Study of Gas Turbine Blade Cooling

By

Mohamed Zafir Bin Mohamad Gazali

Dissertation submitted in partial fulfillment of

the requirements for the

Bachelor of Engineering (Hons)

(Mechanical Engineering)

JANUARY 2009

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

ABSTRACT

Gas turbine engines are being used in most of land based power plant. The increasing demand for energy has lead scientists and researchers to enhance the performance of the gas turbine engines. High turbine inlet temperatures are required to obtain high thermal efficiency and thus higher power output can be produced. Turbine blade could experience high thermal stress due to limitation of its material physical properties. A lot of studies had been carried out to solve this problem such as by developing new material for turbine blade. One of the most effective solutions is to introduce the cooling process in the turbine blade itself. The main focus of this project is to study the heat transfer and temperature profile along the turbine blade surface while the blade is cooled internally (impingement cooling). To observe the cooling process in the gas turbine, a turbine blade geometry model has been created using preprocessor software called GAMBIT. FLUENT software is used to simulate the cooling process of the turbine blade. The simulation result is finally being validated by using published data. The results comprise of the temperature distribution along the surface of the turbine blade and the temperature distribution on the surface of cooling passages. Temperature distribution on the previously mentioned surfaces can be studied by observing the temperature contour which is generated by FLUENT. The turbine blade is convectively cooled by the compressed air that is bled from the fourteenth stage compressor. It is indicated that the temperature of the turbine blade in the hot gas mainstream has been decreased during operation. This research has proven that the cooling process of the turbine blade occurs by the presence of the cooling air which flows through the cooling passages.

ACKNOWLEDGMENTS

First and foremost, I would like to praise to God the Almighty for His guidance. With His guidance and blessings bestowed upon me, I managed to overcome all obstacles in completing this project. Here, I would like to use this special opportunity to express my heartfelt gratitude to everyone that has contributed to the success of the project.

My deepest appreciation and gratitude goes to my Final Year Project Supervisor, Ir Idris Bin Ibrahim for his supervision, commitment, professionalism, advice and guidance throughout the completion of final year project.

Last but not least, special thanks to those who had helped me directly or indirectly in undertaking this project throughout the year end. The contributions and insights are highly appreciated.

TABLE OF CONTENT

LIST OF FIGURES

CHAPTER 1
INTRODUCTION

1.1 Background of study

Gas turbines are the most crucial part for aircraft propulsion and in land-based power generation. Higher turbine rotor inlet temperatures will produce higher thermal efficiency and power output of gas turbines. For land-based gas turbines, which include power generation (300MW combined cycle), marine propulsion, and industrial applications such as pumping and cogeneration (less than 30MW) the rotor inlet temperature (RIT) must be maintained at the level of 2500°F-2600°F due to the restriction of NO_x pollution problem. Hence, the critical issue for land-based gas turbines is how to further improve the efficiency of gas turbine combined cycle.

Study has proven that thermal efficiency of an advanced gas turbine engine can be improved by increasing the rotor inlet temperature. Consequently, the advanced gas turbine engine can produce higher power output. High rotor inlet temperature can harm the turbine blade life due to thermal stress. Therefore, cooling system for the turbine blade must be created so that the turbine blade can withstand high operating temperature. Engineers and researchers have been working hard all these years to study and investigate the best turbine blade cooling technique which produces high cooling efficiency. Their investigations involved in fluid dynamic and heat transfer area of study, the turbine blade cooling passage design, and as well as the types of coolant being used. This effort has been carried out thoroughly and results in the discovery of new evolving technology.

1.2 Problem Statement

The thermal efficiency of a heat engine can be raised by increasing its maximum operating temperature. The temperature of the gas leaving the combustion chamber will be designed to be as high as the first row of blades can withstand. A lot of research and study have been done for blade material to develop nimonic steels and thermal barrier coatings. One of the best ways to improve the efficiency of gas turbine is by cooling the blades with high pressure atmospheric air that has bypassed the combustion chamber. Cooling of the turbine blade using high pressure air will result in the development of different temperature profile of the turbine blades. This project is stated to investigate this temperature profile of the turbine blade using computational fluid dynamic approach.

1.3 Objective and Scope of Study

The main objectives of this research are:

- Develop Computer Fluid Dynamic (CFD) geometry model of the turbine blade by using GAMBIT software
- Simulation of the turbine blade impingement cooling process by using FLUENT software
- Validation of simulation result using published data

The scope of study of this research is to study the temperature profile along the turbine blade surface and study the phenomena of heat transfer during the cooling process.

CHAPTER 2
THEORY AND LITERATURE REVIEW

2.1 Introduction

In order to improve thermal efficiency and power output, the advanced gas turbine engines shall operate at high temperatures (1731-1426°C) [2]. When the turbine inlet temperature is raised, the heat transferred to the turbine blades increases. It is important to limit the level and variation in temperature within the blade material to achieve the reasonable durability results. This is because; by not limiting level and variation in temperature within blade material may lead to thermal stresses.

It can be seen that in Figure 2.1 the engines track fairly close to the ideal performance line based on the diagram which plots specific core power production as a function of turbine rotor inlet temperature [2]. It represents the cycle power output with 100% of turbine efficiency without leakage or cooling flows. Hence, it is proven that by increasing the rotor inlet temperature (RIT) in advanced gas turbine, better turbine engine performance can be achieved. The rotor inlet temperature in advanced gas turbine is much higher than the melting point of the blade material [2]. Therefore, it is compulsory to cool the blades for safe operation. Extracted air from the compressor of the engine is used to cool the blades either externally or internally.
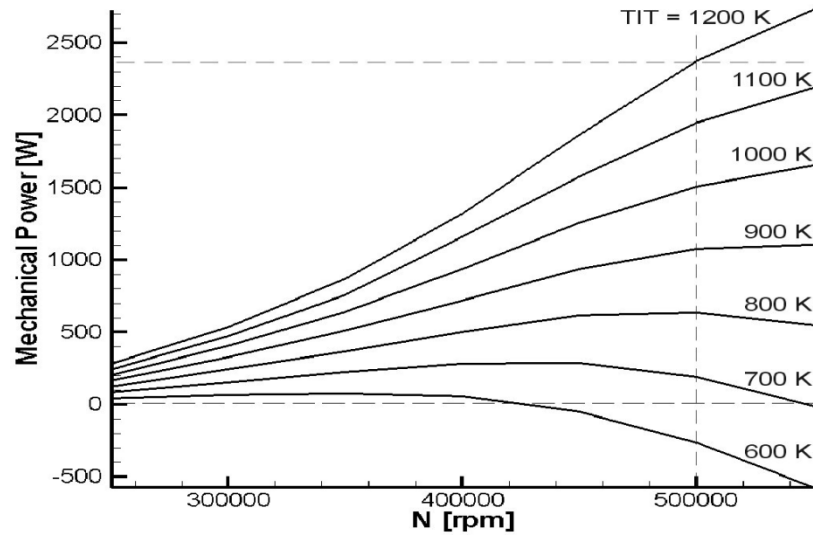
Figure 2.1: Relationship between turbine inlet temperature and mechanical power

2.2 Turbine Blade Cooling Principle

There are two distinct methods of turbine blade cooling; Air cooling and Liquid cooling. The air cooling can be divided into external and internal cooling. The external air cooling comprises the application of transpiration cooling where the cooling air is forced through a porous blade wall [1]. This method is the most economical in air cooling process because the effusing layer of air insulates the outer surface of the turbine blade from the hot gas stream. The heat is also uniformly removed.

Spray cooling method is proven not practical because there are some difficulties in channeling liquid to and from the blades. Besides, it is difficult to eliminate corrosion or the formation of deposits in open systems and there will be inadequate secondary surface cooling area at the base of the blades for closed systems. The proven practical methods are air cooling, internal liquid cooling and forced convection. The blade temperature can be reduced by 200°C and 300°C when there is 1.5-2 % of the air mass flow used for cooling per blade row [1].

4

Figure 2.2 shows various types of gas turbine blade cooling techniques that are used in the advanced gas turbine (AGT) engine. Each of the cooling technique produces different cooling effect and they are tested experimentally using different kind of test rigs. There are two major types of cooling technique which are air cooling and liquid cooling. In this project, the internal cooling technique will be the main focus of the investigation due to its practicality and effectiveness.
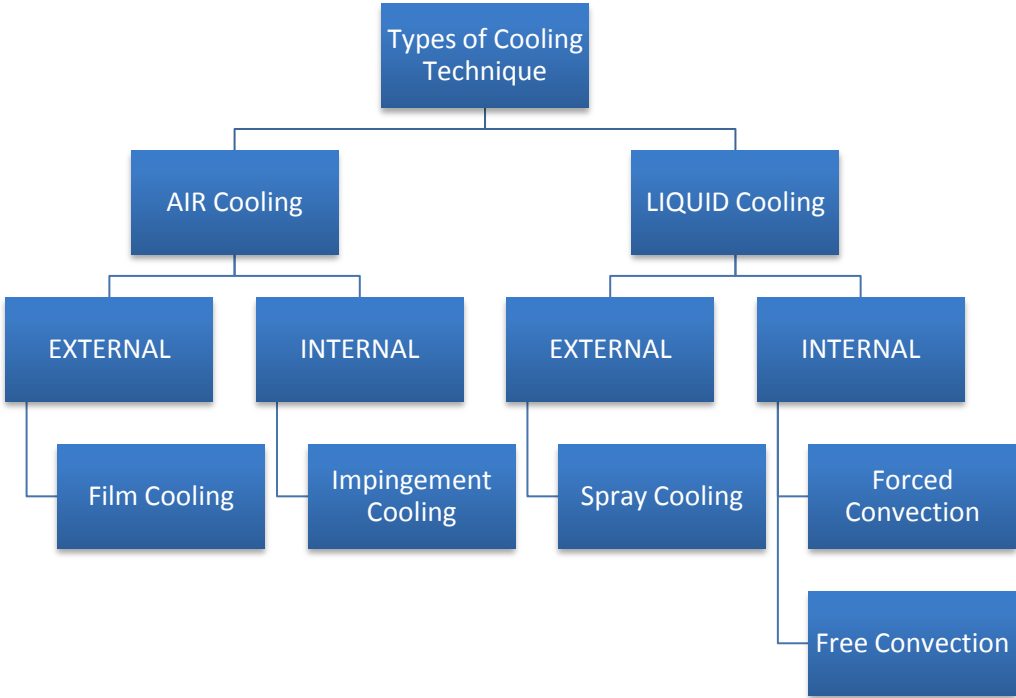

Figure 2.2: Types of cooling technique [1].

There are two main aspects in cooled turbine design. The first aspect is to determine an aerodynamic design that requires the least amount for a specific cooling performance. The cooling performance parameter used is *blade relative temperature.*

Blade relative temperature $=$ $(T_b - T_{cr}) / (T_g - T_{cr})$ (1)

where T_b = mean blade temperature

T_{cr} = coolant temperature at inlet (i.e. at the root radius r_r)

T_g = mean effective gas temperature relative to the blade (= static temperature + 0.85 x dynamic temperature

Secondly, the cycle efficiency losses resulted from cooling process. The examples of the losses are

a) The reduction in turbine mass flow.
b) The expansion is not adiabatic.
c) Pressure loss and reduction in enthalpy due to the mixing of spent cooling air with the hot main gas stream.

The hot turbine blade is convectively cooled internally and externally to maintain its crystalline structure integrity. Heat transfer process is the key to investigate the convective turbine blade cooling process. Ainley [9] stated that the blade temperature $T_b(y)$ increases with spanwise distance from the root, (y), not attaining a maximum until the end of the blade $(y = H)$. Ainley [9] studied the convectively cooled, thin walled blades of constant section, subject to uniform inlet gas temperature T_g. The basic equations for the heat transferred from gas to coolant per unit span (q) as

$$q = \lambda_{met} \; A_{met} \; \frac{d^2 T_b}{dy^2} + h_g S_g \; (T_g - T_b)$$
$$= h_{cl} S_{cl} \; (T_b - T_{cl}) \tag{2}$$

Where, λ_{met} is the metal conductivity and A_{met} is the metal area in a blade cross section.

And

$$q = m_{cl} C_{p,cl} \frac{dT_{cl}}{dy} = h_{cl} S_{cl} (T_b - T_{cl})$$ (3)

$$\frac{T_g - T_b}{T_g - T_{cl,in}} = \left[\frac{\overline{X}}{1 + \overline{X}}\right] e^{-k\left(\frac{y}{H}\right)}$$ (4)

$$\frac{T_g - T_{cl}}{T_g - T_{cl,in}} = e^{-k\left(\frac{y}{H}\right)}$$ (5)

Where, $k = \frac{\overline{X}}{[(1 + \overline{X})w^+]}$ and the $\overline{X}$ indicates the mean spanwise value of X.

J. Horlock et al. [3] stated that in the conventional analyses, the cooling efficiency is defined as

$$(\eta_c) = \frac{T_{cl,out} - T_{cl,in}}{T_b - T_{cl,in}}$$

$$(\eta_c) = 1 - \exp\left(\frac{-X}{w^+}\right)$$ (6)

$$(w^+) = \frac{\varepsilon_o}{\eta_c (1 - \varepsilon_o)}$$ (7)

Where, ε_o is the cooling effectiveness

$$\varepsilon_o = \frac{T_g - T_b}{T_g - T_{cl,in}}$$ (8)

Horlock J.H. et al [3] has produced the paper of turbine blade cooling and investigated on the blade temperature distribution. Figure 2.3 shows the temperature distribution of a thick walled blade being convectively cooled while operating at the temperature of 1700K. Meanwhile the coolant inlet temperature is set at 700K. Horlock J.H. et al [3] stated that the external blade temperature drops from 1100K to 1057K.
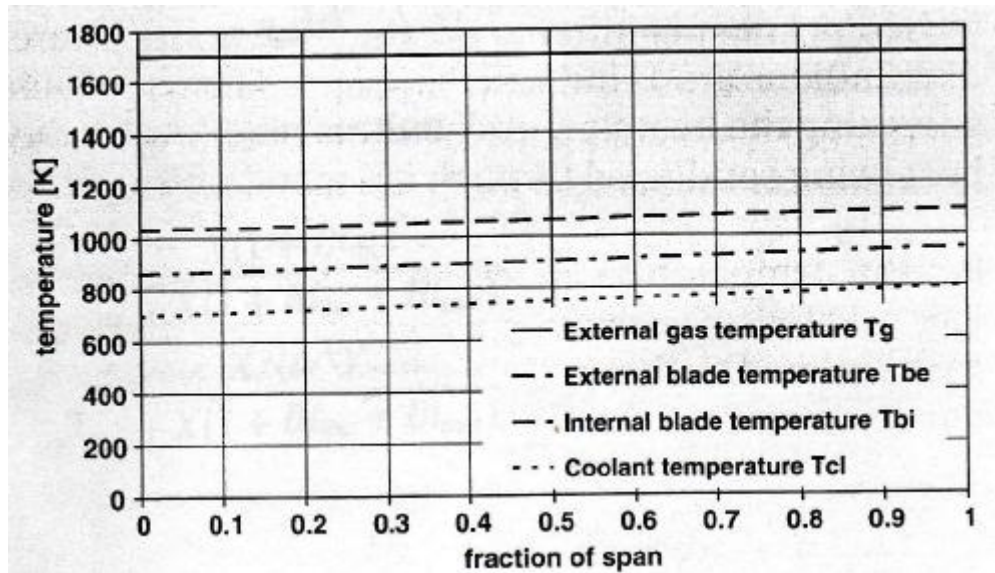
Figure 2.3: Temperature distribution of thick walled blade

2.3 Impingement Cooling Technique

A turbine blade with impingement cooling is provided with at least one partition in the interior of the hollow turbine blade. It is purposely designed to form a cooling-air chamber with cooling air is supplied from the 14th stage compressor. With the formation of an impingement air cooling chamber, the partition is provided with a plurality of impingement air channels to apply cooling air to the remotely adjacent inner surface of the hot outer wall sections.

The stator vanes and rotor blades in the turbine stage downstream of the combustion chamber are subjected to high mechanical stresses due to high operating temperature and limited thermal loadability components. In order to prevent transgression of the material-specific temperature limit, the respective components and, in particular, their thermally highly loaded areas are, as is generally known, cooled with cooling air tapped from the compressor.
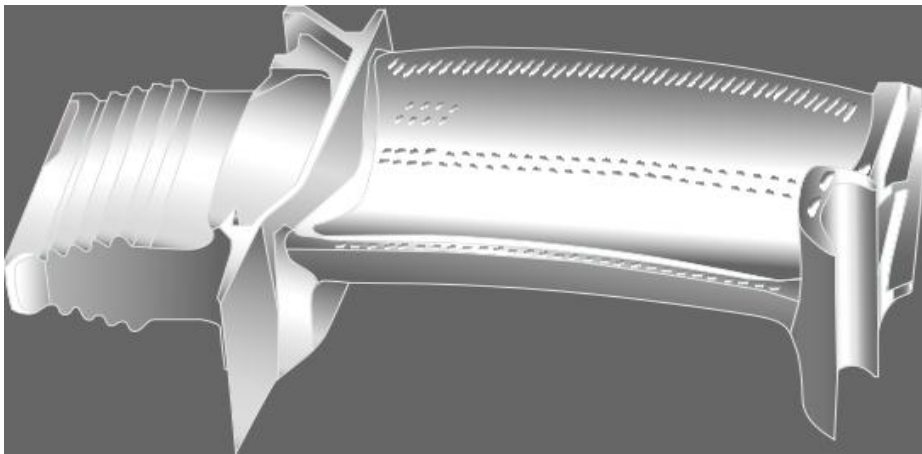
Figure 2.4: Cross section of turbine blade with cooling passage.

In impingement cooling process of a turbine blade, the longitudinal partitions are arranged in the inner of a hollow blade confined by two side walls which, together with a side wall section, form a long cooling air supply and distribution chamber (cooling air chamber) and, adjacent to the cooling air chamber, several impingement air cooling chambers. Via the impingement air channels, the cooling air introduced into the cooling air chamber flows simultaneously into the adjacent impingement air cooling chambers, thereby intensely cooling the inner surfaces of the thermally highly loaded areas of the outer walls of the turbine blade from the inside and enabling the gas turbine to be operated with high efficiency at maximum combustion temperatures and without material damage. The impingement air channels are straight-lined, but inclined within the partition to ensure a favorable angle of impingement of the impingement cooling air onto the inner surfaces of the outer walls.

In addition, the air exiting from the impingement air cooling chambers via air channels in the sidewalls of the turbine blade creates a barrier layer between the blade material and the hot gas which further reduces the thermal load of the turbine blade. While the impingement air channels reduce the load-carrying area of the partitions supporting the outer walls, load peaks occur in the area of the impingement air channels which results in high local mechanical stresses and, in consequence, a reduction of the life of the turbine blade. Furthermore, appropriately large dimensioning of the thickness of the partitions, which would decrease the local load peaks, is to be ruled out for reasons of weight and associated loads.

2.4 Film Cooling Technique

The fourteenth stage bleed air is used to cool the inner walls of the nozzle guide vane (NGV) and the other blades. The air then provides film cooling when it is injected out through leading edge holes, gill holes, mid-chord holes and trailing edge slots and produce a protection layer against the hot gas mainstream.
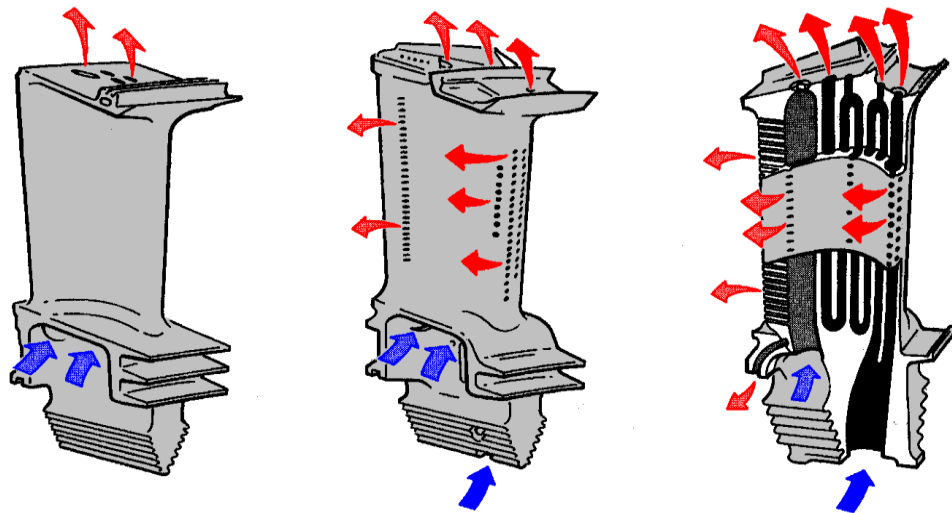

Figure 2.5: Film cooling technique – coolant air is injected out through leading edge holes, gill holes, midchord holes and trailing edge slots.

10

Film cooling depends on several aspects which are:

- Coolant to hot mainstream pressure ratio (p_c / p_t)
- Temperature ratio (T_c / T_g)
- Film cooling hole location, configuration and distribution

The coolant to hot mainstream pressure ratio is related to the coolant mainstream flux ratio or also known as blowing ratio. The coolant to hot mainstream temperature ratio is related to coolant to hot mainstream density ratio. These ratios are very useful to quantify film cooling effectiveness because these ratios will result in the ratio of the coolant to hot mainstream thermal capacitance.

Generally, the higher pressure ratio may produce better film-cooling protection at a specific temperature ratio. For example it can reduce the heat transfer to the turbine blade. Lower temperature ratio will result in better film-cooling protection at a specific pressure ratio. However, the film-cooling protection will be reduced when there is too high pressure ratio due to jet penetration into mainstream.

Thus it is crucial to optimize the amount of blade film cooling under engine operating conditions at Reynolds number of approximately equal to ~10^6 and Mach number is approximately 0.9 at exit conditions. The effects of turbulence and unsteady wakes on the blade can affect the film-cooling performance. In addition, the film-hole location, distribution, angle, and shape also can affect the film-cooling performance. These effects are tested experimentally and are run through some simulations.

Dutta et al. [2] studied the effect of rotation in a triangular two pass duct with smooth walls. They found that as the rotation number increased, the trailing surface of the duct experienced heat transfer enhancement, whereas the heat transfer from leading surface decreased as compared to stationary channels. Griffith et al. [4] studied narrow rectangular channels (AR = 4:1) oriented at 135 degree with respect to the plane of rotation. They found the duct orientation has a significant effect on the leading, inner, and outer surfaces of the smooth rotating channel. However, the effect of rotation on the

trailing surface is minimized. This study also showed Nusselt number ratio can vary as much as 25% in the spanwise direction.

Stephen et al. [13] studied inclined ribs in a straight non-rotating square duct to predict numerically the flow and heat transfer in the non-rotating square duct. Zerkle et al. [14] employing a high-Reynolds-number k-e turbulence model with wall function, performed a numerical prediction of flow and heat transfer in a rectangular duct with 90 degree ribs with and without rotation. However, their calculations used periodicity and neglected buoyancy effects. They suggested that a low- Reynold number turbulence model is necessary to simulate real gas turbine engine conditions and a Reynolds-stress model is required to capture anisotropic effects. Bonhoff et al. [15] calculated the heat transfer coefficients and flowfields for rotating U-shaped coolant channels with 45-deg angled ribs. They used a Reynolds-stress turbulence model with wall functions in the FLUENT computational-fluid –dynamic code.

CHAPTER 3

METHODOLOGY

3.1 Project Works

To start the project the author shall identify the list of work that will be done during the project. In addition, the author must get familiarize with the software to be used in carrying this project. Figure 3.1 below shows the methodology planned for the implementation of this project.
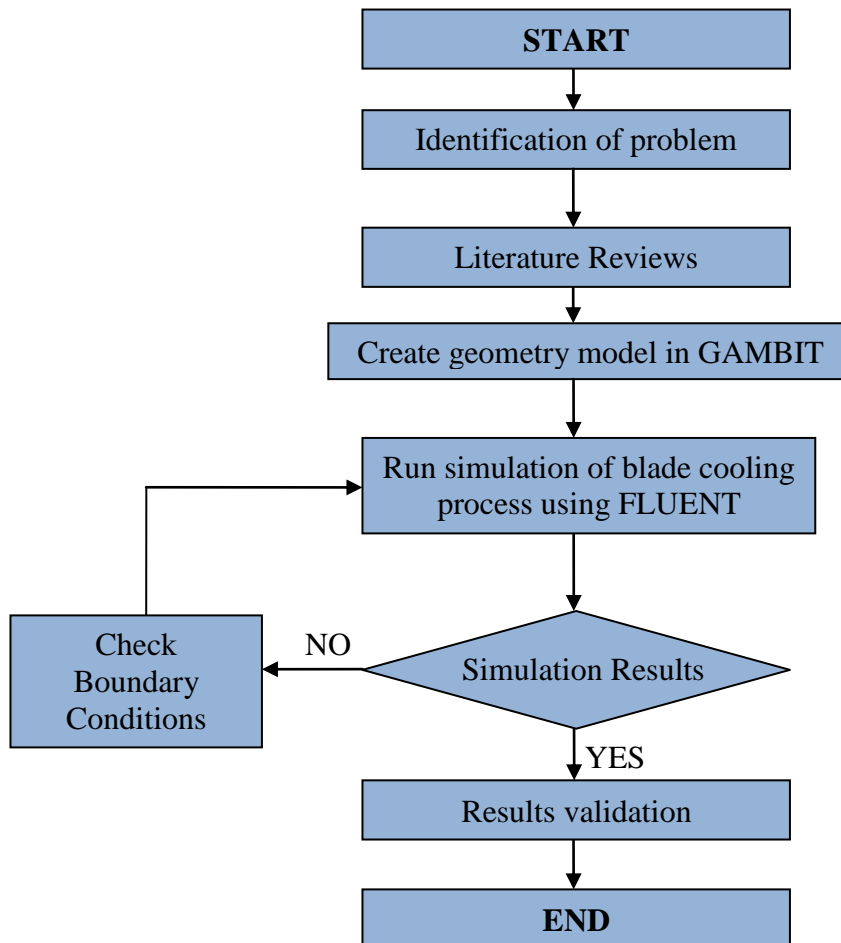
```
                    ┌─────────────────────┐
                    │       START         │
                    └─────────────────────┘
                              │
                    ┌─────────────────────┐
                    │ Identification of   │
                    │      problem        │
                    └─────────────────────┘
                              │
                    ┌─────────────────────┐
                    │ Literature Reviews  │
                    └─────────────────────┘
                              │
                  ┌───────────────────────────┐
                  │ Create geometry model     │
                  │ in GAMBIT                 │
                  └───────────────────────────┘
                              │
          ┌──────────────────────────────────┐
          │ Run simulation of blade cooling  │
    ┌────▶│ process using FLUENT             │
    │     └──────────────────────────────────┘
    │                       │
┌────────┐   NO      ◇──────────────────◇
│ Check  │◀──────────  Simulation Results
│Boundary│            ◇──────────────────◇
│Conditions│                  │ YES
└────────┘                     │
                    ┌─────────────────────┐
                    │ Results validation  │
                    └─────────────────────┘
                              │
                    ┌─────────────────────┐
                    │        END          │
                    └─────────────────────┘
```

Figure 3.1: Flowchart of the methodology of the project.

3.2 Introduction to Fluent Software

Fluent is a computational fluid dynamic (CFD)software created by ANSYS, Inc. for simulating fluid flow, heat and mass transfer, and a related phenomena that involves turbulence, reactions, and multiphase flow. Furthermore, Fluent's CFD technology with the help of Gambit software (a single interface for geometry creation and meshing brings together most of Fluent's preprocessing technologies in one environment), can be a great helping hand in meshing, structural dynamics, optimization, and multiphysics technology that is able to deliver a comprehensive set of computer-aided engineering (CAE) tools.

These software products from Fluent allow researchers to simulate flows with complex rheology and deforming surfaces. Traditional build-and-test methods cause interruptions on the product line. CFD, however, gives a great view on current phase or situation inside the equipment. The multiphysics capabilities of Fluent will enable researchers to improve their product development processes, reduce time-to-market for new products, and improve product innovation and performance.

3.3 Case Study

The study focuses on two types of cooling technique which are film-cooling and impingement cooling. This is because these techniques are interrelated with each other. As mentioned previously, coolant air is bled from the fourteenth stage compressor to the turbine blade. The coolant air acts to cool the inner surfaces of the turbine blade via the cooling passage. Then the coolant air is injected out through the gill holes, midchord holes, leading edge holes and trailing edge slots at the surface of the airfoils.

The coolant that is injected out produce a protection layer on the airfoil surface against the heat flow from the hot mainstream; therefore, the parameters that affect the heat transfer must be investigated. Those parameters are Reynolds number, Mach number, coolant to pressure ratio and coolant to temperature ratio. Afterwards, the information will be used for the modeling work.

3.4 Modeling Geometry Using GAMBIT Software

A geometry model of a turbine blade must be created first in order to conduct a simulation of the turbine blade cooling process. In this study GAMBIT software is used to model the geometry of the turbine blade as well as used to generate a mesh of a geometry model created initially. A geometry model must undergo the mesh process first before being read and solved by a solver which is FLUENT software. Mesh is defined as a volume or face of a geometry that is divided into discrete cells. For this project, it is to be clear that there is no blade design process involved and the study is focused in internal cooling of the existing blade and thus, blade design process will not be considered. In internal cooling, there is a cooling slot in the turbine blade that allows the coolant air to cool the internal part of turbine blade and some the air is ejected out to produce film cooling on the turbine blade surface.

Assumptions are made before proceeding with the simulation so that the main area of concern can be focused. The turbine blade is assumed to be stationary in this study, hence the Coriolis effect is negligible. This is because the turbulence flow and Coriolis Effect produce significant effect on the cooling process. Only one turbine blade is used in the simulation so that the flow of the hot gas mainstream on the turbine blade and the cooling process can be monitored easily. The turbine blade dimension is taken from the UTP fluid mechanics laboratory where the real turbine blade dimension is measured.

Steps in modeling a geometry using GAMBIT software:

1. In an external flow such as that over a turbine blade, it is important to define a farfield boundary and mesh the region between the turbine blade geometry and the farfield boundary. It is a good idea to place the farfield boundary well away from the turbine blade since the ambient conditions are used to define the boundary conditions at the farfield.

2. To specify the turbine blade geometry, the dimension of the real turbine blade is acquired from the laboratory. The dimension of the blade is measured and it is transformed into vertices form in Gambit software. With the vertices, geometry of the turbine blade could be created.

3. In order to make the cooling process of the turbine blade take place like the real condition, three volumes of cooling passages were created in the turbine blade volume geometry. Meanwhile, a big rectangle volume is created as hot mainstream gas which surrounds the turbine blade.

4. Next, the boundary types need to be specified. This means all faces of the volumes need to be assigned to the right boundary types; for example the turbine blade face is assigned to WALL boundary type.
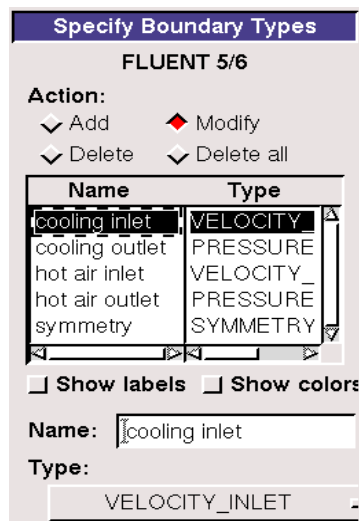

Figure 3.2: Boundary types

16

5. Then the continuum types need to be specified for each volume. For example, the volume of the turbine blade geometry is indicated as a SOLID type. Meanwhile the hot mainstream volume and cooling passage are indicated as FLUID type.
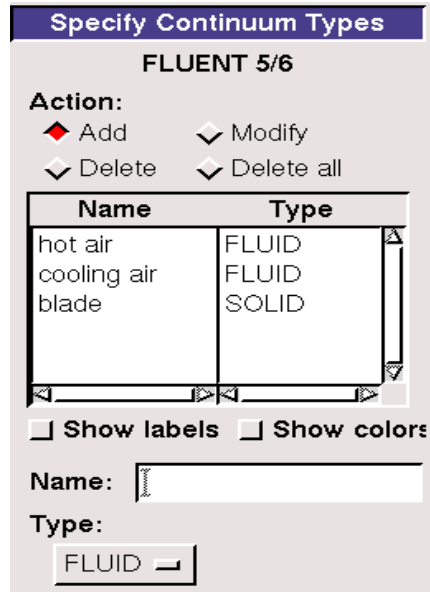

Figure 3.3: Continuum types

6. Finally, the meshed geometry is saved and exported as.msh file. (Refer to APPENDIX A-1 to see the complete geometry of turbine blade modeled in GAMBIT)

3.5 Solving Problem Using Fluent Software

Fluent is a type of solver used in solving a problem regarding fluid dynamic. The "3ddp" option is used to select the 3-dimensional, double-precision solver. In the double-precision solver, each floating point number is represented using 64 bits in contrast to the single-precision solver which uses 32 bits. The extra bits increase not only the precision but also the range of magnitudes that can be represented. The downside of using double precision is that it requires more memory. After the geometry model has been created and mesh in Gambit, the .msh file is being imported to FLUENT software to be read.

In all of the approaches used, the same basic procedure is followed to solve a problem. During preprocessing, the geometry (physical bounds) of the problem is defined which means the volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non uniform. Then, the physical modeling is defined which is for example, the equations of motions that involved enthalpy, radiation and species conservation. The starting point is the conservation of momentum's flux in a turbulent jet.

where r is the radius of the jet. Once α the opening angle , x_o the initial position on the x–axis and v_0 the initial velocity are introduced, the section A at position x is

$$A = \pi r^2 \tag{9}$$

$$A(x) = \pi \left(r_0 + (x - x_0)\tan\left(\tfrac{1}{2}\alpha\right)\right)^2 \tag{10}$$

The conservation of the total momentum flux states that

$$\rho v_o^2 A_0 = \rho v(x)^2 A(x) \tag{11}$$

where $v(x)$ is the velocity at position x and A_0 the initial section. Due to the turbulent transfer, the density ρ is the same on both the two sides of equation (3). The trajectory of the jet as a function of the time is easily deduced from equation (3)

$$x = -\frac{-x_0(\tan(\frac{1}{2})\alpha) + r_0 - \sqrt[2]{r_0(r_0 + 2\tan\left(\frac{1}{2}\right)\alpha)v_0 t}}{\tan\left(\frac{1}{2}\right)\alpha} \qquad (12)$$

The velocity turns out to be

$$v(t) = \frac{v_0 r_o}{\sqrt[2]{r_0(r_0 + 2\tan\left(\frac{1}{2}\right)\alpha)v_0 t}} \qquad (13)$$

Next, the Boundary conditions are defined. This involves specifying the fluid behavior and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined. The simulation is started and the equations are solved iteratively as a steady-state or transient. Finally a postprocessor is used for the analysis and visualization of the resulting solution.

Air is selected as *Fluid* Materials. The density of the air is set to ideal-gas and Cp is set to constant and equal to 1006.43 j/kg-k. Molecular Weight is constant and equal to 28.966 kg/kgmol. By choosing the ideal-gas option means that FLUENT will utilize the ideal gas equation of state to relate density to the static pressure and temperature.

It can be seen that for all flows, FLUENT uses the gauge pressure internally in order to achieve minimum round-off errors. Any time an absolute pressure is needed, as in the ideal gas law, it is generated by adding the operating pressure to the gauge pressure:

Absolute pressure = Gauge pressure + operating pressure

Round-off errors exist when pressure changes Δp in the flow are much smaller than the pressure values p. One then gets small differences of large numbers. In the supersonic flow, the final result will be containing with significant variation in the absolute pressure so that pressure changes Δp are comparable to pressure levels p. So we can work in terms of absolute pressure without being hassled by pesky round-off errors. To have FLUENT work in terms of the absolute pressure, set the Operating Pressure to 0 Pa. The Gauge Pressure is set to 101325 Pa, while the Mach number is set to 3. The temperature is set to 300K so that it can be assumed as ambient temperature.

Furthermore, Titanium is selected for *Solid* Materials. The density of the titanium is set to 4850 kg/m^3 and C_p is 544.25 j/kg-k. Meanwhile the thermal conductivity for titanium is 7.44w/m-k). After both materials have been specified accordingly to their types, the boundary condition and operating condition need to be specified based on the real condition in the gas turbine engine. Since the data of the operating condition of the gas turbine engine is not available yet, the data is obtained from the relevant sources.

Steps involved in solving problem using FLUENT software:

1. First, need to check the grid to make sure that there are no errors. Any errors in the grid would be reported at this time. The grids will be displayed in graphic window. **Figure 3.4** shows the turbine blade grid where the green color sections represent the three cooling passages in the turbine blade.
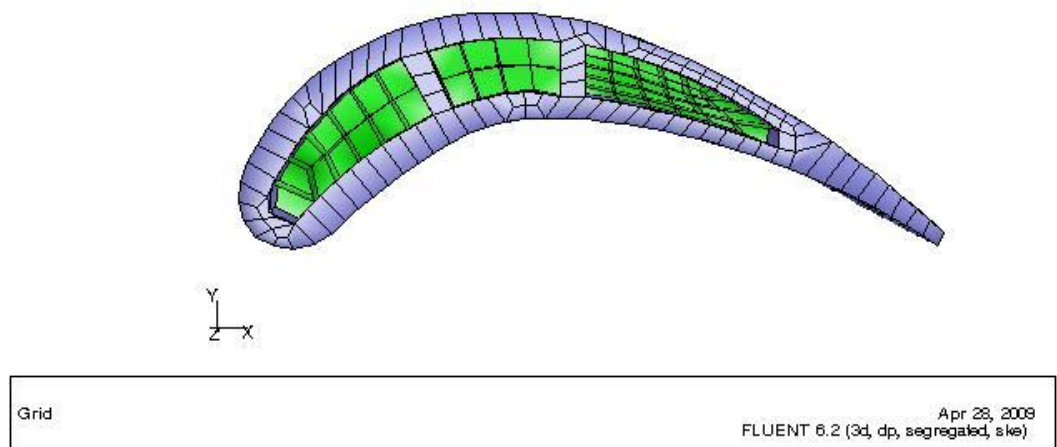

Figure 3.4: Turbine blade grid display

2. Viscous model is the crucial part to be defined. In this study k-epsilon viscous model was chosen due to the turbulent flow properties having a significant effect in solving this problem. **Figure 3.5** shows the viscous model type being used.

3. Air is chosen to be the coolant type while titanium was chosen as the turbine blade material. **Figure 3.6** and **Figure 3.7** show the material properties of the blade material and the type of coolant being used.
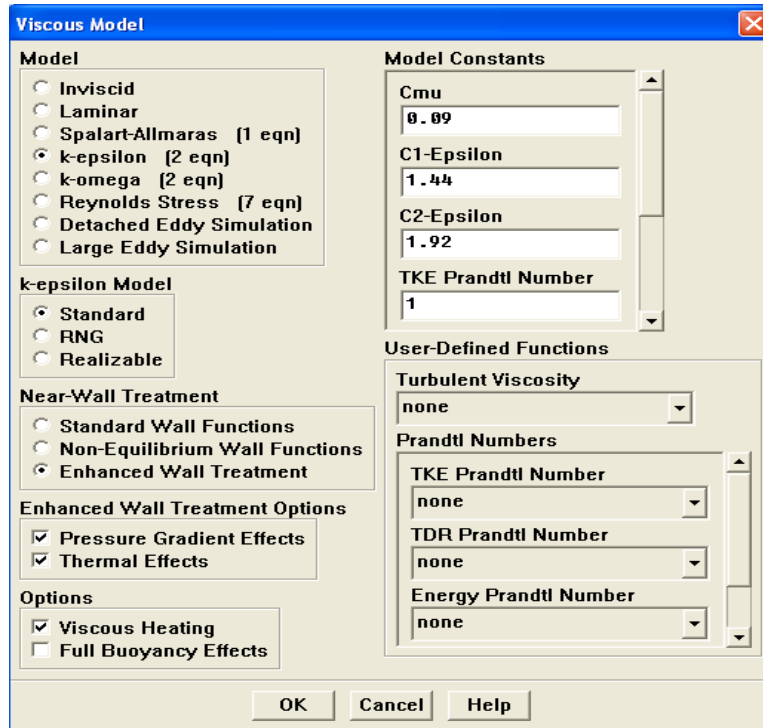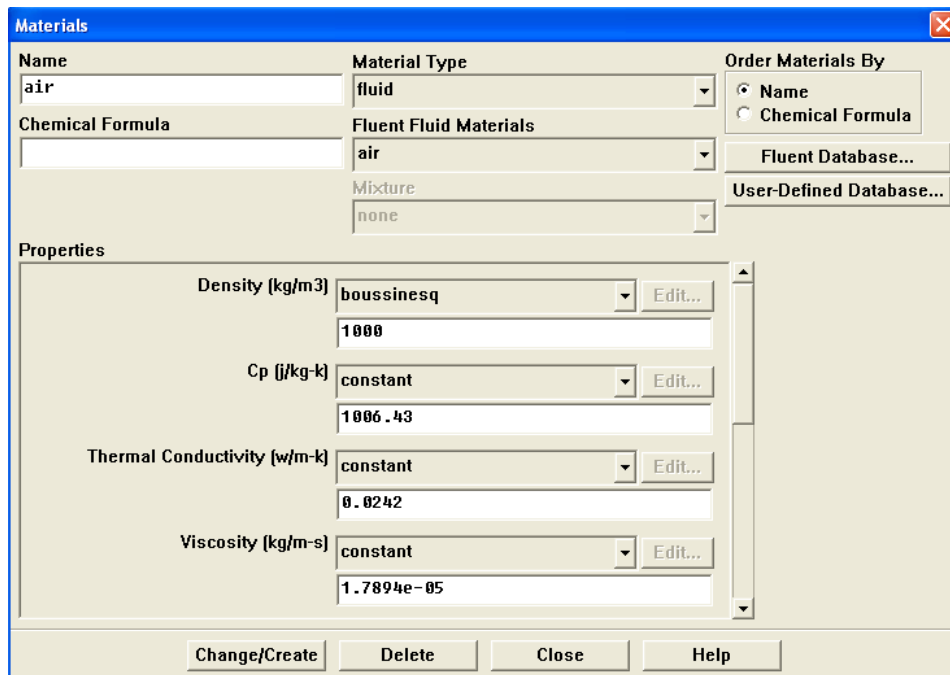
Figure 3.5: Viscous model type

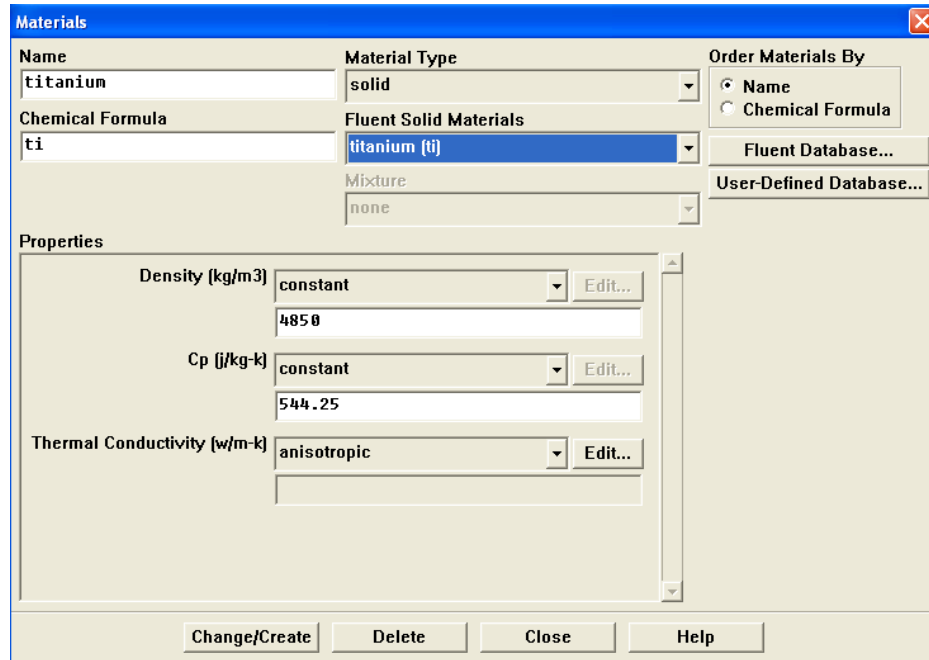**Figure 3.6: Material properties of air**

Figure 3.7: Material properties of turbine blade

4. Then, the boundary conditions and the operating conditions are defined as shown in Figure 3.8 and Figure 3.9.
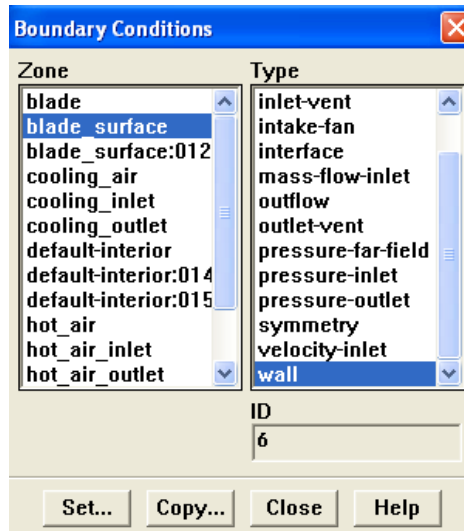
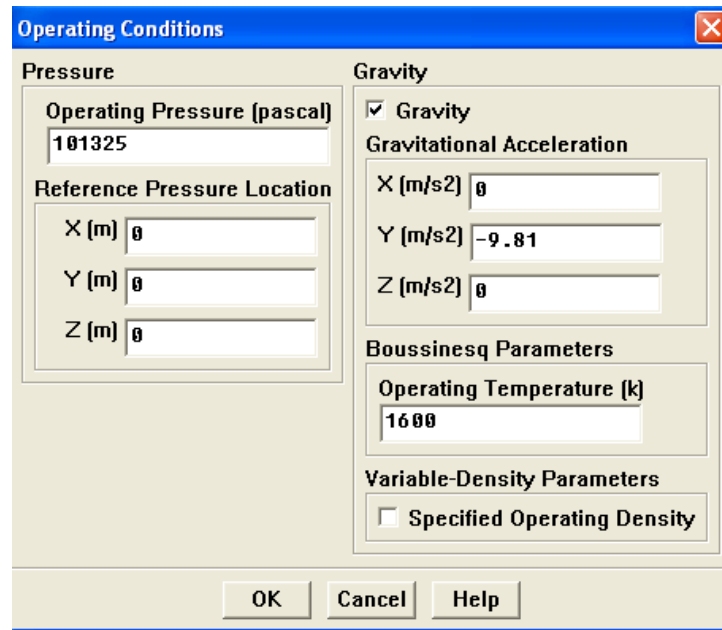

Figure 3.8: Boundary conditions

Figure 3.9: Operating Conditions

5. To solve the problem, initial guess need to be set up. The flow field is initialized to the values stated in the problem or available data.

6. The convergence criterion is set up. FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. The residuals will be plotted in the graphic windows as they are calculated. This completes the problem specification.

7. Next is to iterate the residuals until the convergence is reached. The residual for each iteration is plotted in the display window as they are calculated. In this study, the iterations used will be 50. The iterations is chosen to be 50 because the energy and momentum equations will start to converge at 25th iteration; so there is no need for redundancy.

8. Finally the results will be analyzed and visualized. The postprocessor analysis will be done at this stage.

CHAPTER 4

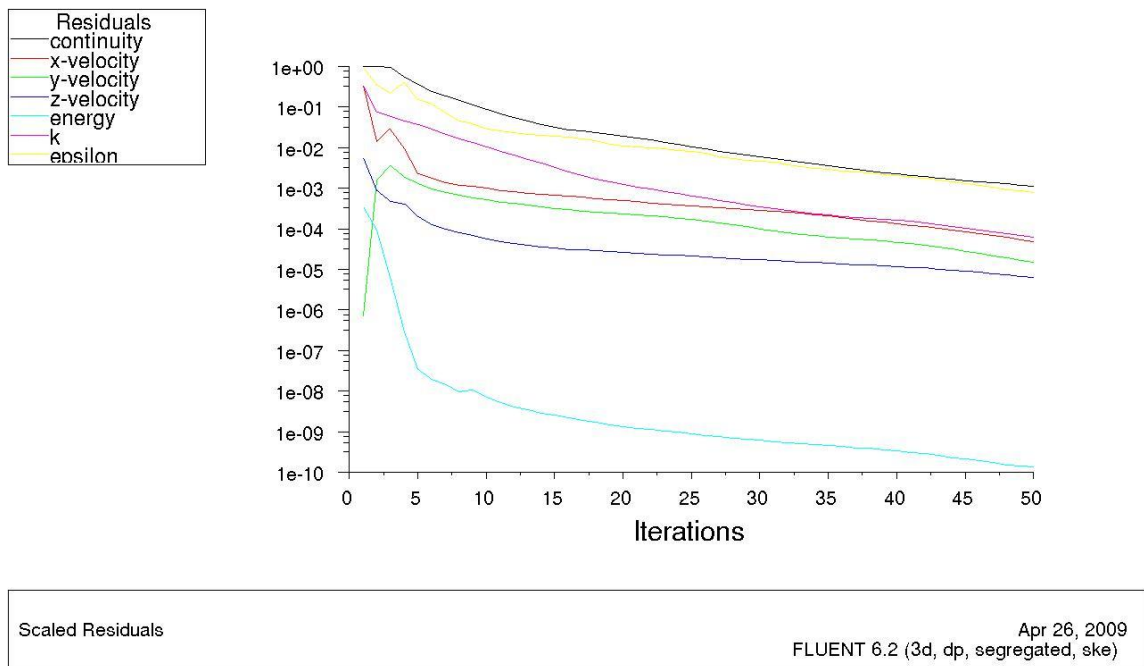# RESULTS AND DISCUSSION

4.1 Iteration result

Figure 4.1: Scaled residuals

Figure 13 represents the scaled residuals. To achieve convergence, all discrete conservation equations such as energy equation and momentum equation are obeyed in all cells to a specified tolerance or the solution no longer changes with subsequent iterations. Overall mass, momentum, energy, and scalar balances must be achieved. When the residuals are decreased by three orders of magnitude, it indicates at least qualitative convergence where the major flow features should be established. For this type of solver used in this study, the scaled energy residual must decrease to 10^{-6} or lower.
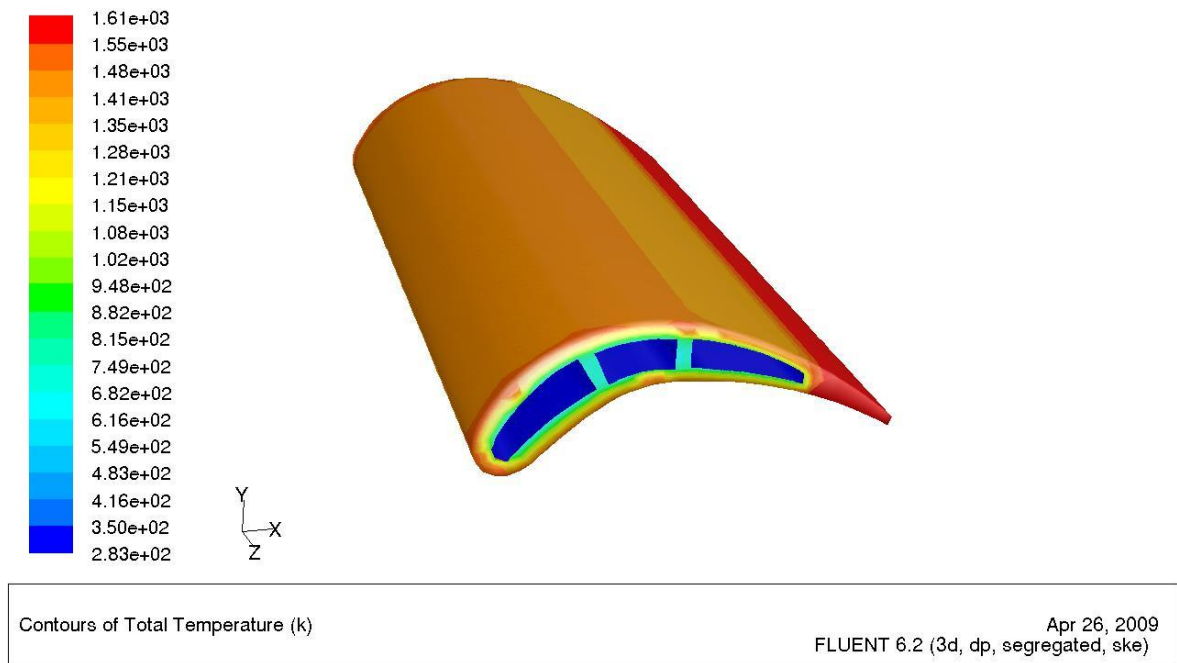
4.2 Simulation Results

Figure 4.2: Temperature distribution on the suction side

Figure 14 shows the temperature distribution on the blade suction side The temperature of the hot gas mainstream is set to be 1600K to achieve optimal thermal efficiency. It can be observed that the temperature on the suction side (exclude blade tip) of the turbine blade lies within the range of 1350 K to 1480K. It is clearly noted that in this simulation the turbine blade is cooled internally and the cooling process takes place through the three cooling passages located at the leading edge, midchord and trailing edge. Trailing edge surface exhibits the lowest temperature which is 1350K.

The cooling air is bled from the fourteenth stage compressor with the temperature of 283K. Heat is convectively transferred from the hot surface of the turbine blade to the cooling air and consequently the temperature of the turbine blade is reduced by more than 200K. This proves that the cooling process takes place and helps to maintain the turbine blade operates at high temperature.
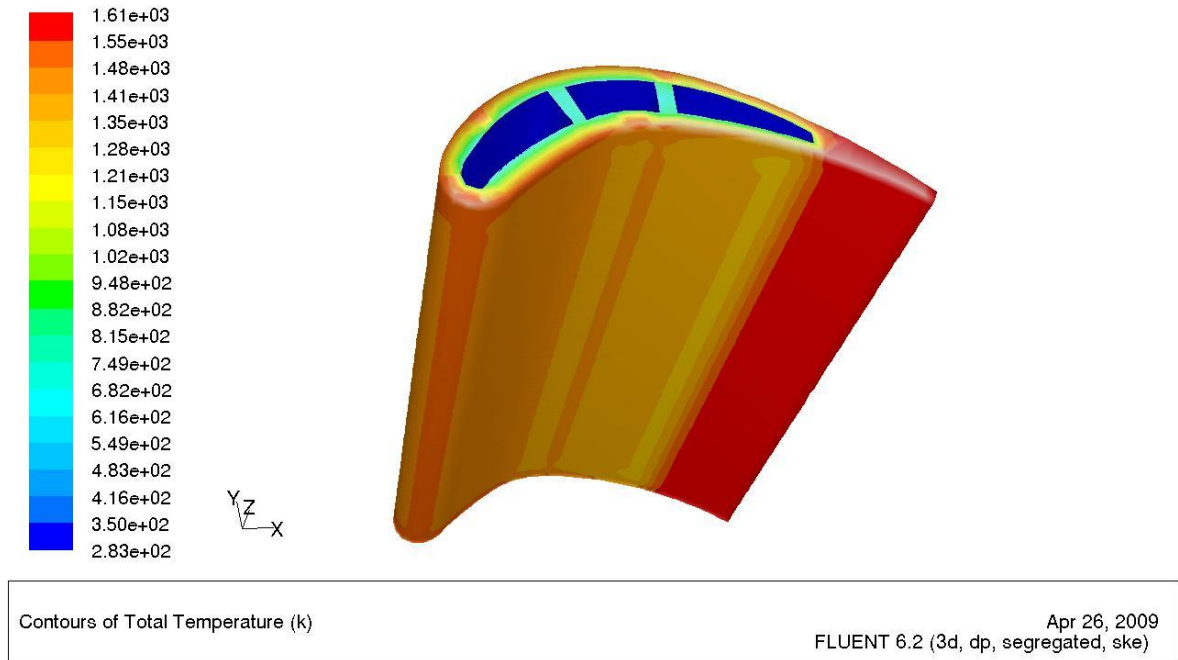
Contours of Total Temperature (k)   Apr 26, 2009
                                          FLUENT 6.2 (3d, dp, segregated, ske)

Figure 4.3: Temperature distribution on blade pressure side

Figure 15 shows the temperature distribution on the blade pressure side. On the pressure side, the temperature of turbine blade (exclude blade tip) lies within the range of 1280K to 1480K. The temperature at the front chord is 1540K which explains that surface area experiences high velocity of hot gas mainstream. Besides, more heat from the hot gas mainstream is concentrated at that leading edge surface even though that area is cooled convectively by the first cooling passage. The area which exhibits the lowest temperature (1280K) lies between the mid chord and the trailing edge.

It is clearly indicated that the turbine blade tip exhibits the highest temperature which is 1540 K. This is because there is no cooling process occur at the end of trailing edge due to dimensional constraint to create another cooling passage at the end of trailing edege. However, in modern turbine blade structure the problem can be solved by introducing film cooling which poduces a thin layer of cooling air along the blade surface to protect the blade from the hot gas mainstream.
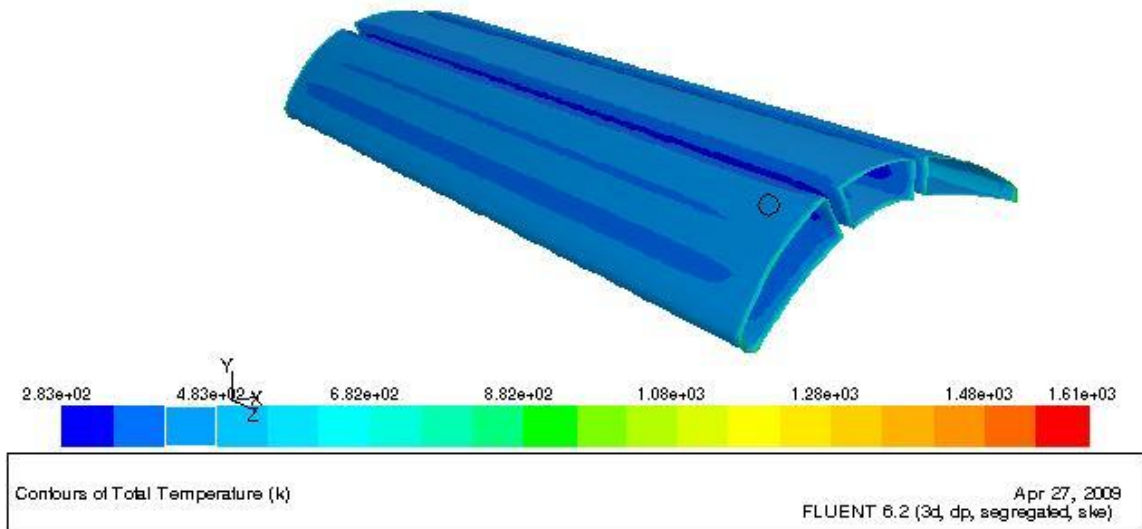
Figure 4.4: Temperature distribution on cooling passage wall

Figure 16 represents the wall of the cooling passages. It can be seen that the temperature of these cooling passages fall in the range of 350K to 420K. Thus, this proves that the heat transfer process occurs along the cooling passage. The cooling air is initially bled from the fourteenth compressor stage with the temperature of 283K. When the cooling air enters the cooling passage, cooling process starts to take place and the temperature of cooling air is increasing along the cooling passage as the convection heat transfer process occurs.
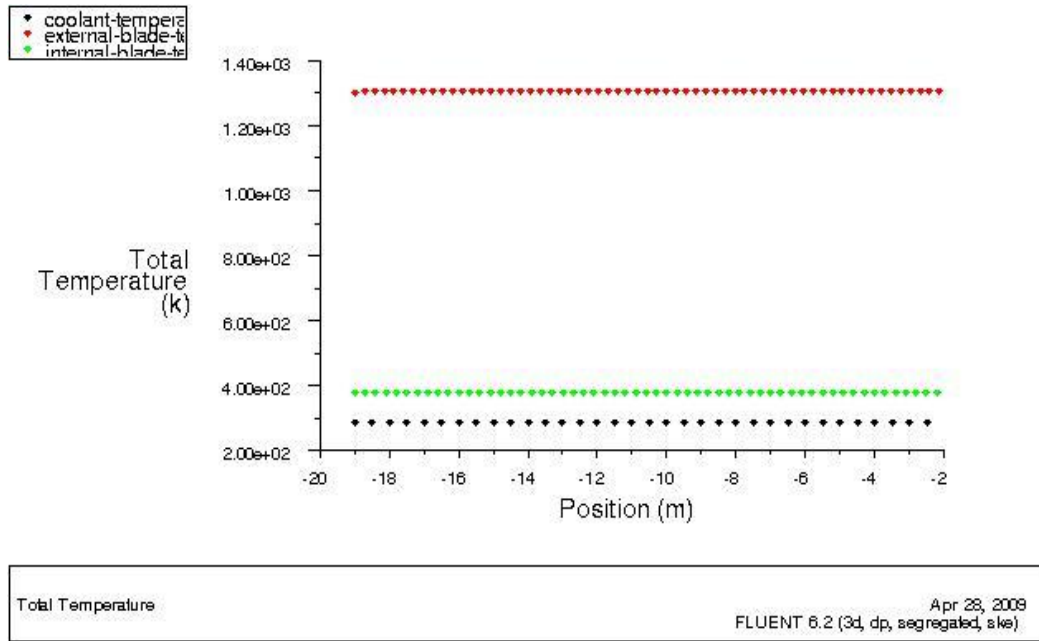
Figure 4.5: Plot of total temperature

Figure 17 shows the plot of total temperature of three different mediums. The red color represents external surface of the blade temperature; the green color represents internal surface of the blade temperature (cooling passage wall temperature), meanwhile black color represents coolant temperature. The result shows that the coolant temperature, internal blade temperature and external blade temperature are constant along the z-axis (spanwise direction

The result is quite inaccurate due to two types of errors which are notified as systematic error and random error. Systematic error could be occurred during the preprocessing (creating geometry and meshing in GAMBIT) stage, which means the meshing type that had been used, is inappropriate. Furthermore, in solving stage (solving using FLUENT) there are a lot of variables needed to be concerned. In this case, systematic error is caused from inaccurately defined boundary conditions and operating condition of the computational domain.

The volume of the far-field boundary used in this study is 20cm x 40cm x 60cm (hot gas mainstream volume); and this might not be inappropriate by comparing to the operating condition that was tested by Horlock J.H..Besides, the boundary condition of turbine blade wall is set to experience only convection heat transfer in this study; but in the real operating condition, there might be radiation heat transfer takes place from the hot gas mainstream to the turbine blade wall.

Random error also leads to the occurrence of the inaccuracy in the results. Random error could be occurred at the initial stage of this study. There must be error occurred while measuring the dimension of the turbine blade. Besides the turbine blade that had been used in this simulation study is different from what Horlock J.H. used in his study. In Horlock J.H. paper, the dimension of the turbine blade and its properties are not mentioned. Hence, different characteristics of turbine blade will produce different results.

CHAPTER 5
CONCLUSION AND RECOMMENDATION

5.1 Conclusion

This research has investigated the cooling process that is being implemented in the advanced gas turbine. Throughout the research period all the information was gathered from all reliable resources such as technical proceedings, journals and online resources. The main focus of this research is to investigate the temperature profile along the turbine blade surface while the cooling process takes place. Therefore the knowledge of heat transfer and fluid dynamics has been fully utilized to conduct the research. The research is conducted by simulating the turbine blade cooling process by using CFD software.

A preprocessor software which is called as GAMBIT has been used to create the meshed geometry of the turbine blade. Meanwhile, FLUENT software is the type of solver that has been used to set up the problem and simulate the cooling process of the turbine blade. The problem has been set up based on the real operating condition during the turbine expansion process. Finally, the result was analyzed and validated by using published data to determine the effectiveness of turbine blade cooling process.

In this study, the operating conditions of the gas turbine engine are set according to the real operating condition. Only a single turbine blade is simulated and the blade is stationary, thus the Coriolis effect can be neglected since there is no rotation involved. The turbine blade is convectively cooled by the compressed air that is bled from the fourteenth stage compressor. The inlet temperature of hot gas mainstream is set at 1600K and the hot gas flows all over the surfaces of the stationary turbine blade.

31

Meanwhile, the cooling air temperature is set at 283K and the coolant air flows through the internal cooling passages so that the convection heat transfer can take place. It is indicated that the temperature of the turbine blade in the hot gas mainstream has been reduced by approximately 200K while operating under high temperature condition. It is indicated that the temperature of those three cooling passages fall in the range of 350K to 420K. Thus, this proves that the heat transfer process occurs along the cooling passage

5.2 Recommendation

The cooling process simulation could not produce the better result due to flaw existence in the geometrical model which has been created in the preprocessor. The cooling passage geometry model needs to be improved so that the cooling process simulation can take place effectively. Besides, the turbine blade dimension that was being used by Horlock J.H. et al [3] is different from what has been simulated in this research. This is because the turbine blade properties such as the exact dimensions and blade material are not stated in the paper. Different properties of blade material and dimension will produce different results of simulation.

This research found that by running simulation based on the real operating conditions is not appropriate since a single turbine blade is used and it is stationary. Therefore to get a better result, a test rig need to be developed to study the blade cooling process experimentally. The test rig must comprise of a single stationary turbine blade and hot gas source and cooling air source. Hence the result of the simulation can be validated by using the experimental test that will be done in the first place. For the betterment of this project research, the test rig which is similar to the simulation conditions is recommended to be developed.

In this research, film cooling effect on the turbine blade has not been studied. In the real turbine blade cooling process, film cooling involves and occurs simultaneously with impingement cooling. Film cooling plays a significant role in the blade cooling

process because it produces a thin protection layer to the turbine blade. Hence, in the future study, the film cooling effect must be investigated. The gill holes must be created on the turbine blade geometry surface that is used in this study. Therefore the cooling air can be ejected out through these holes from the internal cooling passages thus producing film cooling layer. It is recommended that the cooling film effect need to be considered to improve the turbine blade cooling efficiency

By introducing higher turbulent intensity in the cooling passage may lead to higher heat transfer. As a result the efficiency of the cooling process can be increased and finally lead to higher thermal efficiency. A lot of studies have been made to create higher turbulent flow in the cooling passage such as by introducing ribs and pin fin in the cooling passage walls. However due to manufacturing constraints, those cooling passages that are designed with ribs and pin fins cannot be created. Therefore, more efforts are recommended to be done to produce new cooling passage design that provides better cooling effectiveness.

REFERENCES

[1] HIH Saravanamuttoo, GFC Rogers & H Cohen, *Gas Turbine Theory*, *5th edition* (Prentice Hall, 2001).

[2] Je – Chin Han, Sandip Dutta and Srinath V.Ekkad, *Gas Turbine Heat Transfer and Cooling Technology*, *1st edition* (Taylor & Francis, 2000).

[3] J. H. Horlock and L.Torbidoni, 2006 " Turbine blade cooling: the blade temperature distribution" *Journal of Power and Energy*

[4] T.S. Griffith, L. Hadhrami, and J-C. Han, 2002, "Heat Transfer in Rotating Rectangular Cooling Channels (AR=4) with Angled Ribs" *Journal of Heat Transfer,* Vol 124, No.4

[5] L.M. Wright, E. Lee and J-C. Han, 2004, " Effect of Rotation on Heat Transfer in Rectangular Channels with Pin Fins" *Journal of Thermophysics and Heat Transfer,* Vol 18, No.2

[6] G. Su, S. Teng, H-C. Chen and J-C. Han, 2004, " Flow and Heat Transfer Computations in Rotating Rectangular with V-shaped Ribs" *Journal of Thermophysics and Heat Transfer,* Vol 18, No.4

[7] M. Al-Qahtani, H-C Chen and J-C. Han, 2005, " Heat Transfer Prediction of Rotating Rectangular Channels Using Reynolds Stress Model" *Journal of Thermophysics and Heat Transfer,* Vol 19, No.1

[8] J.A. Parsons and J-C. Han, 2004, " Jet-Impingement Heat Transfer in Rotating Channels with Staggered Extraction Flow" *Journal of Thermophysics and Heat Transfer,* Vol 19, No.2

[9] Ainley, D.G. *Internal air cooling for turbine general design study*. Aeronautical Research Reports and Memorandum 3013, 1957.

[10] Nirmalan, N. V. and Hylton, L.D., 1990. "An Experimental Study of Turbine Heat Transfer with Leading Edge and Downstream Film Cooling." *ASME Journal of Turbomachinery,* Vol. 112

[11] Goldstein, R.J., 1971. "Film Cooling." *Advances in Heat Transfer*, Vol. 7

[12] Abuaf, N., Bunker, R., and Lee, C.P., 1997 " Heat Transfer and Film Cooling Effectiveness in a Linear Airfoil Cascade." *ASME Journal of Turbomachinery*, Vol. 119

[13] Stephens, M. A., Chyu, M. K. and Shih T.I.-P "Computation of Convective Heat Transfer in a Square Duct with Inclined Ribs of Rounded Cross Section," American Society of Mechanical Engineers, Paper 96-WA/HT-12, Nov. 1996

[14] Prakash, C. and Zerkle, R., "Prediction of Turbulent Flow and Heat Transfer in Ribbed Rectangular Duct with and Without Rotation," *Journal of Turbomachinery*, Vol. 177, No. 2, 1995, pp.255-264.

[15] Bonhoff, B., Tomm, U., Johnson., B.V., and Jennions, I., "Heat Transfer Prediction for U Shaped Coolant Channels with Skewed Ribs and with Smooth Walls," American Society of Mechanical Engineers. Paper 97-GT-162. June 1997.

[16] 1 August 2008 <http://cfd.me.umist.ac.uk/tmcfd/bladeposter/blade1.html>

APPENDICES

Coolant inlet

Coolant outlet

Hot Gas Mainstream
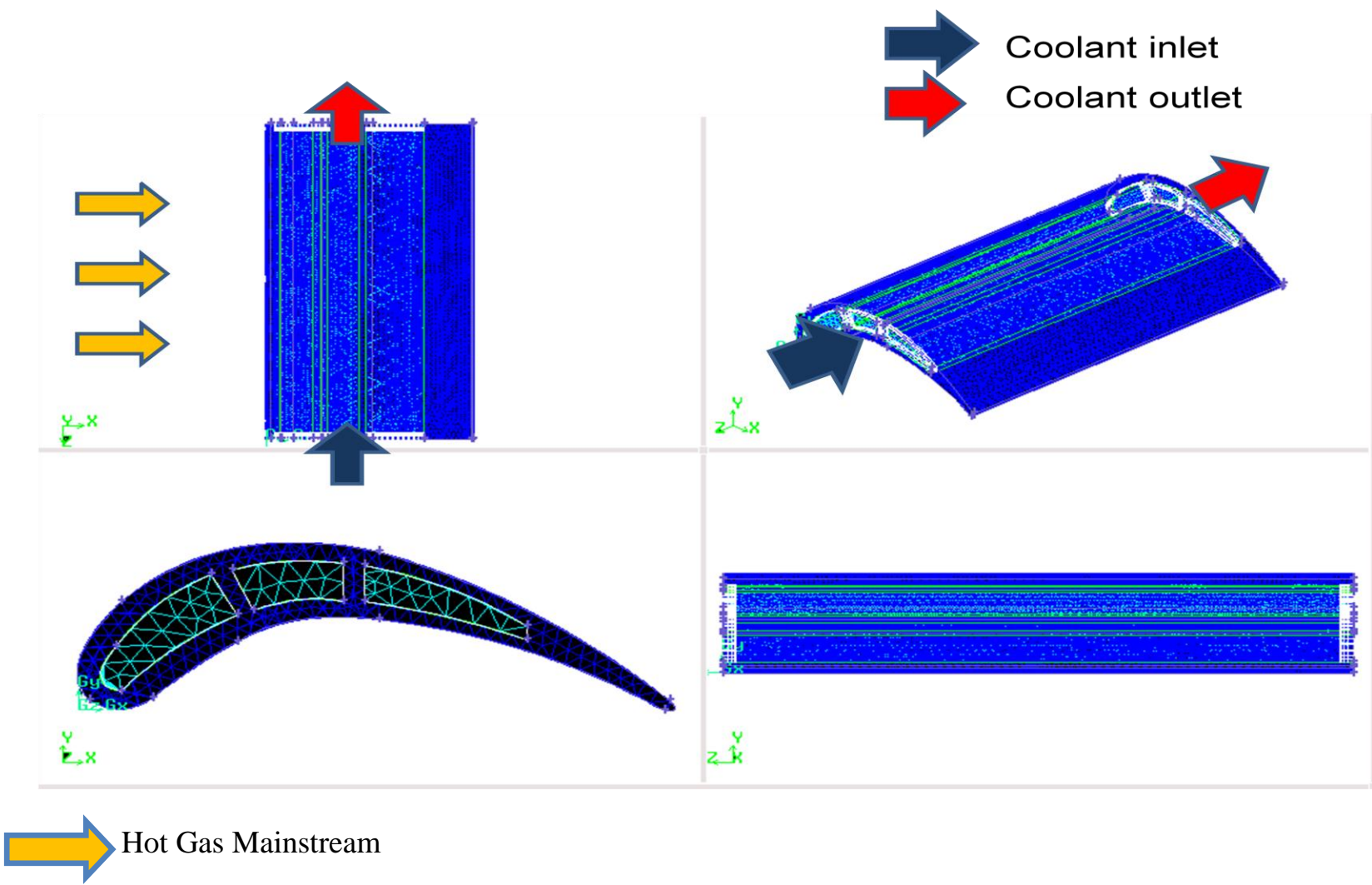
APPENDIX 1: Geometry of turbine blade modeled using GAMBIT