الجُمهُورِيَّة الجَزائِرِيَّة الدِّيمُقرَاطِيَّة الشَّعبِيَّةf

وزارة التعليم العالي والبحث العلمي

MOHAMED BOUDIAF UNIVERSITY - M'SILA

Faculty of Technology

Mechanical Engineering Department

N°: CE-01

Domain: Technical Science

Sector: Mechanical Engineering

Option: Energy

Memory presented for obtaining

Academic Master's degree

Presented by:

Berroussi Abd El Fettah & Louassaa Es-saadi

THE EFFECT OF THE ORIFICES SHAPE ON THE COOLING FILM ON GAS TURBINE BLADE (3D NUMERICAL STUDY)

Before the jury composed of:

NAME and First name	Grade	Quality
	Professor	President
BERKACHE AMAR	MCA	Frame
	MCA	Examiner

2023/2024

Thanks

First of all, we thank God for giving us the strength and patience to finish this Master thesis.

We would like to thank all the teachers in Mechanical Engineering, especially those who contributed to our master's degree.

At the end of our work, we would like to thank our supervisor Mr BERKACHE AMAR for his help and support during the preparation of this thesis.

May God reward him well and we apologise for any wrongdoing.

Finally, we would like to extend our sincere thanks and appreciation to Mr BELKHIRI for his help, advice, and guidance to reach the best possible results, May God reward him as well

Dedication

I dedicate this work to:

My mother and father above all, may ALLAH protect them and prolong their lives, and provide them with health and well-being, who have helped me throughout this long study.

To my brother and sisters, my family, my friends and DJOUD as well, and my wife whom I haven't had a chance to meet yet. And all those dear to me for their support.

BERROUSSI ABD EL FETTAH

I dedicate this humble work to:

To my parents, my wife, my children, and all those who helped us to accomplishing this work.

In particular, my colleague and brother Abdel Fattah and Mr Amar BERKACHE, who has been a lamp for us and a support for us, as well as being patient with us.

Without forgetting Mr. BELKHIRI for his help, advice, and guidance us

We ask God for further development and success for the University of Msila and success for every student who endeavored with his knowledge to benefit Islam and Muslims.

Louassaa Es-saadi

الملخص:

تُستخدم التوربينات الغازية على نطاق واسع في التطبيقات الصناعية، وإنتاج الطاقة الأرضية، ودفع الطيران. تُعد التطورات التكنولوجية في تبريد التوربينات أساسية في رفع إنتاج الطاقة والكفاءة الحرارية للتوربينات الغازية الحديثة. ومن أجل إزالة الحرارة المنقولة من السطح الخارجي، يتم ضخ سائل التبريد من خلال أنفاق أفعوانية متعددة مع تحسينات في الأضلاع لتبريد شفرات التوربينات الغازية داخلياً. ومن خلال دفع الهواء الأكثر برودة قليلاً من ممرات المبرد الداخلية إلى خارج سطح الشفرة، فإن التبريد الغشائي وهي تقنية تبريد خارجي لشفرات التوربينات يخلق حاجزاً بين تدفق مسار الغاز الساخن وسطح الشفرة. بحثت الدراسة الحالية عدديًا في تأثيرات تكوين الثقب وزاوية الميل على فعالية تبريد الفيلم وترسيب الجسيمات عند نسب نفخ مختلفة، والتنبؤ بمختلف المعلمات باستخدام **CFD**.

تُظهر النتائج أنه من بين أشكال الفتحات الثلاث (أسطواني، مستطيل، هلالي)، عند نسبة النفخ **M=2**، تُظهر الفتحة الهلالية أفضل أداء تبريد. بالنسبة لتأثير زاوية الميل على أداء التبريد، فإن فعالية تبريد الفيلم للتكوينات الثلاثة المختلفة للفتحات عند زوايا ميل مختلفة (30°،60°)، وقد اتفقت زاوية 30° مع أفضل تأثير تبريد للفيلم.

الكلمات المفتاحية: التبريد الغشائي، التوربينات الغازية، التوربينات الغازية، الشفرة، **CFD**، دراسة عددية، زاوية الميل، أشكال الفتحة، نسبة النفخ.

Abstract

Gas turbines are widely used in industrial applications, terrestrial power production, and aviation propulsion. Technological advances in turbine cooling are key to increasing modern gas turbines power output and thermal efficiency. To remove heat transferred from the outside surface, coolant is pumped through multiple serpentine tunnels with rib enhancements to internally cool the gas turbine blades. By forcing slightly cooler air from the internal coolant passages out of the blade surface, membrane cooling, an external cooling technique for turbine blades, creates a barrier between the flow of the hot gas stream and the blade surface. The present study numerically investigated the effects of hole configuration and inclination angle on film cooling effectiveness, particle deposition at various blowing ratios, and prediction of multiple parameters using CFD.

The results show that, among the three-hole shapes (Cylindrique, Rectangular, Crescent), At the blowing ratio of M=2, the crescent hole shows the best cooling performance.

For the effect of inclination angle on cooling performance, the film cooling effectiveness of the three different hole configurations at various inclination angles (30◦,60°), the angle of 30° accorded with the best film cooling effect.

Keyword : Film cooling, gas turbine, blade, CFD, numerical study, inclination angle, hole shapes, blowing ratio.

SOMMAIRE

List of Figures

GENERAL INTRODUCTION

By raising the turbine inlet temperature, the gas turbine industry constantly aims to increase the gas turbine engine's thermal efficiency. However, raising the working temperature causes a few serious issues. For instance, turbine blades cannot tolerate heat stressors and temperatures this high. The gas turbine's design operating temperature is far higher than the melting point of most materials [1].

Sophisticated cooling systems are used in modern gas turbines to help prevent thermal breakdown of the vanes and blades [1]. It has been proven to be one of the primary systems for keeping the gas turbine's temperature stable. Blades down to levels suitable for safe operation. Turbine cooling has permitted an additional 250° increase in melting point, and advancements in blade materials have allowed an increase of about 200°. The temperature of the blade material is lowered below its melting point using various external and internal cooling methods [2].

As a result of higher thermal loads, cooling the exposed combustor and downstream turbine surfaces has become crucial, with external film cooling being a vital subject [2]. Cooling the blades is essential for ensuring safe functioning. The engine's compressor extracts air to cool the blades. The extraction process reduces thermal efficiency and power production. Optimizing cooling technologies for a specific turbine blade geometry is crucial for engine operation. Gas turbine cooling technology is complex and differs across manufacturers [3].

Recent developments in turbine blade film cooling have focused on improving the efficiency and effectiveness of this cooling method to maintain the integrity of turbine blades operating at high temperatures. Here are some key advancements:

Passive Strategies for Film Cooling Enhancement

- **New Concepts in Film Cooling**:

NASA Glenn Research Center has developed two innovative approaches to enhance film cooling. The first involves using a shaped recess to create a counter-rotating vortex pair, splitting the incoming flow into high-energy and low-energy streams. The second innovation employs various techniques to mitigate aerodynamic losses that can occur during film cooling, such as using cooling holes that blow in the upstream direction and fences to bend the cooling flow back toward the downstream direction.

- **Shaped Film Cooling Holes:**

By lowering jet lift-off and increasing the lateral spreading of the coolant jets, novel shaped film cooling hole designs, such as fan-shaped and relaxed holes, have been designed to increase the effectiveness of film cooling.

- **Upstream Ramps and Shallow Trenches:**

Placing upstream ramps or shallow trenches ahead of the film cooling holes can create counter-rotating vortex pairs that split the coolant flow into high-energy and low-energy streams, helping maintain the cooling jets on the blade surface

- **Mesh-Fed Slots:**

Mesh-fed slots have shown promise in enhancing film cooling effectiveness compared to discrete holes by providing a more uniform coolant distributioncomplex flow phenomena involved in film cooling and to design more efficient cooling schemes.

Active Strategies for Film Cooling Enhancement

- **Pulsation Modulating Devices:**

One active tactic to enhance the effectiveness of film cooling has been the employment of pulsation-modifying devices to provide an irregular coolant injection.

- **Plasma Actuators:**

Plasma actuators have been explored as a means to actively control the coolant jets and enhance film cooling effectiveness.

Other Advancements

- **Optimization for High-Temperature Operation:**

To ensure effective cooling while allowing high-temperature operation without sacrificing blade integrity, researchers have concentrated on enhancing film cooling for high-temperature operation.

- **Numerical Modeling and Simulation:**

Advancements in numerical modeling and simulation techniques, such as large eddy simulation (LES) and support vector machine (SVM) methods, have improved the prediction of film cooling effectiveness and heat transfer coefficients.

These recent developments in film cooling demonstrate the ongoing efforts to enhance the efficiency and effectiveness of turbine blade cooling, enabling the safe and reliable operation of gas turbines at higher temperatures for improved performance and efficiency.

The focus of this investigation is to study the geometries of blades of different shapes such as cylindrical, elliptical, conical, etc. on membrane cooling heat transfer, cooling efficiency, and prediction of various parameters using CFD, this investigation is divided into four chapters:

After giving a of the cooling systems, Chapter 1 is General information about gas turbines. Chapter 2 Discusses the need for turbine cooling, gas turbine heat-transfer problems, and the effect of the shape of the cooling orifices on the cooling film of a gas turbine blade.

Chapter 3 includes a numerical study using CFD, and in the last chapter 4, we discuss our findings.

Chapter I

General overview of gas turbines

I.1 Introduction:

The gas turbine is a power plant that generates a significant quantity of energy despite its size and weight. Over the past 40 years, gas turbines have gained popularity in the power, petrochemical, and utility industries worldwide. The compact size, low weight, and diverse fuel applications make it an ideal power plant for offshore sites. Gas turbines can operate on natural gas, diesel fuel, naphtha, methane, low-Btu gases, vaporized fuel oils, and biomass gases [4].

Gas turbine technology has grown significantly over the previous 20 years. Growth is driven by advancements in materials technology, coatings, and cooling systems. By increasing the compressor pressure ratio, the gas turbine's thermal efficiency has enhanced from 15% to more than 45% [4].

I.2. Origin and history:

The first steam engine was patented in 1629 by an Italian mechanic, Giovanni Branca, a machine in which a jet was directed towards a horizontal wheel connected to a gear system to drive a press. After Branca, several mathematicians contributed to improving this technology.

In the years 1820-1833 Claude Bourdin, a French professor, built several hydraulic installations of hydraulic machines which he named Turbine, a word derived from the Latin turbins meaning which turns, but he did not succeed in passing to the stage of industrially stable machines. and it wasn't until 1830 that Benoît Fourneyron, a brilliant pupil of Bourdin's improved on his teacher's design and installed an industrial turbine with a power of 50 HP. In 1844, Fourneyron's turbines were installed in Europe and the United States, where improvements were made. improvements were made.

Gas turbines began to appear between the end of the 19th century and the beginning of the 20th century. the beginning of the 20th century, the first gas turbine capable of providing work was built in 1903 by the mechanical engineer Elling in Norway. In 1910 the mechanical engineer Henri Coanda tested an airplane in which the engine was a basic gas turbine consisting of a compressor, a combustion chamber and a nozzle, but it wasn't until 1930 that the idea of the gas turbine was presented in the United States, France, Great Britain Italy , and Germany.

In addition to these inventions, there were those developed by Secundo Compini's team who built a turboprop engine in 1940, and the contribution made by Frank Whittle of England in 1769 who is credited with the practical creation of the modern gas turbine [5].

I.3. Definition of gas turbine:

An industrial gas turbine is a thermal machine that converts a fuel's chemical energy into mechanical energy by using air as the conversion medium. This mechanical energy can be used to power a variety of rotating machinery. The turbine mentioned in this tutorial is a two-shaft turbine that is used to power a compressor unit or to drive a mechanical load.

A mechanical drive is used to power a load, such as a gas compressor or a powerful water pump [6].

In its most basic form, a gas turbine consists of an axial compressor that draws in air at atmospheric pressure and a combustion chamber, where the compressed air is heated to constant pressure by the combustion of a specific amount of fuel (natural gas, diesel or kerosene); and finally, a turbine that expands the gases to atmospheric pressure. Finally, the gases are expanded to atmospheric pressure by a turbine using natural gas, diesel, or paraffin. atmospheric pressure [7].

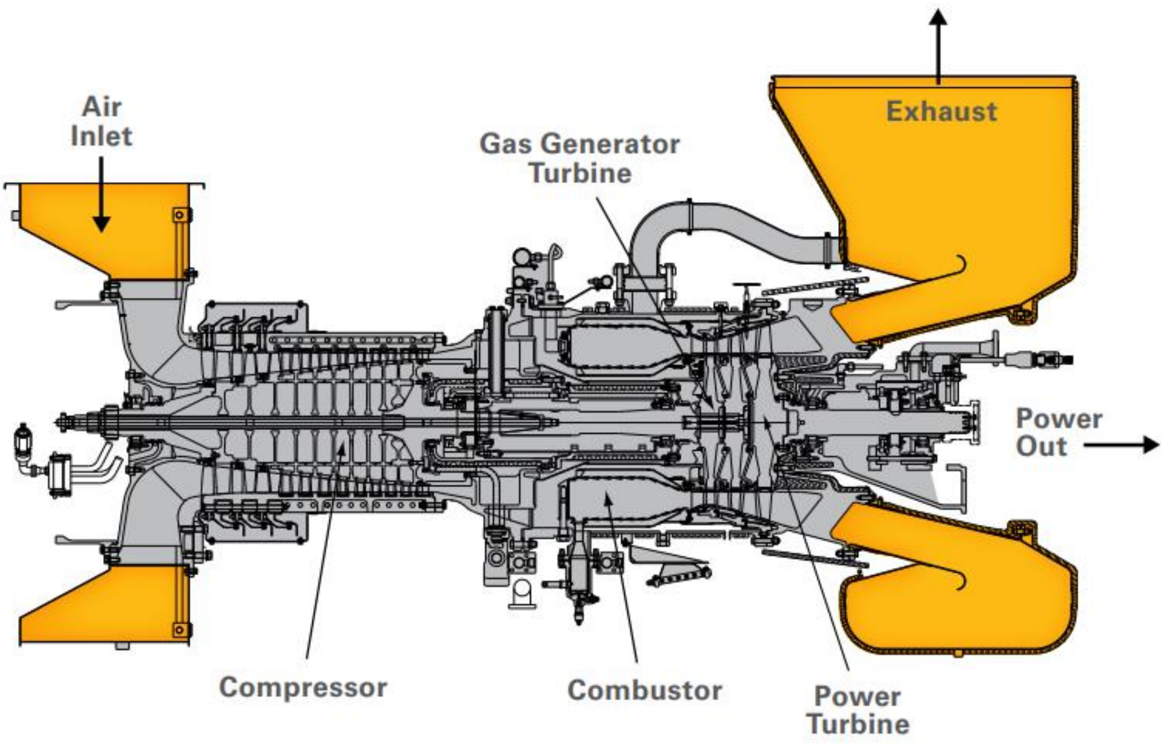


Figure I-1: Simple Brayton Cycle Axial Flow Gas Turbine (Introduction to Gas Turbine Theory (Klaus Brun).

I.4. Components of a gas turbine:

I.4.1. Compressors:

Gas turbine engines use either centrifugal or axial flow compressors to compress air before it expands through the turbine. Both types are powered by the engine turbine and often connected directly to the turbine shaft.

Centrifugal flow compressors (Figure 1-2) are single or two-stage units that use an impeller to accelerate air and a diffuser to increase pressure. The axial flow compressor (Figure 1-3) is a multi-stage equipment that uses revolving (rotor) blades and stationary (stator) vanes to accelerate and diffuse air until the desired pressure is achieved. For small engines, an axial compressor may be employed to increase the inlet pressure to the centrifugal pump [8].

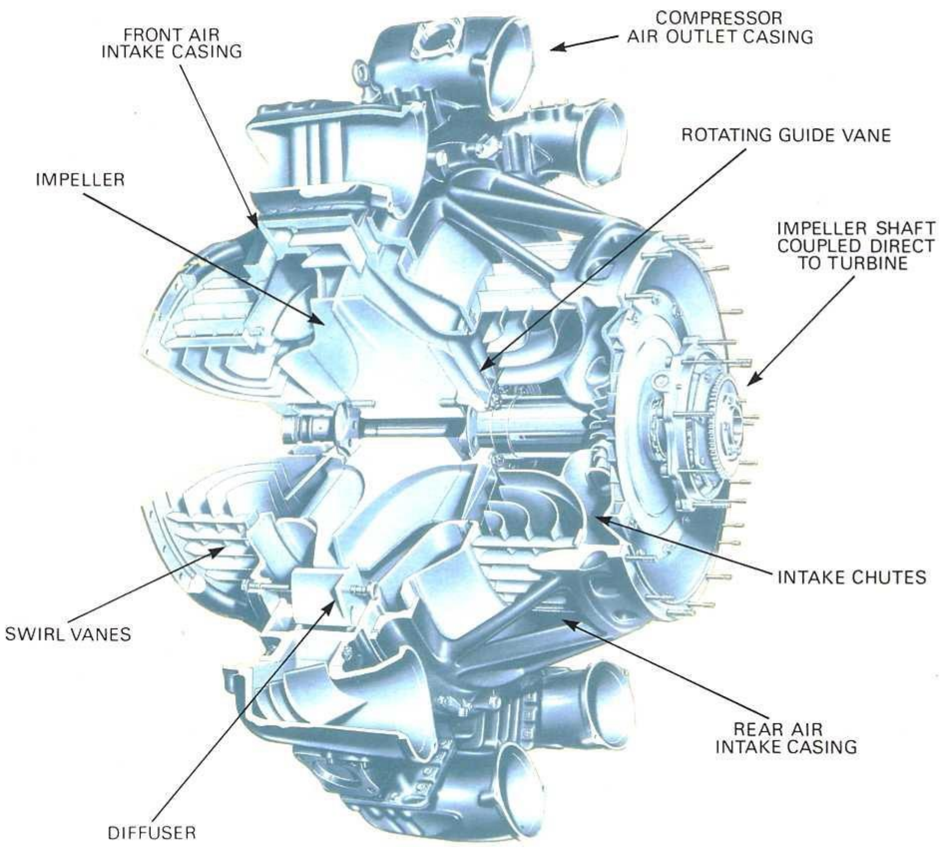


Figure I-2: A typical centrifugal flow compressor (Rolls-Royce Ltd. (2005). The jet engine. Rolls-Royce).

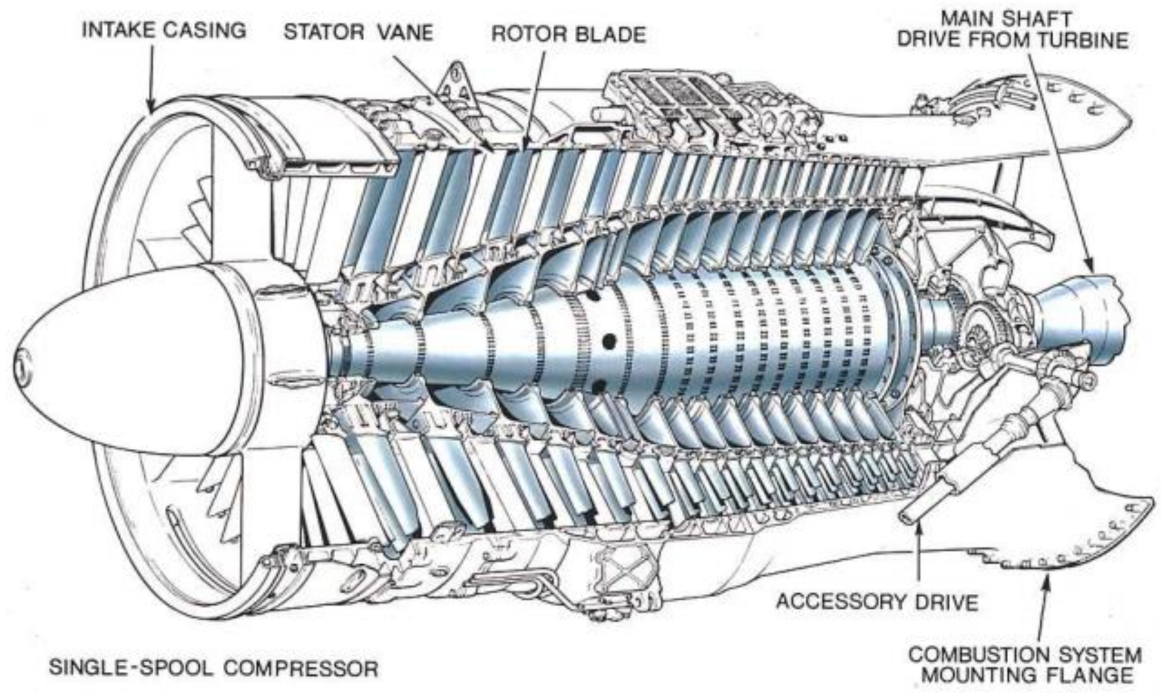


Figure I-3: A typical axial flow compressor (Rolls-Royce Ltd. (2005). The jet engine. Rolls-Royce).

I.4.2. Combustion chambers:

The combustor, often known as the burner, is the next major aerodynamic component that airflow encounters after leaving the compressor. The combustor adds heat energy to the flow, increasing the temperature of the air traveling past it [9].

Gas turbine combustors have the same function: they raise the temperature of high-pressure gas. The gas turbine combustor utilizes only 10% of its air for burning. The remainder of the air is utilized for chilling and mixing. New combustors circulate steam for cooling purposes [4].

The air from the compressor must be dispersed before entering the combustor. The air leaving the compressor has a velocity of 400-600 ft/sec (122-183 m/sec), but the combustor velocity should not exceed 50 ft/sec (15.2 m/sec). Even at modest velocity, caution must be exercised to prevent the flame from spreading downstream [4].

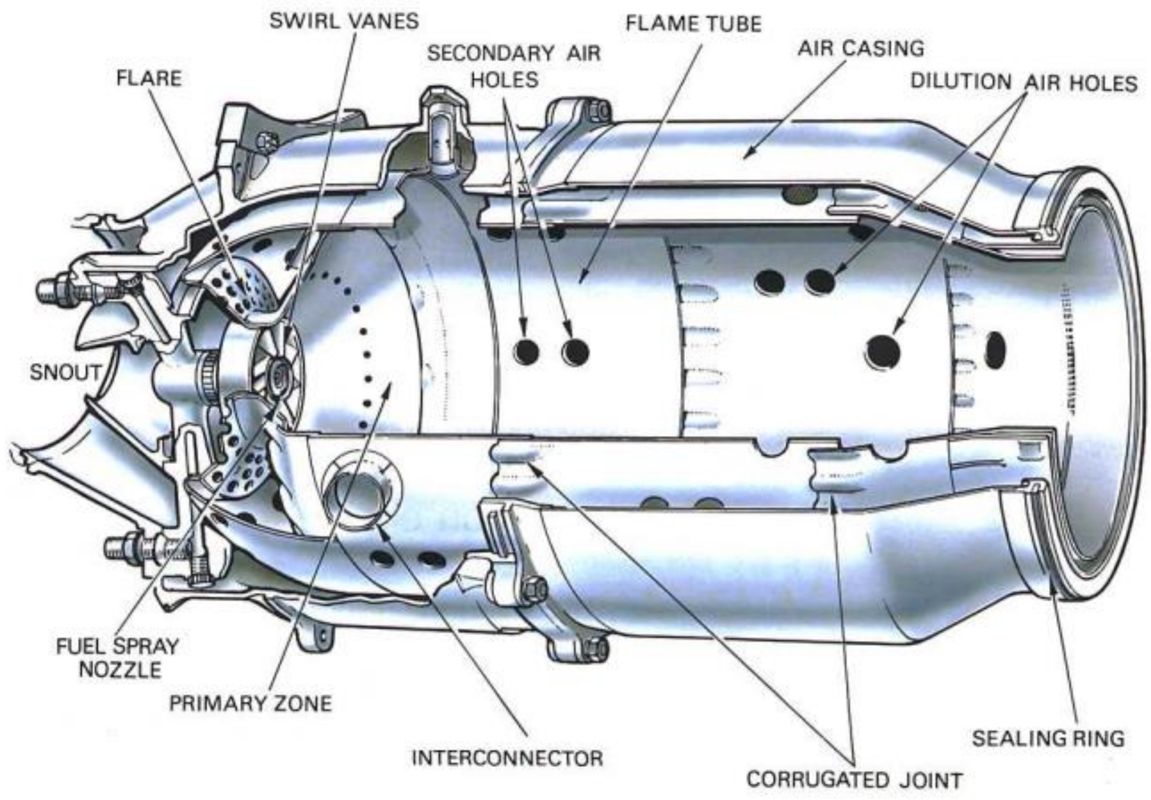


Figure I-4: An early combustion chamber (Rolls-Royce Ltd. (2005). The jet engine. Rolls-Royce).

I.4.3. Turbines:

The gas combination from the combustor contains potential energy in two forms: heat and pressure. The gas turbine converts fluid energy into mechanical shaft energy.

A gas turbine consists of both a gas producer and a power turbine, as previously stated. The gas producer absorbs energy from the flow to drive the compressor, while the power turbine converts the rest into shaft rotational output energy [9].

Gas turbines employ two different types of turbines. There are two types: axial flow and radial inflow. An axial-flow turbine is employed in over 95% of all applications [4].

Axial-flow and radial-inflow turbines can be classified as impulse or reaction units. In contrast to reaction turbines, which take a partial drop through both the impeller blades and the nozzles, impulse turbines absorb the complete enthalpy decrease [4].

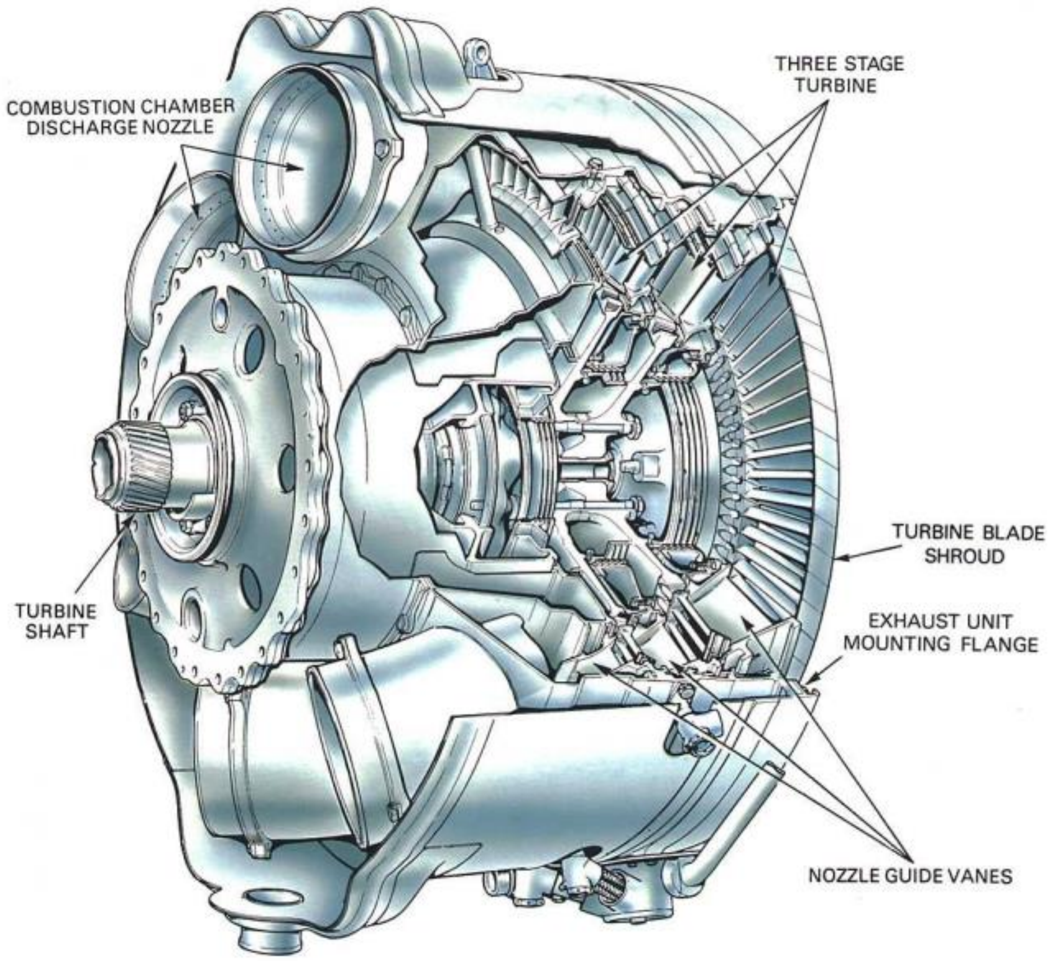


Figure I-5: A triple-stage turbine with single shaft system (Rolls-Royce Ltd. (2005). The jet engine. Rolls-Royce).

I.5. Principles of gas turbines :

Jet engines move the airplane forward with a great force that is produced by a tremendous thrust and causes the plane to fly very fast.

All jet engines, which are also called gas turbines, work on the same principle. The engine sucks air in at the front with a fan. A compressor raises the pressure of the air. The

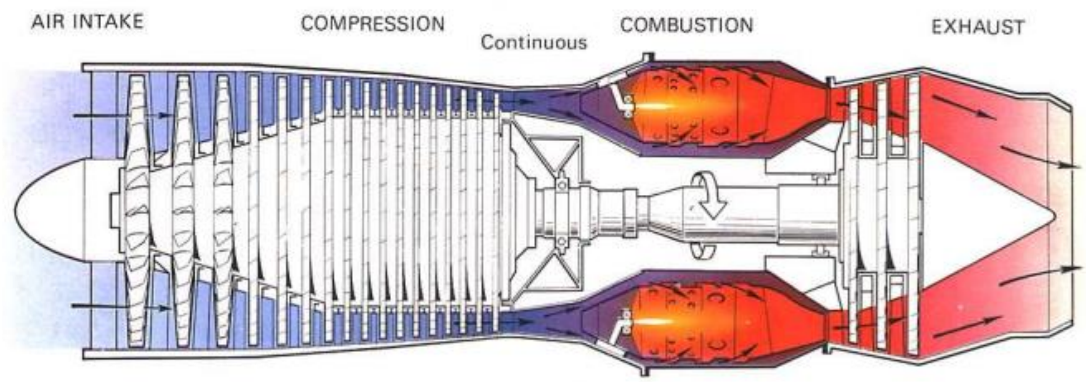


Figure I-6: the working cycle of a turbo-jet engine (themechanicalengineering.com).

compressor is made with many blades attached to a shaft. The blades spin at high speed and compress or squeeze the air. The compressed air is then sprayed with fuel and an electric spark lights the mixture. The burning gases expand and blast out through the nozzle, at the back of the engine. As the jets of gas shoot backward, the engine and the aircraft are thrust forward. As the hot air goes to the nozzle, it passes through another group of blades called the turbine. The turbine is attached to the same shaft as the compressor. Spinning the turbine causes the compressor to spin [11].

I.6. Classification of gas turbine engines:

Gas turbine engines power aircraft and helicopters, as well as various industrial applications such as automobiles, tanks, marine vessels, and electricity generation.

I.6.1. Turbojet engines:

Turbojet engines (Figure 1- 6) were the first jet engines used to power aircraft in the 1940s. The turbojet engine revolutionized air transportation. It significantly cut the cost of air travel while improving airplane safety. The turbojet also enabled higher speeds, including supersonic speeds. The higher thrust per unit weight ratio compared to piston-driven engines resulted in increased range, payload, and lower maintenance costs. Turbojet engines are used in both military fighters and speedy business jets [10].

I.6.2. Turboprop:

A turboprop engine is a jet engine attached to a propeller. The turbine at the back is turned by the hot gases, and this turns a shaft that drives the propeller. Some small airliners and transport aircraft are powered by turboprops.

Like the turbojet, the turboprop engine consists of a compressor, combustion chamber, and turbine, the air and gas pressure are used to run the turbine, which then creates power to drive the compressor. Compared with a turbojet engine, the turboprop has better propulsion efficiency at flight speeds below about 500 miles per hour. Modern turboprop engines are equipped with propellers that have a smaller diameter but a larger number of blades for efficient operation at much higher flight speeds. To accommodate the higher flight speeds, the blades are scimitar-shaped with swept-back leading edges at the blade tips [11].

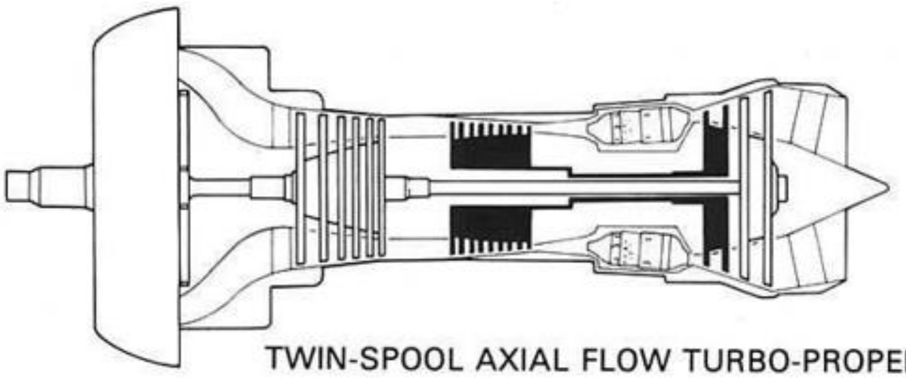


Figure I-7: twin spool axial flow turbo-propeller (Rolls-Royce Ltd. (2005). The jet engine. Rolls-Royce).

I.6.3. Turboshaft:

Turboshaft engines are defined here as those used to power helicopters. A turboshaft is similar to a turboprop, except the latter generates residual propulsion thrust to augment the shaft-driven propeller [10].

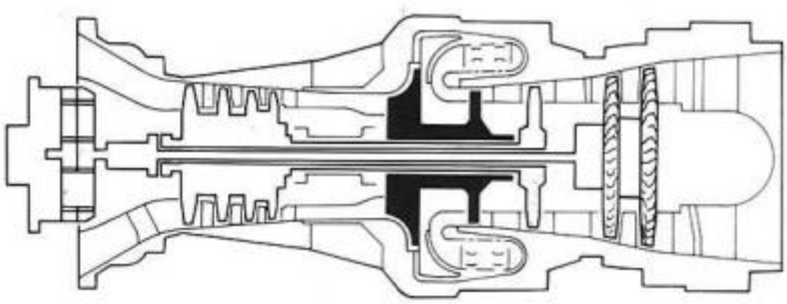


Figure I-8: twin spool turboshaft (Rolls-Royce Ltd. (2005). The jet engine. Rolls-Royce).

I.6.4.Turbofan:

A turbofan engine has a large fan at the front, which sucks in air. Most of the air flows around the outside of the engine, making it quieter and giving more thrust at low speeds. Most of today's airliners are powered by turbofans. In a turbojet, all the air entering the intake passes through the gas generator, composed of the compressor, combustion chamber, and turbine. In a turbofan engine, only a portion of the incoming air goes into the combustion chamber. The remainder passes through a fan, or low-pressure compressor, and is ejected directly as a "cold" jet or mixed with the gas-generator exhaust to produce a "hot" jet. This sort of bypass system is to increase thrust without increasing fuel consumption. It achieves this by increasing the air-mass flow and reducing the velocity within the same total energy supply [11].

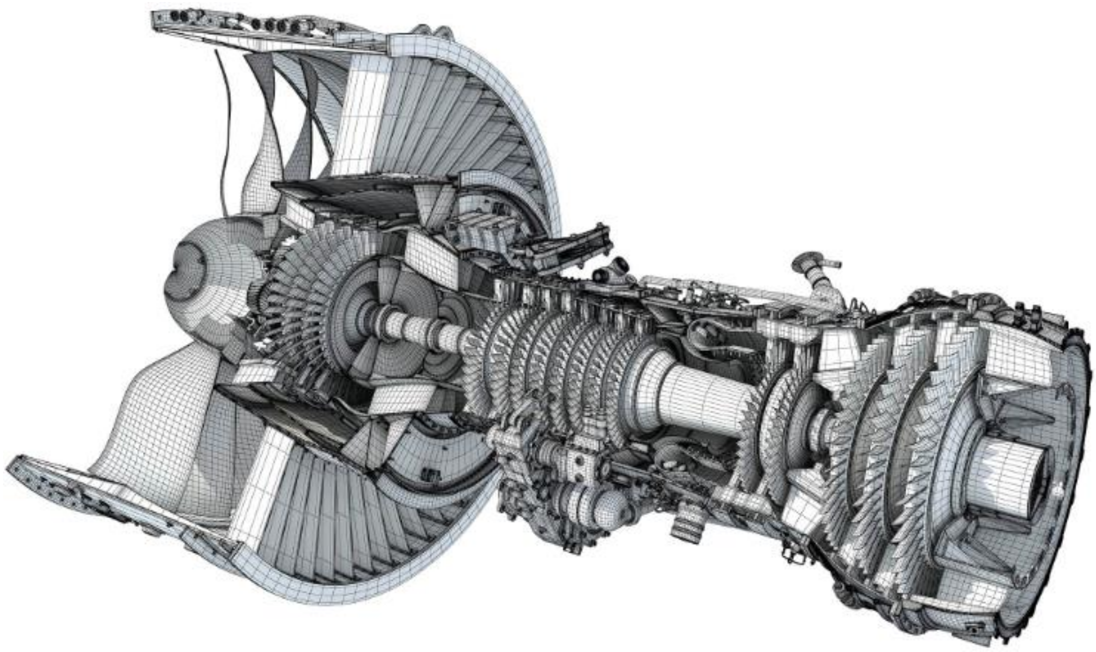


Figure I-9: Turbofan engine (turbosquid.com)

I.6.5. Ramjet:

The ramjet is the simplest jet engine and has no moving parts. The speed of the jet "rams" or forces air into the engine. It is essentially a turbojet in which rotating machinery has been omitted. Its application is restricted because its compression ratio depends wholly on forward speed. The ramjet develops no static thrust and tiny thrust below the speed of sound. consequently, a ramjet vehicle requires some form of assisted takeoff, such as another aircraft. It has been used primarily in guided missile systems. Space vehicles use this type of jet [11].

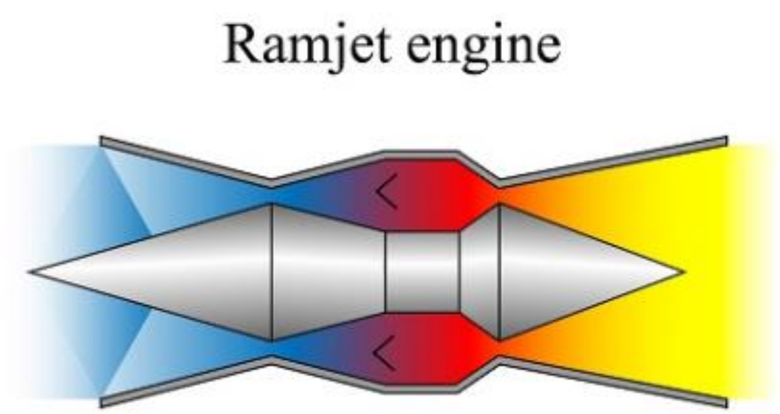


Figure I-10: Ramjet engine (aviationvoice.com).

I.6.6. Advanced ducted fan:

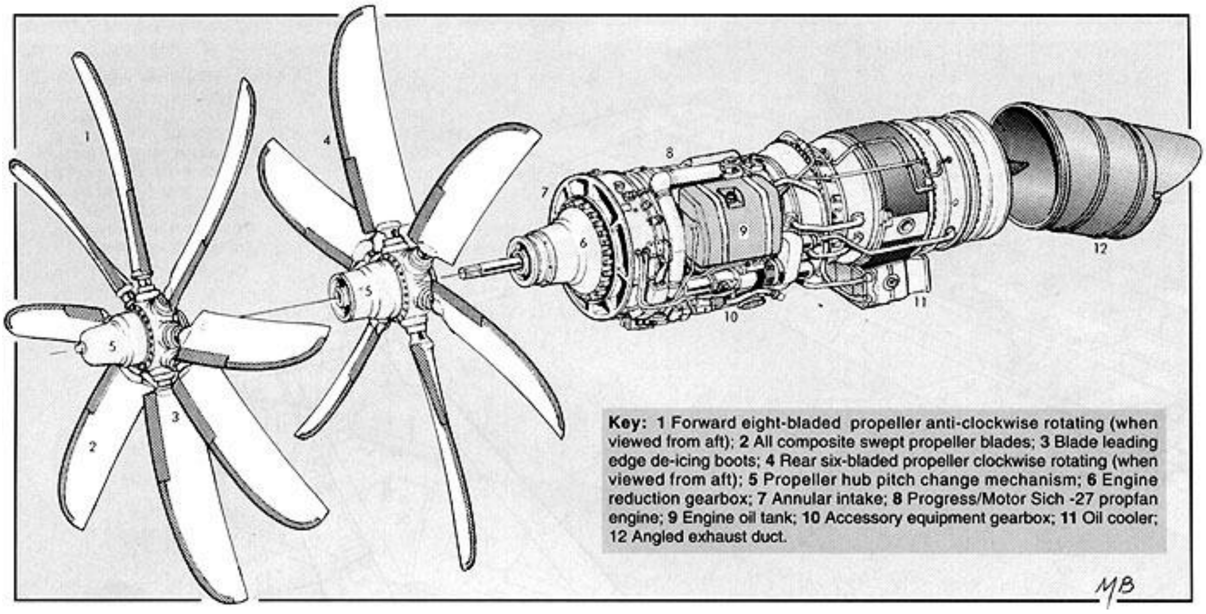


Figure I-11: Counterrotating propfan

These designs are essentially turbofans with large, swept fan blades that have pitch control and reduction gearing similar to propfans, but the fans are enclosed in ducts like turbofans.

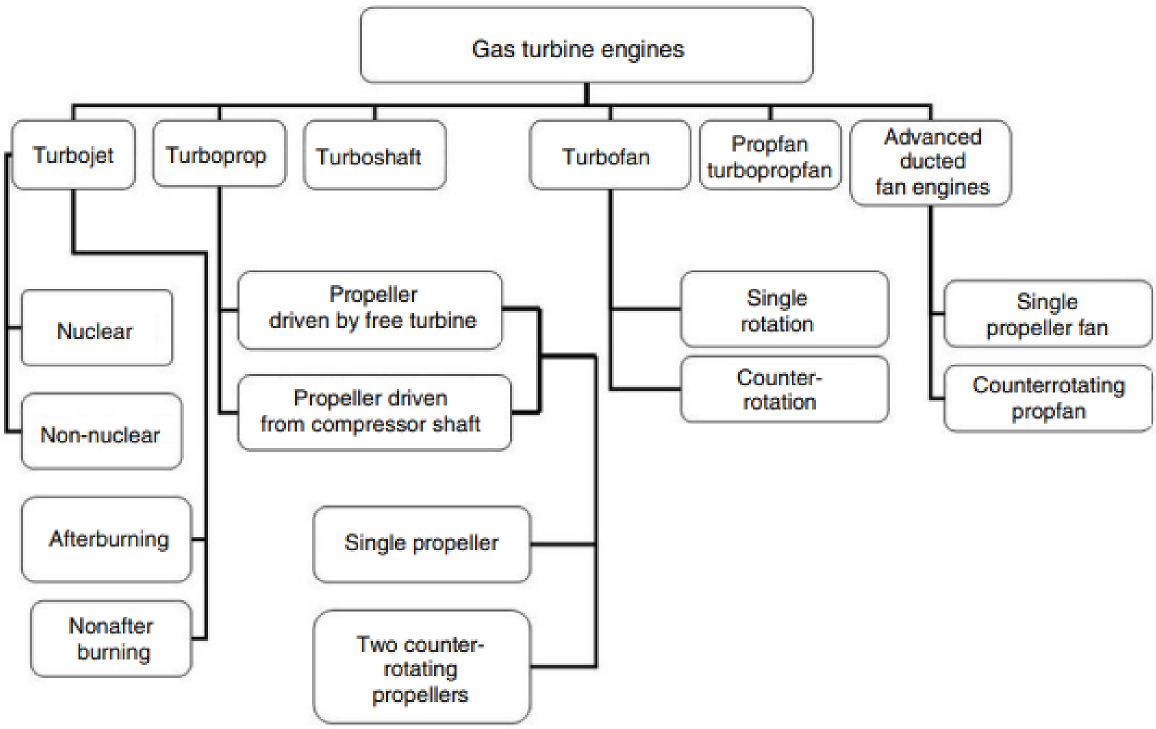


Figure I -12: Classification of gas turbine engines.

I.7. Applications of Gas Turbines :

I.7.1. Supercharging:

Gas turbines are utilized for supercharging. A tiny gas turbine powered by hot exhaust gases operates the compressor in aircraft petrol engines and heavy-duty diesel engines [12].

I.7.2. Turbojet and Turbo Propeller Engines:

Every turbojet and turbo propeller engine contains a gas turbine. In turbojet engines, the turbine only drives the air compressor, however in turboprop engines, it may also drive the propeller. Gas expansion can occur in either a single turbine or a combination of low- and high-pressure turbines. These turbines are designed to run at temperatures ranging from 800°C to 1000°C [12].

I.7.3. Marine Field:

The gas turbine can also be employed in marine applications. These are used for ship propulsion or onboard power generation [12].

I.7.4. Railway:

The gas turbines can also be used for rail propulsion [12].

I.7.5. Road Transport:

Gas turbines are popular for electric power generation due to their ability to start and ramp up to full load fast, as well as their low installation and maintenance costs. When compared to a steam power plant, a gas turbine power station consumes far less water for a given efficiency [12].

I.7.6. Industry:

Gas turbines are also used for industrial reasons, such as blasting air for blast furnaces in the steel, oil, and chemical sectors [12].

I.8. Advantages of Gas Turbines:

I.8.1. Installed cost:

Gas turbine packages are lightweight, compact, and completely self-contained. They fit well with manufacturing packaging, which includes controls, subsystems, and driving equipment. Cranes and huge buildings are unnecessary. Water cooling is eliminated. Due to reduced vibration and lack of reciprocating forces. Foundations can be made of light slabs [12].

I.8.2. Low maintenance:

The rotor is the only moving portion of the engine, spinning in its bearings. Vibration is nearly gone. Oil use and pollution are minimal. Oil does not enter the combustion zone. There are no water-cooling systems to maintain. There are lengthy durations between overhauls.

Most turbine manufacturers also provide factory exchange programs, which allow for frequent factory overhauls [12].

I.8.3. Fuel Consumption:

Modern turbines consume approximately 11.000 to 13,000 Btu/BHP hour. However, many oilfield applications might benefit from exhaust heat. The vast majority do not, because fuel is very inexpensive and the actual savings are minimal [12].

I.8.4. Automation:

Turbine start cycles are automatic, and parts reach operational temperature in seconds. Turbines are ideal for remote and telemetry control [12].

I.8.5. Flexibility:

Turbines can be used in both constant and variable speed applications, as previously discussed. Turbines can be configured to provide the maximum possible horsepower. even as the working circumstances and environmental temps shift. They can maintain continuous suction and discharge pressures/flows. The turbine's air compressor can provide pneumatic control air. Turbines use natural gas, diesel, propane, butane, jet fuel, and other petroleum products [12].

I.8.6. Quick Start:

Turbines contain minimal moving parts. The heated components have little mass. As a result, there is less starting friction. Because parts reach working temperature in seconds, turbines can often handle full load in a minute and are unaffected by cold weather. Turbines can be started and stopped selectively to save fuel and meet changing horsepower requirements [12].

Chapter II

Turbine Blade Cooling

II.1. Introduction:

Jet in crossflow film cooling is a typical method of film cooling. This is commonly accomplished by blowing cool air through a hole in the blade surface. The hole is angled in the flow direction, and as the injected air mixes with the primary gases, a jet forms. This jet will then move downstream to the blade surface. This type of film cooling is commonly utilized; however, it is not as successful as it could be because the injected air is quickly convected away from the surface by the mainstream gas. Shaped perforations in the blade surface are an enhancement to this method, as they channel the jet in a more beneficial direction. This is the method that this investigation will be concerned with.

Today, many gas turbine blades have cooling systems. This is required because the heat load that a blade experiences while in service might be rather considerable. Gas turbine blades are frequently subjected to thermal stresses exceeding 150°C beyond their material melting temperature. The blade must be cooled since the melting temperatures of the Nickel superalloys utilized exceed 1000°C. The cooling process involves diverting some of the air from the compressor and blowing it through the blades.

This air flows through the blade's numerous cooling tubes, heating up and chilling it. One of the most prevalent methods for cooling blades is film cooling. This involves blowing a small layer of colder air over the blade surface, resulting in a protective layer. The cooling air comes from the engine's main gas stream and is frequently at temperatures that exceed the blade's material capacity. This is why the air must form a protective layer and not mix with the surrounding gases [13].

Figure 2-1 shows the common cooling technology with three major internal cooling zones in a turbine blade with strategic film cooling in the leading edge, pressure and suction surfaces, and blade tip region. The leading edge is cooled by jet impingement with film cooling, the middle portion is cooled by serpentine rib-roughened passages with local film cooling, and the trailing edge is cooled by pin fins with trailing edge injection [14].

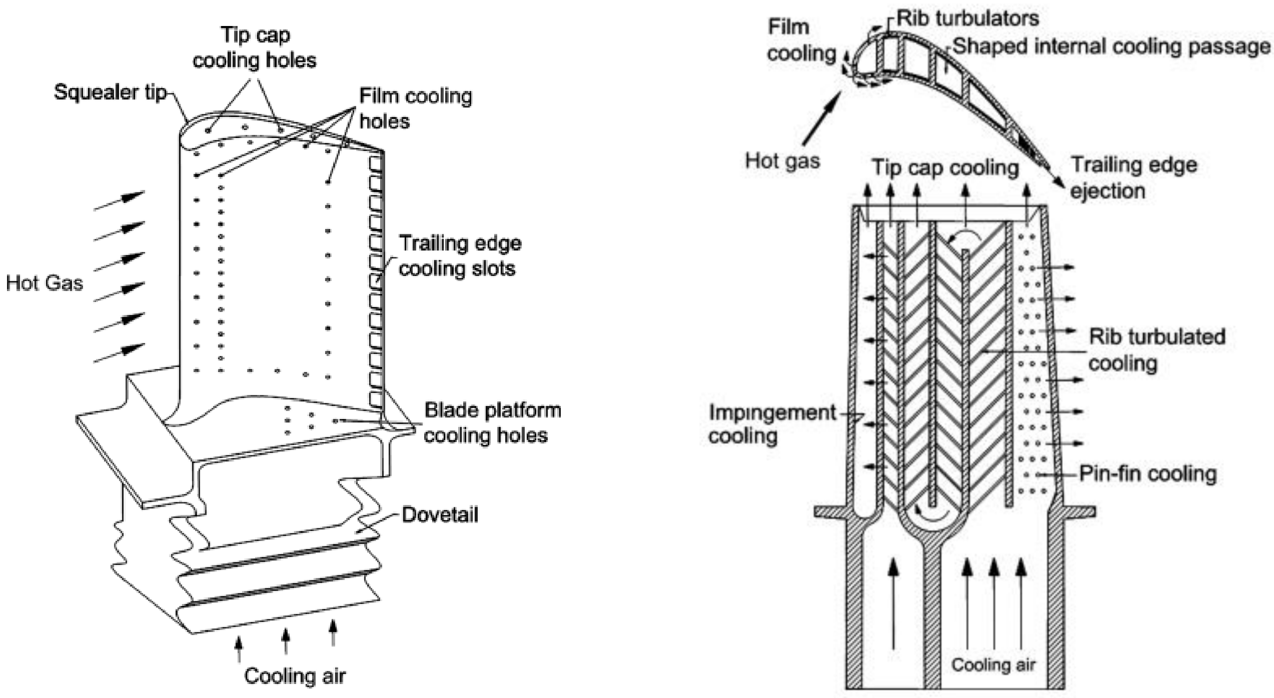


Figure II- 1: The schematic of a modern gas turbine blade with common cooling techniques. (Jäschke, J., Ehrhardt, M., Günther, M., & Jacob, B. (2022). A port-Hamiltonian formulation of coupled heat transfer.)

II.2. Background:

In film cooling, relatively cool air is injected from the inside of the blade to the outside surface, which forms a protective layer between the blade surface and hot mainstreams. Performance of film cooling primarily depends on the coolant-to-mainstream pressure ratio (blowing ratio), temperature ratio (density ratio), film hole geometry (hole size, spacing, shape, angle from the surface, and number of rows), and location (leading edge, trailing edge, pressure and suction sides, end-wall, and blade tip) under representative engine flow conditions (Reynolds number, Mach number, combustion generated high free-stream turbulence, and unsteady wake flow). Recently, there have been many studies in turbine blade film cooling. For example, Abhari and Epstein (1994) studied rotor blade film cooling, Nirmalan and Hylton (1990) reported on linear cascade nozzle guide vane film cooling, Camci and Arts (1985) studied the linear cascade blade film cooling, Harasgama and Burton (1992) presented annual cascade nozzle guide vane end-wall film cooling, Mehendale and Han (1992) studied the modeled blade leading edge film cooling, and Kim et al. (1995)

reported on the modeled blade tip film cooling. A detailed literature survey in film cooling studies can be found in Han and Ekkad (2001), Han et al. (2000), and Dunn (2001). T [15].

II.3. Importance of Cooling Film in Gas Turbine Blade:

Gas turbine engines are employed as aviation propulsion systems and mechanical drive units. Currently, there are two main types of gas turbines. These are the gas turbines used in aviation engines, which are used to generate electricity and operate oil and gas systems mechanically. In the case of an airplane turbine engine, gas energy is transformed into kinetic energy and shaft power.

The mechanical drive unit converts the gas energy into rotational energy for the mechanical drive. Most of the energy produced by the engine is waste heat from the combustor, and turbine components are utilized to extract as much power from the petrol as possible and turn it into mechanical energy. A turbine engine performs best when the gas expands isentropically. This means that there is no loss of efficiency from heat transfer between the gas and its surroundings.

This means there is no efficiency loss from heat transfer between the gas and its surroundings. We aim to maximize heat transfer between the gas and turbine components. This is extremely impractical, yet to maximize efficiency, turbine components are designed with the maximum heat transfer feasible while maintaining the smallest temperature differential between the gas and the component. As a result, there is an extensive study into the cooling of gas turbine blades and vanes, with some estimates indicating that cooling accounts for up to 20% of global gas turbine research investment (Prudhomme and Soechting, 1994).

II.4. Turbine Blade:

The turbine blades are aerofoil-shaped, creating tunnels between blades to accelerate flow up to the 'throat', where the area is smallest. The velocity reaches the required level at exit, causing the desired reaction.

The area of each blade cross-section is determined by the allowed tension in the material and the size of the cooling holes. To achieve high efficiency, thin trailing edges are necessary. However, a trade-off must be made to prevent blade splitting due to temperature variations during engine operation.

The attachment technique of blades to the turbine disc is crucial as it affects the limiting rim speed due to stress in the disc near the fixing or blade roots. The early Whittle engine employed the de Laval bulb root fastening to attach the blades. However, the 'fir-tree' fixing is now often used in gas turbine engines. This form of fastening requires precise machining to ensure equal loading for all.

The serrations, When the turbine is stationary, the blade's serrations are loose, but when it rotates, centrifugal stress stiffens the root. Figures 2-3 illustrate various techniques of blade attachment [10].

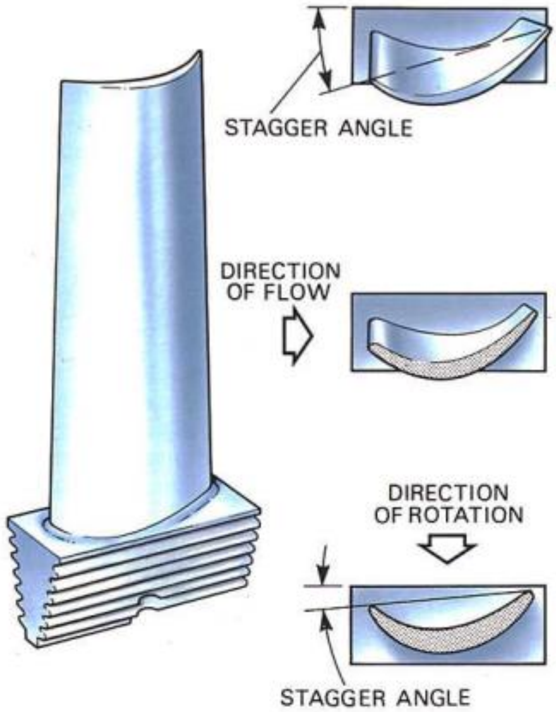


Figure II- 2: A typical turbine blade showing a twisted contour (Rolls-Royce Ltd. (2005). The jet engine. Rolls-Royce)

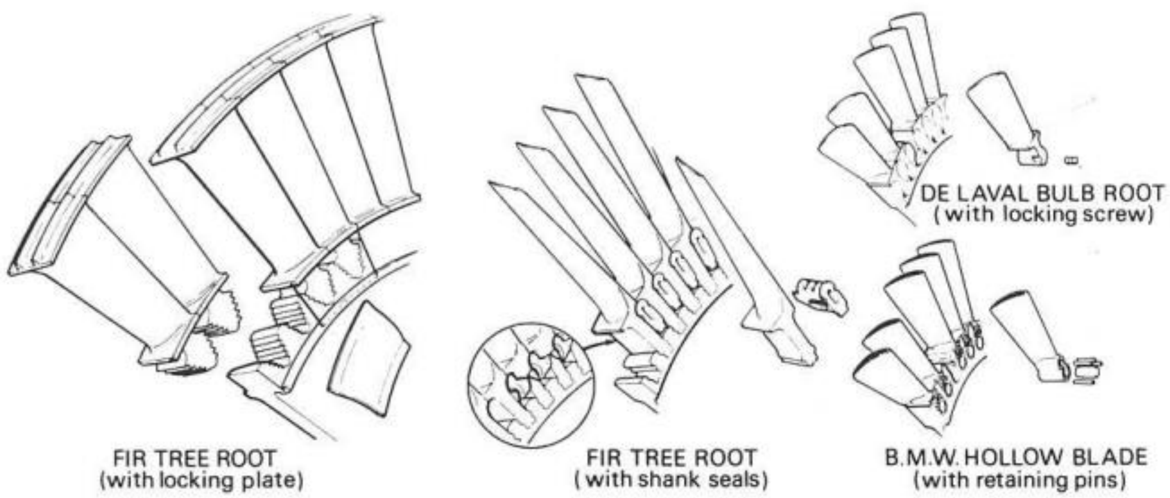


Figure II- 3: Various methods of attaching blades to turbine discs (Rolls-Royce Ltd. (2005). The jet engine. Rolls-Royce)

II.4. Choice of refrigerant fluid:

There are two ways of cooling the blades. The choice of coolant divides the cooling system Into two categories:

- Liquid-cooled.
- Gas cooling.

For cooling systems using a liquid refrigerant, the desirable liquid is: water

Gas cooling is only provided by air drawn from the engine cycle at the compressor.

compressor.

II.4.1. Air Cooling:

The quantity of thrust produced by an engine is limited by the heat that the turbine can withstand. One way to increase this capability is to cool the hot surfaces. The idea is to give the most amount of cooling with the least amount of coolant. Lewis examined three forms of convective cooling of turbine blades: heat removal at the blade root, airflow through hollow blades, and liquid coolant flow through the hollow blade. The lab was also looking into different heat-resistant materials, but cooling was a more cost-effective solution. The cheapest kind of cooling is air cooling, which diverts excess air flow from the compressor into hollow turbine blades to remove heat.

As engines increase speed, the inlet air temperature rises, which diminishes the effectiveness of air cooling. Nonetheless, internal convective air cooling was the primary engine cooling method in the 1960s and 1970s. Since the 1970s, it has been used in conjunction with external film cooling and impingement. Film cooling ejects cooling air from holes in the

blade, resulting in a thin layer of cool air on the blade surface. Impingement systems shoot cooling air against the inside of the blade walls to facilitate heat transfer. Cooling has allowed engines to operate at temperatures above the temperature limits of their materials, thus producing more thrust.

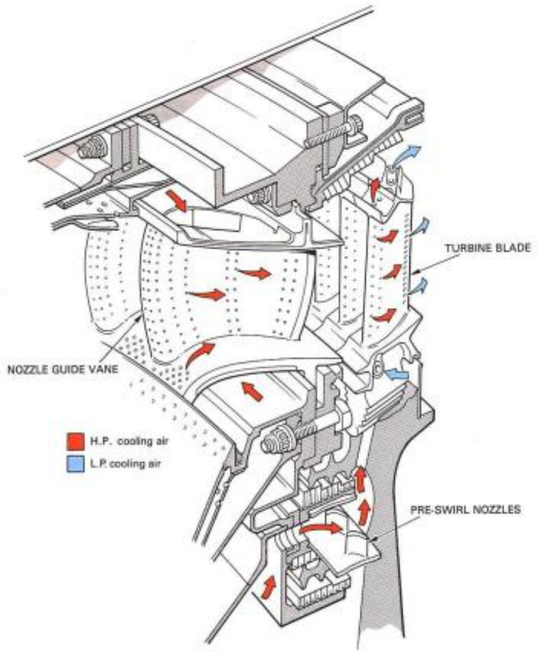


Figure II- 4: Nozzle guide vane and turbine blade cooling arrangement (Rolls-Royce Ltd. (2005). The jet engine. Rolls-Royce)

II.4.2. Liquid Cooling:

Liquid-cooling systems are the most effective way to control turbine blade temperatures, but they require additional components and add weight to the engine. Many liquid-cooling methods exist, such as film cooling and blades with cooling tubes. However, the original form of liquid cooling included spraying water into the airflow upstream of the turbine. The water strikes the heated blades and evaporates, removing the heat as it escapes the tailpipe. Early spray-cooling systems were tested on jet engines in the United Kingdom. Although the cooling was irregular, it was enough to justify additional investigation.

II.5. Cooling:

Gas turbine blades are cooled using many ways, including convection, film, transpiration, cooling effusion, and pin fin cooling, classified as internal or exterior. All methods use cooler air (typically bled from the compressor) to remove heat from turbine blades, however, their specifics vary [16].

II.5.1. Internal cooling:

II.5.1.1. Impingement cooling:

Jet impingement cooling uses cold airflow and convective heat transfer to effectively cool high-flow zones. Jet impingement cooling is the most effective technology for improving local heat transfer coefficients. Impingement jet heat transfer is ideal for high heat loads, such as on the leading edge of turbine blades. The turbine blades' interior compartments create cooling fields through their inserts. Impingement jets are created through creative structural design. Efficiency depends on jet flow angle, impingement hole form and arrangement, target wall roughness, and jet spacing [17].

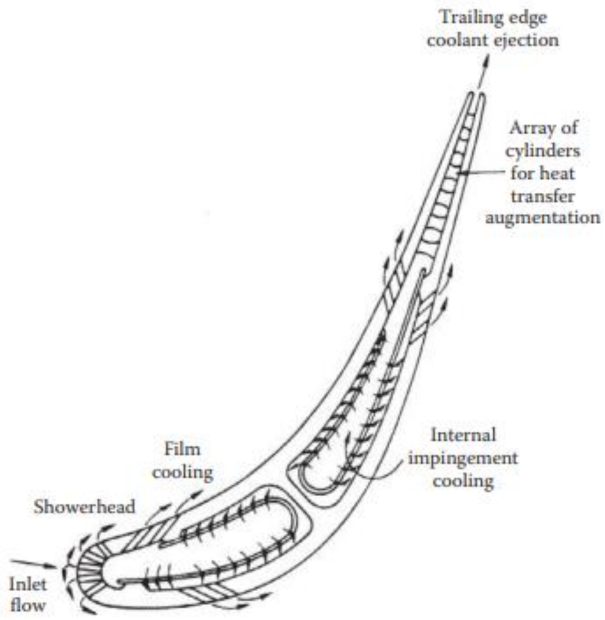


Figure II- 5: Schematic of impingement cooling arrangement in a first-stage turbine inlet guide vane (Han, J.-C. (2004). Recent Studies in Turbine Blade Cooling

II.5.1.2. Convection cooling:

The cooling system circulates air through internal passageways in the blade. Heat is transmitted via conduction through the blade and convection into the air flowing inside.

The blade. To maximize internal surface area, cooling routes are often serpentine with numerous tiny fins. The internal passageways in the blade can be round or elliptical. Air is cooled by traveling via tunnels from the hub to the blade tip. An air compressor provides cooling air. Gas turbines use a cooling tunnel to mix hot fluid from the outside with the main stream near the blade tips [16].

II.5.1.3. Rib tubulated cooling:

A turbine rotor blade has three main internal cooling zones, depending on the cooling mechanism used. The cooling system includes jet impingement at the leading edge, pin fins at the trailing edge, and serpentine rib-roughened coolant passageways in the middle. Advanced gas turbine blades use rib turbulence promoters on internal cooling passageways to improve heat transmission. Turbine blades transfer thermal energy from their outward pressure and suction surfaces to their inner zones, which are then cooled internally [18].

II.5.1.4. Pin fin cooling:

Because of the limits within the very tiny distances, cooling with pin-fins is frequently utilized in these locations to enhance heat transfer from a closed plate. Anchors often have a height-to-width ratio between ½ and 4. Heat is transported to the same pin fin configurations from the other color and wall, resulting in a graceful channel with varied pins. When investigating pin-fin cooling, it's important to examine parameters like pin array type, as the spacing between the pins affects the heat transfer within the channel.

The pin size and form improve heat transfer in this cooling approach. Pin fins at the tip's corners and periphery direct vortices toward the wall, promoting turbulent mixing of hot and cold fluids [19, 20].

II.5.1.5. Dimple cooling:

Dimpled cooling could be a very interesting method because of the consequences of relative losses (compared to pins) and heat transfer with a low degree of reduction. Concave tones create a separation of the flow and the regeneration of two vortices. Areas where heat transfer is a testament to the high ground flow of the ground in real-time downstream. The heat transfer within the marker channel is usually twice as large as the heat transfer during the alternating channel for double to four times the loss of the smooth channel [20].

II.5.2. External cooling:

II.5.2.1. Swirl cooling:

In a broader sense, the surface of the concavity flows in one large area called the vortex technology, which includes multiple pathways which means the establishment of vertical flow or rotating turbines. Another rising vortex technology is that the use of different wall jets is embedded in the bursiform cooling pads, or in parts of the inner wall to make the most complete movements. This cooling method is commonly referred to as swirl cooling, and in addition, the cooling of the hurricane provides equilibrium with equal heat transfer in which a direct impact is applied [20,21].

II.5.2.2. Lattice cooling:

This cooling system uses cross-cutting channels to build sub-structures. The two sections of the channels below appear to be contradicting or clipped in style. Cooling flow in a vast network, like the root zone of blades, demands couplings in upper channels but minimal coupling in lower channels. This cooling approach involves cooling from the edges of the tent network where underground channels meet at "dead ends" or "bind" walls, such as the inner rib or outer wall design aerofoils [22].

II.5.2.3. Transpiration cooling:

Transpiration cooling is a new external cooling approach that combines the benefits of film cooling with porous media properties. Transpiration cooling uses porous media to disperse cooling fluids and provide heat exchange space. This allows for large and reusable thermal protection while also separating the cooling structure from the heated component, as shown in Figures 2-6. It may therefore become a priority research field for external cooling systems in the future as one of the most efficient external cooling techniques for advanced gas turbines with active thermal protection [23].

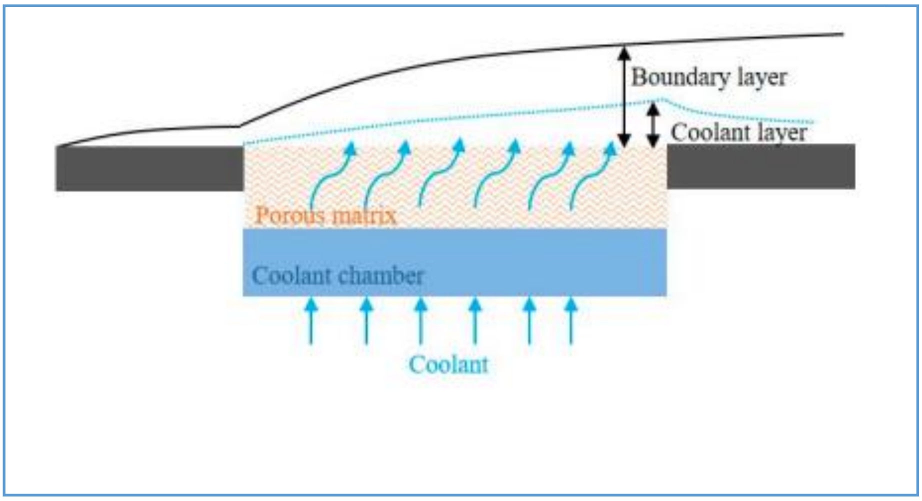


Figure II- 6: Schematic diagram of transpiration cooling (Xu, L., Sun, Z., Ruan, Q., Xi, L., Gao, J., & Li, Y. (2023). Development Trend of Cooling Technology for Turbine Blades at Super-High Temperature of above 2000 K.

II.5.2.4. Film cooling :

Film cooling (also called thin film cooling), a widely used type, allows for higher cooling effectiveness than either convection or impingement cooling. This technique consists of pumping the cooling air out of the blade through multiple small holes or slots in the structure. A thin layer (the film) of cooling air is then created on the external surface of the blade, reducing the heat transfer from the main flow, whose temperature (1300 – 1800 kelvins) can exceed the melting point of the blade material (1300 – 1400 kelvins). The ability of the film cooling system to cool the surface is typically evaluated using a parameter called cooling effectiveness. Higher cooling effectiveness (with a maximum value of one) indicates that the blade material temperature is closer to the coolant temperature. In locations where the blade temperature approaches the hot gas temperature, the cooling effectiveness approaches to zero. The cooling effectiveness is mainly affected by the coolant flow parameters and the injection geometry. Coolant flow parameters include the velocity, density, blowing, and momentum ratios which are calculated using the coolant and mainstream flow characteristics. Injection geometry parameters consist of hole or slot geometry (i.e. cylindrical, Shaped holes or slots) and injection angle [16].

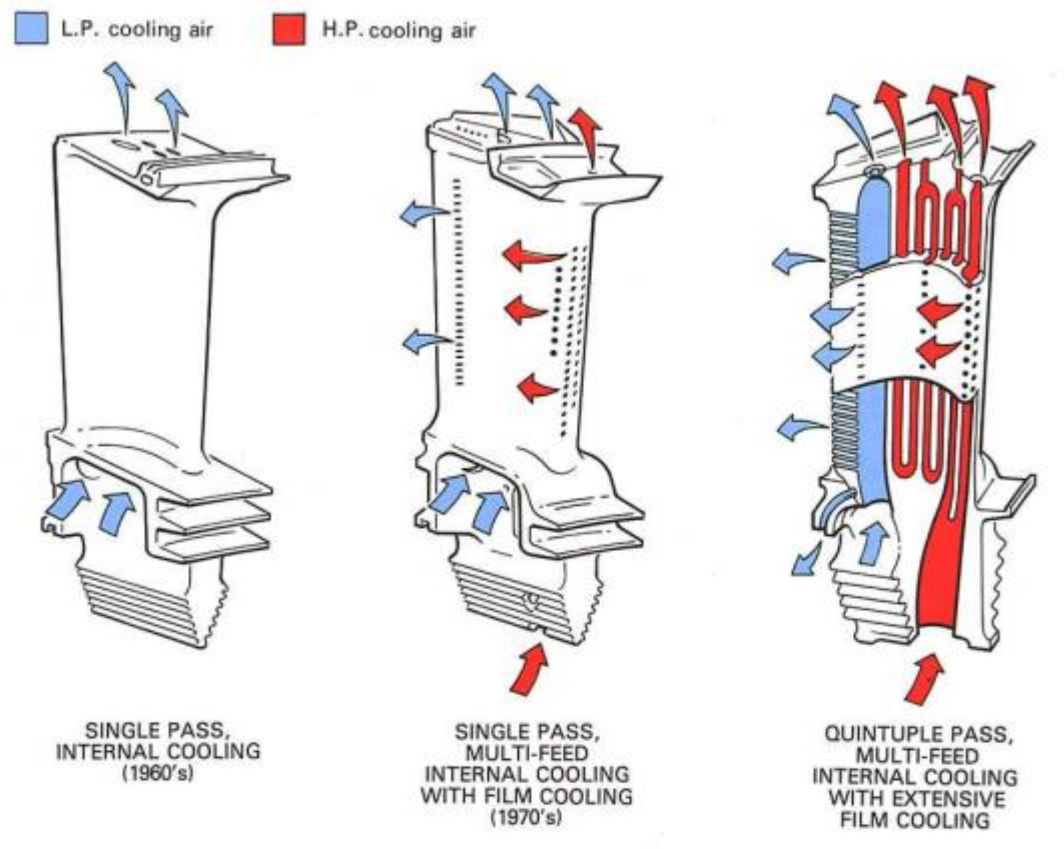


Figure II- 7: Development of high-pressure turbine blade cooling (aeromodelbasic.blogspot.com).

II.6. Film-Cooling Effectiveness:

II.6.1. Unsteady Effect on Turbine Blade Film Cooling:

To build an effective cooling system, it's important to understand the hot gas flow dynamics within turbine stages. Long-term research has focused on turbine blade heat transmission and film cooling. Recent research has focused on the effect of combustion-generated high turbulence on turbine stator heat transfer with or without film cooling, the effect of upstream unsteady wake on downstream rotor blade heat transfer with or without film cooling, and the effect of surface roughness, thermal barrier coatings, and spallations on blade heat transfer with or without film cooling. Recent research has also focused on delivering very precise heat transfer coefficient and film cooling effectiveness distributions on turbine blades under unsteady high turbulence flow conditions utilizing the transient liquid crystal imaging technique.

One important finding from these studies is that unsteady high turbulent flows do not dramatically change heat transfer coefficient distributions on the film-cooled blade, but significantly reduce the film cooling effectiveness. This finding is crucial because the turbine blade might not be well protected by film cooling under hostile turbo machinery flow environments [24].

II.6.2. Effect of Blowing Ratio:

The impact of the blowing ratio on film cooling effectiveness is important. Film cooling efficacy gradually increases with an increase in blowing ratio until a certain point, at which time it begins to decline, and the peak moves downstream. Coolant flow increases initially, but decreases with increased blowing ratios as the jet separates from the surface. High blowing ratios cause the coolant to "jet" or "blow off," enabling hot gas to cover the surface and diminishing effectiveness [25].

II.6.3. Effect of Reynolds number:

The film cooling effectiveness can be influenced by the Reynolds number. As the Reynolds number increases, the film cooling effectiveness tends to decrease [26].

II.6.4. Effect of Coolant Supply Geometry:

Film cooling is a cooling method that prevents gas turbines from overheating. The efficacy of film cooling is determined by a variety of parameters, including coolant supply geometry. For example, in-hole roughness has been demonstrated to alter the hole's cooling effectiveness, increasing convective benefit while decreasing film benefit. Bogard and Thole found three primary elements that influence film cooling: cold air/mainstream conditions, hole geometry/arrangement, and blade geometry. The cooling efficiency of a coolant supply can be influenced by its geometry; certain designs, including those with cooling holes curved like lobes, have been shown to increase the efficiency of film cooling.

II.6.5. Film Hole Shape Effect on Turbine Blade Film Cooling:

Recently, an attempt was made to improve the cooling effectiveness and lifetime of gas turbine blades by contouring the film hole shape. Film cooling holes with a diffuser-shaped extension at their exit point are thought to improve film cooling performance on a gas turbine blade. Compared to a typical cylindrical hole, the greater cross-sectional area at the hole exit reduces coolant velocity for the same blowing ratio.

The momentum flux of the jet exiting the shaped hole and the penetration of the jet into the mainstream will be reduced, which results in an increased film cooling effectiveness.

Furthermore, lateral expansion of the hole provides an improved lateral spreading of the jet, which leads to a better lateral film cooling coverage of the blade. A few previous studies have shown that expanding the exit of the cooling hole improves film-cooling performance compared to a standard cylindrical hole. For example, Schmidt et al. (1996) and Sen et al. (1996) compared a forward-expanded hole to a cylindrical hole, both of them having compound angle injection for flat-plate film cooling. They found that there is a larger lateral spreading of the forward expanded hole even though the spatially averaged effectiveness is the same for both cases. Gritsch et al. (1998a, 1998b) studied detailed measurements of the flat-plate film cooling effectiveness and heat transfer coefficients downstream of a single film-cooling hole by using an IR camera method. They reported that, compared to the cylindrical hole, the two types of expanded holes in their study (the fan-shaped hole and the laidback fan-shaped hole) show significantly improved thermal protection of the surface downstream of the injection, particularly at the high blowing ratios.

II.7. Conservation laws:

II.7.1. Conservation of mass:

Translates the conservation of the mass of the fluid in movement, applied to a fluid particle this law leads to the equation known under the name of equation of continuity. Leads to the equation known as the continuity equation, expressed in differential form in Cartesian coordinates as follows:

$$\frac{\partial p}{\partial t} + \frac{\partial}{\partial x_j}\left(pu_j\right) = \qquad j{:}1,2,3 \qquad \textbf{(II-1)}$$

II.7.2. Conservation of Momentum:

Based on Newton's 2nd law applied to a fluid particle, this principle leads to the famous Navier Stokes equations, which can be written in differential form in Cartesian coordinates as follows in Cartesian coordinates as follows:

$$\frac{\partial p}{\partial t}\left(pu_j\right) + \frac{\partial}{\partial x_j}\left(pu_j u_j\right) = -\frac{\partial p}{\partial x_j} + \frac{\partial t}{\partial x_j} \qquad \textbf{(II-2)}$$

Where τ_{ij} represents the stress tensor given for a Newtonian fluid by the relation as follows:

$$t_i = \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_i}{\partial x_j}\right) - \frac{2}{3}\mu\frac{\partial u_i}{\partial x_j}\ .\delta_g \qquad \textbf{(II-3)}$$

$$\text{As } \delta_g = \begin{cases} 1 & i \neq j \\ 0 & i = j \end{cases}$$

II.7.3. Energy conservation:

Based on the first law of thermodynamics, it translates the conservation of energy (Energy neither appears nor disappears from anything) for a moving fluid as follows:

$$\frac{\partial}{\partial t}(pH - p) + \frac{\partial}{\partial x_j}\left(pu_jH\right) = -\frac{\partial q_j}{\partial x_j}(u_j\tau) \quad \text{(II-4)}$$

Where H is the total heat loss, q_j is the heat flux, and are given by:

$$H = h + \frac{1}{2}u_ju_j \quad \text{(II-5)}$$

$$q_j = -\lambda\frac{\partial T}{\partial t} = -\frac{u}{P_r}\frac{\partial h}{\partial x_j} \quad \text{(II-6)}$$

In addition to these equations, there is an equation linking density to thermodynamic variables. For a perfect gas, the following relationship is used:

$$P = \rho . r . T \quad \text{(II-7)}$$

Chapter III

Numerical Simulation

III.1. Introduction:

Numerical simulation is an increasingly significant technique in scientific study.
Digital simulation has revolutionized investigative methods by allowing the many stages of a process's functioning to be calculated.
Gambit and Fluent are commercially licensed software packages that can be used to carry out 2D or 3D fluid mechanics simulations, from mesh construction with Gambit to solving the Gambit to solving the Navier Stokes equation and post-processing with Fluent.
Widely used in industry (automotive, aeronautics, space... etc.) because of their powerful graphical interface and wealth of options, they can be used to carry out simulations on all types of complex geometry (fixed or mobile) using fixed or adaptive meshes and with a variety of physical models (two-phase, turbulent, etc.).

III.2. Presentation of the GAMBIT software:

The user can create the shape of the calculation domain and divide it into smaller control volumes or calculation cells. All of these elementary volumes combine to produce the mesh. This level also defines the relevant boundary conditions.
It can be used to construct a geometry and generate a mesh for it If required, geometry from another CAD program can be imported into this pre-processor. It generates ***.msh** files for Fluent.

III.2.1. How to create geometry in Gambit:

A flowchart illustrating the creation of a blade profile geometry in our work, by the Gambit work, using the Gambit processor, is shown in the figure:

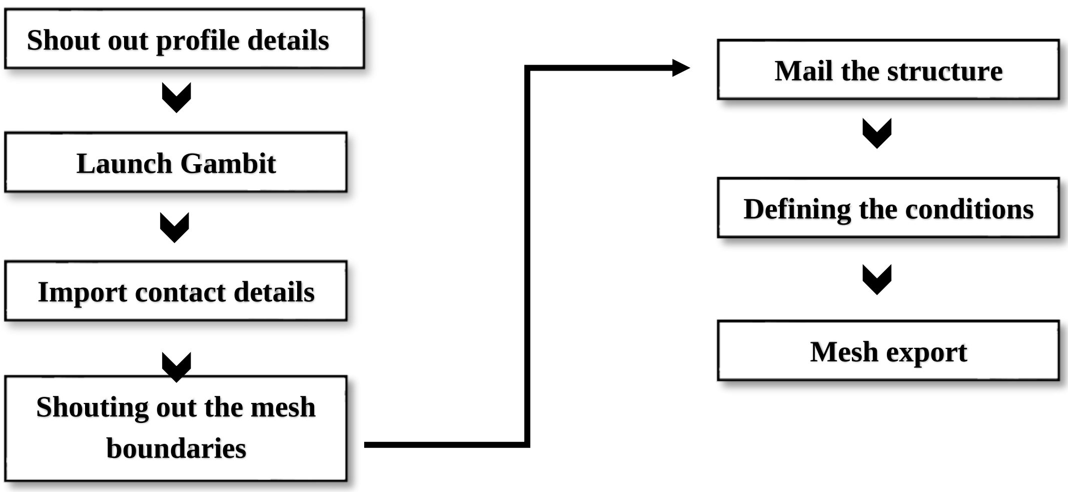


Figure III- 1: The process of building a mesh on Gambit.

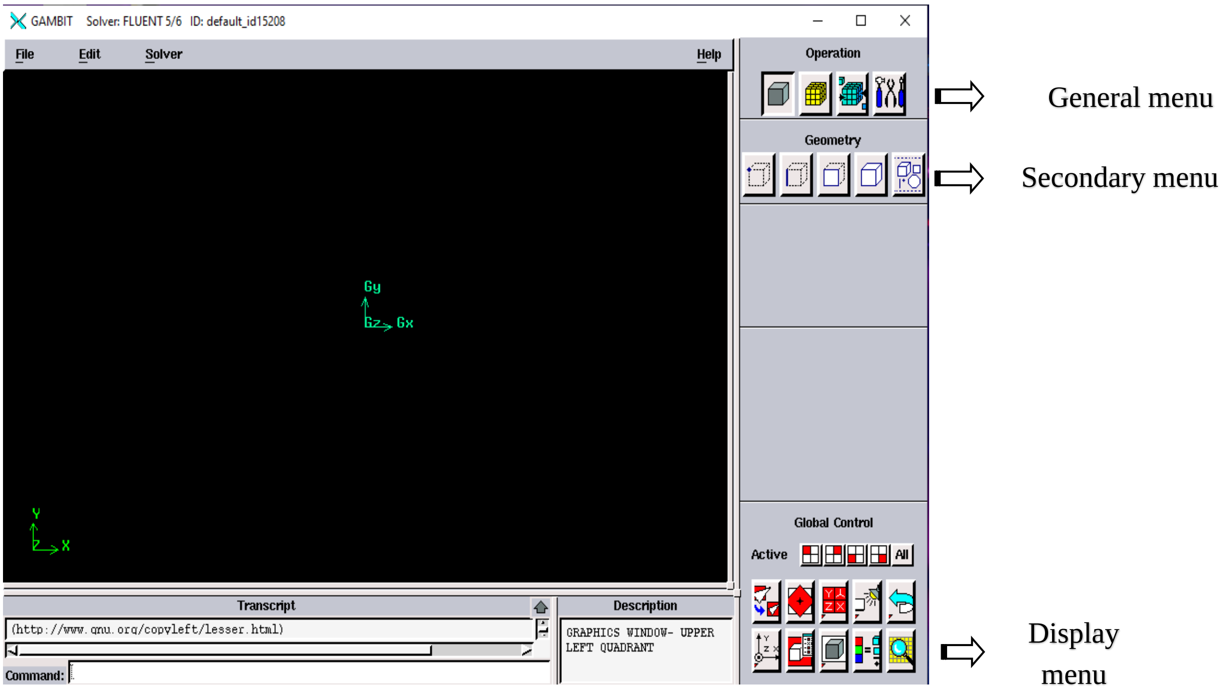


Figure III- 2: Gambit overview.

❖ Create geometry elements menu:

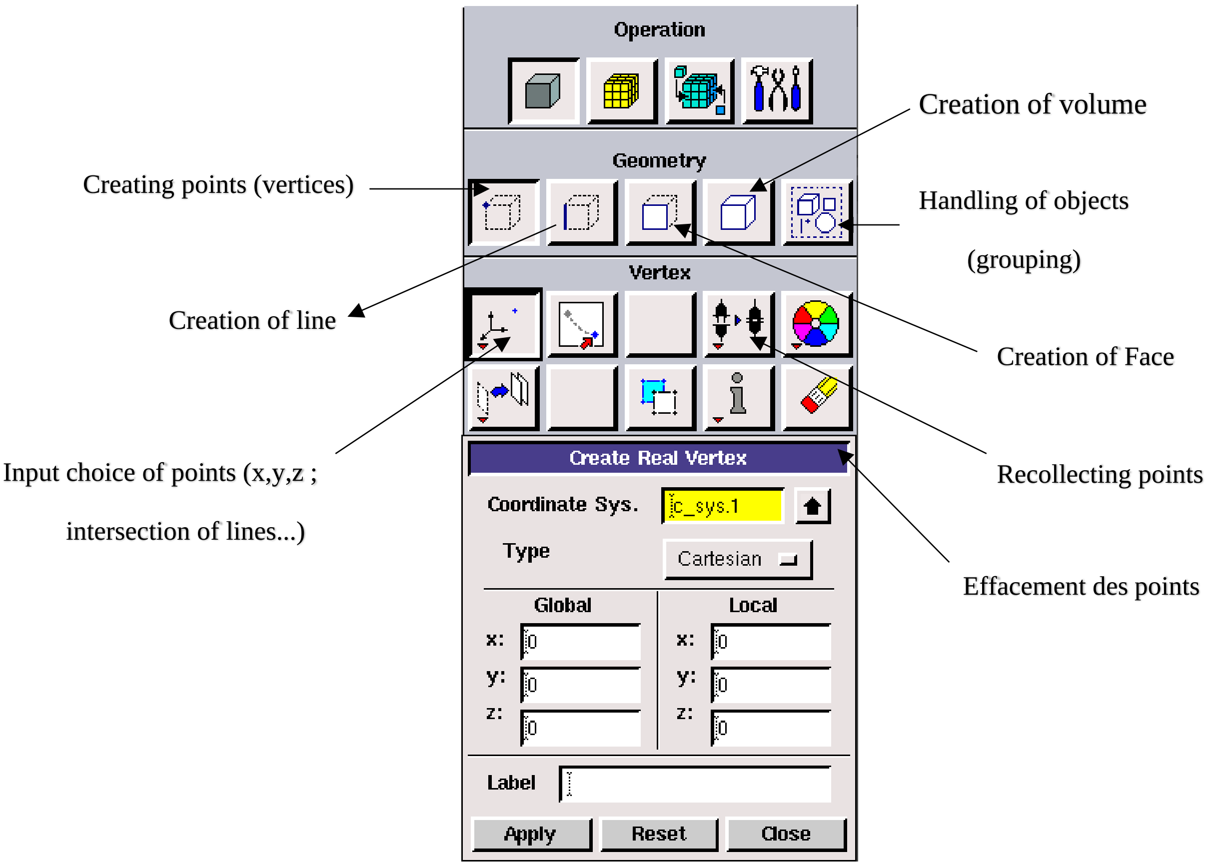
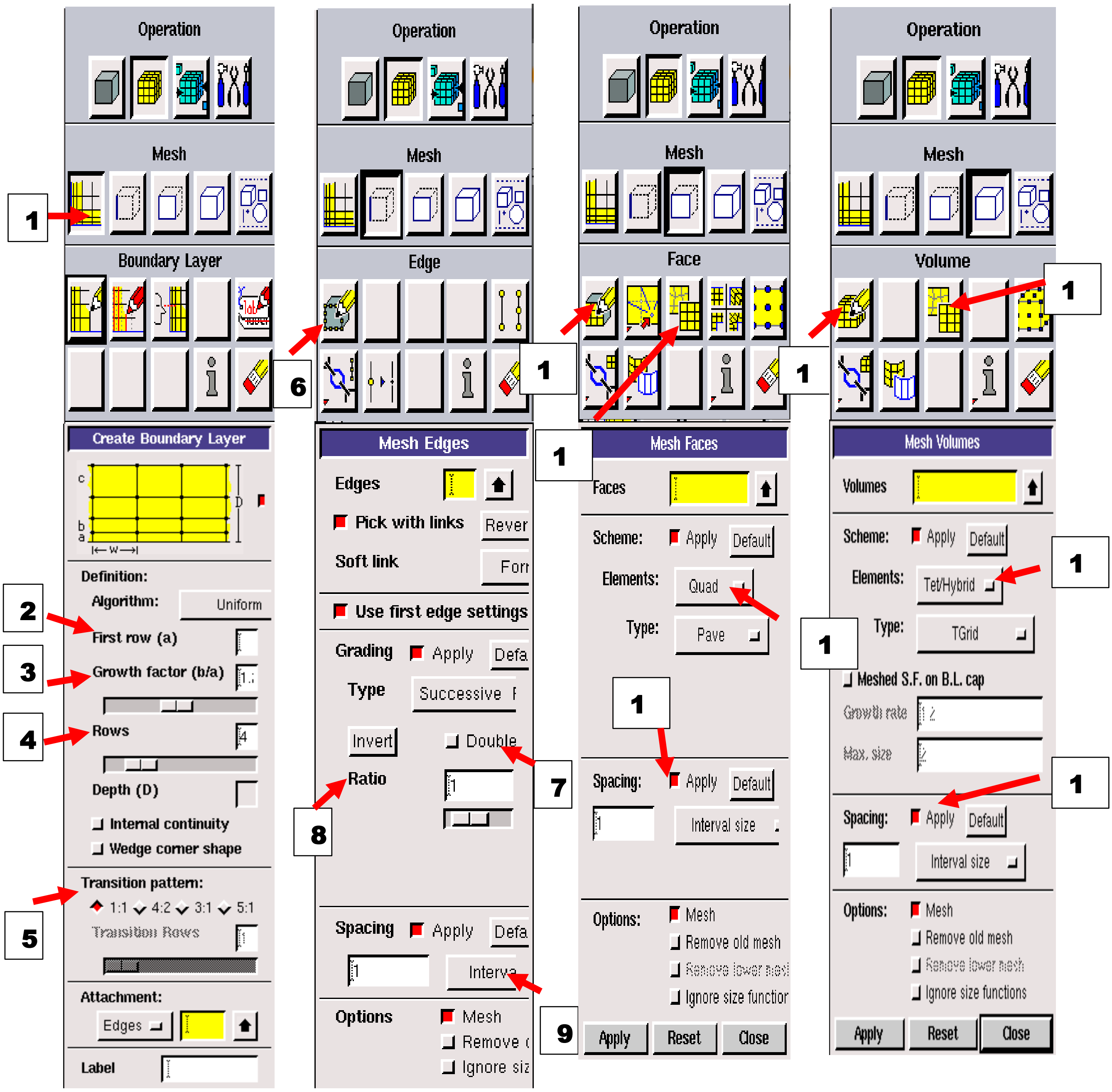


Figure III- 3: geometry elements menu

❖ Mesh geometry:

Mesh construction is crucial in numerical simulations because it affects the estimated solution. Gambit allows you to automatically mesh based on a variety of mesh types and algorithms. While meshing with a large number of components can improve computational accuracy, it can also strain memory resources and slow down the system. Therefore, it's crucial to find a balance between these two factors.



1. Creating and modifying a boundary layer mesh

2

Figure III- 4: Gambits main mesh menus.

3. Mesh expansion factor
4. Number of specific mesh lines
5. Mesh type (4 specific mesh types are available)
6. Meshing a line
7. Application of a double ratio which increases the density of points either on the sides or in the center of the lines
8. Using a ratio for meshing
9. Choice of mesh option: number of nodes or interval size between the two
10. Face mesh
11. Smoothing distorted meshes
12. Choice of mesh type: Tri, Quad
13. If checked, a regular mesh is created using the parameters below. If not, a mesh is created from the nodes defined on the edges.
14. Volume mesh
15. Smoothing distorted meshes
16. Choice of mesh type: Tetra, Hex
17. If checked, a regular mesh is created using the parameters below. If not, a mesh is created from the nodes defined on the edges.

❖ Boundary conditions:

Several types of boundary conditions are available, depending on the problem being addressed. À note that mesh space is assumed to be fluid by default. The name given to the boundary is very important because it will be used in Fluent, and if the names are not clear there is a risk of mixing up the boundaries.

The Gambit mesh generator can generate meshes that many solvers can use. solvers can use, so we need to specify which solver we want to process the mesh file with. mesh file. The choice of boundary type varies according to the solver being considered to solve the problem.

❖ **Velocity inlet:** Used for incompressible or moderately compressible flows, when the when the inlet velocity is known.

❖ **Pressure Inlet:** Used for compressible and incompressible flows.

- ❖ **Mass Flow Inlet:** A known mass flow rate is imposed at the inlet. to use Mass Flow Inlet in incompressible flow.
- ❖ **Pressure Outlet:** This type is used to define the static pressure at the outlet. The use of the "Pressure Outlet" condition instead of "Outflow" often results in result in better convergence.
- ❖ **Outflow:** This condition is used to model fluid outflows for which the details of the velocity and pressure at the outlet are not known a priori. the details of the velocity and pressure at the outlet.
- ❖ **Wall:** This type is used to delimit solid regions from fluid regions. In general, the properties of a smooth wall are used

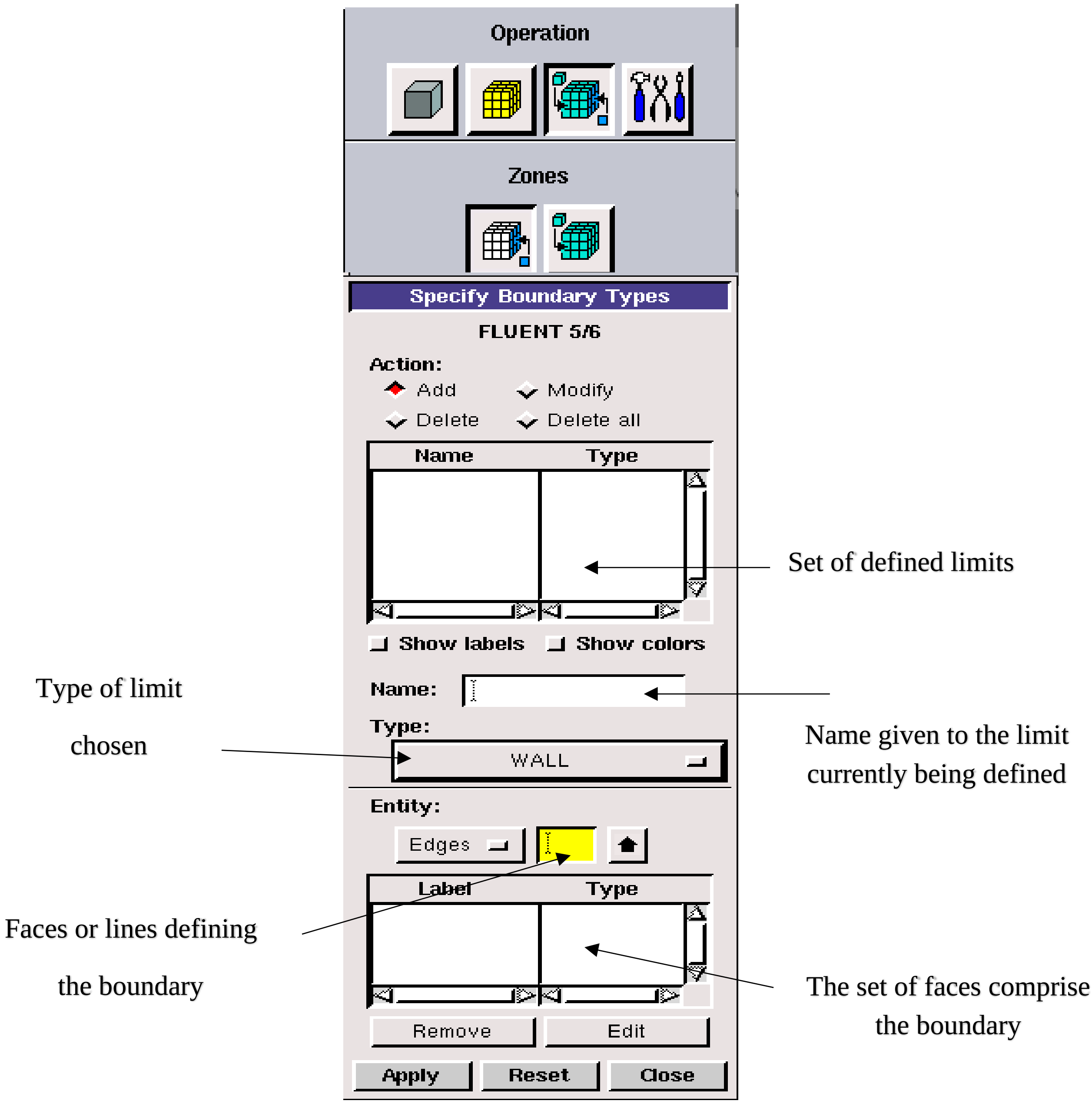


Figure III- 5: Boundary conditions

III.2.3. Exporting the mesh produced by Gambit to Fluent:

Once the geometry has been created and the boundary conditions defined, the mesh is exporting the mesh to FLUENT.

By choosing the path: file → Export→ Mesh so that FLUENT can read and use it. Save the session, and then close GAMBIT.

III.3 Blade geometry and calculation range:

The software will display a NACA 6512 profile curve with a chord of 1m. convert the curve into a sketch that you can then manipulate in:

GAMBIT ➤ **File** ➤ **Import** ➤ **Import ICEM Input** ➤ **NACA 6512**

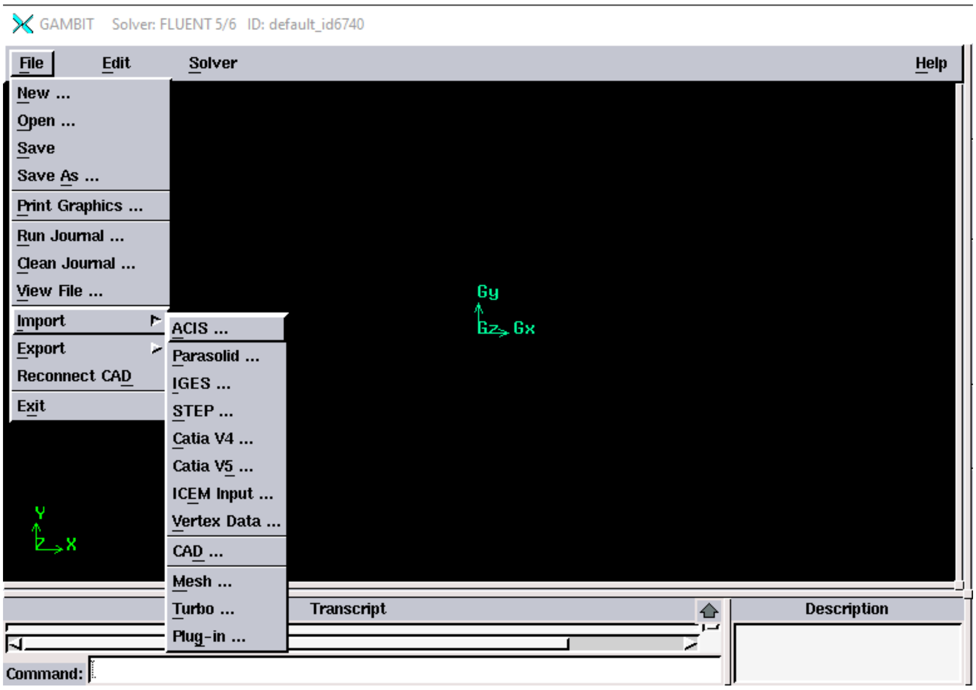


Figure III- 6: Create geometry elements menu.

III.3.1. Drawing of blade geometry by gambit:

We are using the NACA 6512 which is a TXT file uploaded from NACA 4 Digit Airflow Generator it included all the data of our blade.

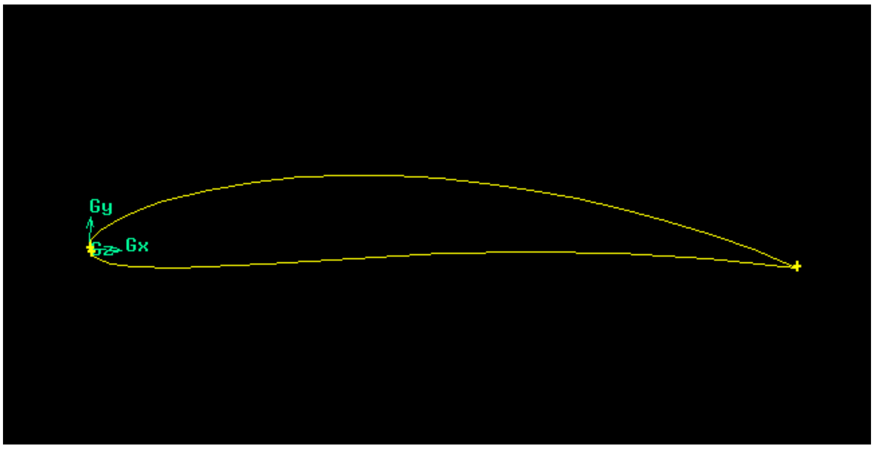


Figure III- 7: 2D profile figure NACA6512.

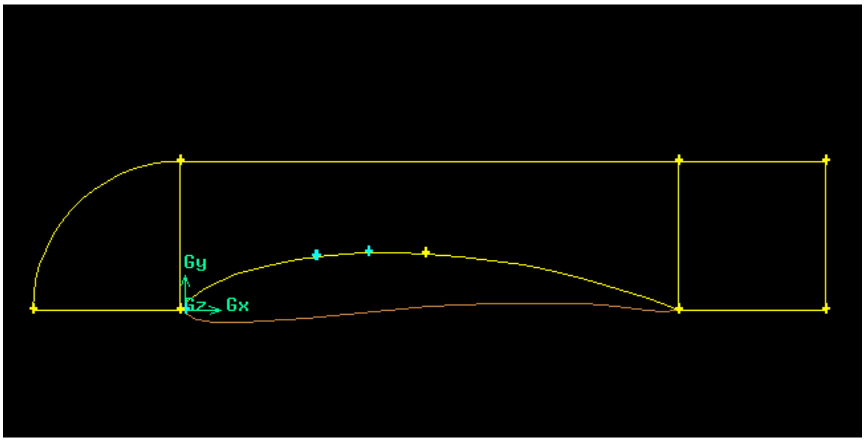


Figure III- 8: Creating control surfaces.

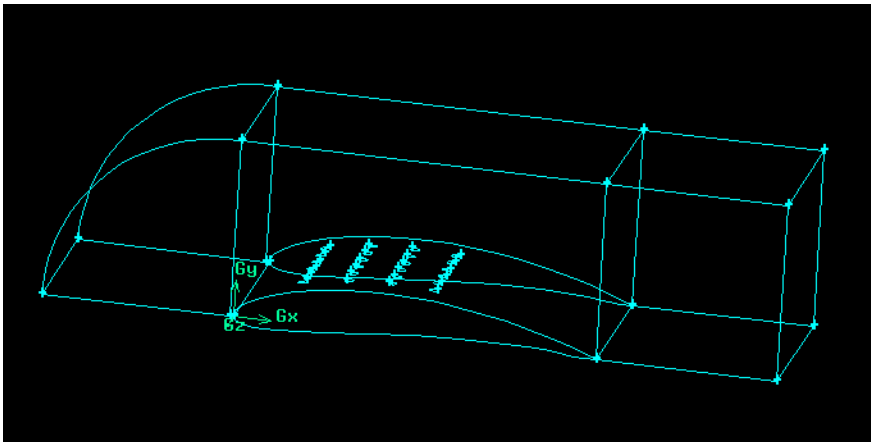


Figure III- 9: Geometry of the dawn in 3D and creating the surface.

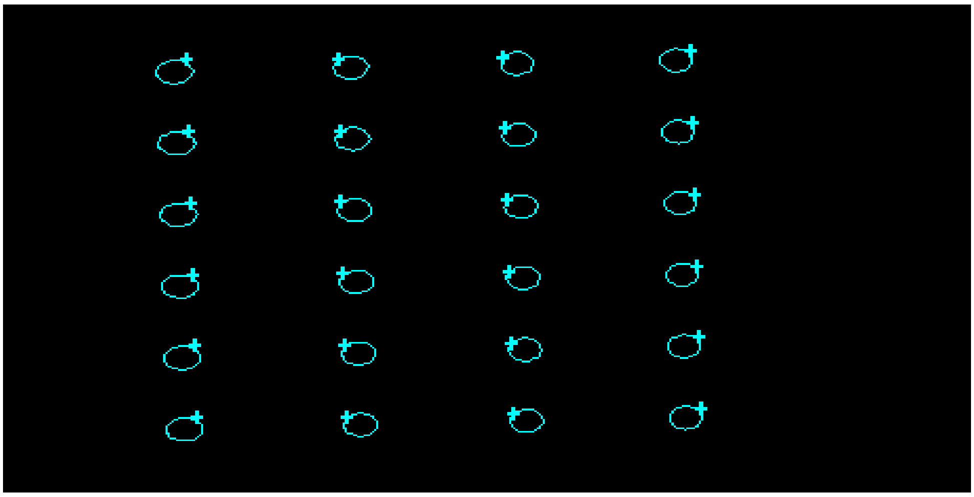


Figure III- 10: Layout of cooling orifices (cylindrical).

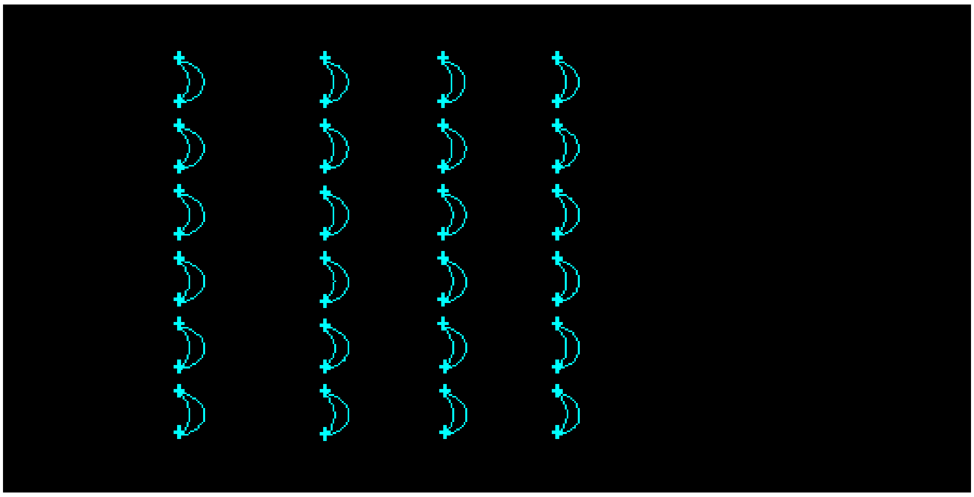


Figure III- 11: Layout of cooling orifices (crescent).

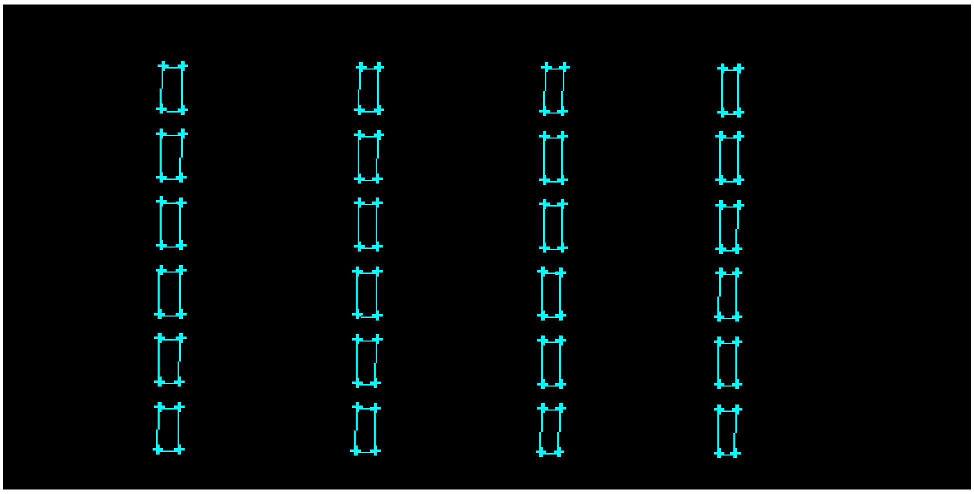


Figure III- 12: Layout of cooling orifices (quadrant).

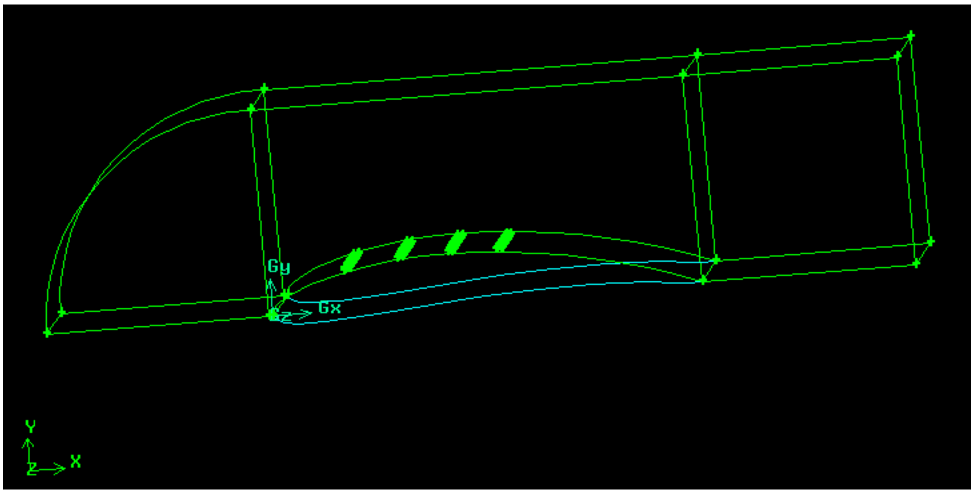


Figure III- 13: Creating the volume.

III.3.2. Mesh:

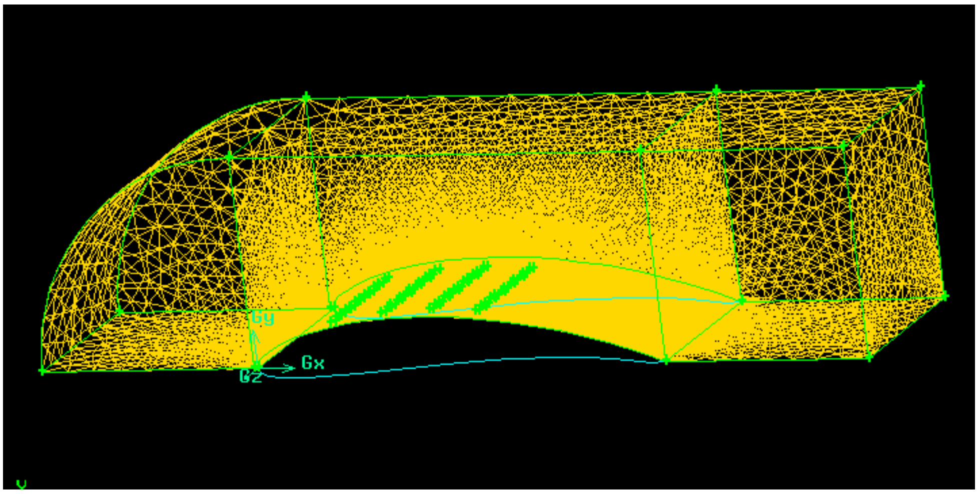


Figure III- 14: Volume mesh.

III.3.3. Boundary conditions:

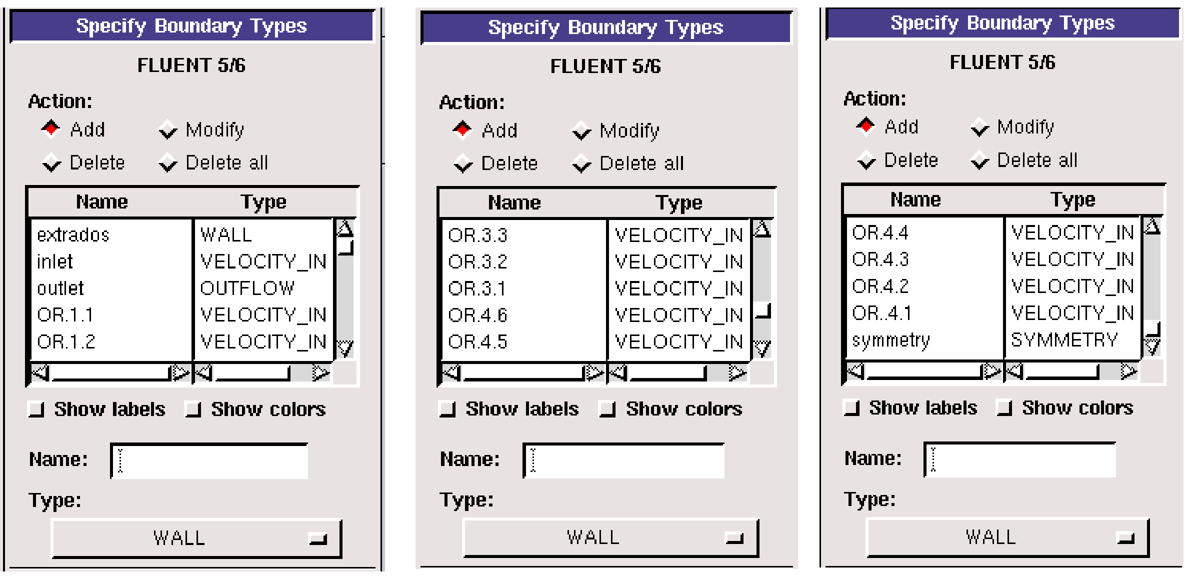


Figure III- 15: Boundary conditions.

❖ Next, we export the mesh as a point.msh

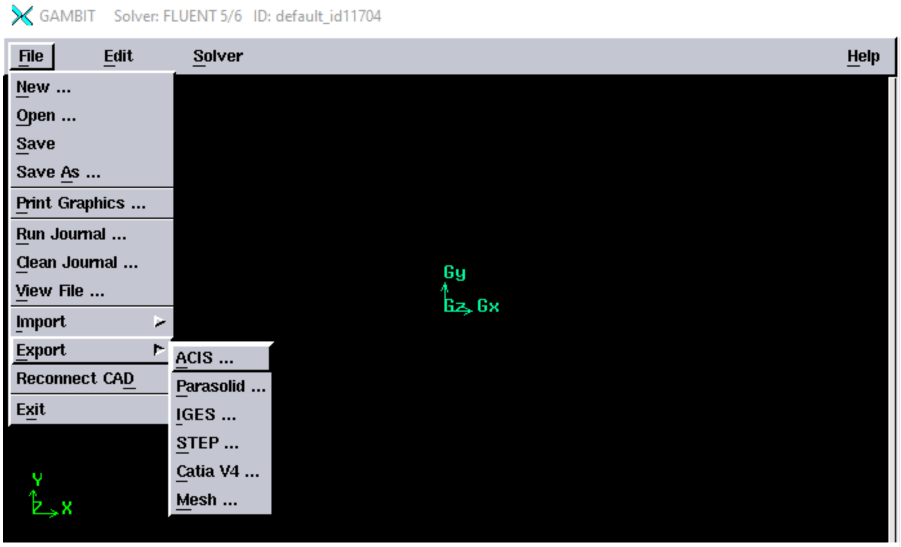


Figure III- 16: Exporting mesh.

III.4.Fluent:

Fluent is a commercial CFD code that is frequently used in industry. It solves various fluid flow and heat transfer concerns. For instance, it can calculate aeroplane wing lift, automotive drag, and electronic circuit cooling with vented air.

When you start the Fluent software, you must choose the dimensions of the calculation domain (2D or 3D), and the precision to be used by the software, single precision or double precision.

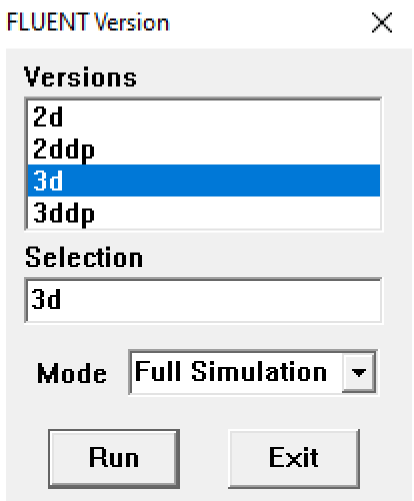


Figure III- 17: Choice of dimensions and precision.

Using Fluent is simple: just follow the order of the menus in general, from left to right and from top to bottom. The main stages of simulation with FLUENT are as follows:

1. Importing geometry (*.msh).
2. Checking the imported mesh.
3. Mesh smoothing (Smooth and swap the grid).
4. Checking the scale.
5. Choice of solver.
6. Grid display.
7. Choice of turbulence model.
8. Definition of fluid characteristics.
9. Operating conditions.
10. Boundary conditions.
11. Choice of convergence criteria.
12. Initializing calculations.
13. Saving the file *.cas
14. Launching the simulation.
15. Post-treatment of the solution.

III.4.1. Importing geometry:

To start the simulation, you need to import the file (*.msh) generated in Gambit.

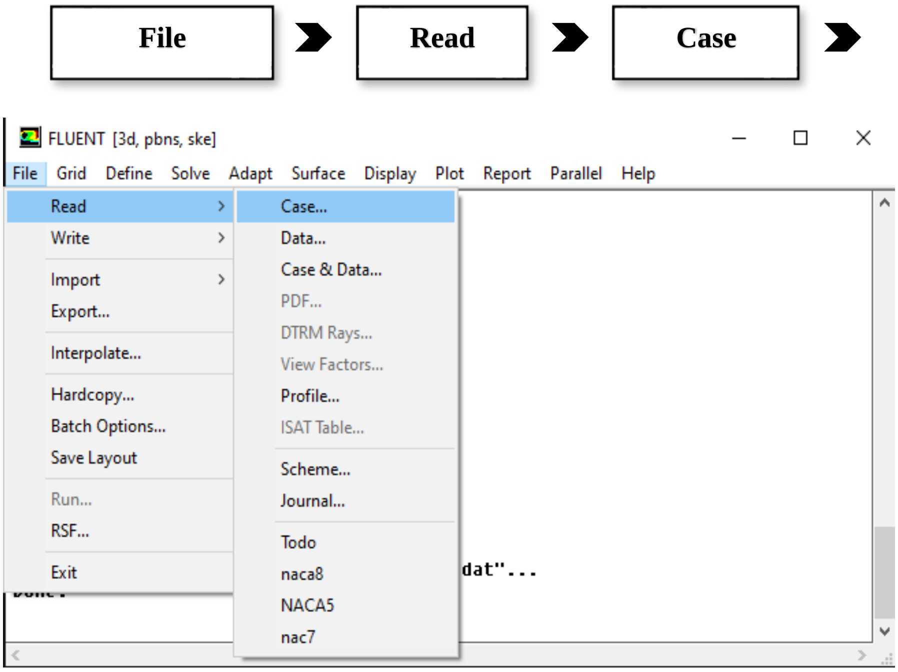


Figure III- 18: Importing geometry.

III.4.2. Checking the imported mesh:

This step is used to check that the imported mesh contains no errors or negative volumes.

Grid ➔ **Check**

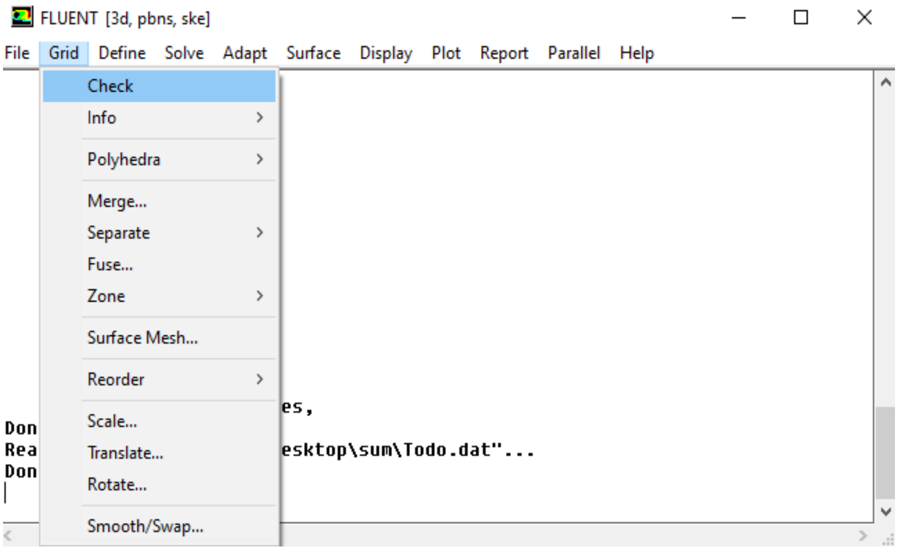


Figure III- 19: Checking the imported mesh.

III.4.3. Mesh smoothing:

Smoothing the mesh might help to ensure its quality. Smooth button, then Swap button. Repeat until FLUENT indicates that zero faces are switched.

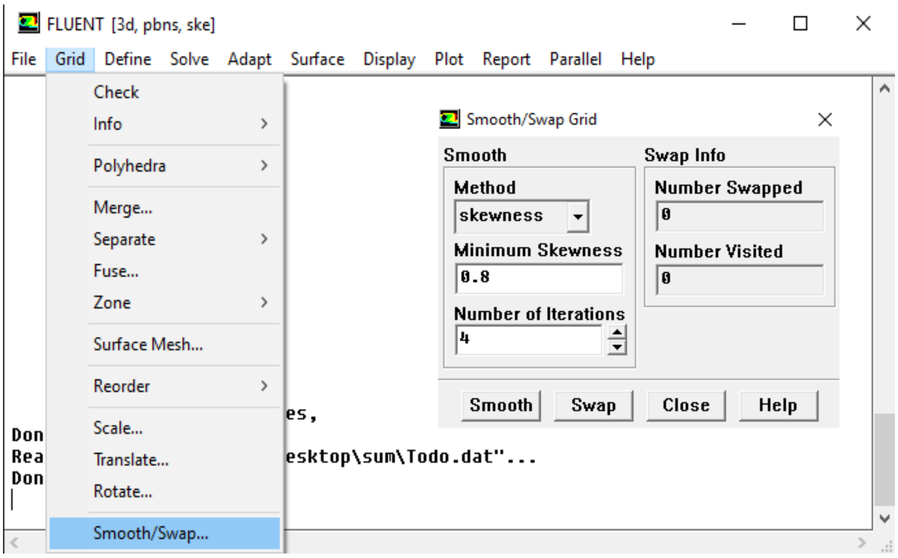


Figure III- 20: Smooth and swap the grid.

III.4.4. Choice of solver:

Define ➤ **Models** ➤ **Solver** ➤

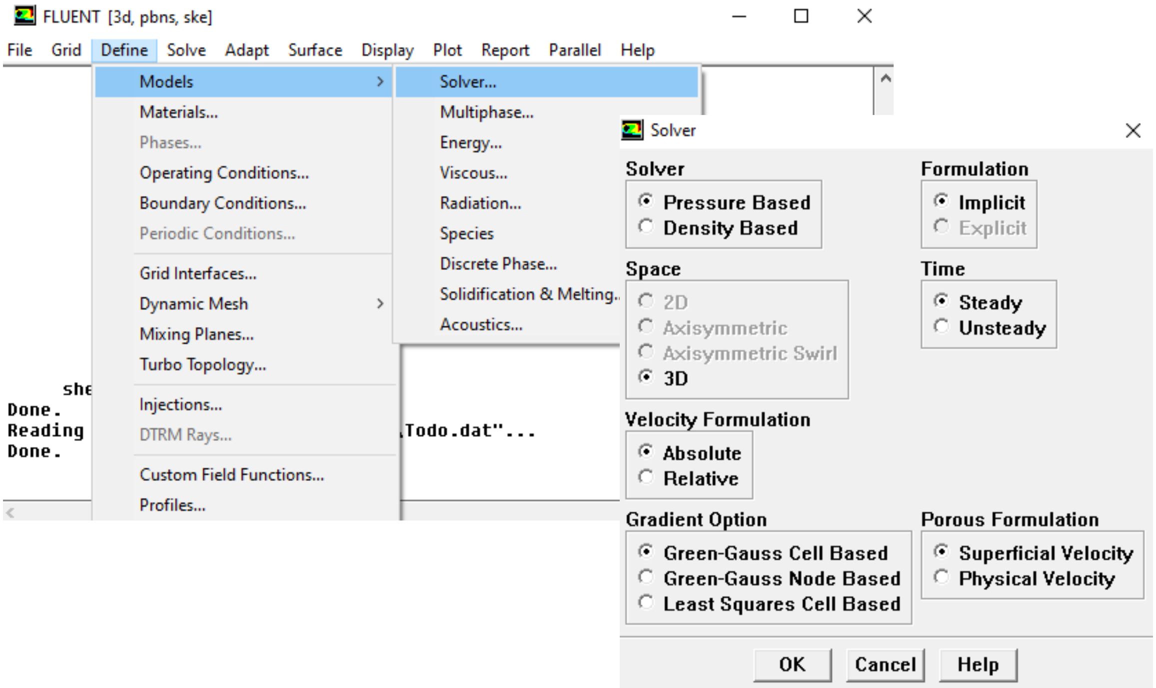


Figure III- 21: Choice of solver.

III.4.5. Authorisation for heat transfer:

Define ➤ **Models** ➤ **Energy** ➤

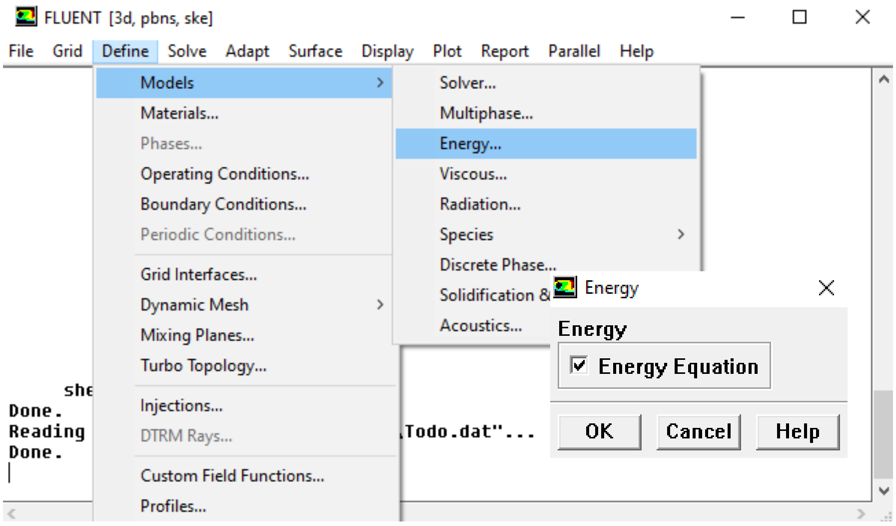


Figure III- 22: Authorisation for heat transfer.

III.4.6. Choice of turbulence model:

Define ➤ **Models** ➤ **Viscous** ➤

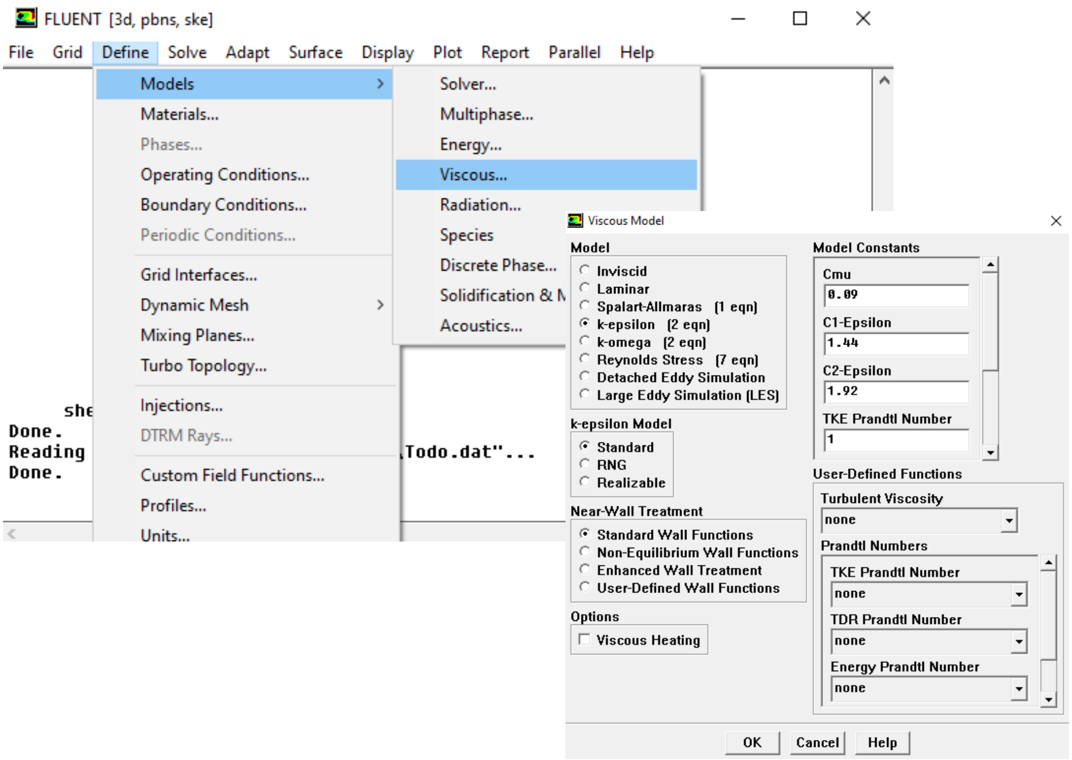


Figure III- 23: Choice of turbulence model.

III.4.7. Definition of fluid characteristics:

Fluid characteristics are loaded from the Fluent data library.

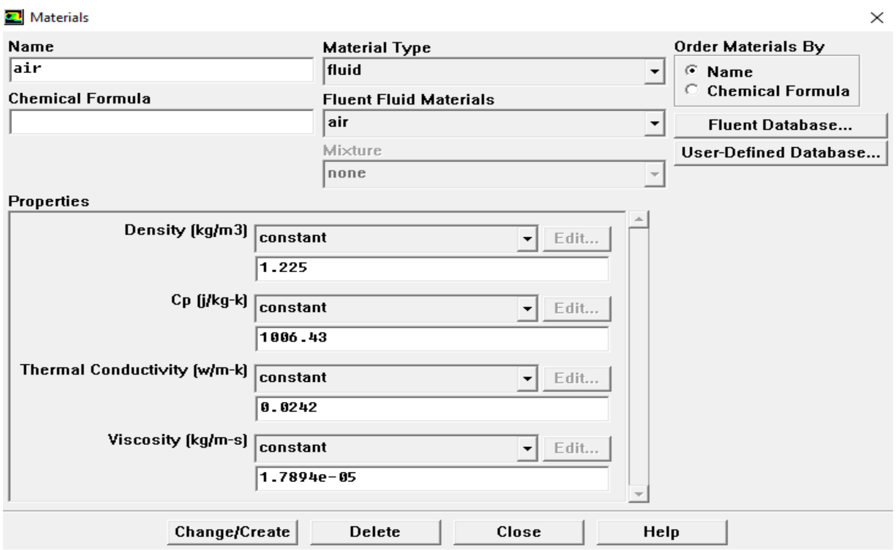


Figure III- 24 : Definition of fluid characteristics.

III.4.8. Operating conditions:

Before selecting the boundary conditions, the value of the reference pressure must first be chosen.

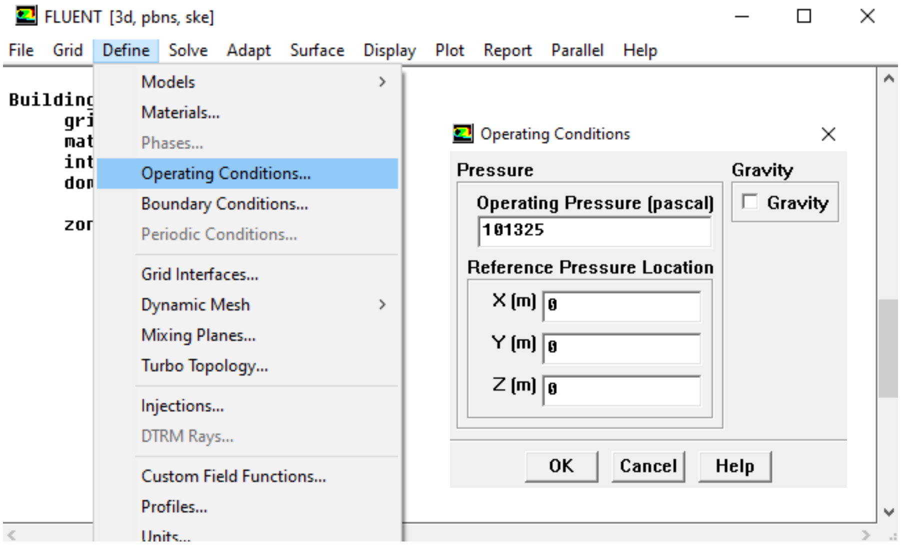
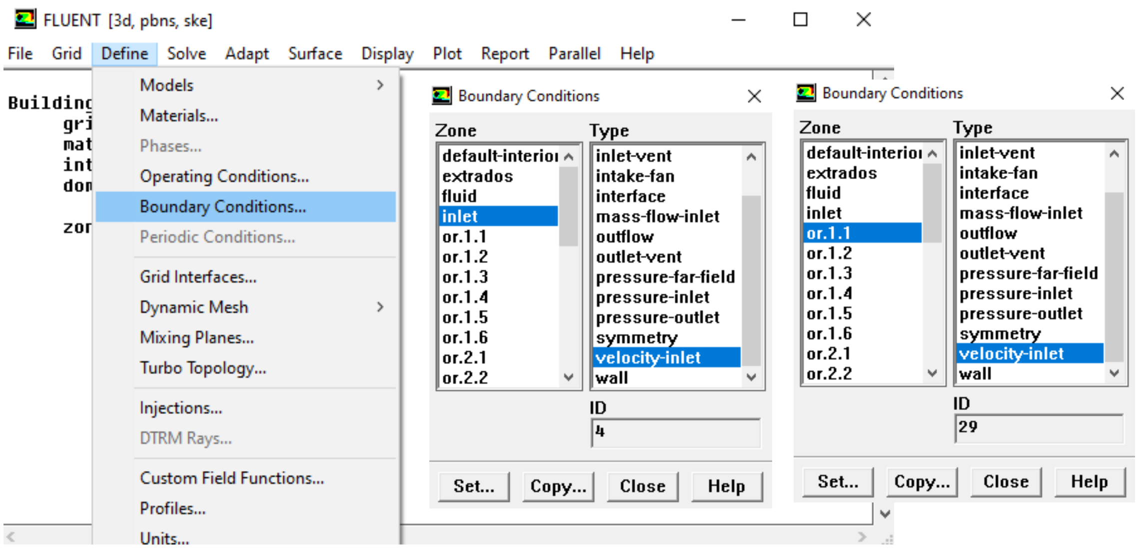


Figure III- 25: Operating conditions.

III.4.9. Boundary conditions:

Define ➔ **Boundary Conditions**



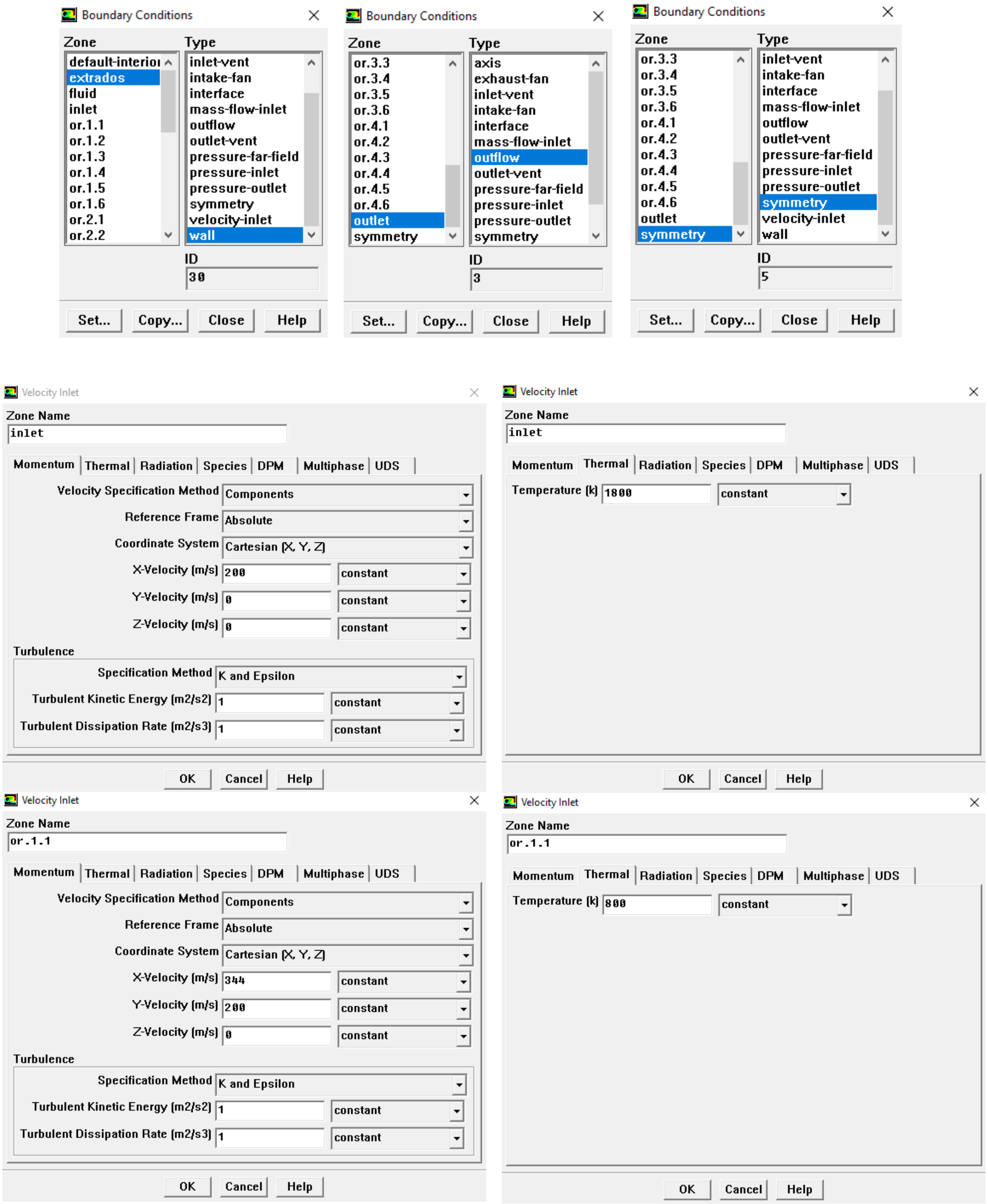


Figure III- 26: Boundary conditions.

III.4.10. Choice of solution control schemes:

Solve ➔ **Controls** ➔ **Solution**

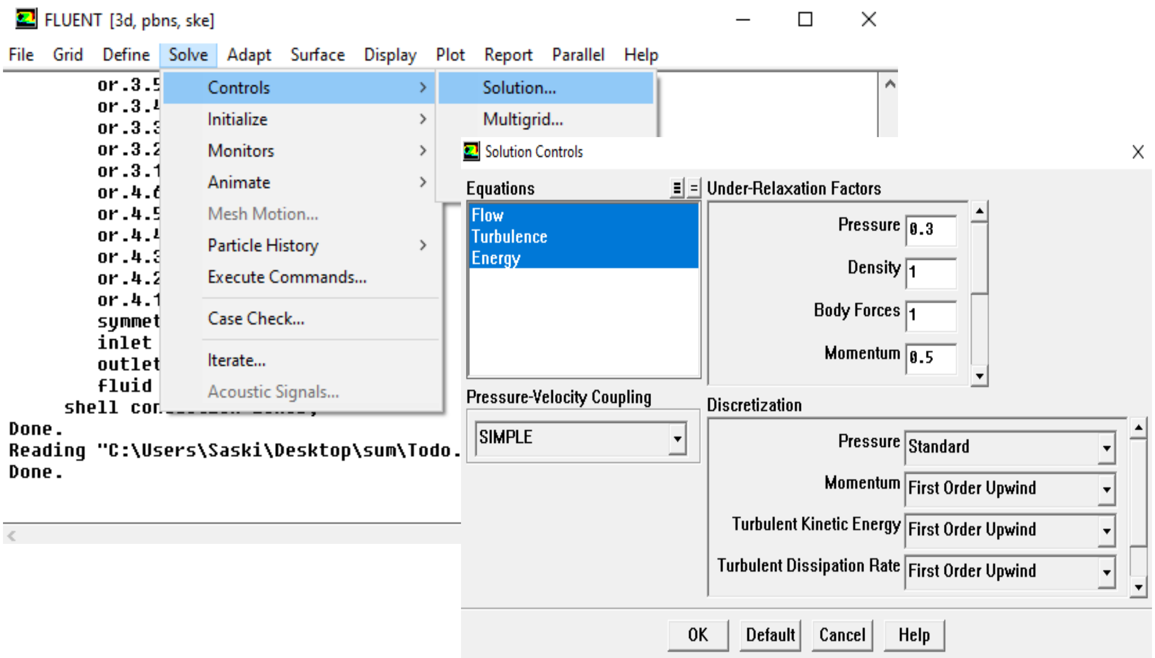


Figure III- 27: Choice of solution.

III.4.11. Initialise data:

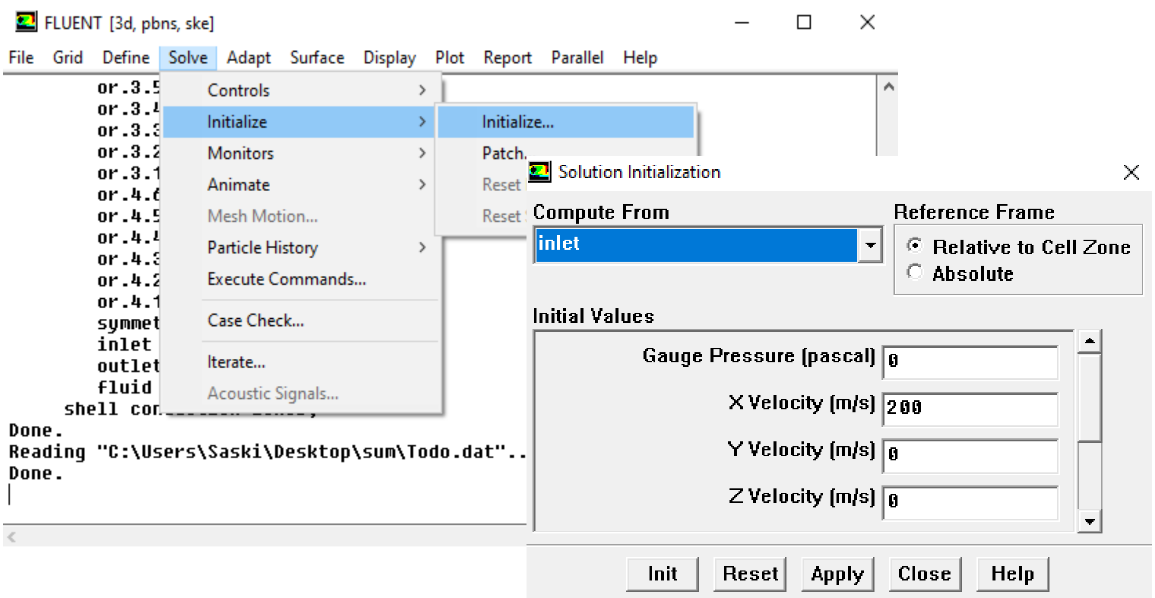


Figure III- 28: Initialise data.

III.4.12. Choice of convergence criteria:

Solve ➤ **Monitors** ➤ **Residual**

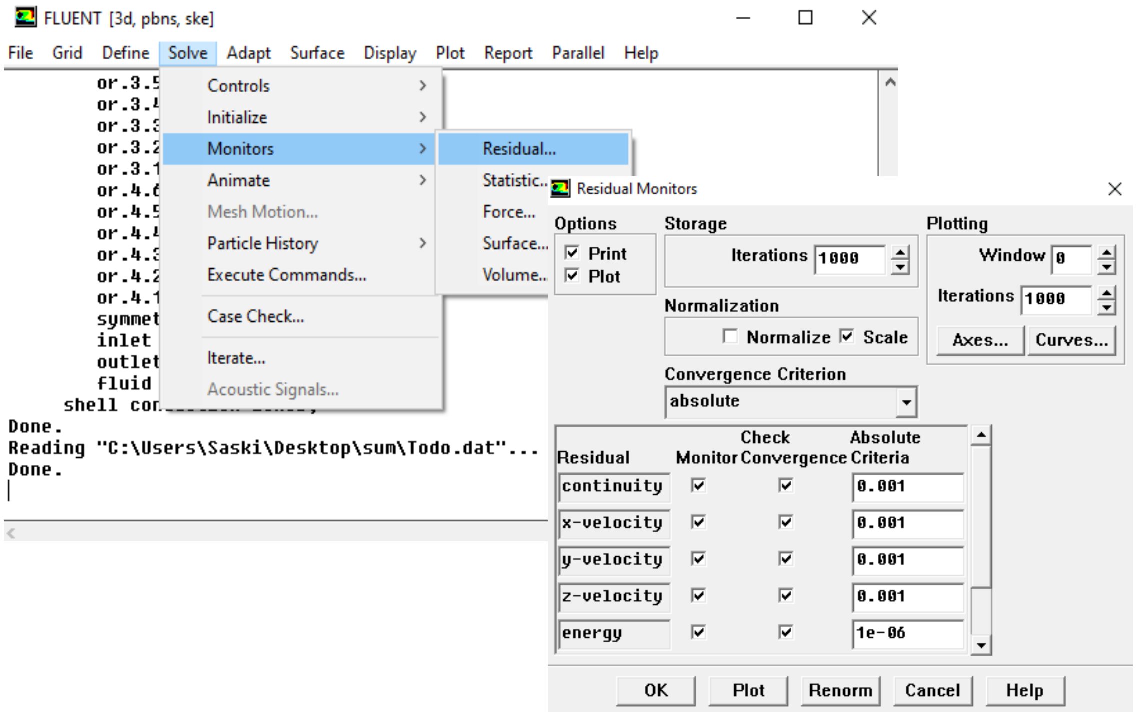


Figure III- 29: Choice of convergence criteria.

III.4.13. Launching the simulation:

Solve ➤ **Iterate**

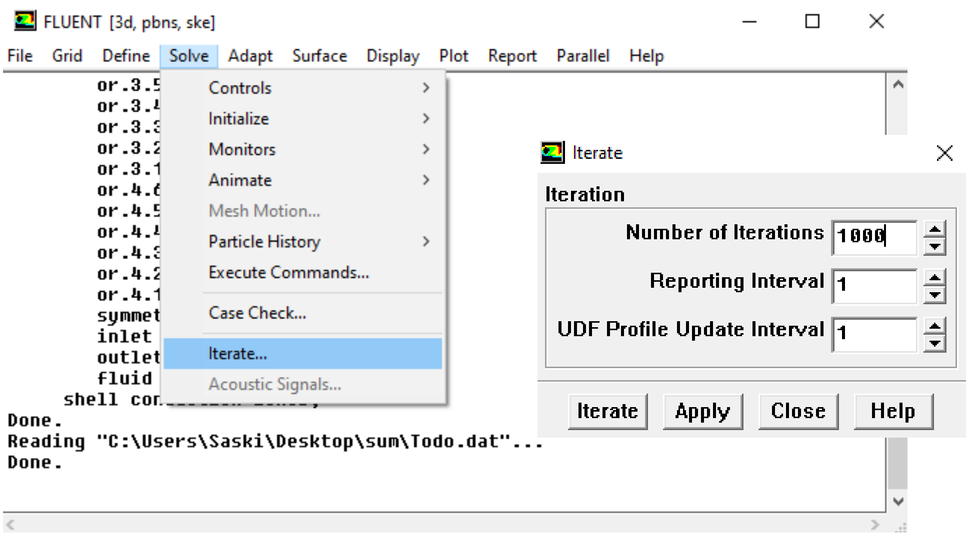


Figure III- 30: Choice of convergence criteria.

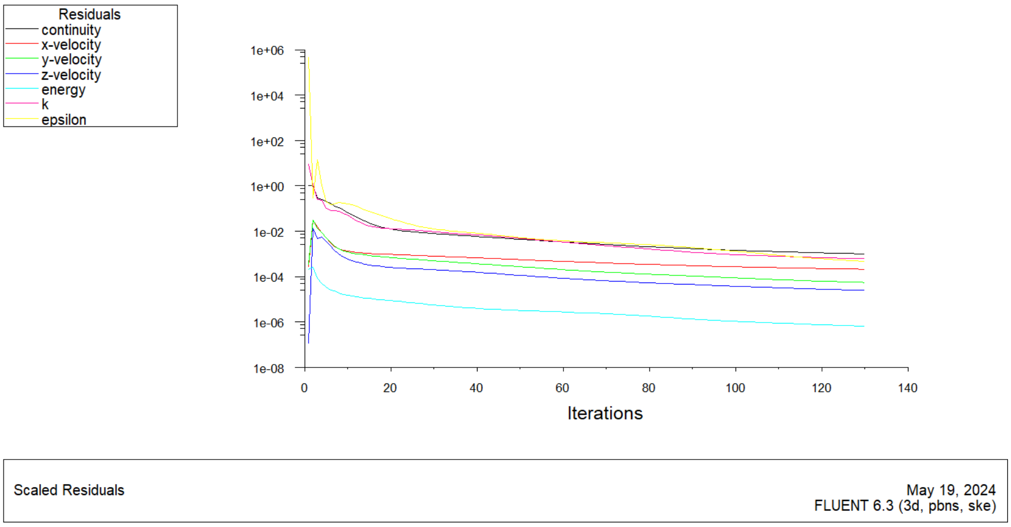


Figure III- 31: Evolution of calculation residuals.

III.4.14. Saving the file:

File ➔ **Write** ➔ **Case**

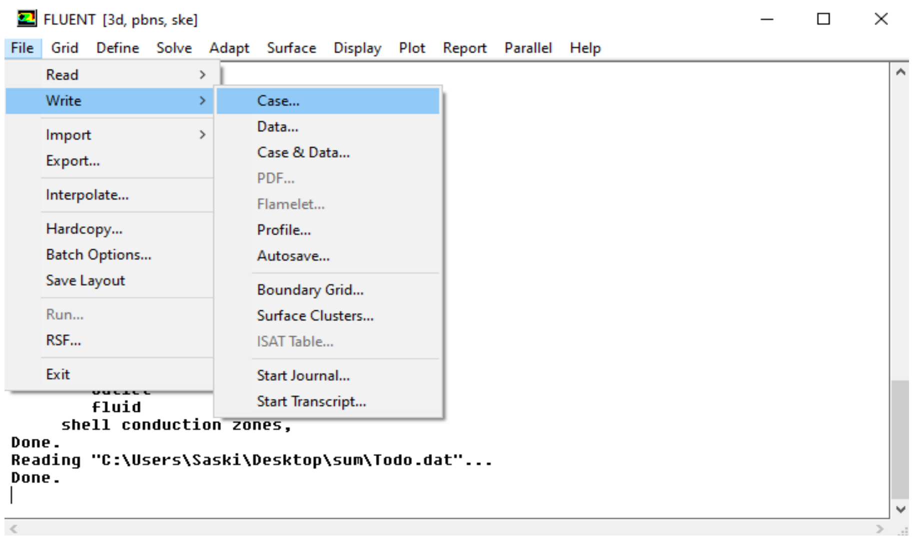


Figure III- 32: Save the file as *Cas.

Chapter IV

Results and Discussions

IV.1. Introduction:

Our work aims to carry out a 3D numerical study of the effect of the shape of the cooling orifices on the cooling film of a gas turbine blade. Film cooling is One of the most prevalent methods for cooling blades. This involves blowing a small layer of colder air over the blade surface, resulting in a protective layer.

To simulate this experience, we will use the CFD program, gambit for drawing the geometry (blade, and the orifice, mash, Boundary condition). Fluent for the calculation and display of the results, in Gambit, we are using the NACA 6512 which is a TXT file uploaded from NACA 4 Digit Airflow Generator it included all the data of our blade, after exporting the file the blade will show in 2D, so we turn it to 3d model the top face of the blade 4 rows each one has 6 orifices, 24 in total inclined by 30 °.

The advantage of the cooling film is that it forms a layer that prevents the hot gases from hitting the surface of the gas turbine guide vanes directly. The surface of the gas turbine guides the vanes, thus preventing the vanes from melting due to the very high temperature of the hot gases (around 2000K), still unfortunately, the cold air coming out of the orifices disturbs the proper flow of the hot gases, creating a compromise between cooling and adequate flow. This creates a compromise between cooling and the appropriate flow of hot air.

For the latter, we will study three studies in which we change the shape of the cooling holes (Cylindrique, Rectangular, Crescent) and compare efficiency and effectiveness in creating a protective layer for the blade

In this chapter, we’re going to discuss the results obtained from the simulation of each case, on note that the most important parameters in this study are flow velocity and temperature distribution on the top of the blade.

IV.2. Effect of turbulence model:

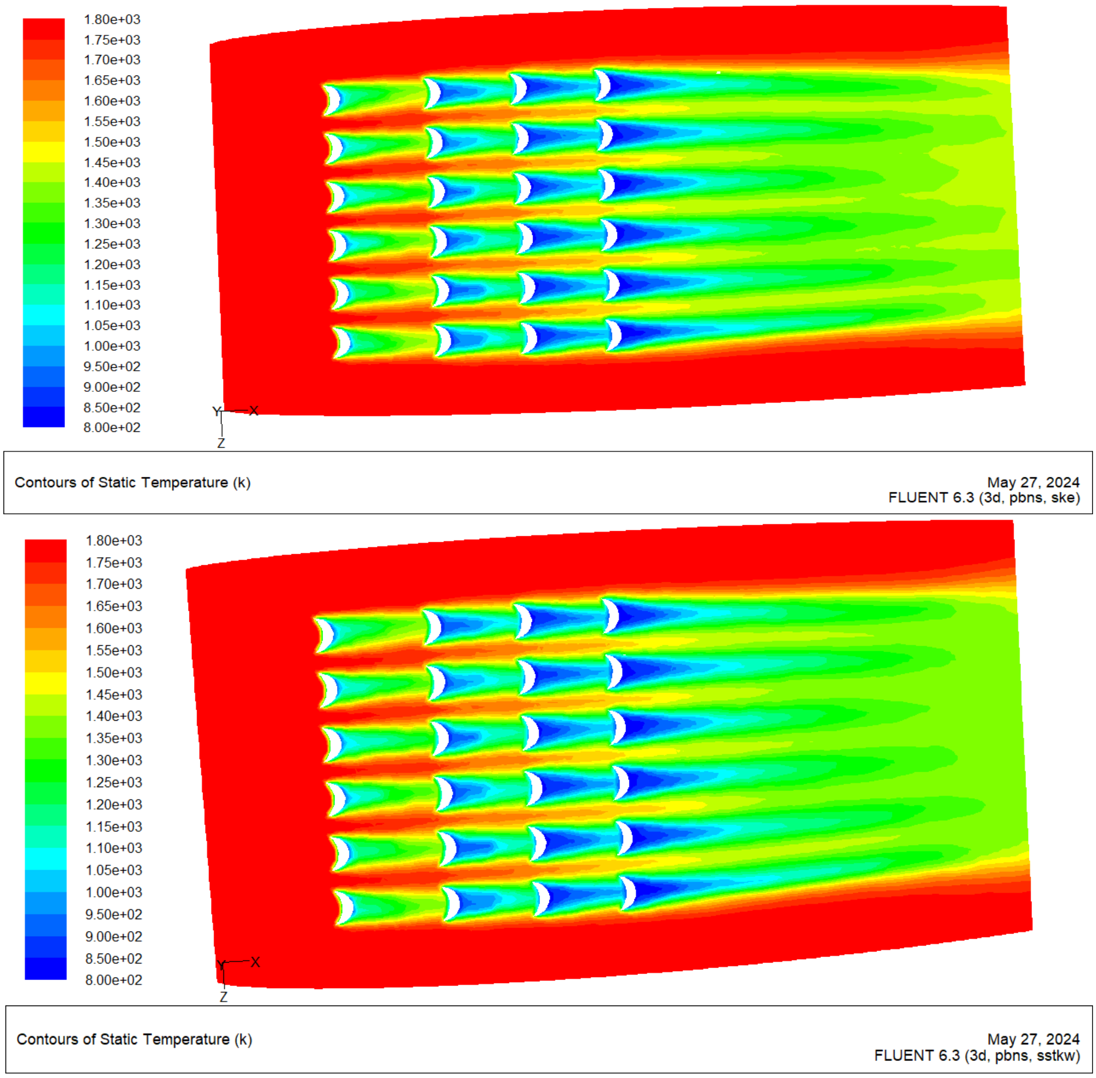


Figure IV- 1: Contours of static Temperature blowing ratio M=2

In This investigation two turbulence models have been chosen the SST and Standard K-ω, both of them are the most recommended models for turbomachinery problems better than realizable K-ω.

Fig. IV.1 presents distributions of the film cooling effectiveness on the curved surface for two different turbulence models the right side presents the k-omega SST model and the left one presents the k-epsilon standard model, it is observed that the lateral diffusion of the coolant air at the end of the blade expanded more efficiently in SST model compared to the k-epsilon standard model. Accordingly, the SST model was selected for this research.

Modèles	**Avantages**	**Inconvénients**
Spalart-Allmaras	Economique (1 equ). Bon pour les écoulements moyennement complexes.	N'est pas largement testé.
STD *k*-ε	Robuste, économique et relativement précis.	Résultats médiocre pour des écoulements complexes (fort gradient de pression, rotation et swirl).
RNG *k*-ε	Bon pour des écoulements moyennement complexes (impact de jet, séparation d'écoulements, écoulements secondaires…)	Limité par l'hypothèse de viscosité turbulente isotrope.
Realizable *k*-ε	Offre les mêmes avantages que le RNG. Recommandé dans le cas des turbomachines.	Limité par l'hypothèse de viscosité turbulente isotrope.
Reynolds Stress Model (RSM)	Le modèle le plus complet Physiquement (transport et l'anisotropie de la turbulence sont tenu en compte)	Requiert plus de temps CPU. Les équations de quantité de mouvement et turbulence sont étroitement liées.
SST et Standard *k*-ω	Modèle le plus recommandé pour les problèmes liés aux turbomachines, meilleur que le Realizable *k*-ε.	Nécessite une plus grande résolution du maillage aux frontières (pas de lois aux murs).

Table 1: Advantages and disadvantages of turbulence models

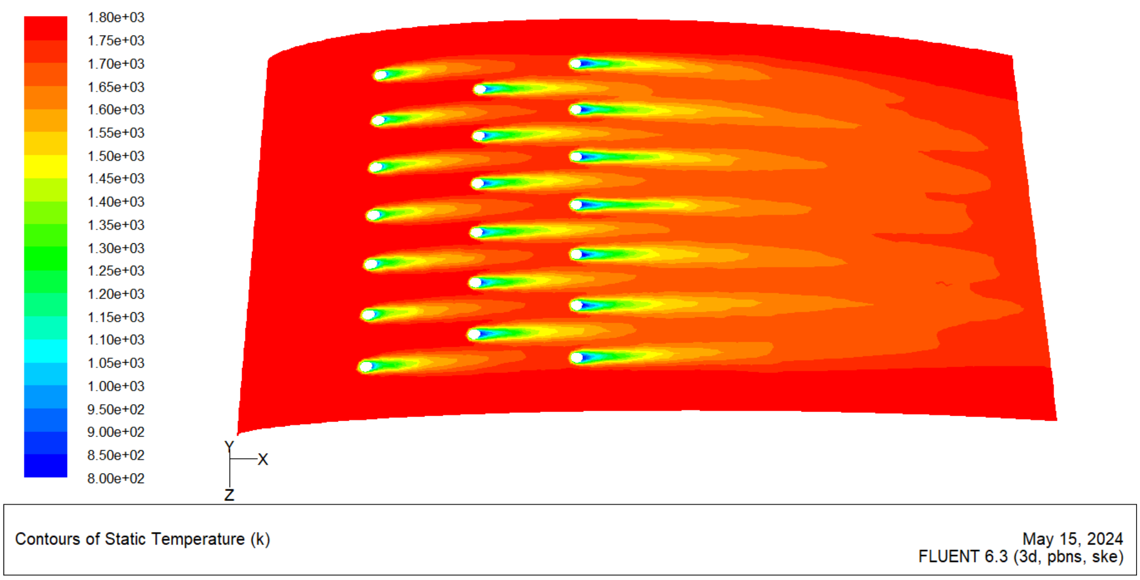
IV.3. Effect of hole configurations on film cooling effectiveness:

To investigate the effect of the hole configuration on the film cooling effectiveness, a detailed analysis is conducted for three different hole configurations with an inclination angle of 30◦ (Cylindrique, Rectangular, Crescent).

Summarize volume mesh:

Table IV- 2: Summarize the volume mesh:

	First Mesh	Second Mesh
Total nodes	64211	72993
Total elements	291461	330997



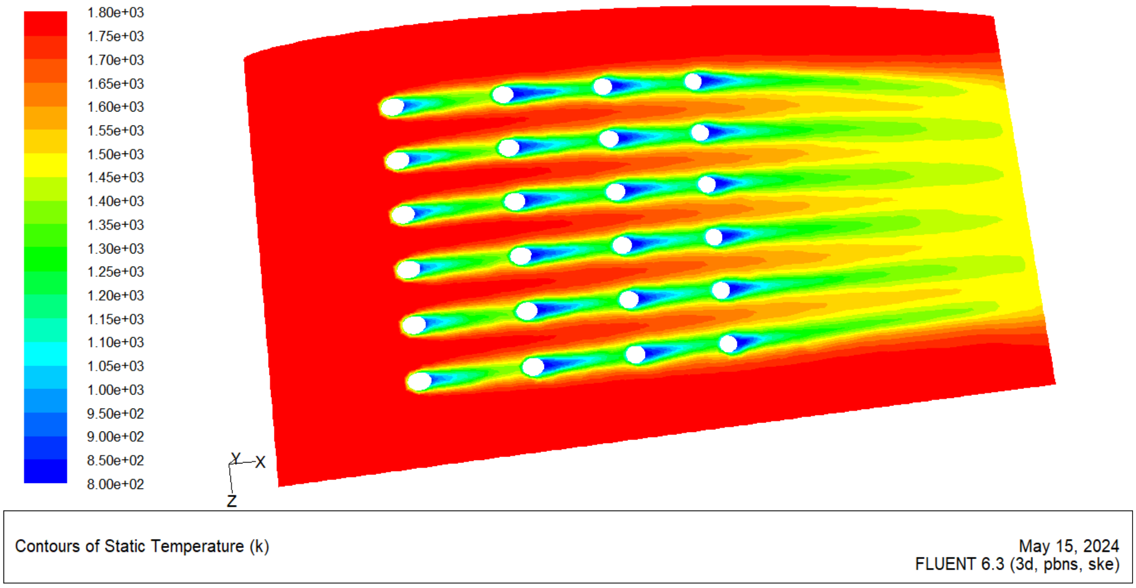


Figure IV- 2: presents distributions of the film cooling effectiveness on the curved surface for two different meshes

Fig. IV.2 presents distributions of the film cooling effectiveness on the curved surface for two different meshes. It is observed that the second mesh expands the lateral diffusion of the coolant air compared to the first one, so we'll generalize it to all cases.

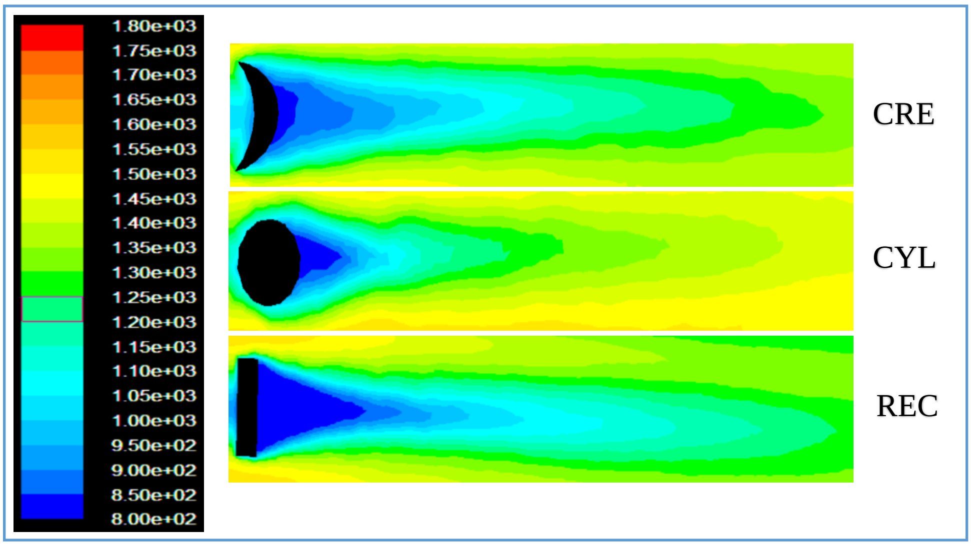


Figure IV- 3: Film cooling effectiveness distribution on the curved surface for three different hole configurations, blowing ratio M

Fig. IV.3 compares the Film cooling effectiveness distribution on the curved surface for three different hole configurations (Cylindrique, Rectangular, Crescent), with blowing ratio M = 2.

It is observed that the crescent hole and the rectangular one expand the coolant air's lateral diffusion compared to the cylindrical hole. This is so because the coverage ratio of the combined hole outlet, the crescent hole, and the Rectangular hole outlet in the lateral direction is higher than that at the cylindrical hole outlet, which causes more lateral diffusion of the coolant air. At the blowing ratio of 2.

IV.4. Effect of Blowing Ratio:

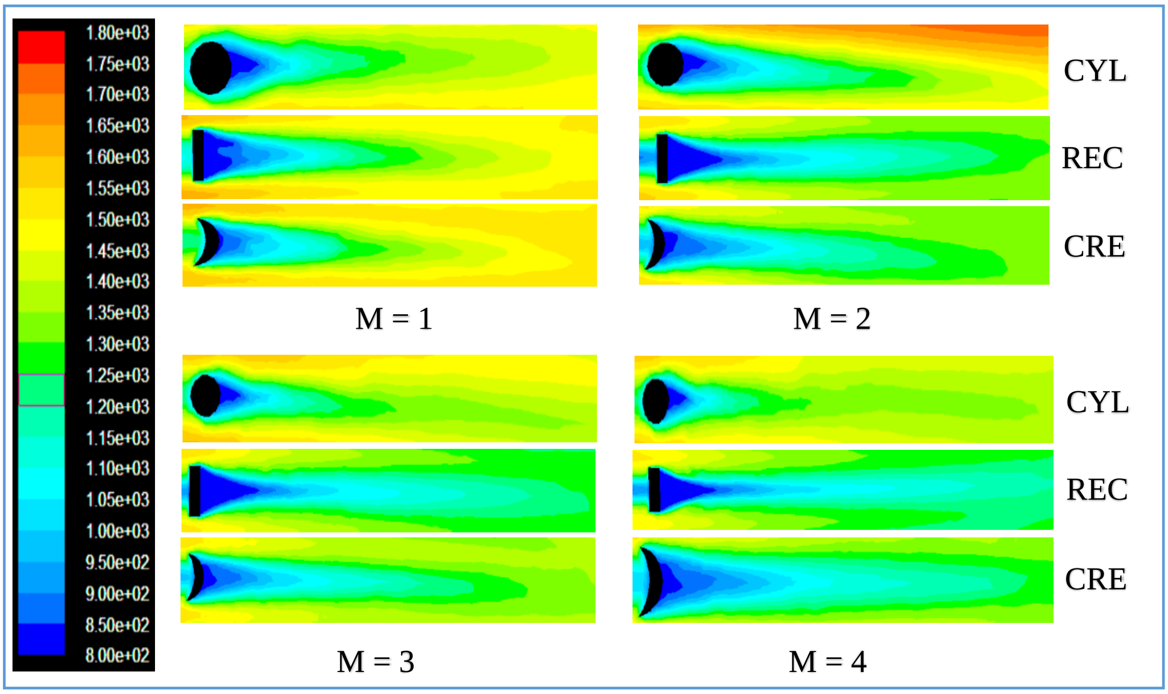


Figure IV- 4: Film cooling effectiveness distribution on the curved surface for three different hole configurations, and for different blowing ratio.

At the blowing ratio of m=1, the film cooling effectiveness near the round-to-rectangular hole is higher than that of the cylindrical hole. At the blowing ratio of m=2, the crescent hole shows the best cooling performance, but the rectangular hole shows a similar effectiveness distribution. In addition, it is indicated that the cylindrical hole has the smallest coverage area of the film cooling compared to the other hole configurations. The film effectiveness of the cylindrical holes decreases at first with the increase of blowing ratio, the case of M=2 provides the best thermal protection over the blade surface compared to all the other three blowing ratios.

Figure IV- 5 presents the Computing stations created above the surface blade for drawing the temperature and velocity curbs.

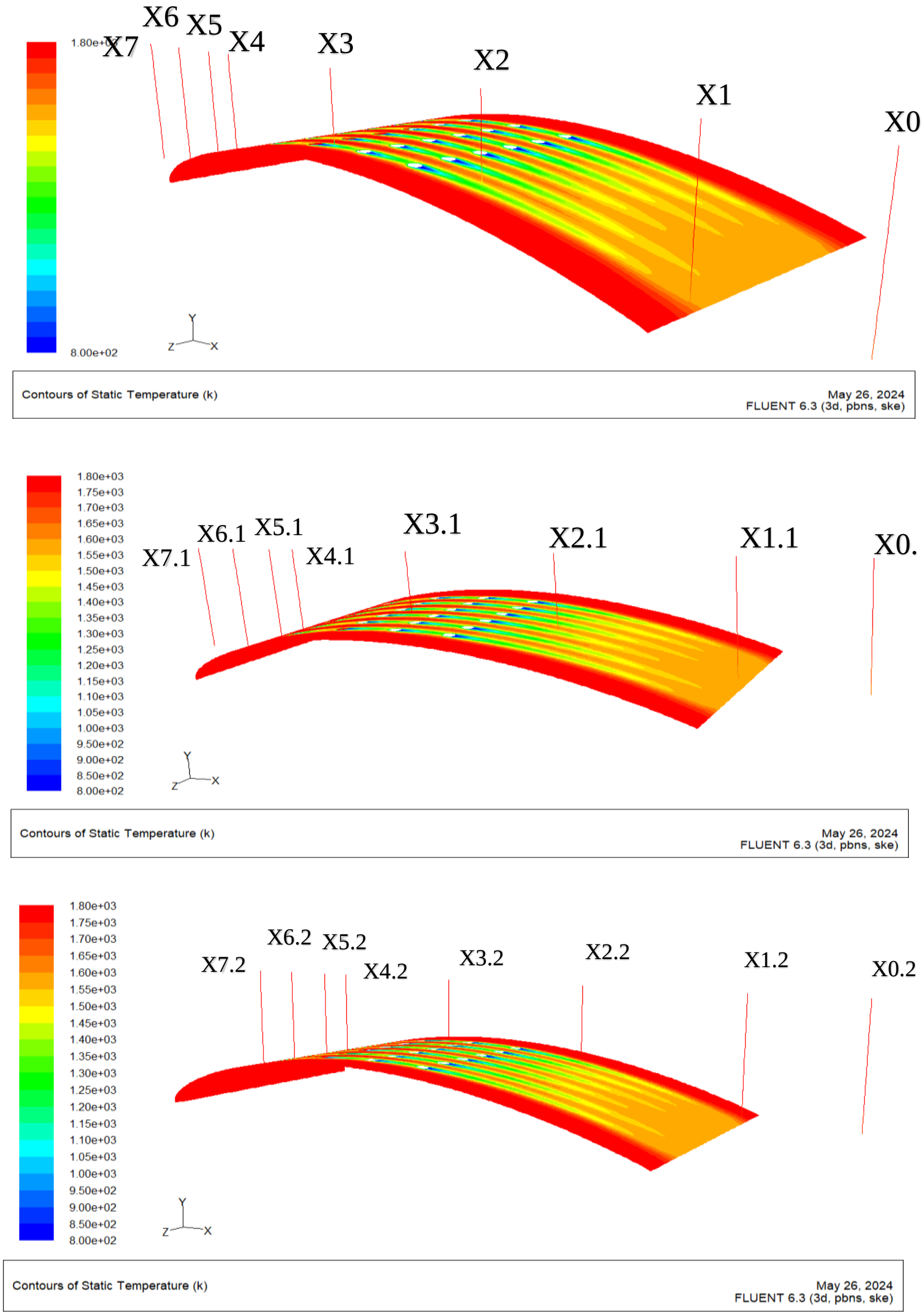


Figure IV- 6: Computing stations.

Table IV- 1: Stations coordinates:

	X	Y	Z
X0	1.2	-0.023685	-0.09375
X1	0.986908	0.00639	-0.09375
X2	0.657535	0.095076	-0.09375
X3	0.341085	0.113682	-0.09375
X4	0.062206	-0.023685	-0.09375
X5	0	-0.023685	-0.09375
X6	-0.1	-0.023685	-0.09375
X7	-0.2	-0.023685	-0.09375
X0.1	1.2	-0.023685	-0.25
X1.1	0.986908	0.00639	-0.25
X2.1	0.657535	0.095076	-0.25
X3.1	0.341085	0.113682	-0.25
X4.1	0.062206	-0.023685	-0.25
X5.1	0	-0.023685	-0.25
X6.1	-0.1	-0.023685	-0.25
X7.1	-0.2	-0.023685	-0.25
X0.2	1.2	-0.023685	-0.4375
X1.2	0.986908	0.00639	-0.4375
X2.2	0.657535	0.095076	-0.4375
X3.2	0.341085	0.113682	-0.4375
X4.2	0.062206	-0.023685	-0.4375
X5.2	0	-0.023685	-0.4375
X6.2	-0.1	-0.023685	-0.4375
X7.2	-0.2	-0.023685	-0.4375

IV.4.1. Variation in temperature at different stations along the longitudinal length Y (for x0, x2.1):

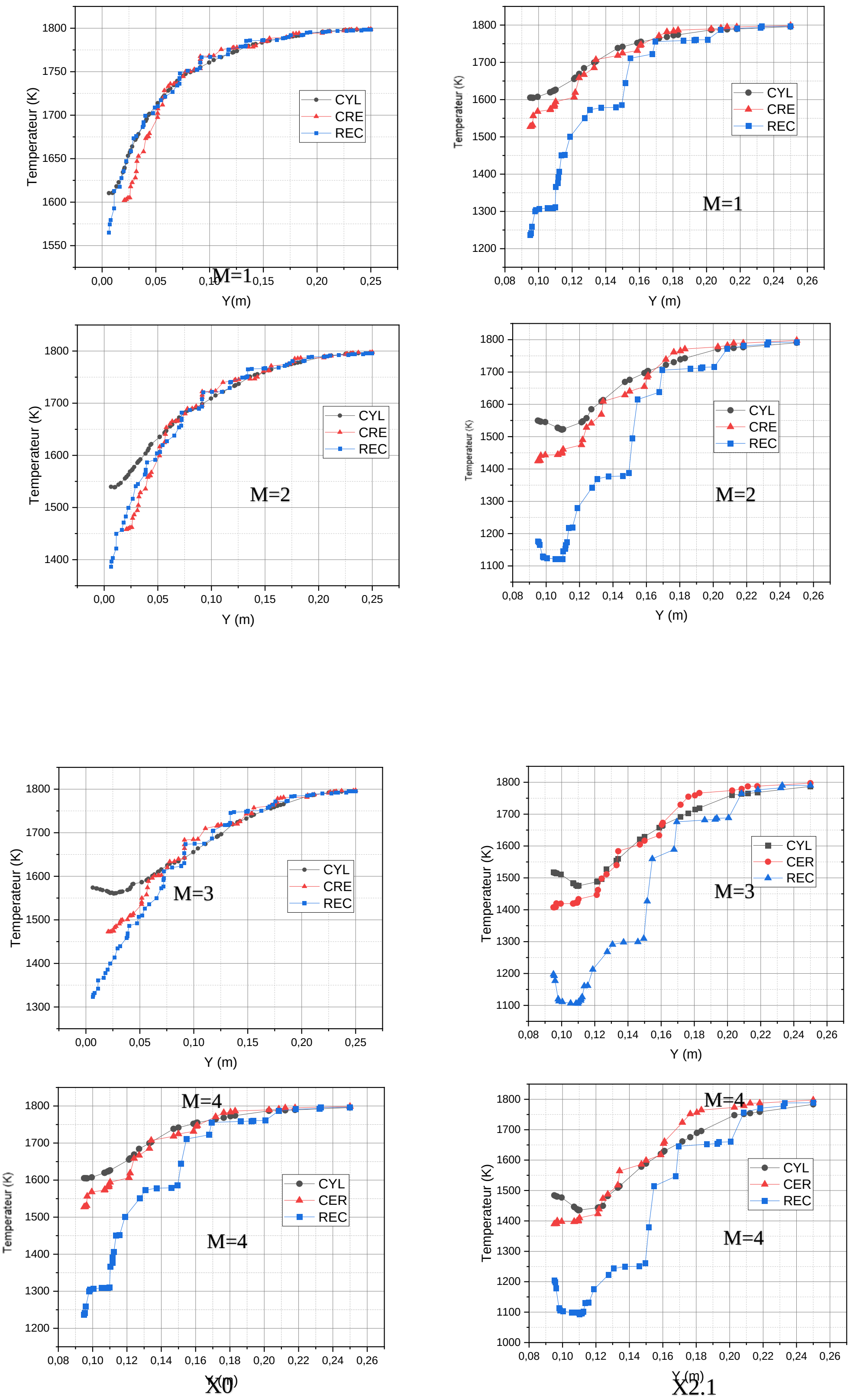


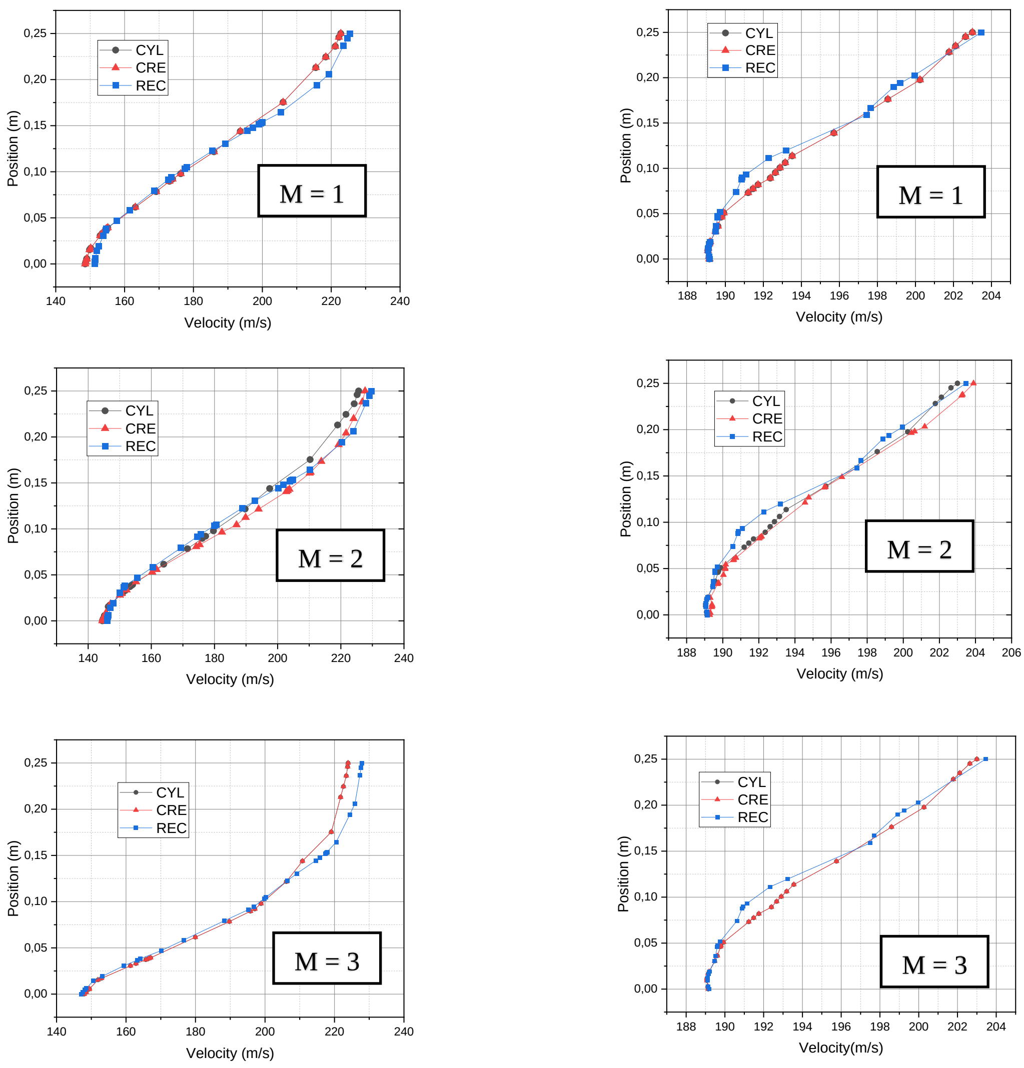
Figure IV- 7: Variation in temperature at different stations along the longitudinal length X (for x0, x2.1)

Figure IV.6 presents the Variation in temperature at different stations along the longitudinal length X at different blowing ratios, we aim to see the effect of the blowing ratio on the temperature distribution, we observe an increase in temperature each time we move away from the surface of the feather and this temperature is due to the value of the hot gases (1500).

Conversely, the temperature gradually decreases on the orifice level indicating that the blade is properly cooled. In blowing ratio M=1 we see the temperature starting from 1550 K, unlike the other cases when the temperature decreases when the blowing ratio increases, indicating that the blowing ratio affects the temperature distribution.

IV.4.2. Variation in magnitude speed as a function of Y along the longitudinal length of the blade (X):

Figure IV.7 presents the Variation in magnitude speed along the position and longitudinal length of the blade X at a different blowing ratio value, we aim to see the effect of the blowing ratio on the magnitude speed, and the results are shown below.



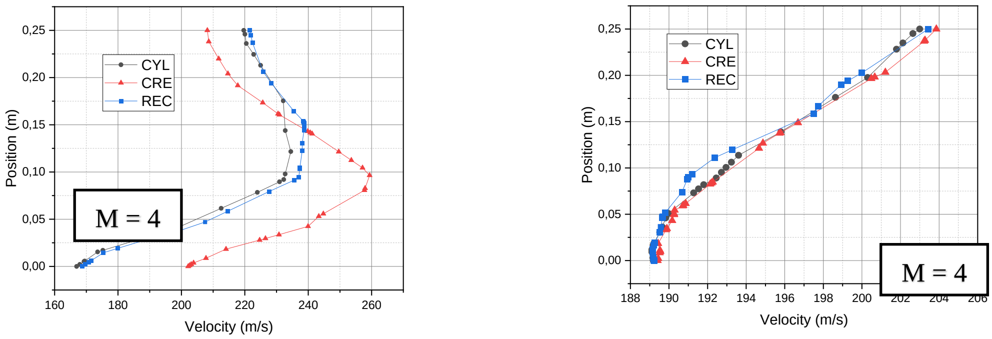


Figure IV- 8: Variation in magnitude along the position and longitudinal length of the blade X (for x0, x2.1)

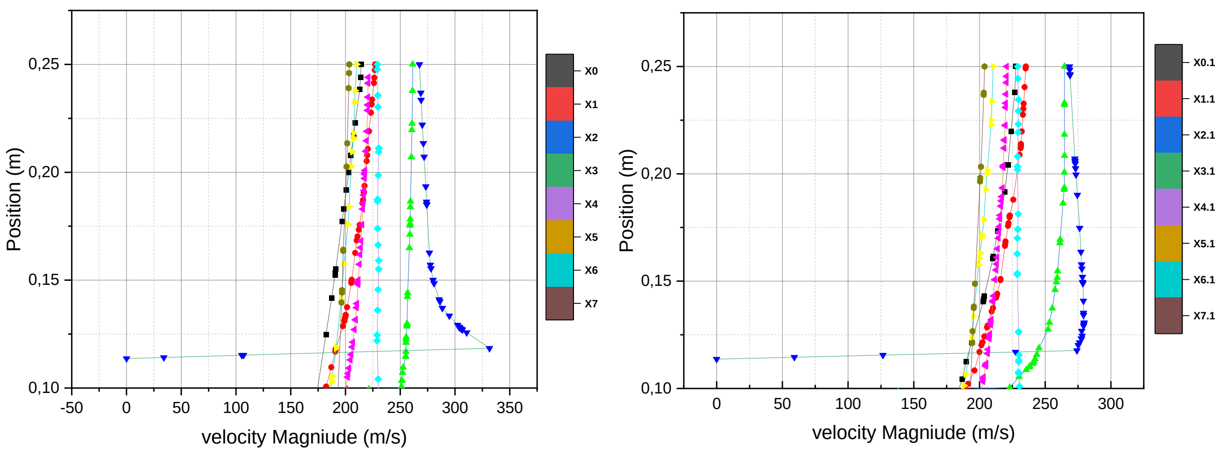
The curves indicate the behaviour of the hot and cold airflow interaction on the surface of the turbine blade. We observe an increase in velocity near the injection port, and the flow exiting the nozzles interferes with the main flow due to the downstream jet effect. The flow then returns to its normal state due to the jet mixing with the main flow of hot gases coming out of the combustion chamber.

Conversely, in M=4 the coolant quickly mixes with the hot flow, causing strong turbulence. It is clearly shown that the vortices close to the orifice not only lift a large part of the cooling flow away from the wall surface; but also bring the cooling flow to the spanwise direction. So that the area of the cooling film is enlarged, but at the same time, the velocity component of the coolant in a streamwise direction is markedly reduced. Due to the turbulence in this area, mass transfer and the convective heat increase in intensity, which expedites the mixing of the two fluids, leading to the instability of the film cooling effect. The film distribution along the streamwise direction at one time becomes more uneven.

Comparing the velocity contours at different blowing ratios, it is found that the film covers a larger area at M=2, and the corresponding cooling effect is of the highest value among the four.

IV.5. velocity contours at different stations:

❖ Crescent shape:



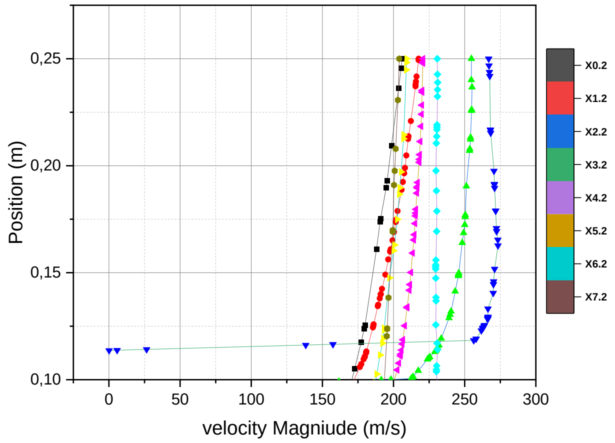


Figure IV- 9: velocity contours at different stations Crescent shape.

❖ Cylindrical shape:

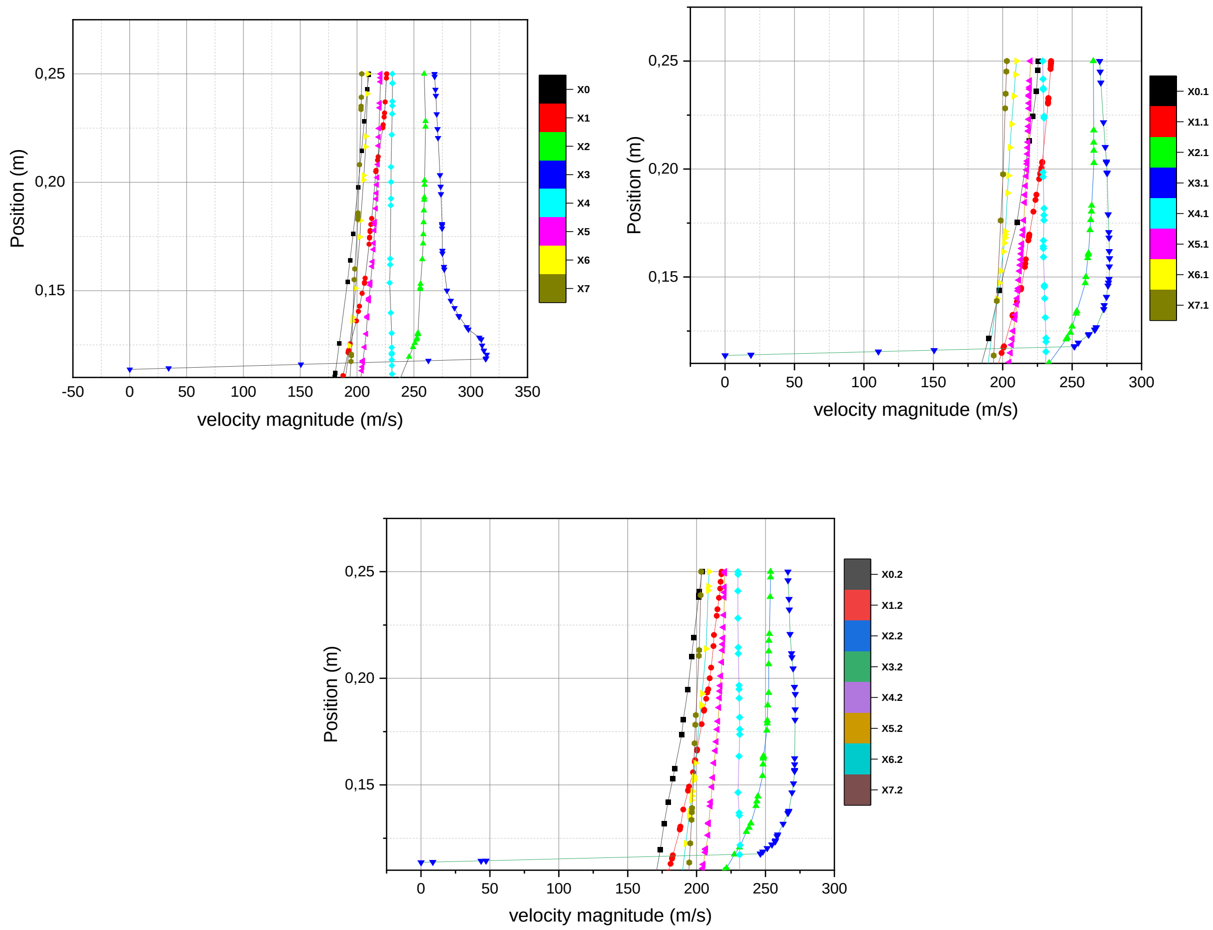


Figure IV- 10: velocity contours at different stations (Cylindrical shape).

❖ Rectangle shape

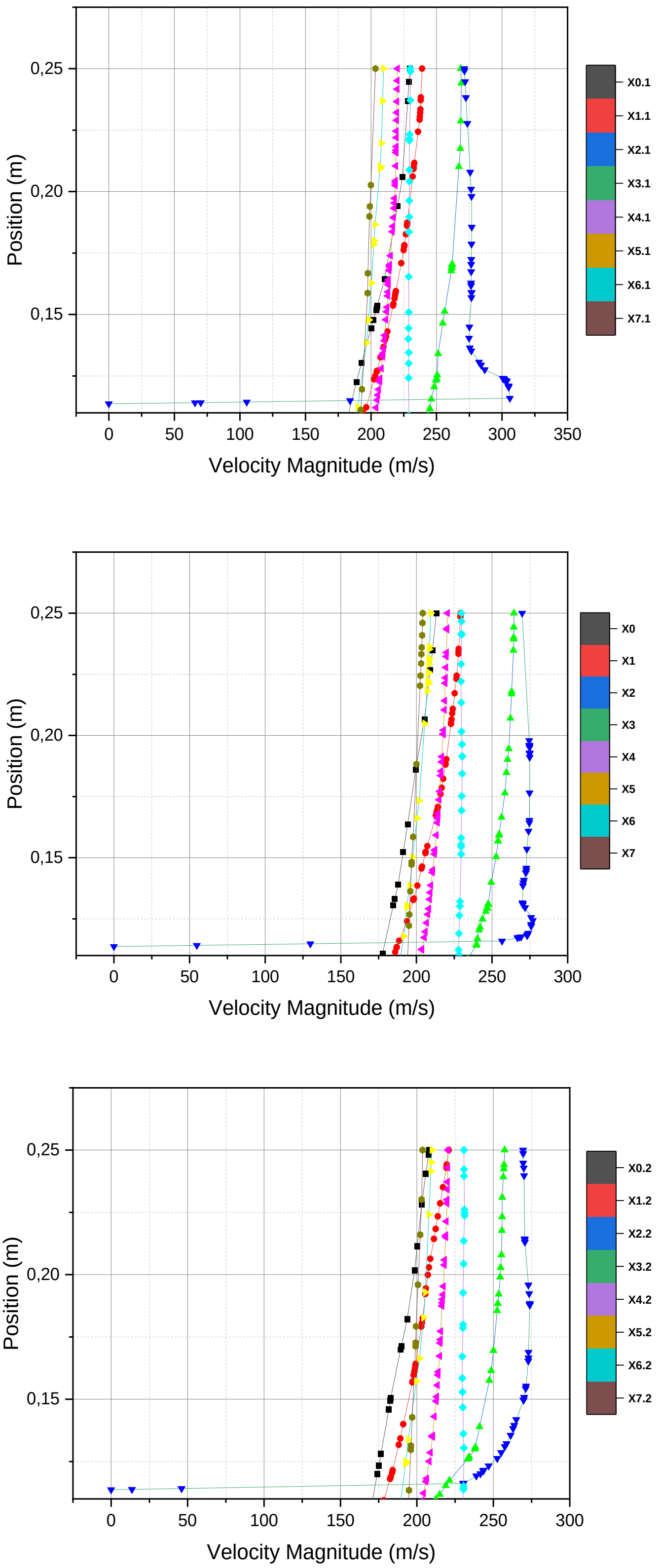


Figure IV- 11: velocity contours at different stations (Rectangle shape).

Figure IV.8, Figure IV.9, and Figure IV.10 present the velocity contours at different stations with different shapes. we aim to see the effect of spacing on the distribution of the velocity magnitude. where there is almost the same distribution at the various stations, where the speed of the cooling airflow increases significantly close to the openings, then the flow starts to become normal with increasing dimension(x).

IV.6. pathlines:

Pathlines are the lines traveled by neutrally buoyant particles in equilibrium with the fluid motion. Pathlines are an excellent tool for the visualization of complex three-dimensional flows. We are using the Tecplot program to display the results after exporting the file from Fluent.

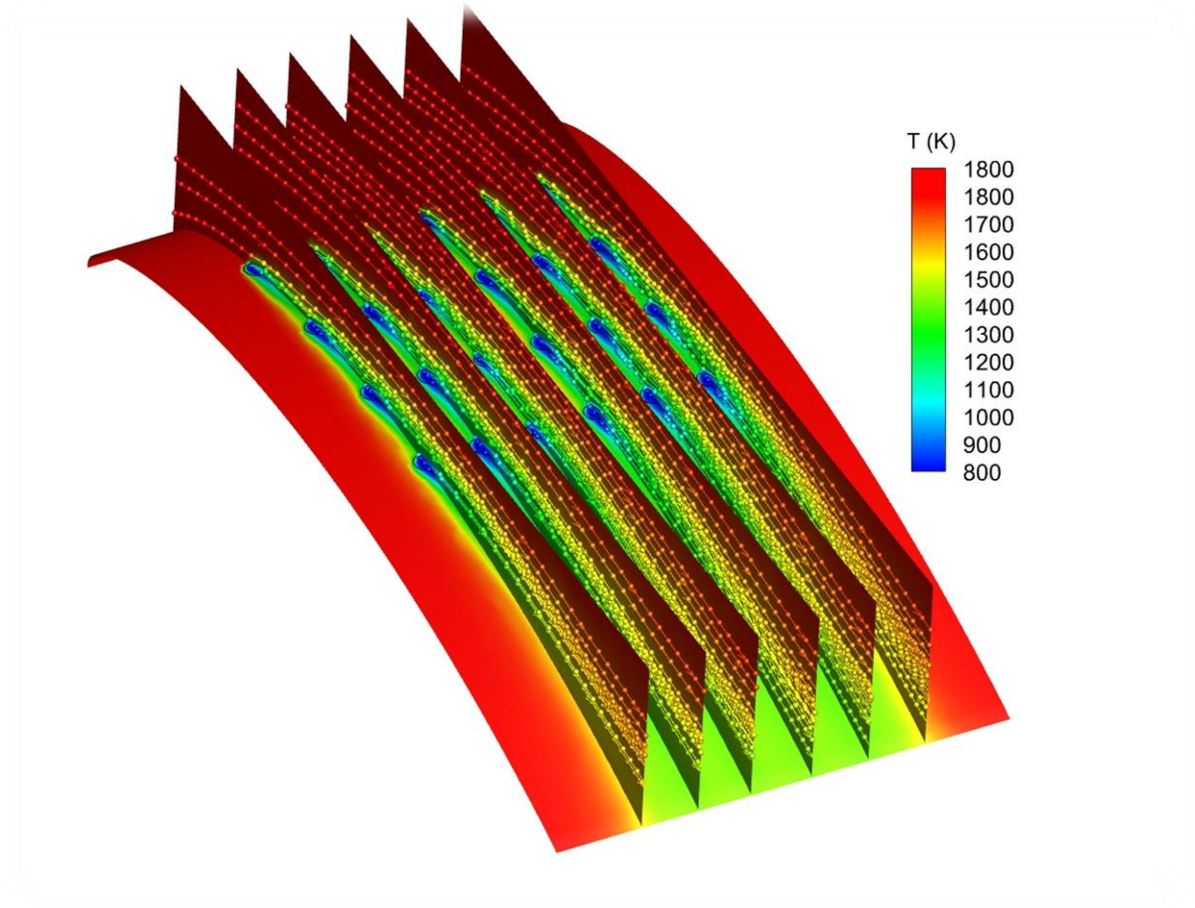


Figure IV- 12: Pathlines using Tecplot (Cylindrical shape 30°).

IV.6.1. Inclination angle:

As tangential coolant injection is challenging to accomplish in real-world applications, different coolant incidence angles are therefore thought to be essential for film cooling.

when the blowing ratio is 2.0 two incidence angles (30° and 60°) are taken as an example to investigate the relationships of the incident angles with the film cooling effect.

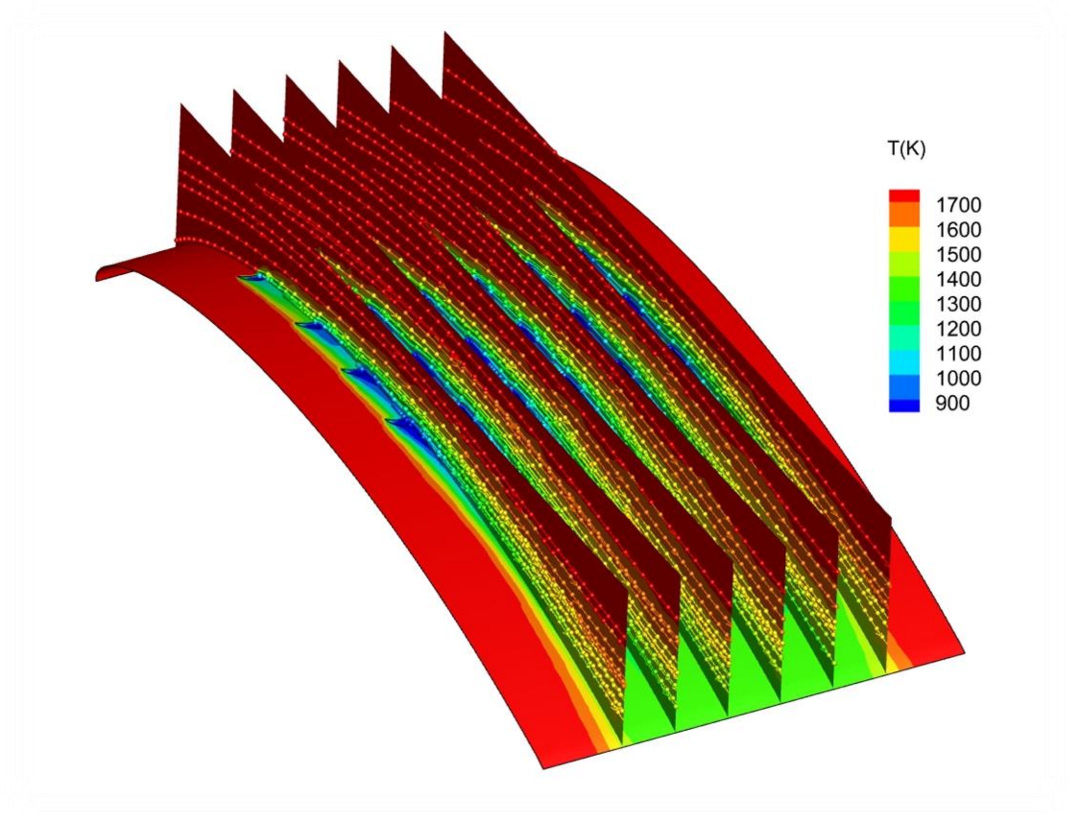


Figure IV- 13: Pathlines using Tecplot (30°).

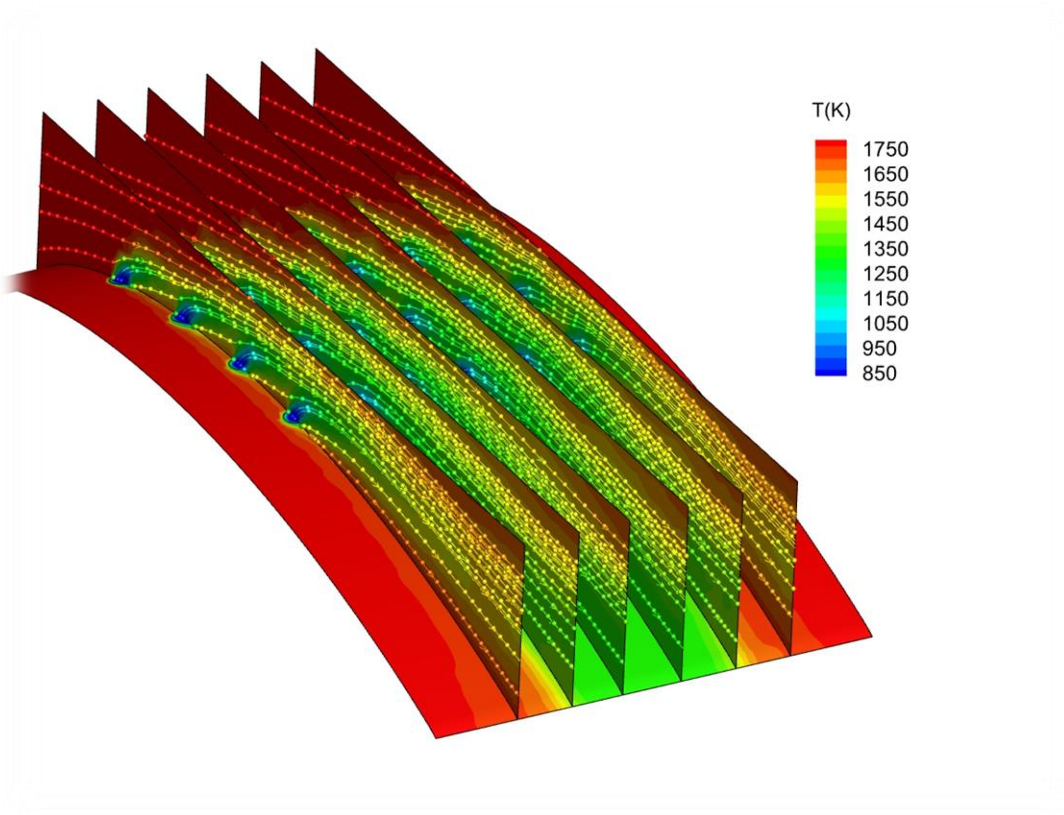


Figure IV-13: Pathlines using Tecplot (60°).

Figure.IV.13 indicates the path lines in the central cross-section of the instantaneous results of 30° incidence after the film has stabilized. The blue area represents the coolant, after the computational domain is injected with coolant, the mainstream with high temperature having a certain speed carries a large part of the injected cooling flow upward, blowing the cooling jet away from the wall and mixing with the most cooling flow. And counter-rotating vortex pair is pulling the coolant out of the cooling channel and spreading the cooling jet upstream and laterally. What can be identified from the bottom section of temperature distribution is that as the distance along the downstream increases, the wall temperature gradually increases. Additionally, as the jet develops downstream, the spanwise coverage area is widened.

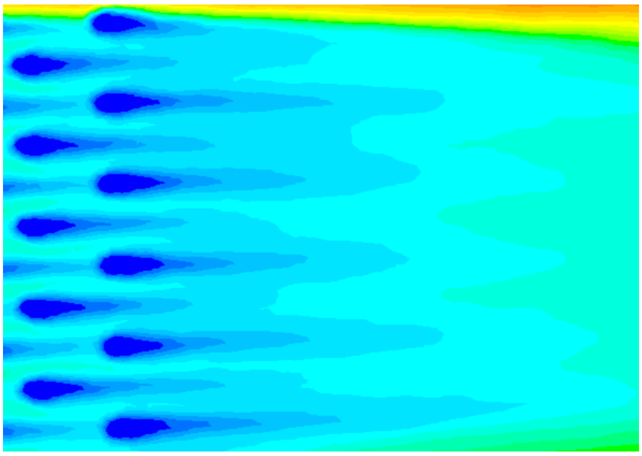
Figure.IV.12 shows that the coolant is enveloped by counter-rotating vortex pairs to cause the jet's upward momentum to grow when the inclination angle is increased to 60°. No cooling film would be developed at the bottom surface due to this lifting itself away from the wall and the separation area expanding.

Comparing the pathlines at different inclination angles, it is found that the film covers a larger area at the angle of 30°, and the corresponding cooling effect is of the highest value among the two. When the inclination angle increases to 60°, the jet velocity component of the cooling flow decreases and there is almost no wall adhesion.

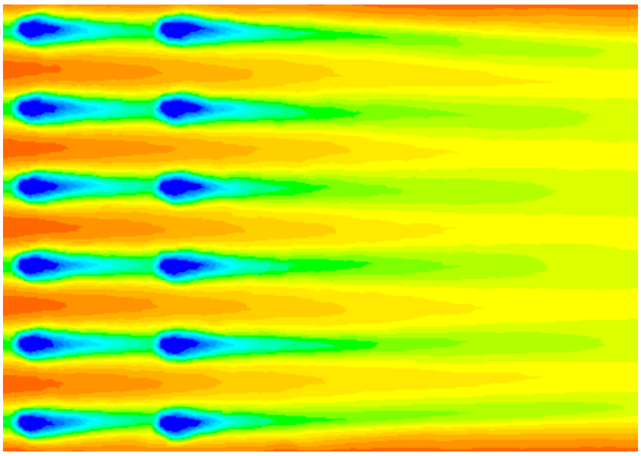
IV.7. Hole arrangement:

In practical application, the whole process of film cooling is often accomplished in the form of porous and a certain hole arrangement, to achieve a better lateral cooling effect.

Additionally, the single row of holes can be added to double rows, and the double rows can be subdivided into two kinds of arrangement methods: alignment and plug. The film cooling effect of double rows of holes including alignment and plug are compared in Figure.IV.14:



a) Double rows of plug-arranged hole



b) Double rows of alignment arranged hole

Figure IV- 14: The temperature distribution and the comparison of film cooling effect of various hole methods.

In the cooling dead zone, the incidence of plug double rows performs better, on average. (the dead zone is a phenomenon that happens in the near-field region where the temperature at the bottom surface is higher than that of the far-field area, inducing low film cooling efficiency). Then those of the alignment double-row incidences, that is because the gap between the holes of plug double-row incidence is filled by the holes from the back row to ensure sufficient gas momentum.

IV.8. General Conclusion:

The film cooling effect of different hole shapes is numerically simulated in this memory. The results show that, within the scope of this investigation, among the three-hole shapes (Cylindrique, Rectangular, Crescent), At the blowing ratio of M=2, the crescent hole shows the best cooling performance. Still, the rectangular hole shows a similar effectiveness distribution. In addition, it is indicated that the cylindrical hole has the smallest coverage area of the film cooling compared to the other hole configurations.

According to the film cooling results under different blowing ratios, it can be appreciated that as the blowing ratio increases, the film coverage area of the near field is narrower, and the cooling efficiency appears a downward trend. So, the conclusion is conducted that the condition at M= 2 performs the best among the four.

For the effect of inclination angle on cooling performance, the film cooling effectiveness of the three different hole configurations at various inclination angles (30◦,60°), the angle of 30° accorded with the best film cooling effect.

As for the two arrangements of the holes (double-row (alignment), double-row (plug)), the cooling effect is the finest when the double-row plug configuration of holes is injected. For the back row of the holes can effectively fill the gap between the holes to ensure sufficient gas momentum in this kind of configuration.

References

[1]: Members, C., Lau, S. C., Morrison, G. L., & Chen, H. C. (2009). EXPERIMENTAL STUDY OF GAS TURBINE BLADE FILM COOLING AND HEAT TRANSFER.

[2]: DrHan. (2004). SHAPED HOLE EFFECTS ON FILM COOLING EFFECTIVENESS AND A COMPARISON OF MULTIPLE EFFECTIVENESS MEASUREMENT TECHNIQUES.

[3]: Choi, J. H., Mhetras, S., Han, J. C., Lau, S. C., & Rudolph, R. (2008). Film cooling and heat transfer on two cutback trailing edge models with internal perforated blockages. Journal of Heat Transfer, 130(1). https://doi.org/10.1115/1.2780174

[4]: Boyce, M. P., Oxford, B., Melbourne, J., & Delhi, N. (2006). Gas Turbine Engineering Handbook Third Edition. http://elsevier.com

[5]: NESRAOUI ILIAS, "ANALYSE ET SIMULATION DE REFROIDISSEMENT DES AUBES D'UNE TURBINE A GAZ. "Master thesis, (2015).

[6]: KLAA EFTIKHAR, "Conception d'une aube de turbine à gaz en matériaux composites." MAGISTÈRE. thesis, UNIVERSITÉ DE BATNA, 2015.

[7]: DJOUDI YOUCEF, "Simulation numérique d'une chambre de combustion d'une turbine à gaz MS 5002B." Mémoire master, Université de Biskra, 2013.

[8]: Rolls-Royce Ltd. (2005). The jet engine. Rolls-Royce.

[9]: Introduction to Gas Turbine Theory (Klaus Brun, (Z-Library). (n.d.).

[10]: elsayed_af_aircraft_propulsion_and_gas_turbine_engines. (n.d.).

[11]: Wang, J., Zhao, Z., Tian, L., Ren, X., & Sundén, B. (2021). Effects of hole configuration on film cooling effectiveness and particle deposition on curved surfaces in gas turbines. Applied Thermal Engineering, 190.

[12]: Han, J.-C. (2004). Recent Studies in Turbine Blade Cooling? The International Journal of Rotating Machinery, 10(6), 443–457. https://doi.org/10.1080/10236210490503978

[13]: Gas Turbine Heat Transfer and Cooling Technology. (n.d.).

[14]: Han, J. C., & Teng, S. (2000). Effect of Film-Hole Shape on Turbine Blade Film Cooling Performance. http://www.sti.nasa.gov

[15]: Abdelghany, E. S., El-Sayed, A. F., Fouad, M. A., & Khalil, E. E. (n.d.). EFFECT OF HOLE-SHAPED ON FILM COOLING EFFECTIVENESS OF GAS TURBINE BLADE.

[16]: Qadir Talal, A., & Rahman, F. (2020). Design and Analysis of a Gas Turbine Blade. International Research Journal of Engineering and Technology. www.irjet.net

[17]: Xing, H.; Du, W.; Sun, P.; Xu, S.; He, D.; Luo, L. Influence of surface curvature and jet-to-surface spacing on heat transfer of impingement cooled turbine leading edge with crossflow and dimple. Int. Commun. Heat Mass Transf. 2022, 135. [CrossRef]

[18]: Gas Turbine Heat Transfer and Cooling Technology, Second Edition (Dutta, Sandip Ekkad, Srinath Han, Je-Ch

[19]: Xu, L., Wang, W., Gao, T., Shi, X., Gao, J., Liang, W. (2014). Experimental study on cooling performance of a steam-cooled turbine blade with five internal cooling smooth channels. Experimental Thermal and Fluid Science,

[20]: Gaikwad, S.S., Sonawane, C.R. (2014). Numerical simulation of gas turbine blade cooling for enhancement of heat transfer of the blade tip. IJRET: International Journal of Research in Engineering and Technology, 3(9): 35-41.

[21]: Xue, S., Ng, W.F. (2018). Turbine blade tip external cooling technologies. Aerospace, 5(3): 90. https://doi.org/10.3390/aerospace5030090

[22]: Xue, S., Ng, W.F. (2018). Turbine blade tip external cooling technologies. Aerospace, 5(3): 90.

[23]: Skamniotis, C.G.; Cocks, A.C.F. Designing against severe stresses at compound cooling holes of double wall transpiration cooled engine components. Aerosp. Sci. Technol. 2021, 116.

[24]: Choi, J. H., Mhetras, S., Han, J. C., Lau, S. C., & Rudolph, R. (2008). Film cooling and heat transfer on two cutback trailing edge models with internal perforated blockages. Journal of Heat Transfer,

[25]: Ahmed el makkky, "ANSYS CFX Flow Modelling in a Gas Turbine Combustion Chamber." First edition.(2012).

[26]: New Concepts in Film Cooling for Turbine Blades. (n.d.). www.nasa.gov

[27]: Dutta, S., Kaur, I., & Singh, P. (2022). Review of Film Cooling in Gas Turbines with an Emphasis on Additive Manufacturing-Based Design Evolutions. In Energies (Vol. 15, Issue 19). MDPI. https://doi.org/10.3390/en15196968

[28]: Chang, J., Xu, J., Duan, X., Wang, K., & Du, Y. (2021). Investigations of effect for different influencing factors on film cooling effectiveness—Steady coolant ejected. Energy Reports, 7, 1453–1465. https://doi.org/10.1016/j.egyr.2021.09.097

[29]: Mme BAKHTI Fatima Zohra, MODULE: CFD et logiciels, Niveau: S3 Master Energétique, Université de M'sila.