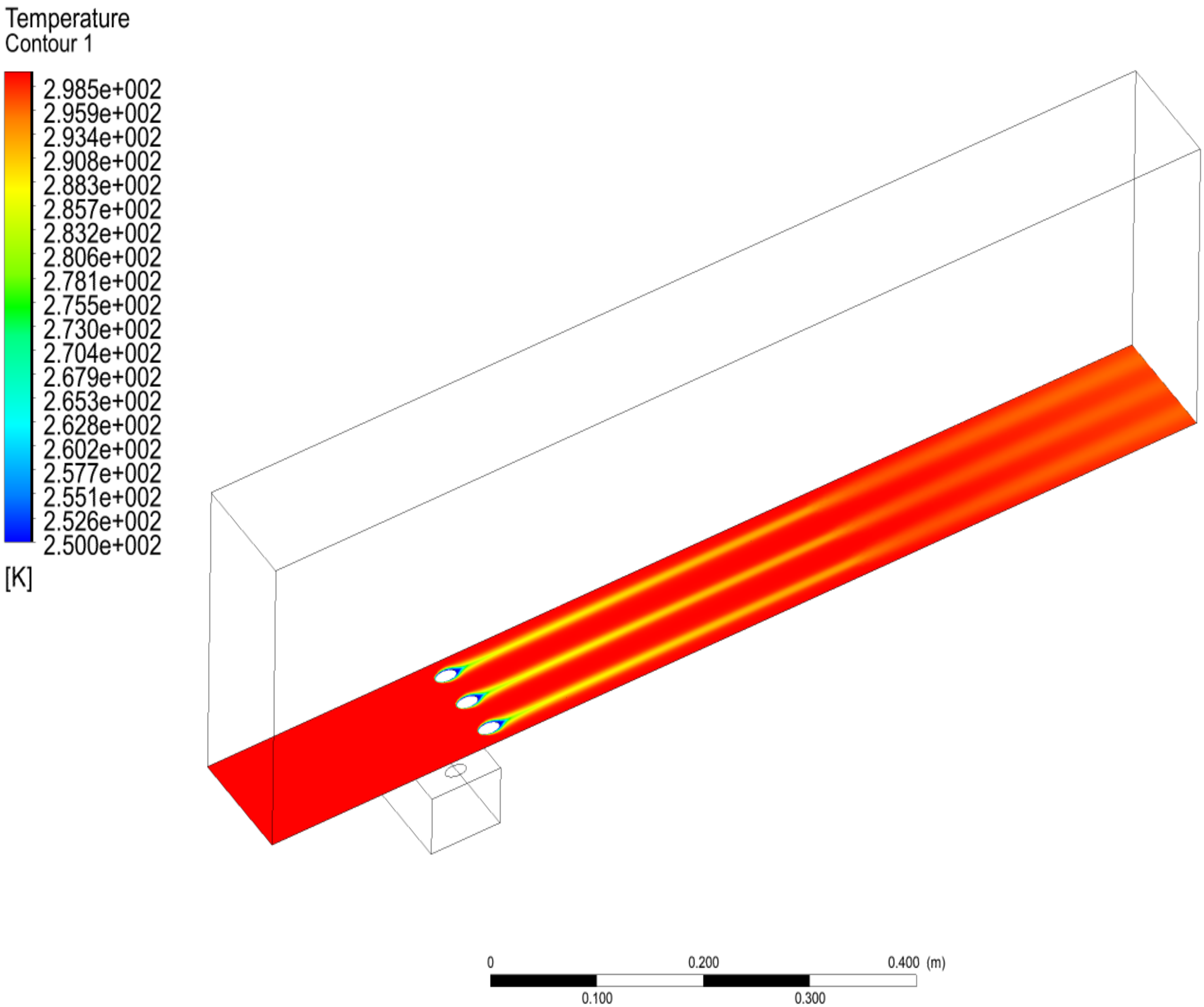
Chapter 4

Results

Film Cooling Effectiveness

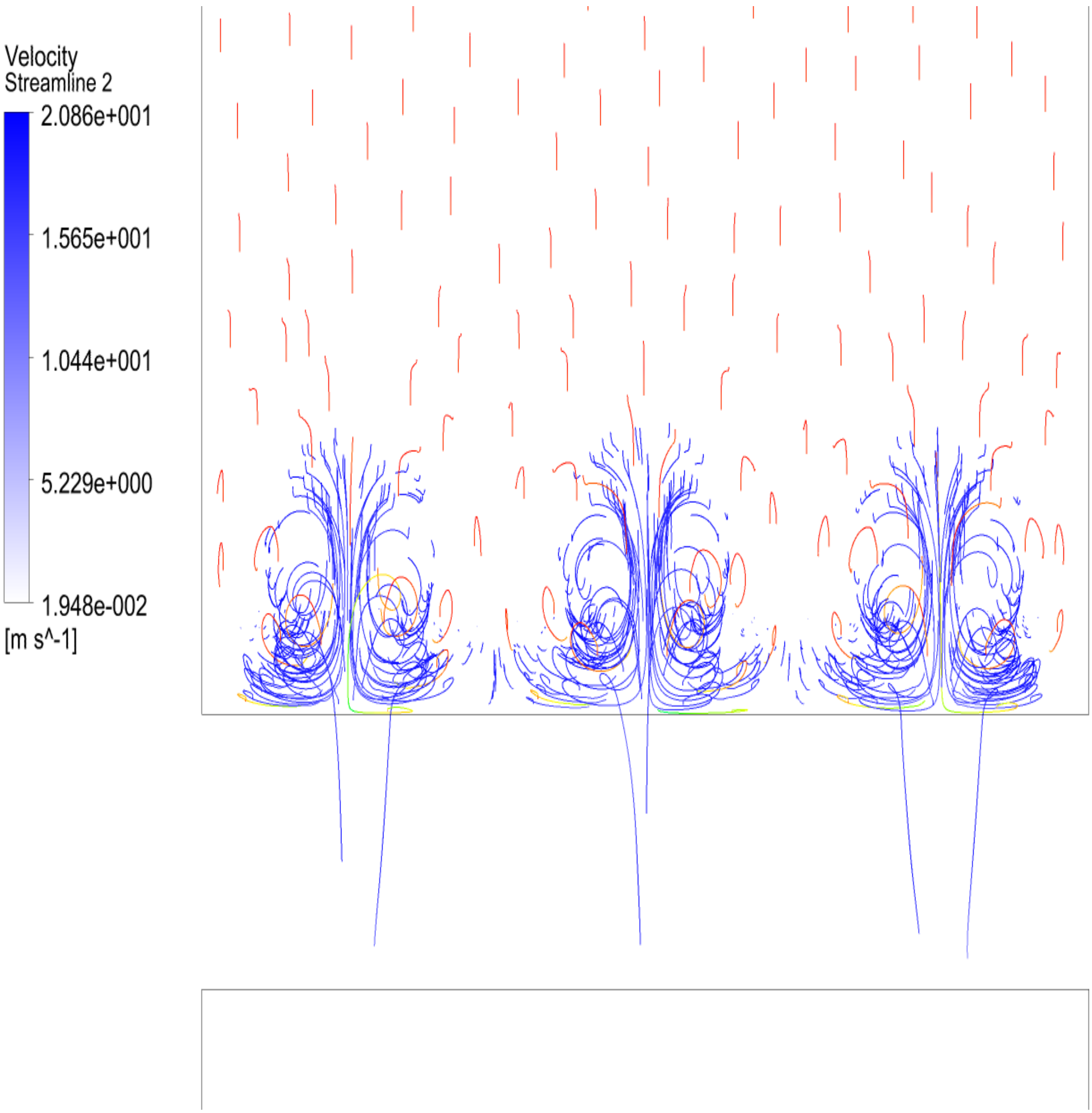


Contours


Temperature
Contour 1
2.985e+002
2.959e+002
2.934e+002
2.908e+002
2.883e+002
2.857e+002
2.832e+002
2.806e+002
2.781e+002
2.755e+002
2.730e+002
2.704e+002
2.679e+002
2.653e+002
2.628e+002
2.602e+002
2.577e+002
2.551e+002
2.526e+002
2.500e+002
[K]
0
0.200
0.400 (m)
0.100
0.300

Vortices

The red lines are the velocity profiles of the mainstream flow, and the blue swirling lines are the vortices of the plenum air created due to the mass momentum interaction with mainstream air.



Discussion

As evident by the charts given above, the reference film effectiveness profile (Sinha Paper) has not been simulated properly. Different turbulence models were used to get the desired profile for validation and a failure to get the appropriated profile means, the simulations ran were not validated.

The desired profile has not been reached because of the following probable reasons:

a. The mass flow rate of the plenum was never specified and as a result remained as a variable. The mass flow rate is necessary to maintain the Blowing Ratio of the experiment and thus maintain the consistency with the paper. However, during the simulations to get the desired outlet velocity from the plenum chamber (10 m/s), the mass flow rate value was iteratively changed. This lead to multiple number of probable solutions to get the desired velocity value, as a results, the graphs widely varied from on simulation to the next.

b. The mesh quality was not satisfactory due to computational limitations, thus invariability in the results is difficult to identify.

Future Prospects

Having read various research papers on this particular field, we have decided to investigate the effect of combining the aspects that resulted in positive outcomes in different research papers, on the overall cooling performance. Several such aspects can be studied:

1. Numerical analysis of the cooling effectiveness of an Anti-Vortex hole configuration, as studied by Timothy W. Repko, Andrew C. Nix in “Flow Visualization of multi-hole film cooling flow under varying freestream turbulence levels”, embedded in a narrow trench as presented in the paper of Katharine L. Harrison and David G. Bogard at ASME turbo expo 2007.
2. We can also investigate the effect of orientation angle and inclination angle on the cooling performance of the previously mentioned AVH configuration, and hence deduce the optimum angles for the setup.
3. We can analyze Diffuser-shaped holes, as presented in “Investigation on the Film Cooling Performance of Diffuser Shaped Holes with Different Inclination Angles” by Ying-Ni Zhai, Cun-Liang Liu, Yi-Hong He and Zhi-Xiang Zhou, embedded in narrow trenches as in the study of Bogard.

References

1) An advanced impingement/film cooling scheme for gas turbines – numerical study
A. Immarigeon and I. Hassan
Department of Mechanical and Industrial Engineering, Concordia University, Montre´al, Canada.

2) Film Cooling Effectiveness of an Advanced-Louver Cooling Scheme for Gas Turbines
X. Z. Zhang and I. Hassan
Concordia University, Montréal, Québec H3G 1M8, Canada.

3) Flow Visualization of Multi-Hole Film-Cooling Flow under Varying Freestream Turbulence Levels
Timothy W. Repko, Andrew C. Nix, S. Can Uysal, Andrew T. Sisler
Department of Mechanical and Aerospace Engineering, West Virginia University, Morgantown, WV, USA.

4) Gas Turbine Film Cooling
D. G. Bogard
University of Texas, Austin, Texas 78712-0292
and
K. A. Thole
Virginia Polytechnic Institute and State University, Blacksburg, Virginia 24060.

5) Investigation on the Film Cooling Performance of Diffuser Shaped Holes with Different Inclination Angles
Ying-Ni Zhai, Cun-Liang Liu*, Yi-Hong He and Zhi-Xiang Zhou

6) Investigations of Trenched Film Hole Orientation Angle on Film Cooling Effectiveness
Krishna Anand V.G and Parammasivam K.M

7) Mechanism of Film Cooling with One Inlet and Double Outlet Hole Injection at Various Turbulence Intensities
Guangchao Li, Yukai Chen, Zhihai Kou, Wei Zhang and Guochen Zhang

8) Unsteady Simulations of a Film Cooling Flow from an Inclined Cylindrical Jet
Sung In Kim and Ibrahim Hassan
Concordia University, Montreal, Quebec H3G 1M8, Canada

9) Film-Cooling Effectiveness Downstream of a Single Row of Holes with Variable Density Ratio
A. K. Sinha, D. G. Bogard, M. E. Crawford
Mechanical Engineering Department,
University of Texas at Austin,
Austin, TX 78712