

T.R.

GEBZE TECHNICAL UNIVERSITY

GRADUATE SCHOOL

**AEROTHERMAL PERFORMANCE OF ROTATING TURBINE
COOLING FEATURES**

İBRAHİM EDHEM TİN

A THESIS OF MASTER OF SCIENCE

DEPARTMENT OF ENERGY TECHNOLOGIES

**IN APPLIED PROPULSION SYSTEM DESIGN & ENGINEERING
FOR AEROSPACE TECHNOLOGIES PROGRAM**

ADVISOR: DR. ERİNÇ ERDEM

JULY 2024

T.R.

GEBZE TECHNICAL UNIVERSITY

GRADUATE SCHOOL

**AEROTHERMAL PERFORMANCE OF ROTATING
TURBINE COOLING FEATURES**

İBRAHİM EDHEM TIN

A THESIS OF MASTER OF SCIENCE

DEPARTMENT OF ENERGY TECHNOLOGIES

**IN APPLIED PROPULSION SYSTEM DESIGN &
ENGINEERING FOR AEROSPACE TECHNOLOGIES
PROGRAM**

ADVISOR: DR. ERİNÇ ERDEM

JULY 2024

T.C.
GEBZE TEKNİK ÜNİVERSİTESİ
LİSANSÜSTÜ EĞİTİM ENSTİTÜSÜ

DÖNER TURBİN SOĞUTMA YAPILARININ
AEROTHERMAL PERFORMANSI

İBRAHİM EDHEM TIN

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

DANIŞMAN: DR. ERİNÇ ERDEM

TEMMUZ 2024



MASTER of SCIENCE JURY APPROVAL FORM

A thesis submitted by İbrahim Edhem Tın, defended on 10/07/2024 before the jury formed with the 08/07/2024 date and 2024/35 numbered decision of the GTU Graduate Administration Board, has been accepted as a MASTER of SCIENCE thesis in the Department of Energy Technologies, Applied Propulsion System Design Engineering for Aerospace Technologies Program.

JURY

MEMBER

(THESIS ADVISOR) : Dr. Erinç ERDEM

MEMBER : Prof. Dr. İlyas Kandemir

MEMBER : Prof. Dr. Kürşad Melih GÜLEREN

APPROVAL

Gebze Technical University Graduate Administration Board

/ / date and / numbered decision.

SIGNATURE/SEAL

ABSTRACT

Turbine cooling is one of the most important technologies in today's gas turbine engines. This technology, which has been integrated into gas turbine engines almost 80 years ago, has been continuously developed for many years. Rib turbulator cooling structures have been one of the most important structures of this technology in the past and today.

In this study, the results of a wide-known study about rib turbulators under stationary and rotary conditions with PIV and TLC measurement in the literature are compared with the RANS CFD results and the findings are reported.

In Chapter 1, the history and critical milestones of gas turbine engines are explained. After this, the operating principles of gas turbine engines and Brayton Cycle are explained. This chapter is concluded by explaining the importance of turbine cooling and its development history.

In Chapter 2, an extensive literature review is presented and the fluid physics in rib turbulator structures is explained case by case through studies. In this section, where basic physics is explained, the effect of design changes in rib turbulator structures on rib performance is also revealed.

In Chapter 3, the base case is explained in detail. The test rig and test campaign are explained. Flow field and heat transfer results are shown. Test geometry and measurement methods are explained.

In Chapter 4, numerical approaches and solutions are shown. The CFD program used, geometry, grid generation and turbulence models are explained respectively. In addition, solver setup, boundary conditions and analysis matrix are shown. Finally, the analysis results in stationary and rotary cases were compared with the test.

In Chapter 5, the results are summarized and discussed.

As a result of the study, it was stated that caution should be exercised in solving reattachment problems with RANS CFD models. Test and CFD results are compatible with each other with constant margins under stabilized rotary and stationary conditions. However, in the case of destabilized rotary, the results vary considerably depending on the CFD setup.

Keywords: Turbine Cooling, Heat Transfer, Computational Fluid Dynamics (CFD), Rib Turbulator

ÖZET

Türbin soğutma bugünün gaz türbinli motorlarındaki en önemli teknolojilerden biridir. Gaz türbinli motorlara yaklaşık 80 sene önce entegre edilen bu teknoloji uzun yıllardır geliştirilmeye devam etmekte. Rib türbülator soğutma yapısı ise bu teknolojinin en önemli parçalarından birisi.

Bu çalışmada, literatürde genişçe kabul edilmiş, sabit ve döner koşulda PIV ile TLC ile ölçümler alınmış bir çalışmanın test dataları ile mevcut çalışma kapsamında yapılan RANS CFD analizlerinin sonuçları karşılaştırılmış ve bulgular raporlanmıştır.

Bölüm 1’de, gaz türbinli motorların tarihsel gelişimi anlatılmıştır. Sonrasında gaz türbinli motorların çalışma prensipleri ve bu motorların çalışma çevrimi olan Brayton Çevrimi aktarılmıştır. Türbin soğutmanın tarihçesi ve önemi de anlatılarak bu bölüm sonlandırılmıştır.

Bölüm 2’de, literatür taraması detaylıca aktarılmıştır. Rib türbülatorlerdeki akış fiziği farklı çalışmalar vasıtasıyla aktarılmıştır. Bu bölümde rib türbülator tasarım parametrelerinin rib performansına etkisi de incelenmiştir.

Bölüm 3’de, baz çalışma detaylıca açıklanmıştır. Test sistemi ve test kampanyası aktarılmıştır. Akış alanı ve ısı transferi test sonuçları gösterilmiştir. Test geometrisi ve ölçüm yöntemleri de anlatıldıktan sonra bu bölüm bitirilmiştir.

Bölüm 4’de, numerik modelleme yaklaşımları ve sonuçlar gösterilmiştir. Kullanılan CFD programı, geometri, mesh ve türbülans modelleri sırasıyla açıklanmıştır. Bunlara ek olarak çözücü ayarları, sınır koşulları ve analiz matriksi de bu bölümde yer almaktadır. Son olarak da durağan ve döner koşuldaki analiz sonuçları ile baz çalışmadaki test sonuçları karşılaştırılmıştır.

Bölüm 5’te, çalışmanın sonuçları özetlenmiş ve tartışılmıştır.

Çalışmanın özeti olarak, yeniden tutunma problemlerinde RANS CFD modellemelerinde dikkatli olunması gerektiği aktarılmıştır. Analiz ve test sonuçları durağan ve stabil döner koşullarda, belli bir marjin ile birbiri ile uyumlu çıkmıştır. Bunla birlikte stabil olmayan döner koşulda sonuçlar analiz ayarlarına bağlı olarak oldukça değişmektedir.

Anahtar Kelimeler: Türbin Soğutma, Isı Transferi, Hesaplamalı Akışkanlar Dinamiği (HAD), Rib türbülator

ACKNOWLEDGEMENTS

This thesis work cost me a lot of extra working hours and weekends. I would like to thank my one and only wife, Esma, for her unwavering support even though she had to stay at home for months with me because of these extra hours. Without her support this work would not have been possible.

I would like to thank my father, who spent years of effort for my education, and my mother, who was always by my side and supported me.

I would like to thank all the instructors who increased our awareness and expertise about gas turbine engines throughout the graduate education. I would also like to thank Mr. Orçun Kor and Mr. Erinç Erdem, who were among these instructors. I learned a lot from both of them in terms of management and technical depth.

I am grateful to Mr. Erinç Erdem for his contributions and guidance during my thesis work and the almost 4-year period I spent at TEI.

I would like to thank TEI for the work opportunities and family environment it offered me.

TABLE OF CONTENTS

	<u>Page</u>
ABSTRACT	v
ÖZET	vi
ACKNOWLEDGEMENTS	vii
TABLE OF CONTENTS	viii
LIST OF SYMBOLS AND ABBREVIATIONS	x
LIST OF FIGURES	xi
LIST OF TABLES	xiv
1. INTRODUCTION	14
1.1. History of Gas Turbines	14
1.2. How It Works	16
1.3. Working Cycle	17
1.4. Usage Applications	19
1.5. Importance Of Turbine Blade Cooling	20
2. LITERATURE REVIEW AND THEORETICAL BACKGROUND	25
2.1. Effects of Rib Layouts and Flow Parameters on Ribbed-Channel Heat Transfer	27
2.1.1. Effect of Rib Spacing on the Ribbed and Adjacent Smooth Sidewalls	27
2.1.2. Angled Ribs	28
2.1.3. Effect of Channel Aspect Ratio with Angled Ribs	31
2.2. Comparison of Different Angled Ribs	32
2.3. High-Performance Ribs	34
2.3.1. V shaped ribs	34
2.3.2. V-Shaped Broken Rib	37
2.4. Effect of High Blockage-Ratio Ribs	38
2.5. Effect of Number of Ribbed Walls	41
2.6. RANS CFD Studies	43
2.7. Rotational Effects on Heat Transfer & Flow Field	44
3. BASE CASE	50
3.1. Test Setup	50
3.2. Heat Transfer Results	54
3.3. Flow Field Measurement	56
4. NUMERICAL APPROACH AND SOLUTIONS	59
4.1. CFD Tools	59
4.2. Geometry	59
4.3. Grid Generation	62
4.4. Turbulence Models	64
4.5. Solver Setup	66
4.6. Boundary Conditions	66
4.7. Analysis Matrix	68

4.8. Solutions & Comparison With Test Data	69
4.8.1. Stationary Cases	70
4.8.2. Cases with Rotation & $Ro: -0.3$	76
4.8.3. Cases with Rotation & $Ro: +0.3$	80
4.9. Bonus – RSM & Periodic Setup Trials	85
4.9.1. RSM Solution	86
4.9.2. Periodic Conditions	87
5. SUMMARY AND DISCUSSION	90
REFERENCES	92
BIOGRAPHY	95

LIST OF SYMBOLS AND ABBREVIATONS

Ω	: Angular Velocity
e/D_h	: Blockage Ratio
Boi	: Body Of Influence
T_b	: Bulk Fluid Temperature
U_b	: Channel Inlet Velocity
CFD	: Computational Fluid Dynamics
μ	: Dynamic Viscosity
ε_{TLC}	: Emissivity Of TLC
EF	: Enhancement Factor
HTC	: Heat Transfer Coefficient
q	: Heat Flux
D_h	: Hydraulic Diameter
Nu	: Nusselt Number
PIV	: Particle Image Velocimetry
P/e	: Pitch-To-Rib Height
Re	: Reynolds Number
RANS	: Reynolds-Averaged Navier Stokes
e	: Rib Height
Ro	: Rotation Number
k	: Thermal Conductivity
σ	: Stefan Boltzmann Constant
TEI	: Tusaş Engine Industries
VKI	: Von Karman Institute
T_w	: Wall Temperature
W/H	: Width-To-Height Ratio

LIST OF FIGURES

	<u>Page</u>
Figure 1.1: Different gas turbine engines [11].	17
Figure 1.2: T-s diagram of Brayton Cycle	18
Figure 1.3: Turbine rotor inlet temperature and specific core power comparison [12].	20
Figure 1.4: Cooling technology through years [12].	22
Figure 1.5: Increment of OPR through years [12].	23
Figure 2.1: Representative view of rib orientations and flow separation [13].	25
Figure 2.2: Representative view of turbine blade internal cooling channel.	26
Figure 2.3: Typical foil-heated experimental setup and rib arrangements [14].	27
Figure 2.4: Impact of P/e on the channel heat-transfer coefficient distributions for rectangular channels [15].	28
Figure 2.5: Effect of rib angle on the channel centerline heat-transfer coefficient distribution in a square channel [16].	29
Figure 2.6: Effect of rib angle on secondary flow and lateral Nusselt number ratio (a) Schematic of secondary flows and (b) comparison of lateral Nusselt number ratios [16].	31
Figure 2.7: Effect of channel aspect ratio on heat-transfer performance [17].	32
Figure 2.8: Effect of rib angle on heat-transfer performance [17].	33
Figure 2.9: Different rib configurations studied by Han et al. [18].	34
Figure 2.10: Heat-transfer performance for different rib configurations [18].	35
Figure 2.11: Heat-transfer correlation for different rib configurations [18].	36
Figure 2.12: Rib configurations used by Han and Zhang [19].	37
Figure 2.13: Average Nusselt number ratio vs average friction factor ratio for broken ribs [19].	38
Figure 2.14: Effect of different rib arrangements on the average Nusselt number ratio [20].	39
Figure 2.15: Effect of rib spacing on average Nusselt number [20].	40
Figure 2.16: Effect of rib angle and rib spacing on average Nusselt number [21].	40
Figure 2.17: Effect of rib angle and rib spacing on friction factor [21].	41
Figure 2.18: Effect of the number of ribbed walls on heat-transfer performance [22].	42
Figure 2.19: CFD and test results comparison [23].	43
Figure 2.20: Schematic view of Coriolis and buoyancy effects on radially inward and outward flows [24].	44
Figure 2.21: Impact of rotation on a wall shear stress [25].	45
Figure 2.22: Schematic view of a two-pass rotating coolant flow distribution [26].	46
Figure 2.23: Serpentine test coupon [27].	47
Figure 2.24: Rotation number effect on Nusselt number ratio for $Re = 25,000$ (a) Trailing surface and (b) Leading surface [27].	48
Figure 3.1: Top view of the table [28].	51
Figure 3.2: Front view of the test section [28].	51
Figure 3.3: Schematic of the rotating facility, lateral view [29].	53
Figure 3.4: Sketch of the PIV layout [29].	54
Figure 3.5: Literature comparison [29].	54
Figure 3.6: Enhancement factor distribution for $Re=30000$, $Ro=0.0$ [28].	55
Figure 3.7: Averaged Enhancement Factor as a function of Re and Ro [28].	56

Figure 3.8: Normalized mean velocity and turbulence intensity measurement at the entrance of channel [29].	57
Figure 3.9: Normalized mean velocity measurement for a channel periodicity a) Ro: -0.3 b) Ro: 0 c) Ro: +0.3 [29].	57
Figure 3.10: Two-dimensional streamlines and contours of mean in-plane velocity: stabilizing rotation a) Ro: -0.3 b) Ro: 0 c) Ro: +0.3 [29]	58
Figure 4.1: Inlet view of fluid domain.	60
Figure 4.2: Side view of whole fluid domain.	60
Figure 4.3: Entrance region of fluid domain.	60
Figure 4.4: Exit region of fluid domain	61
Figure 4.5: Dimensions of BOI	61
Figure 4.6: Schematic of BCs	67
Figure 4.7: Notation of analysis matrix.	69
Figure 4.8: HTC EF distribution at Re: 15000&Ro: 0 [28].	70
Figure 4.9: Streamlines in mid-plane at Re: 15000&Ro: 0 [29].	70
Figure 4.10: Streamlines in mid-plane at Re: 15000&Ro: 0 for different mesh and turbulence model conditions	71
Figure 4.11: Normalized axial velocity at x/h:1 (black dots) and x/h:11 (red dots) for periodicity check at Re: 15000&Ro: 0 for different mesh and turbulence model conditions.	73
Figure 4.12: HTC EF distribution at Re: 15000&Ro: 0 for different mesh and turbulence model conditions.	74
Figure 4.13: HTC EF distribution at Re: 15000&Ro: -0.3 [28]	76
Figure 4.14: Streamlines in mid-plane at Re: 15000&Ro: -0.3 [29]	76
Figure 4.15: Streamlines in mid-plane at Re: 15000&Ro: -0.3 for different mesh and turbulence model conditions	77
Figure 4.16: Normalized axial velocity at x/h:1 (black dots) and x/h:11 (red dots) for periodicity check at Re: 15000&Ro: -0.3 for different mesh and turbulence model conditions	78
Figure 4.17: HTC EF distribution at Re: 15000&Ro: -0.3 for different mesh and turbulence model conditions.	79
Figure 4.18: HTC EF distribution at Re: 15000&Ro: +0.3 [28].	80
Figure 4.19: Streamlines in mid-plane at Re: 15000&Ro: 0.3 [29].	81
Figure 4.20: Streamlines in mid-plane at Re: 15000&Ro: 0.3 for different mesh and turbulence model conditions	81
Figure 4.21: Normalized axial velocity at x/h:1 (black dots) and x/h:11 (red dots) for periodicity check at Re: 15000&Ro: 0.3 for different mesh and turbulence model conditions.	83
Figure 4.22: HTC EF distribution at Re: 15000&Ro: 0.3 for different mesh and turbulence model conditions.	84
Figure 4.23: HTC enhancement distribution of RSM solution in a stationary condition.	86
Figure 4.24: Streamline of RSM solution in a stationary condition	87
Figure 4.25: Periodic solution domain.	87
Figure 4.26: HTC enhancement distribution of periodic boundary solution in a stationary condition.	88
Figure 4.27: Streamline of periodic boundary solution in a stationary condition.	88

LIST OF TABLES

	<u>Page</u>
Table 3.1: Test Matrix [28].	52
Table 4.1: Side views of meshes.	62
Table 4.2: Various views of meshes.	63
Table 4.3: Mesh parameters.	64
Table 4.4: Mesh metrics.	64
Table 4.5: Solver setup.	66
Table 4.6: Boundary conditions.	67
Table 4.7: Analysis matrix.	68
Table 4.8: CFD reattachment length comparison with the test.	72
Table 4.9: HTC EF CFD comparison with the test at Re: 15000&Ro: 0.	75
Table 4.10: CFD reattachment length comparison with the test at Re: 15000&Ro: -0.3.	77
Table 4.11: HTC EF CFD comparison with the test at Re: 15000&Ro: -0.3.	80
Table 4.12: CFD reattachment length comparison with the test at Re: 15000&Ro: 0.3.	82
Table 4.13: HTC EF CFD comparison with the test at Re: 15000&Ro: 0.3.	85

1. INTRODUCTION

1.1. History of Gas Turbines

The rapid adoption of gas turbine engines for powering aircraft stands as a remarkable transformation within aviation history. Prior to the 1950s, this technology remained largely unknown and its potential for aircraft propulsion was hardly considered. Although the idea of jet engines had intrigued aeronautical engineers for years, early aircraft were simply too slow, and existing piston engines couldn't generate the high-speed airflow needed for efficient jet propulsion. These limitations posed significant challenges that seemed insurmountable at the time.

The timeline starts from 1500s, more than 5 centuries ago with Leonardo DaVinci's the smoke jack.

Here is the list of crucial milestones of gas turbine development through the history, including very first ideas.

- Leonardo da Vinci envisioned an early form of gas turbine in 1500 with his "Smoke Jack." Hot air rising from a fireplace spun a single-bladed turbine, connected by gears to a roasting spit, making it the first known example of using hot gas to produce mechanical energy.
- Back in 1791, Englishman John Barber snagged a patent for a remarkable invention - the first true gas turbine! This innovative design, remarkably, contained many base items found in modern gas turbines [1].
- In 1894, Charles Parsons revolutionized maritime technology with his patented design for a steam turbine-powered ship [2].
- In 1899, the first gas turbine engine in America patented by Charles Gordon Curtis [3].
- Sanford Alexander Moss, after submitting his thesis on gas turbines in 1900, joined General Electric's Steam Turbine department in 1903. There, his

expertise in gas turbines found practical application in the development of the turbocharger.

- In 1903, amidst limited understanding of aerodynamics, Norwegian inventor Ægidius Elling made a significant technical breakthrough. He built the first gas turbine capable of generating more power than it consumed. This ingenious machine, utilizing both rotary compressors and turbines, pumped out an impressive 11 horsepower, marking a significant milestone in early gas turbine history [4].
- Franz Stolze's gas turbine engine, based on his 1873 patent, was built in Berlin in 1904. However, it lacked the efficiency to operate independently, marking an interesting attempt in the early development of gas turbine technology [5].
- During the 1920s, significant progress was made in gas turbine technology. A.A. Griffith laid the groundwork by developing a formal theory of gas flow around airfoils, crucial for turbine design. He published his findings in the influential book "An Aerodynamic Theory of Turbine Design" in 1926. In parallel, the Royal Aeronautical Establishment made practical advancements by building and testing working models of axial turbines suitable for propelling aircraft [6].
- In 1930, undeterred by the Royal Air Force's lack of enthusiasm, Frank Whittle filed a patent for his innovative design: a centrifugal gas turbine specifically intended for jet propulsion [7].
- The first-ever industrial gas turbine with constant flow was developed by the Brown Boveri Company and started to giving service in US [8].
- In 1937, both Britain and Germany took major steps towards jet engine technology. In England, Frank Whittle's prototype turbojet engine successfully ran for the first time, while in Germany, Hans von Ohain's Heinkel HeS 1 engine achieved to steady operation [9].
- 1939 marked the first flight of the Heinkel He 178, the pioneering jet aircraft that paved the way for a new era in aviation.

- Brown Boveri unveiled the first 4 MW gas turbine for an emergency power plant in Neuchâtel, Switzerland, in 1939, a major advancement in generating electricity [10].
- 1944 witnessed the mass production of the Junkers Jumo 004 engine, propelling the first German jet fighters like the Messerschmitt Me 262.

1.2. How It Works

Imagine blowing up a balloon: the air rushing out propels it forward. That's essentially what happens in planes, both with jet engines and propellers. Both rely on Newton's third law of motion, where pushing air backwards (action) results in the plane moving forward (reaction).

While the jet pushes out, it's not the force on air that creates thrust. Instead, jet engines like rockets and turbojets speed up air or gas and push it out. The engine's power depends on how much stuff (mass) is pushed out and how much faster it goes. Basically, you can get the same thrust by gently speeding up a lot of air or really speeding up a little bit.

The mechanical configuration of the gas turbine engine is straightforward, comprising only two primary rotating components: a compressor and a turbine, along with one or multiple combustion chambers. The schematic representation of the mechanical arrangement for different gas turbine engines is illustrated in **Figure 1.1**.

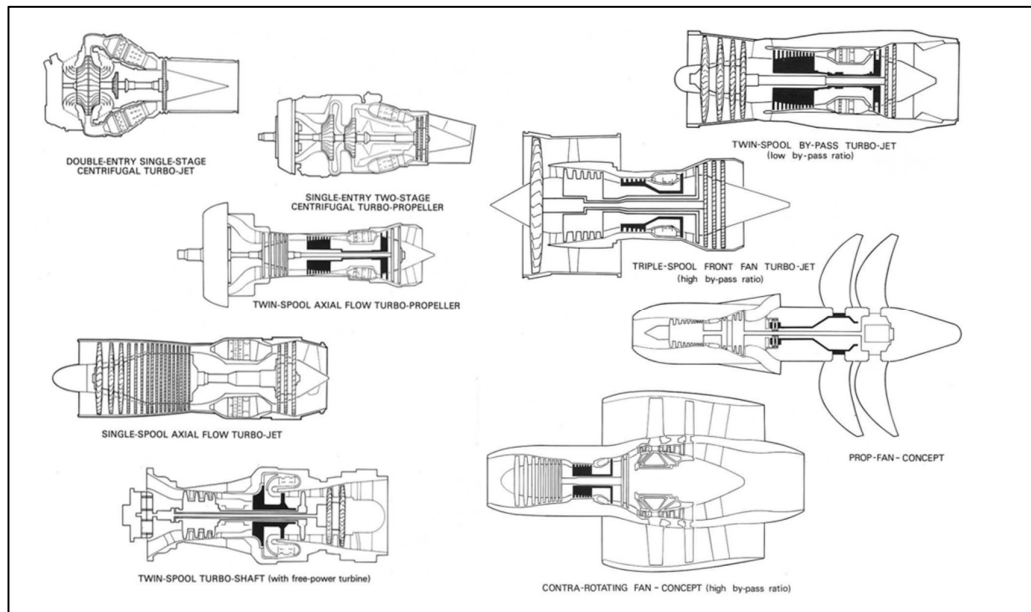


Figure 1.1: Different gas turbine engines [11].

Turbo-prop engines combining propellers and turbines are less attractive due to newer designs. Bypass, ducted fan, and propfan engines handle more air with slower exhaust, achieving similar or better efficiency than propellers and turbines, or pure jet engines, respectively.

1.3. Working Cycle

Contemporary Brayton engines are mostly turbine-based, unlike the original piston design by Brayton himself. Here's how the 19th-century Brayton engine worked: air got sucked into a piston compressor where it was squeezed (ideally a perfect process). Then the compressed air moved to a mixing chamber where fuel was added (constant pressure process). This mix ignited in a separate cylinder, releasing energy and pushing a piston outward as the hot air expanded (again, ideally a perfect process). Some of this work from the piston was used to power the compressor through a crankshaft.

Because of the continuous operation of the turbine engine and the open nature of the combustion chamber, the air pressure doesn't increase as in piston engines during combustion; instead, the volume expands. This process is referred to as heating at constant pressure. In these conditions, there are no intense or fluctuating pressures to endure, unlike piston engines with peak pressures exceeding 7 MPa. These peak

pressures necessitate heavy-duty construction and the use of high-octane fuels for piston engines, whereas turbine engines can utilize lighter fabricated combustion chambers and lower octane fuels.

The figure shows a T-s diagram explaining how a Brayton cycle works in a turbine engine. Imagine air flowing freely at point 0. As it enters the engine (station 2), the air slows down slightly. Ideally, the compressor (between stations 2 and 3) would squeeze the air perfectly, raising both pressure and temperature. In reality, some energy is lost, so the line on the diagram bends slightly. Fuel is then added in the burner (between stations 3 and 4) at a constant pressure, causing a temperature rise depending on the fuel used. The hot exhaust then spins the turbine (between stations 4 and 5), transferring some of its energy to power the compressor. Ideally, the temperature drop in the turbine would exactly match the temperature gain in the compressor, and both would be on the same shaft. Finally, the nozzle (between stations 5 and 8) accelerates the flow back to its original pressure, matching the free-flowing air outside the engine.

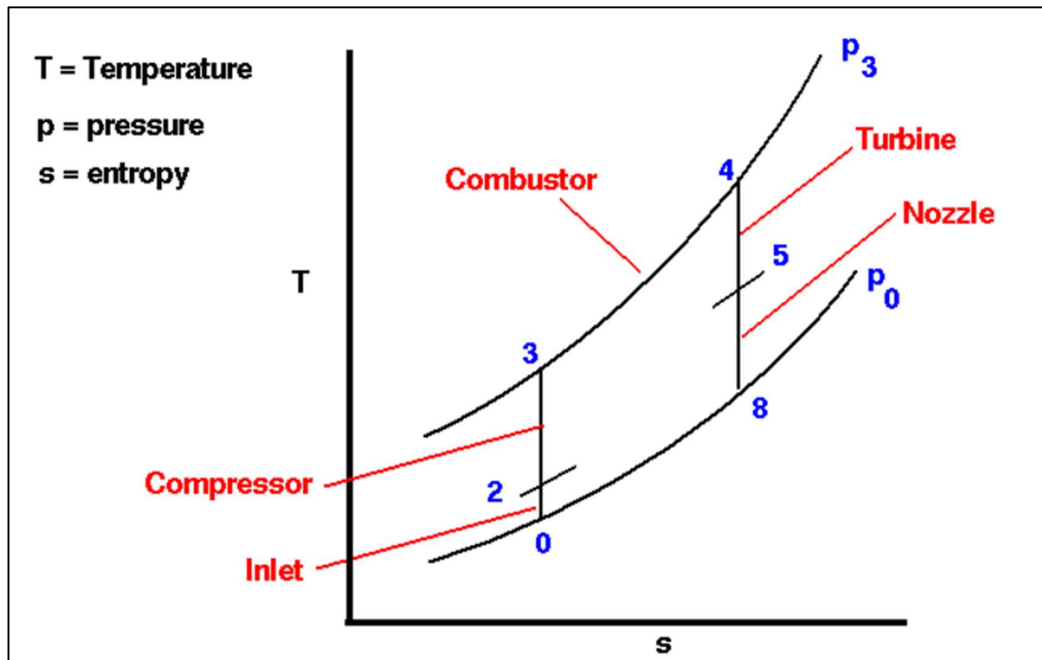


Figure 1.2: T-s diagram of Brayton Cycle.

Since the gas turbine engine is a heat engine, the higher combustion temperature leads higher expansion of the gases. However, an exit temperature of a combustion chamber must not surpass the allowable turbine entry temperature which arises from the design

and materials of the turbine airfoils. The air-cooled turbine airfoils allows a much higher turbine entry temperature and that brings higher thermal efficiency and power.

1.4. Usage Applications

A turboprop engine is a type of turbine engine that propels an aircraft's propeller by utilizing a reduction gear mechanism, which reduces the high speed of the turbine section (often tens of thousands of revolutions per minute) to the lower speeds required for efficient propeller operation (typically in the low thousands). The advantage of employing a turboprop engine lies in leveraging the high power-to-weight ratio of turbine engines to drive the propeller, enabling the use of smaller yet more powerful engines.

Aeroderivative gas turbines are typically adapted from on hand aircraft engines, resulting in smaller and light units compared to industrial gas turbines. They find widespread use in electricity generation because of their capability to be rapidly shut down and adjust to load changes, which exceeds the performance of industrial models. Additionally, they are favored in the marine sector for their weight reduction benefits. Examples include models like the General Electric LM2500 and LM6000.

Turboshaft engines are workhorses in many applications. They power compressors that pump gas in stations and liquefy natural gas for transport. In aviation, they're the go-to engine for almost all helicopters and even provide extra power in big jets. These engines have a core shaft with a compressor and turbine (the gas generator) and a separate combustor where fuel is burned. There's also a special power turbine that spins on its own, usually to turn the helicopter's main rotor blades. Because these two parts can spin independently, it gives engineers more freedom to design them for specific tasks.

Industrial gas turbines are distinct from aeronautical versions due to their heavier construction of frames, bearings, and blading. Moreover, they are intricately linked with the equipment they operate, typically an electric generator, as well as secondary energy systems used to harness leftover energy, primarily heat.

Another significant benefit lies in their capacity for swift activation and deactivation, enabling power provision during peak or unforeseen demand spikes. Given the comparatively inferior efficiency of standalone gas turbine power plants in contrast to

combined cycle facilities, they predominantly function as peaking power stations. These peaking facilities typically operate for several hours daily, extending to a few dozen hours annually, contingent upon electricity requirements and regional generation capabilities. In areas grappling with insufficient base-load and load-following power plant capacities or enjoying low fuel expenses, gas turbine power plants may sustain operations for the majority of the day. Big, single-use gas turbine power plants typically generate between 100 and 400 megawatts of electricity. However, they only convert 35 to 40% of the fuel's energy into usable electricity, meaning a significant portion of the energy is lost as heat.

1.5. Importance Of Turbine Blade Cooling

Gas turbines are used for both jet engines in airplanes and for generating electricity or powering industrial machinery on the ground. The hotter the gas entering the turbine blades (higher turbine rotor inlet temperatures), the more efficient the engine is at converting fuel into power and the more power it can produce. This is depicted in **Figure 1.3.**, where the relationship between specific core power production (linked to specific thrust) and turbine RIT is graphically represented.

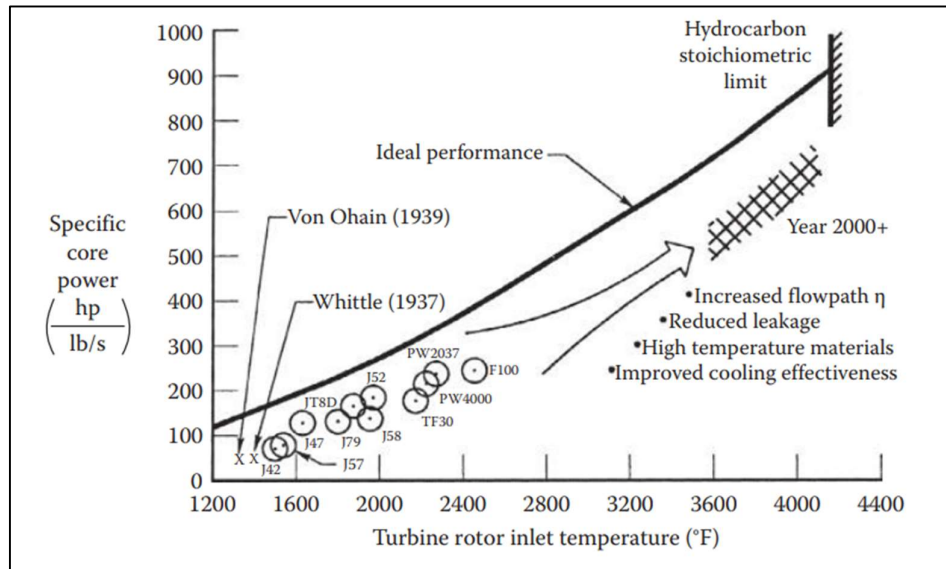


Figure 1.3: Turbine rotor inlet temperature and specific core power comparison [12].

Aircraft engines usually perform quite close to the perfect performance line, which represents a scenario where turbines operate with maximum efficiency and no loss or cooling. It's clear that increasing the temperature at the turbine inlet is crucial for improving gas turbine engine performance. **Figure 1.4.** illustrates that in advanced gas turbines, the temperature at the turbine inlet is much higher than where the blade material becomes soft. Hence, turbine blades have to be cooled. To achieve a doubling of engine power in aircraft gas turbines, the turbine inlet temperature (TIT) needs to escalate from the current 1600°C to 2000°C, while conserving the same proportion of cooling air. Moreover, the compressor pressure ratio should advance from the current 20 times to double it, as depicted in **Figure 1.5.** This means that in the future, aircraft gas turbines will have higher turbine inlet temperatures, but they'll still use the same amount of warm air for cooling from the high-pressure compressor. So, it's crucial to develop materials that can handle these high temperatures, like thermal barrier coatings or advanced cooling techniques, to ensure that future gas turbines perform well and produce high power.

State-of-the-art gas turbine engines operate at high temperatures, typically ranging from approximately 1400°C to 1550°C, to enhance both thermal efficiency and power generation. With the increase in turbine inlet temperature, there is a corresponding rise in the heat transferred to the turbine blades. It is essential to control the magnitude and fluctuations of temperature within the blade material to maintain durability objectives within reasonable limits, as excessive thermal stresses can compromise performance.

Gas turbines are pushing the limits of metal strength with their super-hot operating temperatures. To keep things safe, engineers need to cool the turbine blades, but this cool air comes from the engine itself, which reduces efficiency. That's why it's important to find the perfect balance between cooling techniques, how the engine runs, and the design of the blades themselves. There are two main ways to cool these blades: internally and externally. Internal cooling is like running water through a maze inside the blade to absorb heat. Tiny channels and special fins help with this. External cooling, also called film cooling, is more like a protective shield. Cool air is blown out of tiny holes on the blade's surface, creating a cool layer that blocks the scorching hot combustion gases.

The cooling system of the engine needs to be carefully planned to ensure that the highest temperatures experienced by the blade surfaces, as well as the temperature

changes across the blades during operation, align with the maximum thermal stress the blades can withstand throughout their lifespan according to the design. Insufficient coolant flow leads to hotter blade temperatures and shortens the lifespan of components. Conversely, excessive coolant flow diminishes engine performance. Ideally, the engine's cooling system should be designed to use as little compressed air as possible. This is because taking air out for cooling reduces the overall efficiency of the engine. By minimizing the amount of air bled from the compressor for cooling, the engine can take full advantage of the benefits of having a hot inlet gas temperature.

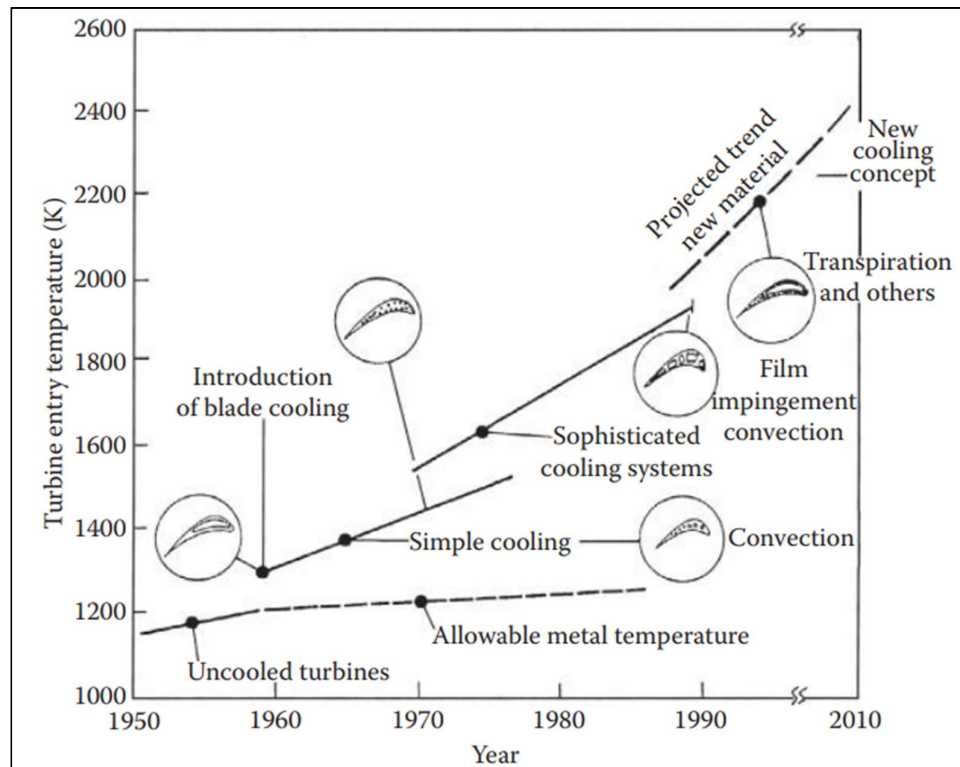


Figure 1.4: Cooling technology through years [12].

Modern gas turbines operate at extremely high temperatures that would damage standard metals. To maintain blade lifespan and safe operation under such heat, advanced cooling techniques are crucial. These include internal cooling methods that circulate cool air within the blades and external film cooling that forms a protective layer on the blades' exterior. Effective turbine cooling requires a deep understanding of how hot gases flow within the engine. This hot gas flow is complex and requires accurate prediction of local heat transfer and blade temperatures. Even slight

deviations in these predictions can significantly impact blade life. Ideally, engineers aim to prevent hot spots on the blades to maximize their lifespan. Current methods for predicting heat transfer across turbine blades under real-world engine conditions are not very precise. This leads to conservative designs that use excessive cooling air, reducing overall engine performance. In essence, improved understanding of hot gas flow is essential for designing more efficient and durable gas turbines.

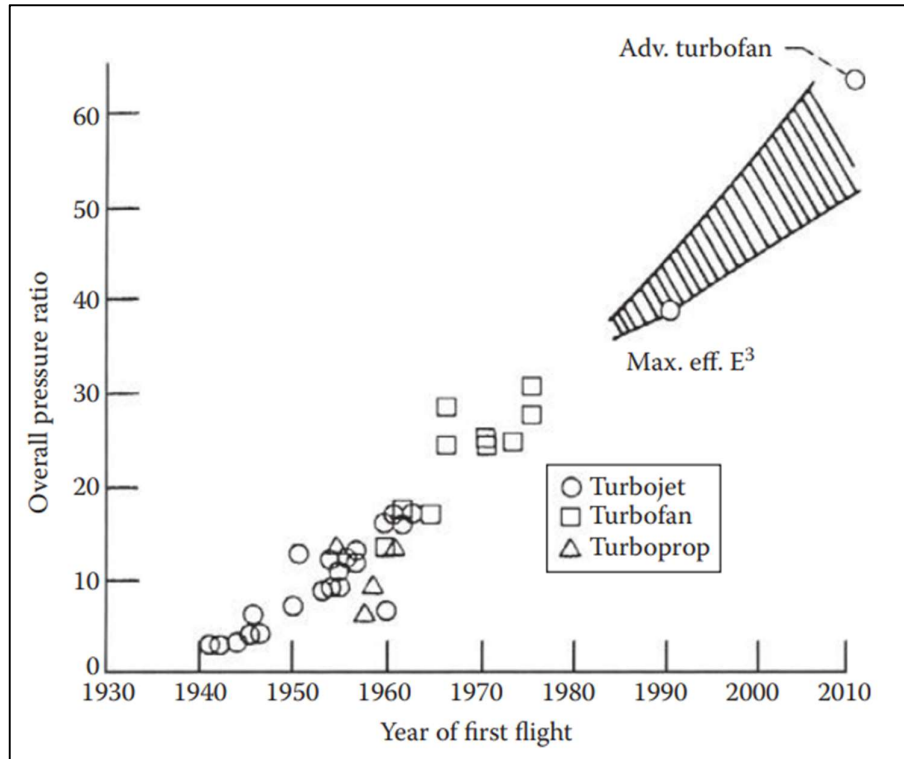


Figure 1.5: Increment of OPR through years [12].

The first parts of the hot gas path, facing the hottest and most turbulent exhaust from the combustor, are the stationary vanes. To design an effective cooling system and prevent overheating, engineers need to understand exactly how heat is distributed across these vanes and the surrounding walls under normal engine operation conditions. After the vanes, the hot gas flows through the rotating blades to generate power. While the gas entering these blades is cooler and less turbulent than at the vanes, it's also moving much faster, two to three times as fast. Additionally, the blades experience uneven air flow coming off the vanes in front of them. A particular concern is hot gas leaking from the high-pressure side of the rotating blade near the tip to the low-pressure side. This can damage the blade tip. To prevent blade failure due to

uneven cooling under high heat stress from rotation, engineers need a deep understanding of the complex 3D flow patterns and heat transfer around the blades, especially near the tips, under typical engine operating conditions. This knowledge is essential for designing a more effective cooling system for these blades.

2. LITERATURE REVIEW AND THEORETICAL BACKGROUND

In modern gas turbine blades, a series of ribbed structures are molded onto both walls of internal channels to increase heat transfer. Numerous fundamental studies have investigated how flow separation caused by ribs enhances heat transfer. Typically, experimental studies use ribs with a square cross-section, where the relative height of the ribs is between 5% and 10% of the hydraulic diameter of the channel. The ratio between the rib pitch and its height (p/e ratio) generally ranges from 7 to 15. Moreover, modern airfoils feature more complex rib shapes, and smaller gas turbines use high-blockage ribs with tighter spacing.

Key features that define a ribbed channel's configuration include geometric elements such as rib height, pitch, and the angle of attack. Common terminology for rib geometry includes "e" for rib height and "P" for the distance between ribs. **Figure 2.1.** shows the flow pattern around ribs, showing that a boundary layer forms both upstream and downstream of the ribs.

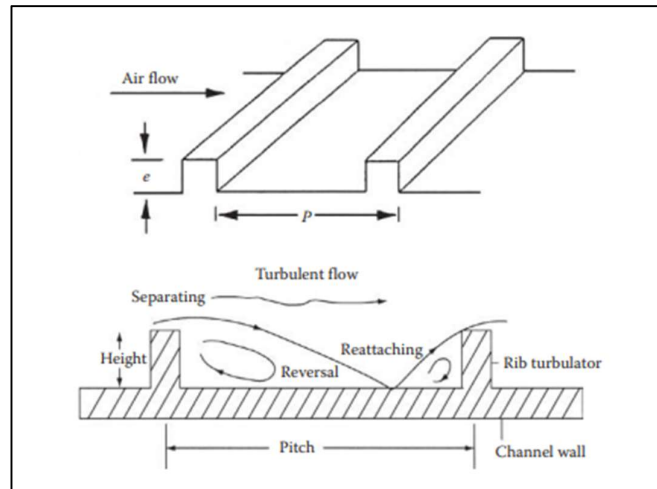


Figure 2.1: Representative view of rib orientations and flow separation [13].

Numerous studies have demonstrated that an additional separation zone might form on top of the rib. These flow separations cause the boundary layer to reattach to the heat-

transfer surface, leading to the heat-transfer coefficient increment. Additionally, the turbulent mixing caused by the separated boundary layer allows heat from the near-surface fluid to dissipate more efficiently into the main flow, further boosting the heat-transfer coefficient. Ribs primarily affect the flow near the wall, which means that the resulting pressure drop is generally manageable for blade cooling designs.

Figure 2.2. illustrates a typical design for cooling channels inside a turbine blade. Due to the blade's curved and uneven shape, the cooling channels near the back (trailing edge) are wider compared to their height (aspect ratio). In contrast, the channels closer to the front (leading edge) are narrower relative to their height. It's also common to find ribs inside these channels on both sides of the blade, the suction side and the pressure side.

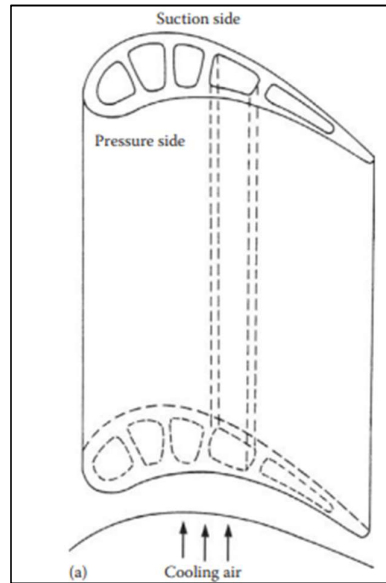


Figure 2.2: Representative view of turbine blade internal cooling channel.

It's important for the reader to understand the experimental setups used to gather the results presented. **Figure 2.3.** demonstrates the cross-section of a typical foil-heated test channel and the rib geometry used by Park et al. [14]. The researchers tested five straight, rectangular channels of different widths and heights (varying aspect ratios). A chamber (plenum) at the channel entrance started the flow and shaped it as it traveled through (hydrodynamically developing). The entrance from the chamber was abrupt. Each side of the rectangular duct was equipped with individual heaters, which were electrical resistive elements providing a uniform heat flux.

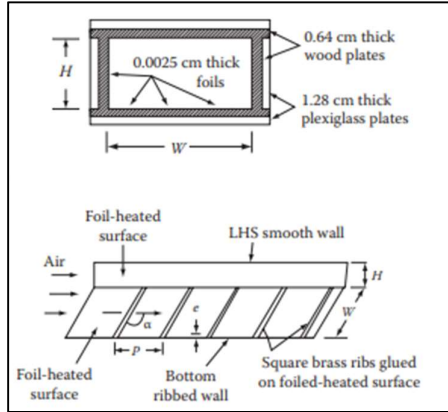


Figure 2.3: Typical foil-heated experimental setup and rib arrangements [14].

2.1. Effects of Rib Layouts and Flow Parameters on Ribbed-Channel Heat Transfer

Heat transfer performance of rib largely depends on the geometry of the ribs and the conditions of the flow. The influence of flow is typically described by its Reynolds number. Beyond that, key geometric factors like the spacing between ribs, the orientation of the ribs, and the aspect ratios of the channels are also critical, as detailed in the upcoming paragraphs.

2.1.1. Effect of Rib Spacing on the Ribbed and Adjacent Smooth Sidewalls

Figure 2.4. illustrates how rib spacing affects the heat-transfer coefficient distribution along the centerline of the channel. The ribs are oriented perpendicular to the flow, with two spacing ratios, $P/e = 10$ and 20 . The graph shows that the local Nusselt number ratios on the smooth walls of ribbed channels decrease markedly as the distance from the inlet increases, for both P/e ratios. These trends are similar to those observed in channels with smooth walls on all sides, but with more fluctuations in the ribbed-channel data, which is also about 20% to 60% higher than the all-smooth-wall data. These fluctuations are caused by the adjacent ribbed walls. Additionally, the local Nusselt number ratios on the ribbed walls exhibit even greater fluctuations. The heat-transfer coefficient gradually decreases with distance from the inlet, settling into a

periodic pattern shortly after the first few ribs (beyond $X/D > 3.0$). For $P/e = 10$, the heat-transfer coefficient rises near the point of flow reattachment, which is downstream of each rib, and then increases again as the flow nears the next rib. The passage delves into the impact of rib spacing on convective heat transfer within a ribbed channel. When the P/e ratio is raised to 20, the overall heat transfer patterns exhibit similarity to those observed at P/e of 10. However, the heat transfer coefficients demonstrate a reduction of 10-20%. This decline is attributed to the increased rib spacing, which fosters the development of a thicker boundary layer subsequent to flow reattachment between the ribs. In the initial segment (X/D less than 1.0), the ratios of Nusselt numbers (a dimensionless quantity characterizing convective heat transfer) on both ribbed and smooth walls display comparable values for both P/e configurations. Nevertheless, further downstream (X/D exceeding 1.0), the ribbed wall Nusselt number ratios diverge from the smooth wall values and can attain values up to twice as high within the fully developed region further downstream.

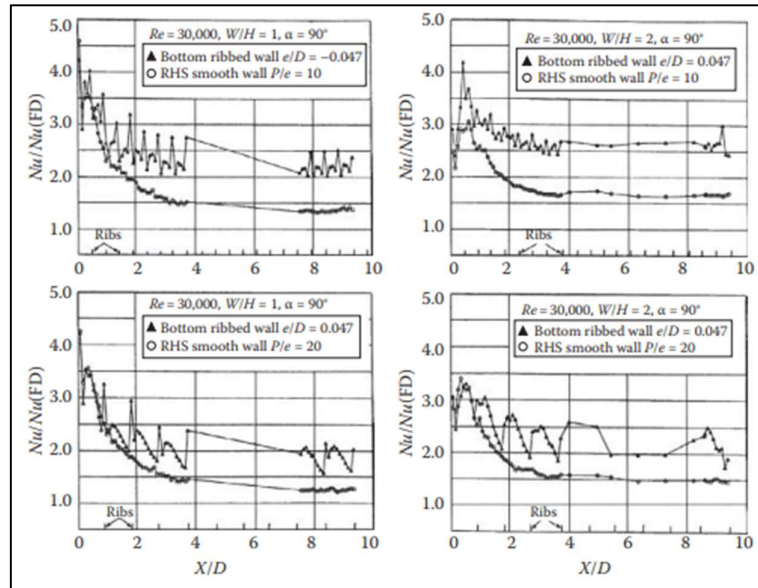


Figure 2.4: Impact of P/e on the channel heat-transfer coefficient distributions for rectangular channels [15].

2.1.2. Angled Ribs

Ribs can also be positioned at an angle relative to the direction of the main flow. Han and Park [16] explored heat transfer using these non-orthogonal ribs. **Figure 2.5.**

illustrates the impact of varying the rib angle of attack on the heat-transfer coefficients along the centerline for a configuration with $P/e = 10$ and $Re = 30,000$. The local heat-transfer patterns on the ribbed wall differ when using different rib angles of attack. In a square channel with a width-to-height ratio of 1 ($W/H = 1$), the Nusselt number ratios along the centerline for a rib angle of $\alpha = 90^\circ$ reach a stable periodic pattern when the axial distance exceeds three times the hydraulic diameter ($X/D > 3$). This periodic pattern in local Nusselt numbers arises from the regular flow separation from ribs and subsequent reattachment between them.

Channels with angled ribs at 60, 45, and 30 degrees transfer heat more effectively in the center (centerline Nusselt number) after a certain distance ($X/D > 3$) compared to a channel with ribs at 90 degrees. Additionally, these angled ribs (60, 45, and 30 degrees) keep improving heat transfer throughout the entire channel (periodic Nusselt number) even beyond that same distance ($X/D > 3$).

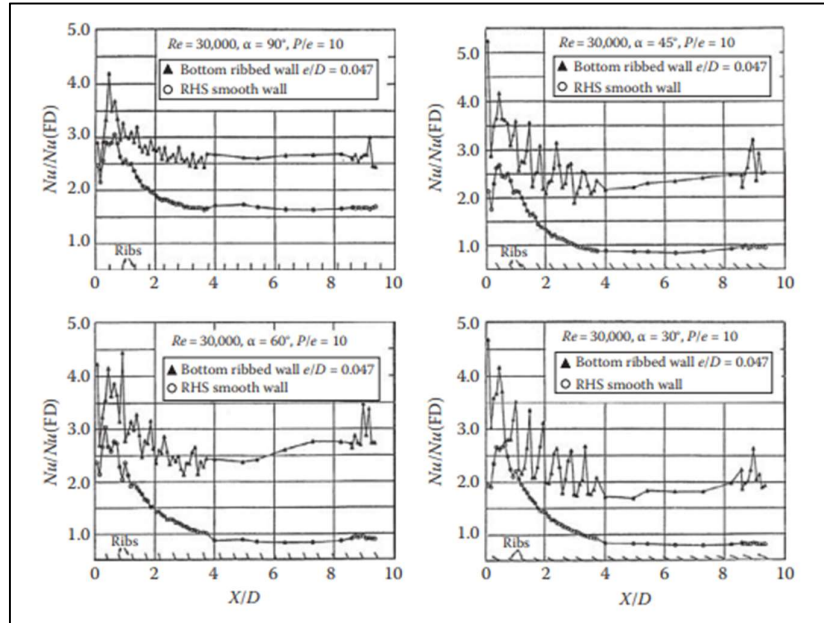


Figure 2.5: Effect of rib angle on the channel centerline heat-transfer coefficient distribution in a square channel [16].

The idea is that the secondary flow created by angled ribs plays a significant role in heat transfer. Han and Park [16] examined the effect of rib angle on the average Nusselt numbers along the streamwise direction. The average Nusselt number ratio along the

flow direction was calculated by averaging the Nusselt number ratios along three lines in each test channel: center, middle, and edge.

In a square channel, heat transfer across the channel width (streamwise) was similar for ribs positioned perpendicular to the flow (at 90 degrees). However, for angled ribs (60, 45, and 30 degrees), the heat transfer across the width varied more. The heat transfer was 30% to 50% higher near the edges compared to the center of the channel.

The angled ribs, as illustrated in **Figure 2.6.** create a swirling flow within the channel that moves from right to left. This swirling motion (secondary flow) causes the heat transfer to be higher near the front (leading edge) of the channel compared to both the center and the back (trailing edge). Interestingly, even the center of the channel with angled ribs transfers heat better than channels with straight ribs (90 degrees). This applies to both the ribbed side and the smooth side of the channel.

Figure 2.6. shows how heat transfer changes in rectangular channels (wider than they are tall) with ribs at different angles (30 to 90 degrees). When the ribs are straight across the flow (90 degrees), the heat transfer pattern across the channel width is similar to square channels (equal width and height). However, for angled ribs (60, 45, and 30 degrees), the flow gets redirected due to the angle. This creates a secondary flow that moves from the left side to the right side along the direction of the ribs. This same pattern of flow and heat transfer is also observed in even wider rectangular channels.

In wider rectangular channels (compared to square channels), the heat transfer in the center of the channel (centerline Nusselt number) is lower for both the ribbed side and the smooth side with angled ribs. This is true when compared to channels with ribs that go straight across the flow (90 degrees).

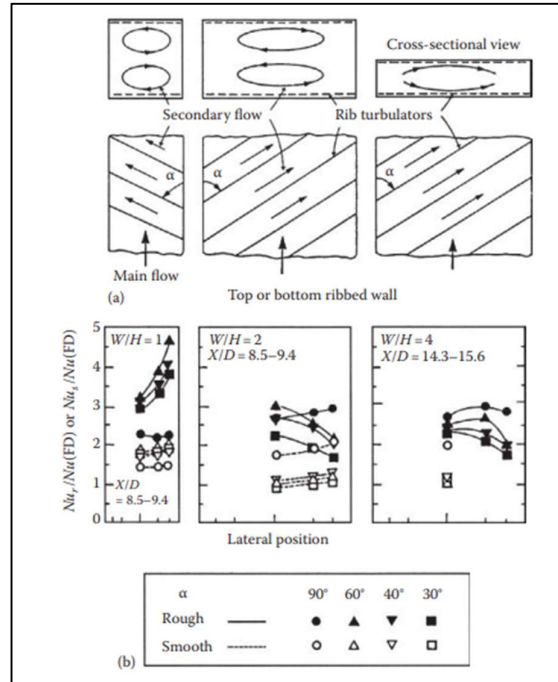


Figure 2.6: Effect of rib angle on secondary flow and lateral Nusselt number ratio
(a) Schematic of secondary flows and (b) comparison of lateral Nusselt number ratios [16].

2.1.3. Effect of Channel Aspect Ratio with Angled Ribs

Figure 2.7. illustrates how varying the aspect ratio of a channel affects heat-transfer enhancement and pressure drop for specific rib angles at a Reynolds number of 30,000. With 60° ribs, the increase in heat transfer on the ribbed side is relatively consistent regardless of the aspect ratio. However, the pressure drop penalty rises sharply, escalating from 2 times to 18 times as the channel's aspect ratio transitions from narrow ($1/4$) to wide (4).

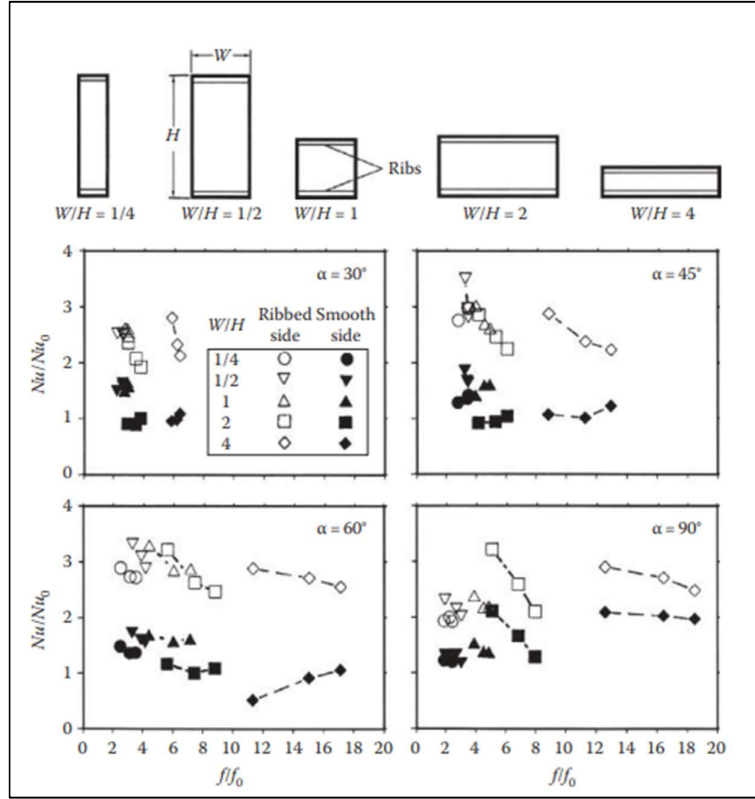


Figure 2.7: Effect of channel aspect ratio on heat-transfer performance [17].

Narrow channels transfer heat more efficiently than wider channels. In narrow channels, 30-degree angled ribs improve heat transfer with a minimal pressure increase. For wider channels, 30-degree ribs cause significantly less pressure loss than other shapes while maintaining similar heat transfer benefits.

2.2. Comparison of Different Angled Ribs

Figure 2.8. presents a comparison of heat-transfer in channels with ribs at different angles and varying widths compared to their heights (aspect ratios). Angled ribs (45° and 60°) in narrow channels ($W/H = 1/4$) significantly improve heat transfer (three times more than a smooth channel) with a moderate pressure increase (two to four times higher). However, wider channels ($W/H = 4$) require angled ribs to achieve similar heat transfer gains, but this comes at a much higher pressure penalty (eight to sixteen times more than a smooth channel). Overall, the study suggests narrow channels with angled ribs perform better than wider channels. In narrow channels, 90° ribs cause

minimal changes in heat transfer and pressure, while 60° ribs provide the most significant improvements in both for very narrow channels ($W/H = 1/2$ and $1/4$).

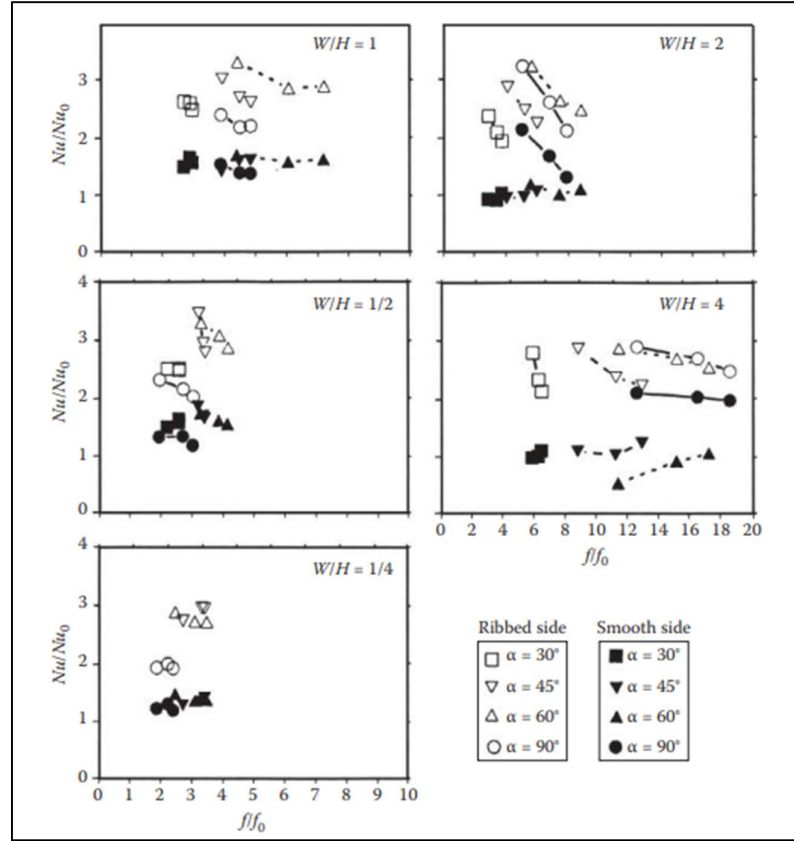


Figure 2.8: Effect of rib angle on heat-transfer performance [17].

The findings suggest that 30-degree angled ribs offer the least improvement in heat transfer, while 60-degree angled and 90-degree straight ribs perform better in wider channels (aspect ratio of 4 and 2). For smooth walls (see Figure 13), adding ribs to square or narrow channels can significantly increase heat transfer by up to 50%. This improvement is greater for angled ribs compared to straight ribs. Angled ribs create a swirling motion within the channel that mixes the fluid more effectively, leading to better heat transfer. Interestingly, the heat transfer on the smooth side of wider channels (aspect ratio of 2 and 4) is higher than in narrow channels. In these wider channels, adding straight ribs (90 degrees) can even double the heat transfer coefficient on the smooth side.

2.3. High-Performance Ribs

Orthogonal ribs are known to improve heat transfer compared to smooth channels, and ribs positioned at an angle relative to the flow direction can increase the heat transfer of ribbed channels even further.

2.3.1. V shaped ribs

Han et al. [18] examined how various high-performance rib arrangements could boost heat transfer. **Figure 2.9.** illustrates the rib layouts they investigated, and **Figure 2.10.** displays the comparison of heat transfer augmentation across different rib designs. A graph that shows how heat transfer and pressure loss are affected by rib angle and flow rate (Reynolds number) in channels. The graph plots Nusselt number ratios (heat transfer) and friction factor ratios (pressure loss) for channels with ribs at 90, 60, and 45 degrees, all across different flow rates. The results show that as the flow rate increases (higher Reynolds number), the heat transfer compared to a smooth channel (Nusselt number ratio) goes down, while the pressure loss caused by the ribs compared to a smooth channel (friction factor ratio) goes up.

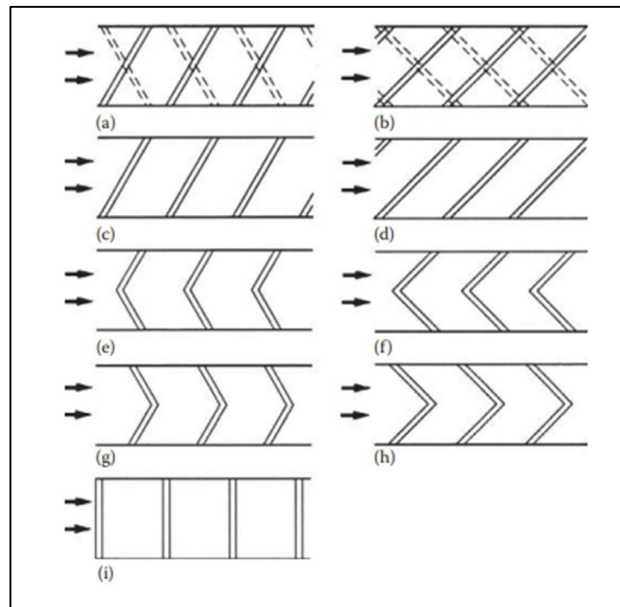


Figure 2.9: Different rib configurations studied by Han et al. [18].

Higher flow rates (higher Reynolds number) lead to diminishing gains in heat transfer efficiency from ribs. Data also shows a trade-off: larger heat transfer increases due to rib configuration (higher Nusselt number ratio) are linked to greater pressure loss (higher friction factor ratio). For instance, the 60° forward V-shaped rib offers a significant heat transfer improvement (2.7-3.5 times better) but with a substantial pressure drop penalty (8-11 times more than a smooth channel). In contrast, the 60° crossed rib provides a more modest heat transfer increase (1.8-2.2 times) with a less severe pressure penalty (5-7 times increase).

The 90° rib achieves heat-transfer gains similar to those of the 60° and 45° crossed ribs, but the friction factor with the 45° crossed rib is lower compared to the 90° and 60° crossed ribs. The 60° inverted V rib delivers marginally less heat transfer but considerably higher pressure drop than the 60° V rib. Parallel ribs generally exhibit a higher heat-transfer coefficient than their corresponding crossed rib variants. The 60° parallel rib offers slightly lower heat transfer compared to the 60° V rib, with a comparable pressure-drop penalty. The performance of the 45° V-shaped rib is approximately on par with that of the 45° parallel rib, both of which outperform the 45° crossed rib, the 45° inverted V rib, and the 90° rib.

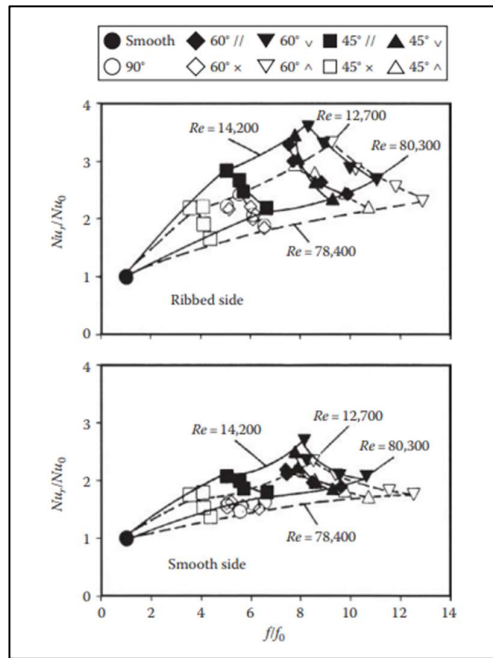


Figure 2.10: Heat-transfer performance for different rib configurations [18].

Figure 2.11. shows a relationship between heat transfer and surface roughness in channels with ribs at different angles (90, 60, and 45 degrees). The graph plots a value called "heat-transfer roughness function" (G) versus another value called "roughness Reynolds number" (e^+). This is done for a range of flow rates (Reynolds numbers). The figure also includes similar lines for an "average heat-transfer roughness function." This average value is based on heat transfer on both the ribbed side and the smooth side of the channel. In all the cases examined (ribs at different angles), the heat transfer roughness function increases as the roughness Reynolds number goes up. This means that rougher surfaces (higher roughness Reynolds number) lead to higher heat transfer roughness function.

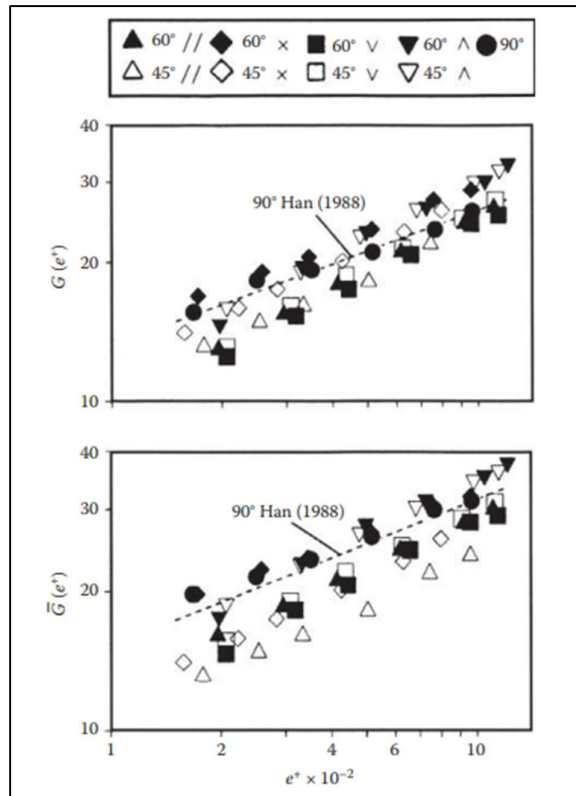


Figure 2.11: Heat-transfer correlation for different rib configurations [18].

2.3.2. V-Shaped Broken Rib

Han and Zhang [19] investigated high-performance broken parallel and V-shaped ribs. **Figure 2.12.** illustrates the rib configurations they studied, while **Figure 2.13.** shows the Nusselt number ratio of ribbed versus smooth sides against the friction factor ratio (heat-transfer performance curve) for parallel broken and V-shaped broken ribs across a range of Reynolds numbers from 15,000 to 80,000. Data for parallel continuous and V-shaped continuous ribs are also included for direct comparison.

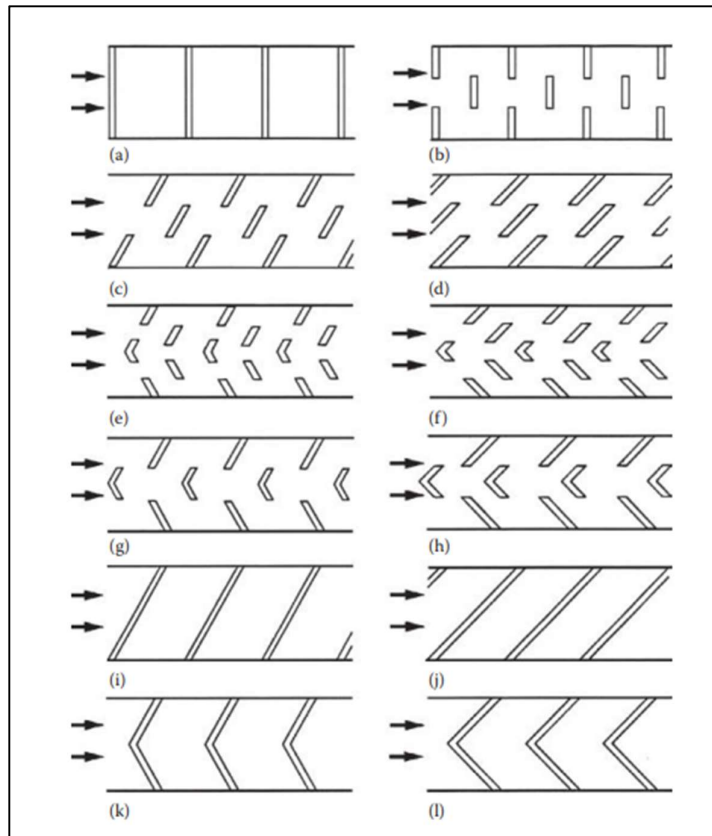


Figure 2.12: Rib configurations used by Han and Zhang [19].

Rising Reynolds numbers lead to decreasing Nusselt number ratios and increasing friction factor ratios, confirming prior observations. Ribbed-side Nusselt number ratios for 60° and 45° broken (parallel or V-shaped) configurations are significantly higher than those for their continuous counterparts. However, friction factor ratios show no difference between broken and continuous rib configurations.

Among broken ribs at 60 and 45 degrees (parallel or V-shaped), heat transfer and pressure drop behave similarly, with the 60° V-shaped broken rib showing the best performance. In contrast, 60 and 45 degree continuous ribs (parallel, V-shaped), along with 90 degree broken ribs, exhibit comparable heat transfer characteristics, with the 60° V-shaped continuous rib offering the highest heat transfer.

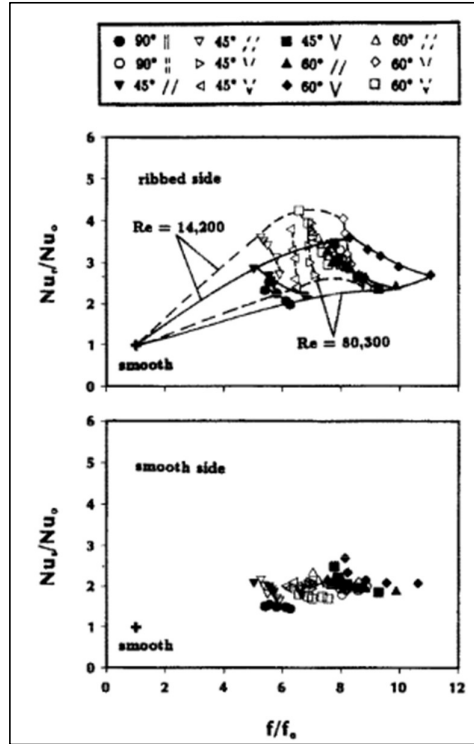


Figure 2.13: Average Nusselt number ratio vs average friction factor ratio for broken ribs [19].

2.4. Effect of High Blockage-Ratio Ribs

Most studies involving ribs typically use a rib size that blocks about 10% of the channel, with a spacing-to-height ratio of 10. But in smaller gas turbine engines, the ribs can be much taller, leading to greater blockages. Additionally, when the ribs are angled relative to the flow, you can fit more ribs into the cooling channel, even with shorter distances between them (smaller p/e ratios). Metzger et al. [20] demonstrated that when more ribs are placed within a given surface area, the heat transfer from the top surface of the rib becomes more significant. **Figure 2.14.** depicts the ratio of the Nusselt number for the top of the rib to that of the space between the ribs. The figure

also reveals that this Nusselt number ratio increases as the pitch-to-height ratio (p/e) rises for shorter ribs, whereas this ratio is less impacted by large blockages (where $e/D = 0.25$) and angled ribs. Metzger et al. further highlighted the relative impact of the rib-top Nusselt number on the overall average Nusselt number.

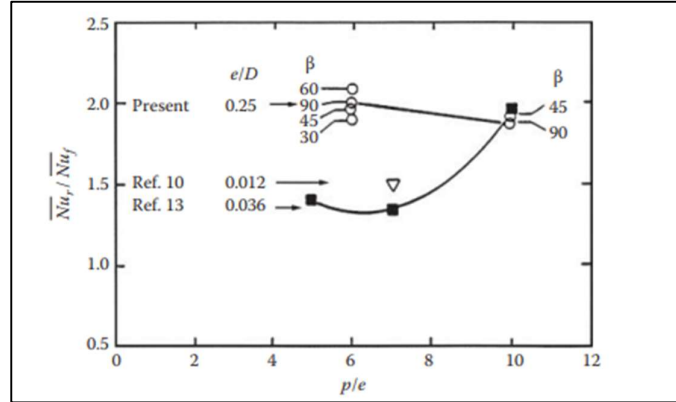


Figure 2.14: Effect of different rib arrangements on the average Nusselt number ratio [20].

The portion of heat transfer represented by the rib-top Nusselt number goes up as the p/e ratio goes down. According to the results, when the p/e ratio is below 6, the heat transfer from the rib-tops can account for more than half of the total heat transfer. The research team at Northeastern University conducted multiple studies on high-blockage ribs. Taslim and Wadsworth [20] explored three rib configurations with blockage ratios (e/D) of 0.133, 0.167, and 0.25, and with pitch-to-height ratios (p/e) of 5, 7, 8.5, and 10. **Figure 2.15.** presents the average Nusselt numbers in a channel with an e/D ratio of 0.25. The experimental setup used a channel with a cross-section of 3.81 cm $\times$ 3.81 cm. The two opposite sides of the channel are lined with ribs in a staggered arrangement, indicating that the ribs are not aligned directly above each other.

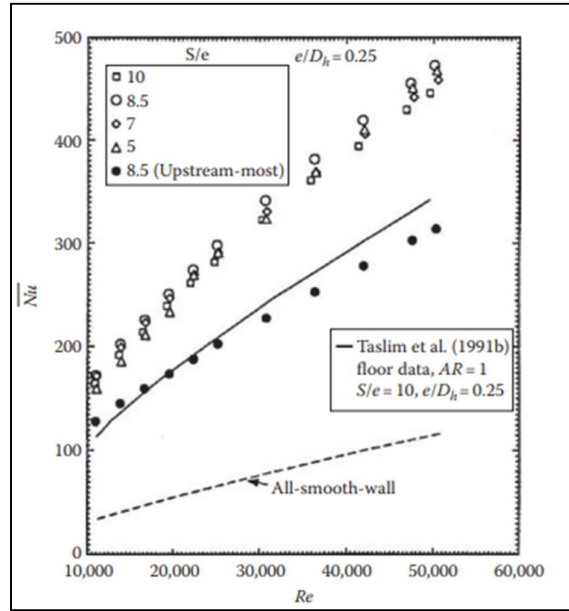


Figure 2.15: Effect of rib spacing on average Nusselt number [20].

Findings from this high-blockage rib study suggest that changes in the average Nusselt number are not heavily influenced by rib spacing. The ribs positioned upstream have a lower heat-transfer coefficient, consistent with data from non-staggered ribs. The higher readings from this study suggest that the staggered rib arrangement boosts flow mixing, which in turn leads to improved heat transfer.

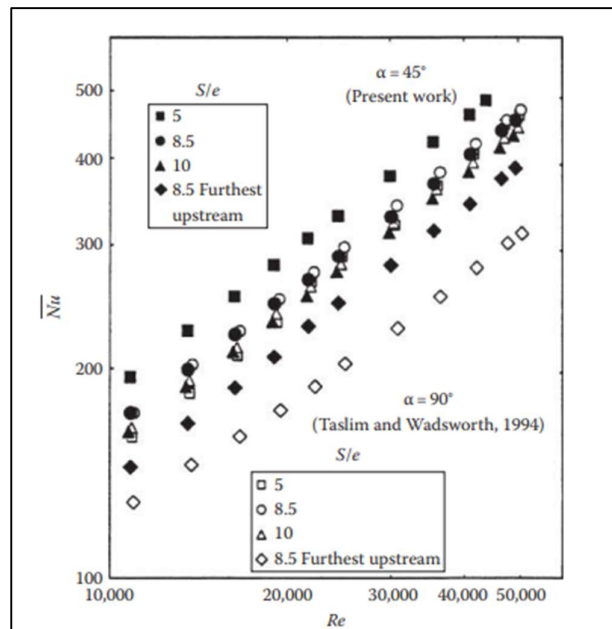


Figure 2.16: Effect of rib angle and rib spacing on average Nusselt number [21].

Taslim and Lengkong [21] investigated the heat-transfer coefficient in a channel featuring high-blockage staggered ribs positioned at a 45° angle. **Figure 2.16.** presents the average Nusselt number in the channel for various rib spacings with an e/D ratio of 0.25. The results indicate that rib spacing has a greater impact when the ribs are angled at 45° . The configuration with a closer spacing of 5 rib heights exhibits the highest heat-transfer coefficient among the group.

Figure 2.17. depicts the average friction factors in the channel for this rib setup. In this angled-rib configuration, a closer spacing lead to a higher friction factor. However, with wider spacings ($S/e = 8.5$ and 10), the friction factor is lower for the angled ribs. The change in the friction factor with varying Reynolds numbers is minimal in these rib configurations.

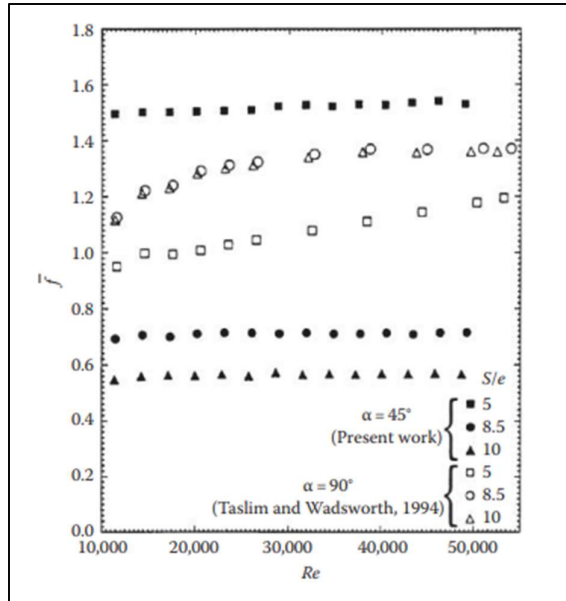


Figure 2.17: Effect of rib angle and rib spacing on friction factor [21].

2.5. Effect of Number of Ribbed Walls

Ribs can change the way the flow moves, potentially affecting both the overall flow distribution and turbulence patterns when they are present. Adding ribs to one wall can impact the heat-transfer coefficient on an adjacent smooth wall. Chandra and Cook [22] examined how the number of ribbed walls influences heat transfer and friction in a square channel. **Figure 2.18.** illustrates the different rib configurations they used.

The purpose of their study was to analyze how the number of ribbed walls affects the average heat-transfer coefficient at the wall centerline.

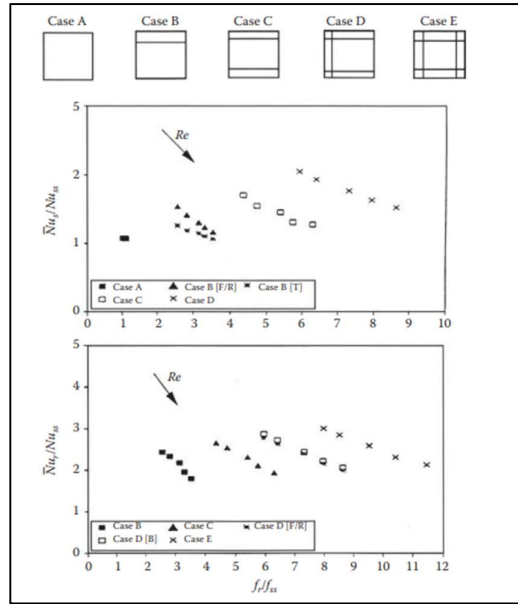


Figure 2.18: Effect of the number of ribbed walls on heat-transfer performance [22].

Since adding a rib to one surface can affect the overall flow distribution and turbulence levels, the smoothness or roughness of adjacent walls can alter the heat-transfer behavior of a surface. **Figure 2.18.** illustrates various scenarios: case A had a square channel with all smooth walls, case B had ribs on one wall, case C had ribs on two opposite walls, case D had ribs on three walls, and case E had ribs on all four walls. It also illustrates the changes in the Nusselt number ratio in relation to the friction factor ratio for both the smooth and ribbed sides of the channel. As mentioned earlier, this type of graph is known as a heat-transfer performance curve. The results indicate that as the Reynolds numbers increase, the Nusselt number ratio tends to decrease, while the friction factor ratio tends to increase. In case C, where there are transverse ribs on two opposite walls, the ribbed-side heat transfer is enhanced by a factor of 2.5–1.8, while the pressure drop increases by 4.5–6.6 times, as the Reynolds number rises. In contrast, case E, with ribs on all four walls, achieves a ribbed-side heat transfer increase of 2–3 times and a pressure drop increase of 8–12 times. The impact of ribbed surfaces on heat transfer diminishes as the Reynolds number increases. In contrast to heat transfer, the friction factor tends to be higher at higher Reynolds numbers. The presence of more ribbed walls within the channel leads to a rise in both the heat transfer

coefficient and pressure drop. This increase in heat transfer applies to both the smooth and ribbed walls. Notably, the smooth wall sees a rise of roughly 13% in its heat transfer coefficient for each additional ribbed surface, while the ribbed wall experiences a 10% increase for each extra rib added.

2.6. RANS CFD Studies

Since almost all CFD studies in the literature are LES, URANS and RANS/LES hybrid models, only the study by Keshmiri et al. [23] was added to the literature review for the sake of comparison.

Keshmiri et al. compared RANS models and LES models in their study in 2016. In this 2D study, they chose a test case with a blockage ratio of 0.1 and a P/e ratio of 9. They examined k- ω SST, v^2f , theta f and RSM models in their RANS analysis as a stationary at 30000 Reynolds number. They used STAR-CD and Code_Saturne CFD codes and ran the analyses.

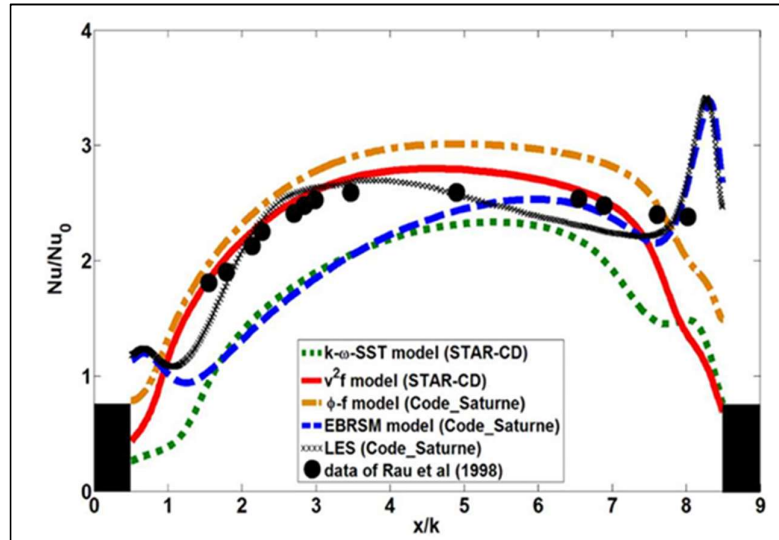


Figure 2.19: CFD and test results comparison [23].

The results of the study are presented in **Figure 2.19**. As can be seen from the study, there is a very low difference between the LES results and the test. However, while there was a serious shift in the RANS models, the v^2f model offered the best results. This study actually reveals that RANS models do not give good results in problems with reattachment flow.

2.7. Rotational Effects on Heat Transfer & Flow Field

The analysis of coolant flow in rotor airfoils is complicated by rotational forces, which significantly impact heat-transfer distribution within the coolant passages. Rotational forces, including Coriolis and rotational buoyancy forces, alter the flow field and thus affect heat transfer. Studies have shown that coolant flowing through a rotor blade cooling passage experiences strong Coriolis and rotational buoyancy forces. These forces induce secondary flow and cause the core flow to be distributed asymmetrically within the channel. In serpentine multipass channels, the flow separates and recirculates in the turn regions, and the flow development downstream of sharp turns significantly affects both the local heat-transfer coefficient distribution and the overall channel heat transfer. Flow in a rotating channel differs markedly from flow in a stationary, non-rotating channel. In rotating channels, secondary flow redistributes velocity and changes the patterns of random velocity fluctuations in turbulent flows. The rotation number, denoted as Ro , is defined by the equation $Ro = \omega D/V$, where ω represents the rotation speed, D is the channel depth, and V is the average axial velocity.

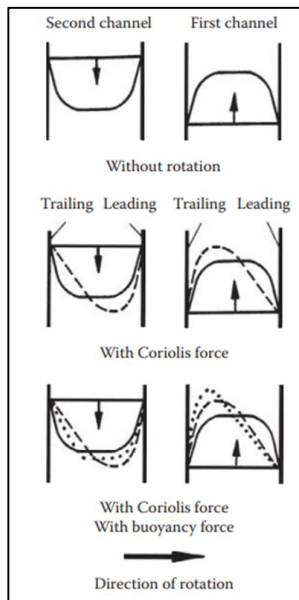


Figure 2.20: Schematic view of Coriolis and buoyancy effects on radially inward and outward flows [24].

Figure 2.20. demonstrates the combined effects of Coriolis forces and rotational buoyancy on flow distribution. In the first channel, the Coriolis force shifts the core flow towards the trailing wall during radial outward flow. If both the trailing and leading walls are symmetrically heated, the faster-moving coolant near the trailing wall will be cooler than the slower-moving coolant near the leading wall. Rotational buoyancy, driven by strong centrifugal forces, pushes the cooler, heavier fluid away from the center of rotation. In the first channel, rotational buoyancy similarly affects the flow as the Coriolis force, increasing flow near the trailing wall. In contrast, in the second channel, the Coriolis force favors the leading side, while rotational buoyancy works to make the flow distribution more uniform within the duct.

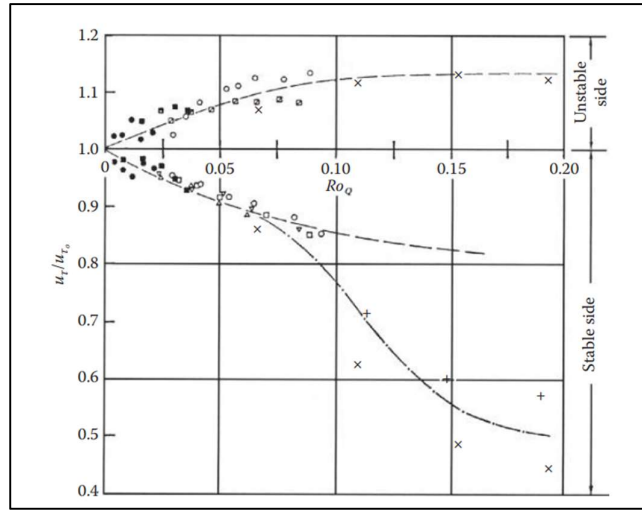


Figure 2.21: Impact of rotation on a wall shear stress [25].

Figure 2.21. illustrates how rotation impacts wall shear stress: it increases on the unstable side and decreases on the stable side as rotational effects intensify. It is important to note that wall shear stress is comparable to surface heat transfer, meaning areas with high shear stress will correspondingly have high heat-transfer coefficients.

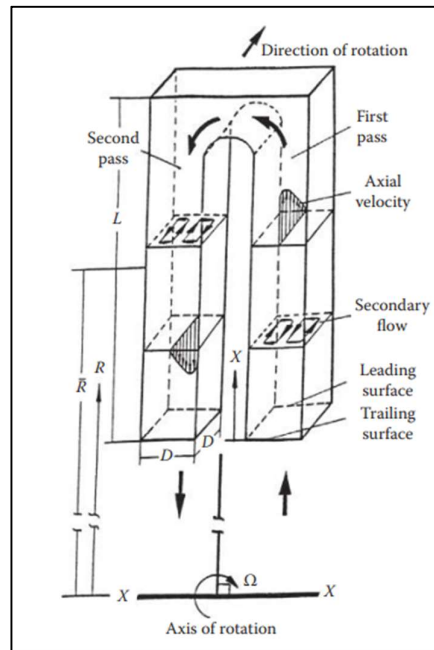


Figure 2.22: Schematic view of a two-pass rotating coolant flow distribution [26].

Secondary flows in a two-pass channel differ between radial outflow and radial inflow passes. Figure 3. illustrates the schematic secondary flow and axial flow distribution in a rotating two-pass channel. In the first pass, the flow is radial outward, and in the second pass, it is radial inward. Since the direction of the Coriolis force depends on both the rotation and flow directions, it changes between the two passes. Although the rotation direction remains constant for both channels, the flow direction reverses from the first channel to the second at the 180° bend. Consequently, the Coriolis force acts in opposite directions in these two channels, resulting in different secondary flows.

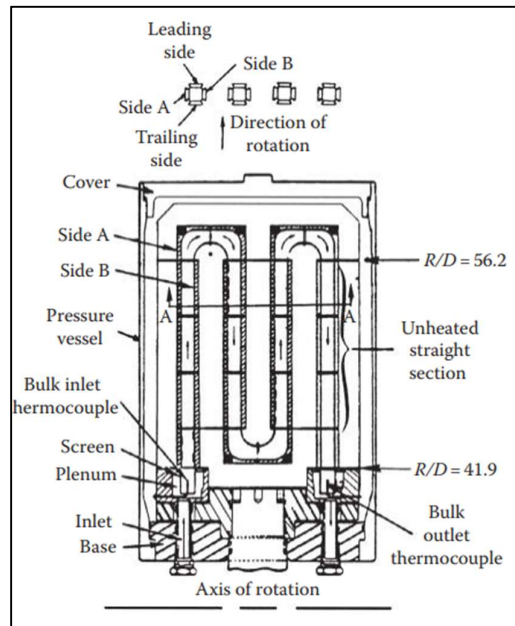


Figure 2.23: Serpentine test coupon [27].

Figure 2.23. depicts a four-pass square cross-section channel used by Wagner et al. (1991). This rotating test facility replicates the conditions found in a serpentine rotor blade coolant passage. The first, second, and third passes were equipped with instruments to study heat transfer, but the presented results focused mainly on the first two passes. The results were given in the nondimensional form of the Nusselt number ratio (Nu/Nu_∞), where Nu_∞ represents the Nusselt number for fully developed pipe flow. Wagner et al. (1991) rotated the test assembly at speeds up to 1100 rpm and operated it at approximately 10 atmospheric pressures.

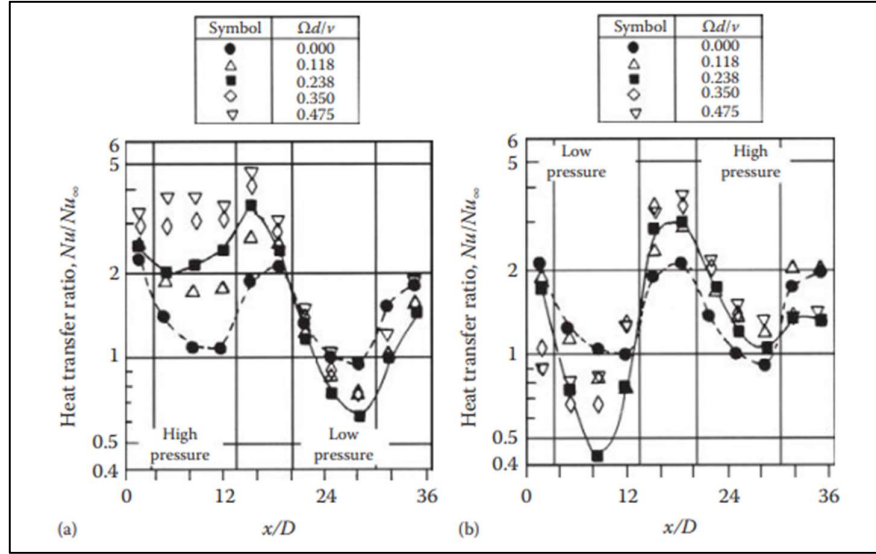


Figure 2.24: Rotation number effect on Nusselt number ratio for $Re = 25,000$ (a) Trailing surface and (b) Leading surface [27].

The rotation number is a measure of the relative strength of the rotational Coriolis force compared to the bulk flow inertial force. A higher rotation number indicates a greater influence of rotational effects on flow and heat transfer. **Figure 2.24.** demonstrates that increasing the rotation rate significantly enhances heat transfer on the trailing surface of the first pass ($x/D < 12$), while the increase is comparatively less on the leading surface of the second pass ($x/D > 20$). In the first pass, heat transfer increases by more than a factor of 3.5 at the highest rotation number, compared to the stationary heat-transfer value. In contrast, the leading surface of the second pass shows an increase of approximately 1.5 times the stationary duct results. Although Coriolis effects on heat transfer are expected to be similar for both passes, the differences in heat-transfer coefficient distribution between the outward (first pass) and inward (second pass) flows are attributed to the varying effects of buoyancy in radial outward and inward flows. The higher heat-transfer coefficients on the trailing surface of the first pass are primarily due to two factors. First, coolant impingement on the trailing surface driven by Coriolis-induced secondary flow, which creates a thinner boundary layer, and second is the coolant temperature profile that generates favorable rotational buoyancy-induced flow near the destabilized thin boundary layer on the trailing surface. Conversely, the reduction in heat-transfer coefficients near the inlet of the channel on the leading surfaces of the first pass is caused by a thicker, stabilized boundary layer. The increase in heat-transfer coefficients near the end of the channel

is due to a stronger centrifugal buoyancy-induced, destabilized wall turbulence boundary layer. Additionally, the effects of the turn can also enhance heat transfer in the later part of the first pass. In the second pass, where the coolant flows radially inward (opposite to the first pass), the Coriolis force acts in the opposite direction, shifting the bulk flow toward the leading surface. This results in cooler fluid moving faster near the leading surface and warmer fluid moving slower near the trailing surface. As a result, the heat-transfer coefficient is higher on the leading surface compared to the trailing surface.

3. BASE CASE

3.1. Test Setup

When selecting the base case, attention was paid to whether the parameters required for validation were clearly provided. Two articles published using the Von Karman Institute RC-1 rotating test system were selected as base cases. Since these articles have the same test coupon and intersecting test conditions, it is possible to examine them comparatively.

The first work is "Two-Dimensional Heat Transfer Distribution Of a Rotating Ribbed Channel At Different Reynolds Numbers " by Mayo and Arts [28]. In this study, the test setup was examined from the heat transfer perspective. The second publication is " Flow field investigation in rotating rib-roughened channel by means of particle image velocimetry " by Coletti et al [29]. In this study, the flow field was examined by taking measurements with PIV.

Extensive studies have been done on the mentioned studies in the literature and they have a high number of citations, and it can be seen that many LES studies have been conducted.

The RC-1 rotating test system consists of a 2.5 meter diameter wooden table, the test section and instrumentation placed on this table. The table is rotated by an electric motor, and the table can rotate clockwise and counterclockwise. Air was used as the fluid and the air was supplied with the help of a blower. Honeycomb and wire screen are positioned right at the entrance of the test section and straighten the air.

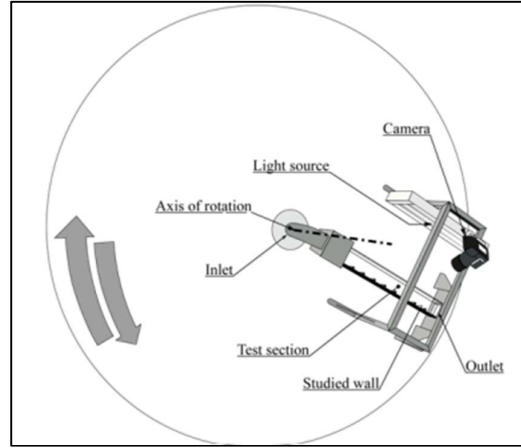


Figure 3.1: Top view of the table [28].

The test system is shown in **Figures 3.1.** and **Figure 3.2.** The sizes of the test system is given in **Figure 3.2.** The test system is 750 mm long and 75*83 mm² in size. The 20 micron thick Inconel sheet was attached to the 75 mm wide walls using epoxy adhesive. The height of each rib H is 8 mm, and the distance P between 2 consecutive ribs is 80 mm. In this case, it can be seen that P/D is 10 while H/D is 0.1. The top of the Inconel sheet was covered with uniform matte black paint, and TLC was applied on it. The regions between the 6th and 7th ribs and the 7th and 8th ribs were examined.

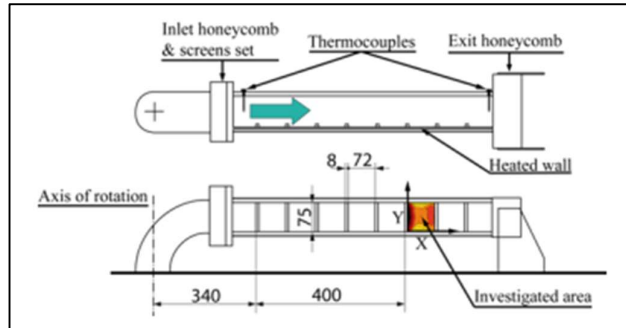


Figure 3.2: Front view of the test section [28].

Special mounts have been designed to withstand the high g force of the test system. The color change of TLC under heat transfer was visualized with a Nikon camera. A fluorescent bulb was used as the light source. The temperatures at the entrance and exit of the channel were measured with two thermocouples.

The test matrix in Mayo's study is given below. The test matrix was created according to Reynolds and Rotation dimensionless numbers. The test matrix can be seen in **Table**

3.1. Reynolds and Rotation numbers are given in equation (3.1) and equation (3.2) respectively. Since the test system can rotate in both directions, whether the rotation number is positive or negative is determined by the direction in which the test system rotates. In case the ribbed wall has a trailing edge, the rotation number is taken as positive, otherwise it is taken as negative.

Table 3.1: Test Matrix [28].

Ro \ Re	0.00	±0.06	±0.12	±0.18	±0.24	±0.30
15000						
20000						
30000						
40000						

$$Re = \frac{\rho U_b D_h}{\mu} \quad (3.1)$$

$$Ro = \frac{\Omega D_h}{U_b} \quad (3.2)$$

The definition of heat transfer coefficient is given in equation (3.3). Some of the energy given by Inconel foil is lost through radiation and conduction. The remaining energy is transferred to the air and the heat transfer coefficient is calculated according to this transferred value. Wall temperature was measured by TLC. The uncertainty due to the measurement method was calculated as 1.7 K.

$$h = \frac{q_{conv}}{T_w - T_b} \quad (3.3)$$

$$q_{conv} = q_{joule} - q_{cond} - q_{rad} \quad (3.4)$$

$$q_{cond} = \frac{k_{xps}}{t_{xps}} (T_w - T_{ext}) \quad (3.5)$$

$$q_{rad} = \varepsilon_{TLC} \sigma (T_w^4 - T_p^4) \quad (3.6)$$

Dittus-Boelter correlation was used when calculating the enhancement factor. Uncertainty analysis was performed according to the Kline and McClintok method. According to this analysis, the enhancement factor uncertainty was calculated as 15%.

$$EF = \frac{Nu}{Nu_0} \quad (3.7)$$

$$Nu_0 = 0.023 x Re^{0.8} x Pr^{0.4} \quad (3.8)$$

The measurement methods used in Coletti et al.'s study are given below. The same test system was used, the TLC module was disassembled, and the PIV measurement module was added to the system.

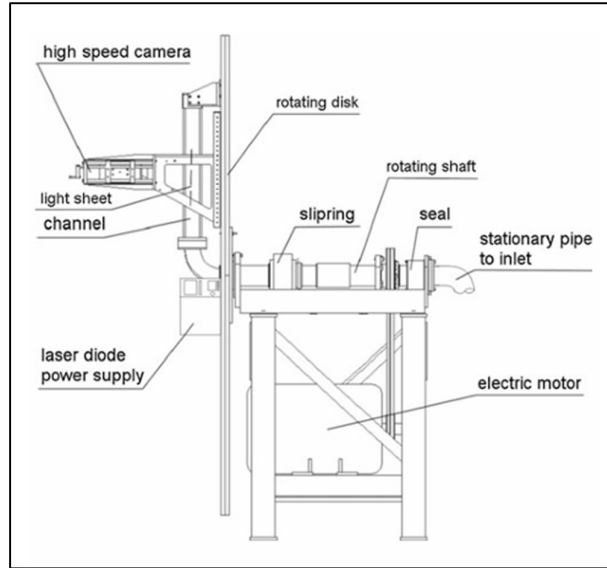


Figure 3.3: Schematic of the rotating facility, lateral view [29].

The side view of the rotating test system is given in detail in **Figure 3.3**. The laser and high-speed camera, which are parts of the PIV module, can be seen in figure.

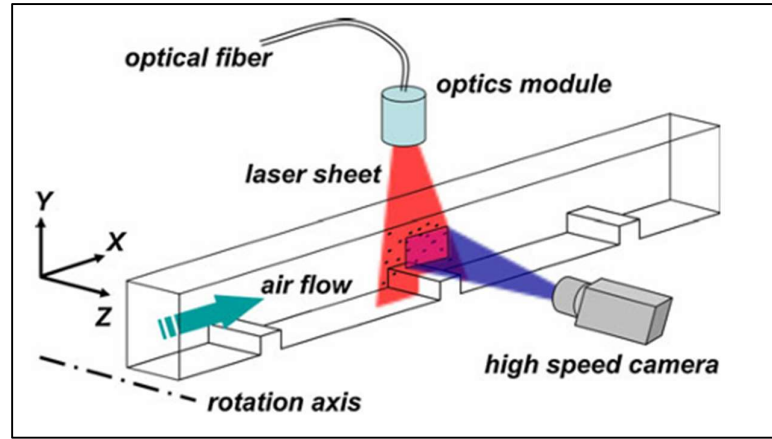


Figure 3.4: Sketch of the PIV layout [29].

The 2-D velocity fields are measured using a PIV system installed on the disk. The light source is a 25 W air-cooled laser diode that operates continuously at a wavelength of 806 nm. PIV layout is given in **Figure 3.4**.

3.2. Heat Transfer Results

The results given in this section are given in terms of enhancement factor. First, stationary tests were conducted to compare with studies in the literature, shown in **Figure 3.5**.

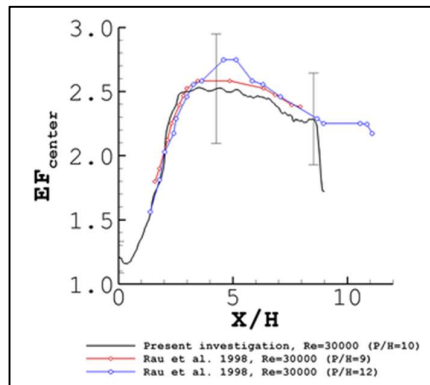


Figure 3.5: Literature comparison [29].

In this study, the average EF was around 2.6 for Re: 15000 and around 2.1 for Re: 30000. Similar values have been seen in studies in literature.

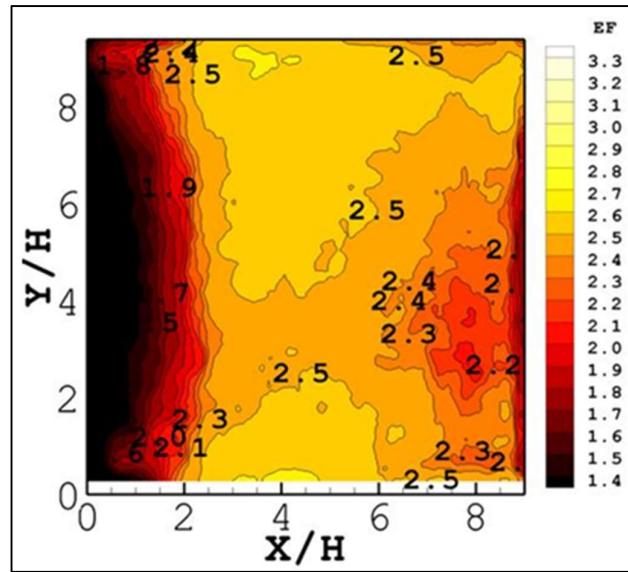


Figure 3.6: Enhancement factor distribution for Re=30000, Ro=0.0 [28].

As an example, **Figure 3.6.** shows the TLC measurement taken at stationary condition at 30 000 Reynolds number. As can be seen, TLC measurement gave a very detailed distribution.

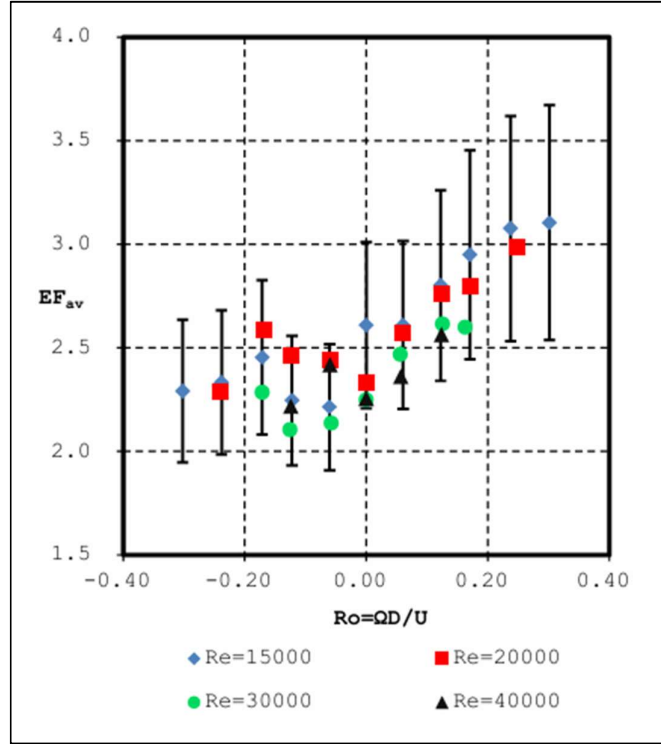


Figure 3.7: Averaged Enhancement Factor as a function of Re and Ro [28].

HTC data extracted at all points in the test matrix are summarized in **Figure 3.7**. In the results section, the data in this table and the CFD results will be compared and analyzed.

3.3. Flow Field Measurement

In the study conducted by Coletti et al. [29], the test matrix was slightly narrowed. Situations where the Rotation number was 0.3, 0 and -0.3 under Reynolds 15 000 condition were examined.

When we examine the mentioned study, we can see that normalized velocity and turbulence intensity values are measured at the entrance. Y/D value represents the normalized channel height.

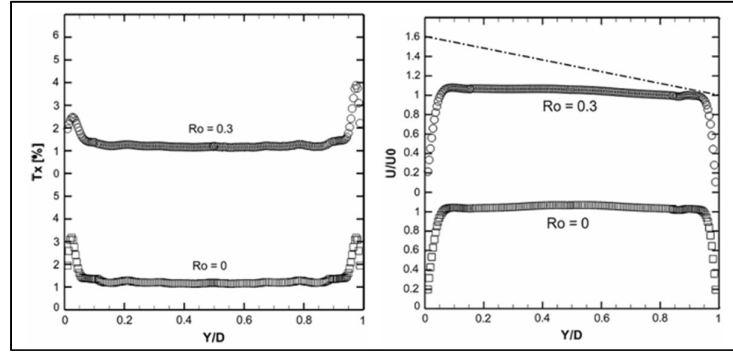


Figure 3.8: Normalized mean velocity and turbulence intensity measurement at the entrance of channel [29].

It was understood whether the flow had a periodic structure along the channel by drawing a normalized average velocity curve at the beginning of the 6th and 7th ribs. These curves can be seen in **Figure 3.8**. While measurements were taken, measurements were taken up to 3 times the rib height and the y-axis was normalized according to this height.

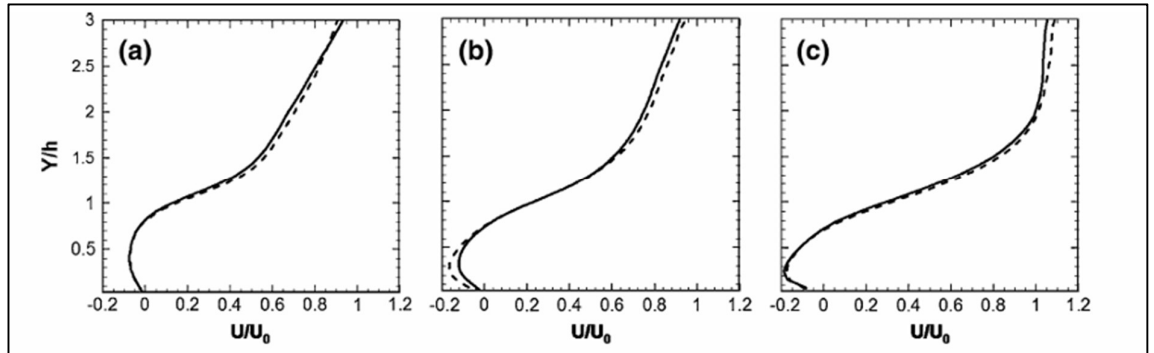


Figure 3.9: Normalized mean velocity measurement for a channel periodicity a) Ro : -0.3 b) Ro : 0 c) Ro : +0.3 [29].

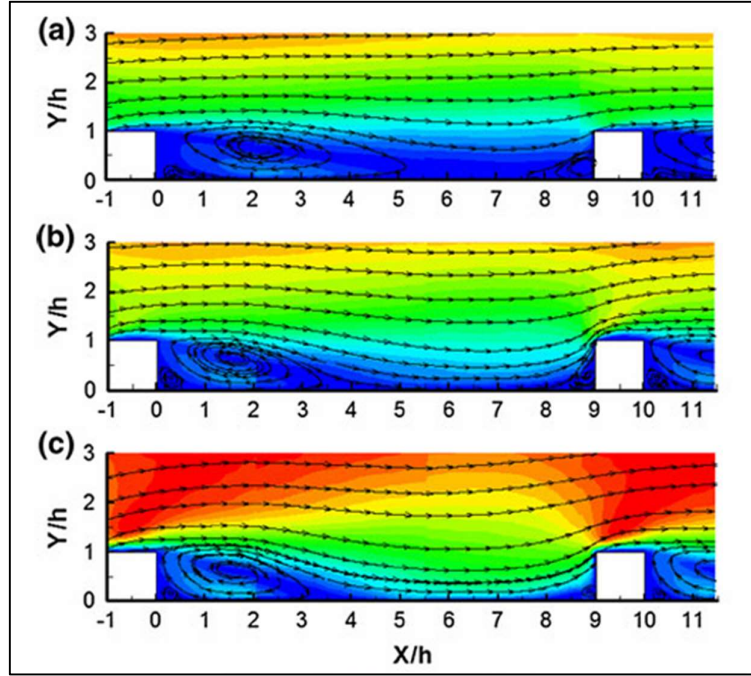


Figure 3.10: Two-dimensional streamlines and contours of mean in-plane velocity: stabilizing rotation a) $Ro: -0.3$ b) $Ro: 0$ c) $Ro: +0.3$ [29].

Figure 3.10. shows streamline contours for different rotation conditions. Reattachment values under these conditions will be examined in detail in the results section. The outputs of these two studies will be shown together with the CFD results in the upcoming sections.

4. NUMERICAL APPROACH AND SOLUTIONS

4.1. CFD Tools

Analyses were made in the commercial Ansys Fluent 2021 R2 package program. Ansys Fluent is multi-purpose CFD software for simulating fluid flow, heat and mass transfer, chemical reactions, and various other phenomena. Fluent solves conservation equations for mass and momentum in all types of flows. For flows that involve heat transfer or compressibility, it also solves an energy conservation equation. The Reynolds-averaged Navier-Stokes (RANS) equations describe the transport of averaged flow properties, modeling the entire spectrum of turbulence scales. This RANS-based approach significantly lowers computational demands and is commonly used in practical engineering applications. Mathematical explanations of the forms of Navier Stokes equations and turbulence models solved by Fluent can be found in the Fluent Theory Guide [30]. With the new versions of Fluent, it is possible to create polyhedral mesh much faster and with higher quality, thanks to the mesh module that comes in the package of the program. In this study, a grid was created with Fluent's built-in mesh module.

4.2. Geometry

While creating the geometry for the CFD study, the parameters in the base case were followed exactly, and no scaling were made. This situation led to a flow domain on the order of meters. The dimensions of the CFD domain are given below. A better mesh was applied to the volume close to the wall in order to capture the flow physics of this region. This was provided through the BOI. The dimensions are given in terms of mm.

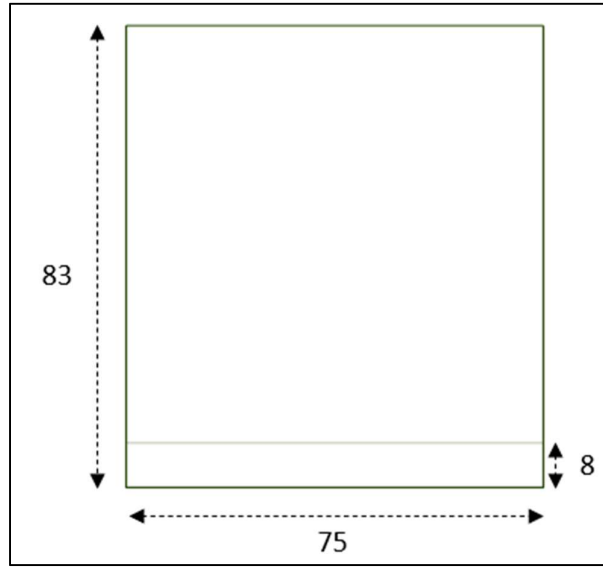


Figure 4.1: Inlet view of fluid domain.

The e/D_h ratio of the channel is 0.1. Since rib is square, its height and width are the same, and the e value represents the height and equals to 8 mm. The width of the channel is 75 mm and the height is 83 mm.

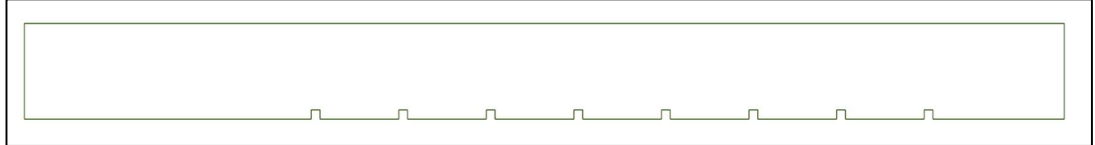


Figure 4.2: Side view of whole fluid domain.

The whole fluid domain can be seen at **Figure 4.2**. The total length of the fluid domain is 950 mm.

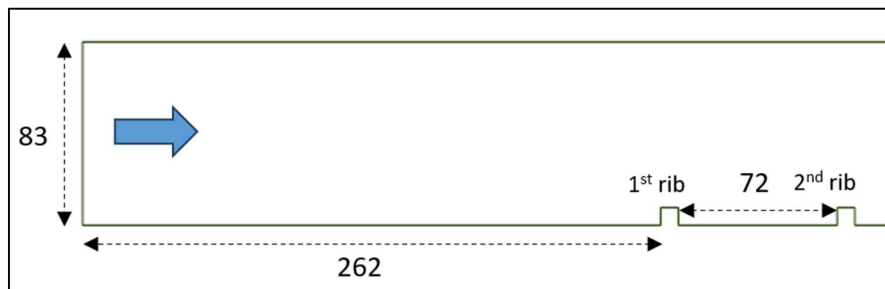


Figure 4.3: Entrance region of fluid domain.

The total length of the entrance region, which is the distance between the channel inlet and the first rib, is 262 mm.

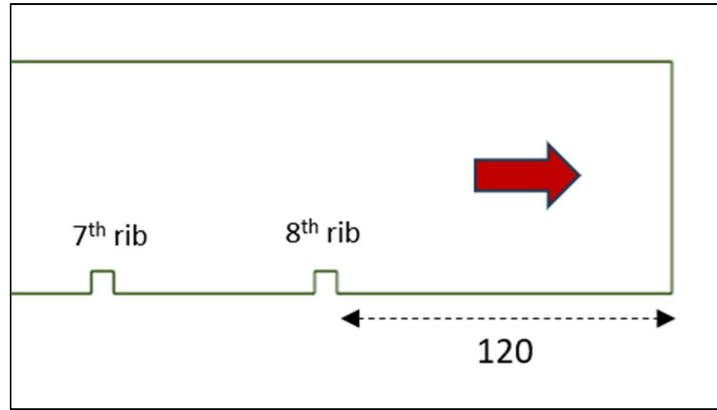


Figure 4.4: Exit region of fluid domain.

The distance between two ribs is 72 mm. However, the distance from the start of one rib to the start of the next rib is 80 mm. In other words, the pitch to rib height ratio, which is the p/e ratio, is 10. The distance from the end of the last rib to the channel exit is 120 mm.

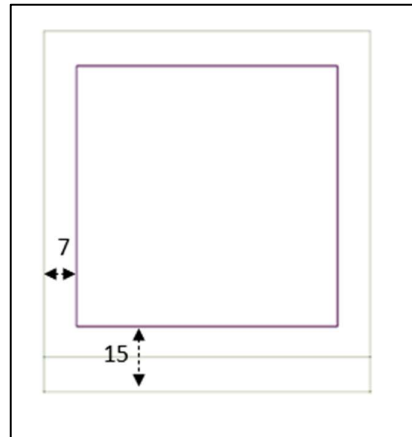


Figure 4.5: Dimensions of BOI.

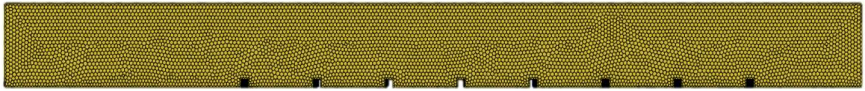
The distance between the BOI domain and the side surfaces of the channel is 7 mm, and the distance to the ribbed surface is 15 mm.

4.3. Grid Generation

In this study, polyhedral mesh was used due to some of its prominent features. Polyhedral meshes offer several significant advantages in CFD analyses. Polyhedral meshes generally produce higher quality elements compared to tetrahedral meshes, with faces that better align with the flow, reducing numerical diffusion and enhancing accuracy. This improved alignment and shape help in better capturing the physics of the problem, leading to more accurate simulations, particularly in complex geometries and turbulent flows. Additionally, polyhedral meshes often require fewer cells to achieve the same level of accuracy as tetrahedral meshes, which can significantly reduce computational cost and memory usage. They also tend to improve the convergence behavior of solvers due to better aspect ratios and more uniform distribution of cell sizes, contributing to more stable and efficient numerical computations. The flexibility of polyhedral meshes allows them to conform to complex geometrical shapes more easily than structured meshes, making them particularly useful for simulations involving intricate geometries. Furthermore, these meshes tend to have lower skewness compared to tetrahedral cells, improving the overall quality of the mesh and reducing errors, thus enhancing the robustness of the simulation.

Studies were carried out with 3 different meshes. After a few trial runs, meshes were created based on y^+ : 5, y^+ : 1 and y^+ : 0.2 these are named coarse, mid and fine, respectively. The metrics and images of each mesh are given in **Table 4.1.** and **Table 4.2.** below.

Table 4.1: Side views of meshes.

Coarse	
Mid	
Fine	

Side views of the meshes are given in **Table 4.1**. It is noticeable that the mesh density increases with the increase in the number of elements.

Table 4.2: Various views of meshes.

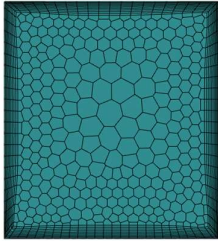
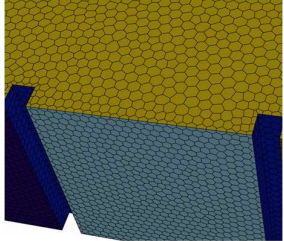
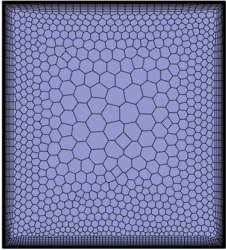
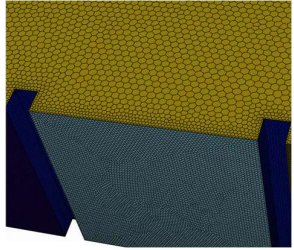
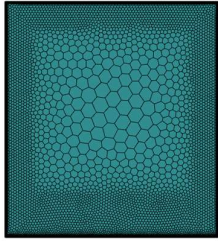
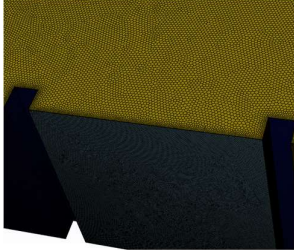
	Front-BOI	Bottom-Isometric
Coarse		
Mid		
Fine		

Table 4.2. shows the front, isometric and side-inflation view of the mesh. In the front view, the effect of BOI can be seen. Face sizing has been given to the ribbed surface to reduce the aspect ratio value. In the isometric image, the density difference between the ribbed surface and the side surfaces can be seen as well. The mesh on the boundary layer can be seen in side-inflation images.

Table 4.3: Mesh parameters.

	BOI size	Face size	Max cell size	First layer thickness	Layer number	Layer growth rate
Coarse	4 mm	3 mm	9 mm	0.3 mm	10	1.1
Mid	2.5 mm	1 mm	4 mm	0.06 mm	14	1.2
Fine	1 mm	0.4 mm	3 mm	0.012 mm	14	1.25

Mesh parameters can be seen in **Table 4.3**. Coarse, mid and fine meshes have an inflation layer with a first layer thickness of 300 microns, 60 microns and 12 microns, respectively. In order to keep the total inflation layer thickness above a certain value, the number of layers and the layer growth rate were increased as the first layer thickness became thinner. Along with the first layer thickness, the face sizing on the ribbed surface was also reduced and the aspect ratio and skewness values of the mesh were preserved.

Table 4.4: Mesh metrics.

	Aspect Ratio	Skewness	Cell Count
Coarse	49	0.51	389 277
Mid	71	0.61	2 523 229
Fine	127	0.59	17 279 929

Mesh metrics of 3 different meshes can be seen in **Table 4.4**. Mesh quality was checked according to 2 main mesh metrics: aspect ratio and skewness. An attempt was made to keep the aspect ratio value below 150 and the skewness value below 0.7.

As said in the previous paragraphs of this section, y^+ : 5 is targeted for coarse mesh, y^+ : 1 for mid mesh, and y^+ : 0.2 for fine mesh. Every 5 times increase in the Y^+ value increased the number of elements by approximately 7 times.

4.4. Turbulence Models

Four different turbulence models with different features were used in CFD analyses. The turbulence models used and brief features of these models are summarized below. Since the most important feature of $k-\omega$ GEKO is tune capability with parameters pre-defined to the model, near wall tune parameter which is called as C_{nw} is changed from 0.5 to 2 for enhancing heat transfer.

- The **$k-\omega$ SST** model is renowned for its robustness in predicting adverse pressure gradients and flow separation, which are common in rib-enhanced cooling channels. It combines the benefits of the $k-\omega$ model near the wall and the $k-\epsilon$ model in the free stream, providing accurate predictions of boundary layer behavior. This model is particularly useful for capturing the complex interactions between the ribs and the cooling flow, ensuring that the heat transfer enhancements and pressure losses are accurately represented.
- The **$k-kl-\omega$** model is a transitional turbulence model that predicts the onset and development of turbulence from laminar flow. Given that the flow over rib turbulators can transition from laminar to turbulent, this model is essential for accurately capturing this behavior. The $k-kl-\omega$ model enables a detailed analysis of how the flow transitions due to the presence of ribs, which is crucial for understanding the enhancement in heat transfer and the accompanying pressure drop.
- The **$k-\epsilon$** model is one of the most widely used turbulence models due to its simplicity and relatively good performance for a wide range of flow types. It is particularly effective for high Reynolds number flows, typical in gas turbine applications. The model provides reliable results for turbulent kinetic energy and dissipation rate, essential for evaluating the effectiveness of rib turbulators in disrupting the boundary layer and enhancing heat transfer. Despite its limitations in near-wall regions, the $k-\epsilon$ model's robustness and computational efficiency make it a valuable tool for preliminary analyses and comparisons.
- The **$k-\omega$ GEKO** (Generalized $k-\omega$) model offers a customizable approach to turbulence modeling. By allowing the adjustment of key parameters, the $k-\omega$ GEKO model can be fine-tuned to specific flow conditions encountered in ribbed cooling channels. This flexibility is crucial for addressing the unique flow features induced by rib turbulators, such as vortex shedding and reattachment points. The model's adaptability ensures that the complex interactions between turbulent structures and heat transfer surfaces are accurately captured, leading to more precise predictions of cooling performance.

These 4 turbulence models have been examined extensively in the analysis matrix.

4.5. Solver Setup

As mentioned before, Ansys Fluent 2021 R2 software was used in the current study. In this study, coupled scheme was used. The thermophysical properties of the air were entered into the program in piecewise-polynomial format depending on the temperature, except viscosity, which obeys the Sutherland law. To examine the effect of turbulence models on the solution, 4 turbulence models were used in the analysis. All parameters in the spatial discretization section are resolved as second order. Solver setup settings are shown in **Table 4.5**.

Table 4.5: Solver setup.

Dimension of problem	3D-stationary-rotating
Fluid	Density: Ideal gas Specific Heat (Cp): Piecewise-polynomial Thermal Conductivity (k): Piecewise-polynomial Viscosity: Sutherland
Turbulence Models	$k-\omega$ SST, $k-kl-\omega$, $k-\epsilon$ Realizable, $k-\omega$ GEKO(c_{nw} : 2)
Discretization Schemes	Pressure-Velocity Coupling: Coupled Gradient: Least Squares Cell Based Pressure: Second Order Density: Second Order Upwind Momentum : Second Order Upwind Swirl Velocity : Second Order Upwind Turbulent Kinetic Energy : Second Order Upwind Specific Dissipation Rate : Second Order Upwind Energy : Second Order Upwind

4.6. Boundary Conditions

Four main boundary condition types are given as boundary conditions in CFD analyses. Since the analyzes were carried out under constant Reynolds number, constant temperature and pressure conditions, the same inlet flow rate, inlet total temperature and outlet static pressure were defined in all analyses. Since all dimensions of geometry and operating conditions were clearly given in the base case, these values were direct boundary condition input to CFD analyses.

Since PIV measurements were taken only for Reynolds number 15k in the base case, CFD analyzes were run at a single Reynolds number. Tests were carried out under atmospheric pressure condition and room temperature. 101325 Pa was assumed for atmospheric pressure and 300 K for room temperature. When we look at the tests with PIV measurements and heat transfer measurements for the rotating case, we see that these two measurements are taken in common at -0.3 and -0.3 rotation numbers. At these rotation numbers, the test rig was rotated at a speed of 110 rpm. The input turbulence value is given as 1% in the base case.

In the tests, a constant heat flux of 835 W/m² was applied to the ribbed surface, and the same value was used in the analyses. The side surfaces were left adiabatic in the analyses.

The specified boundary conditions are summarized in **Table 4.6**.

Table 4.6: Boundary conditions.

Boundary Conditions			
Mass flow inlet	0.0223 kg/s		
Pressure outlet (static)	101325 Pa		
Inlet Total Temperature	300 K		
Inlet Turbulence Intensity	1%		
Rotation (rpm/radian/s)	+110/+11.57	0/0	-110/-11.57
Heat transfer	835 W/m ² (ribbed surface)		Adiabatic (lateral surfaces)

In **Figure 4.6**, the boundary conditions are shown on the figure. The direction of rotation in which the flow tends to stick to the ribbed surface is indicated by positive notation in the Rotation number, the opposite direction is indicated by negative notation.

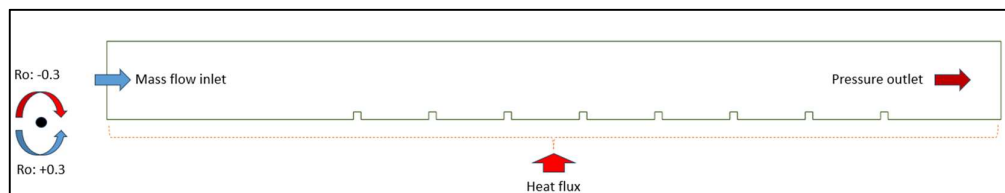


Figure 4.6: Schematic of BCs.

4.7. Analysis Matrix

The analysis matrix is designed so that the test and analysis results can be compared comprehensively. 3 meshes, 4 turbulence models, one Reynolds number and 3 Rotation numbers were examined. Runs were taken in 36 conditions in total, these conditions can be seen in **Table 4.7**.

Table 4.7: Analysis matrix.

	Coarse			Mid			Fine		
Rotation Number	+0.3	0	-0.3	+0.3	0	-0.3	+0.3	0	-0.3
$k-\omega$ SST	PCS153F	PCS150F	PCS15-3F	PMS153F	PMS150F	PMS15-3F	PFS153F	PFS150F	PFS15-3F
$k-kl-\omega$	PCK153F	PCK150F	PCK15-3F	PMK153F	PMK150F	PMK15-3F	PFK153F	PFK150F	PFK15-3F
$k-\epsilon$	PCR153F	PCR150F	PCR15-3F	PMR153F	PMR150F	PMR15-3F	PFR153F	PFR150F	PFR15-3F
$k-\omega$ GEKO	PCG153F	PCG150F	PCG15-3F	PMG153F	PMG150F	PMG15-3F	PFG153F	PFG150F	PFG15-3F

Each condition is numbered with a code name, from now on the results will be analyzed according to these code names. The "P" at the beginning of each codename refers to the polyhedral mesh. The second character represents the mesh used in the analysis condition. The characters "C" for coarse mesh, "M" for mid mesh and "F" for fine mesh are used, respectively. The third character represents the turbulence model. The characters "K" for $k-kl-\omega$, "R" for $k-\omega$ SST, "E" for $k-\epsilon$ and "G" for $k-\omega$ GEKO are used, respectively. The number 15 in the middle of the code refers to the Reynolds number in the analysis. The numbers -3, 0 and 3 following the number 15 represent the Rotation numbers. Finally, the "F" at the end of the code indicates that the analyzes were performed with the Fluent program. Notation has been shown below for clear understanding.

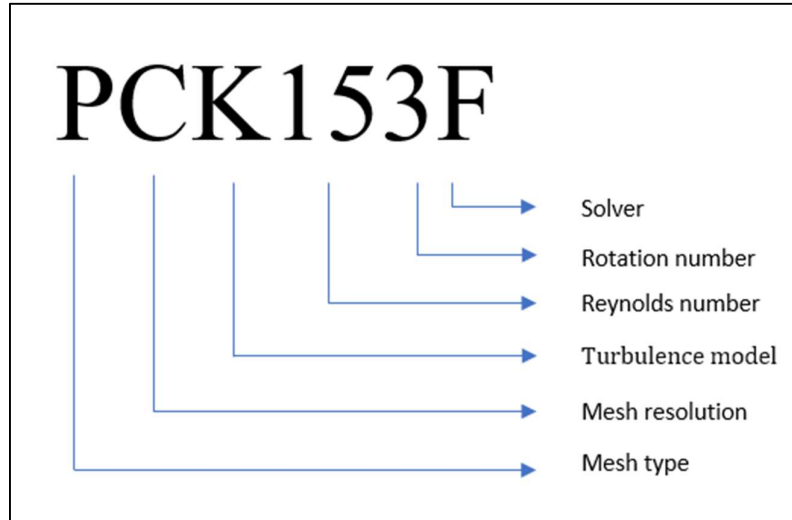


Figure 4.7. Notation of analysis matrix.

In the remaining part of the report, the results will be shown according to this coding scheme.

4.8. Solutions & Comparison With Test Data

In this section, first the analysis and test results in the stationary situation and then in the rotary situation will be examined and compared. It has been observed that in the stationary case, the analysis results do not change dramatically with the mesh and turbulence model. However, in the rotary section, the results were found to be highly dependent on the mesh and turbulence model. There is a significant shift between the test and analysis results. The reasons for these will be examined in the following paragraphs.

The convergence of the analyzes was decided by looking at some heat transfer and flow parameters in addition to the classical residuals. The inlet and outlet mass flow rate difference and inlet pressure were plotted in each iteration and added to the analysis setup as a convergence criterion. The analysis was stopped after the residuals settled, and convergence criteria were reached.

4.8.1. Stationary Cases

Test data in the condition of $Re: 15000$ and $Ro: 0$ in base cases are given below. If we first examine the heat transfer, the distribution of the heat transfer coefficient between the 6th and 7th ribs is as follows.

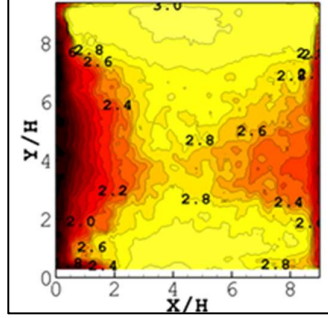


Figure 4.8: HTC EF distribution at $Re: 15000$ & $Ro: 0$ [28].

The average HTC enhancement factor (EF) value is 2.6. The EF value increases with the impingement effect towards the side walls, however, it is seen that the heat transfer decreases due to the recirculation zones at the beginning and end regions of the ribs down to the 1.4 EF. The EF exhibits a minimum just behind the rib due to a recirculation bubble where fluid velocity is quite low. Following this, the flow reattaches, leading to a minimum boundary layer thickness, which increases the EF. The highest EF value in the channel's symmetry plane occurs between $3H$ and $5H$ behind the rib, where the EF remains nearly constant at 2.5.

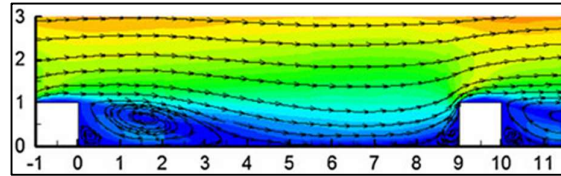


Figure 4.9: Streamlines in mid-plane at $Re: 15000$ & $Ro: 0$ [29].

Streamline data measured with PIV can be seen in **Figure 4.9**. The x and y axis of the figure are normalized according to the rib height. The data is normalized and clearly shows the end of the recirculation zone originating from the 6th rib and the point where the flow reattaches. Reattachment length was reported by the authors as $3.85h$.

In the next section, the average EF value, EF distribution contours, streamline, reattachment lengths and periodicity curves obtained in the analyzes will be given for each case.

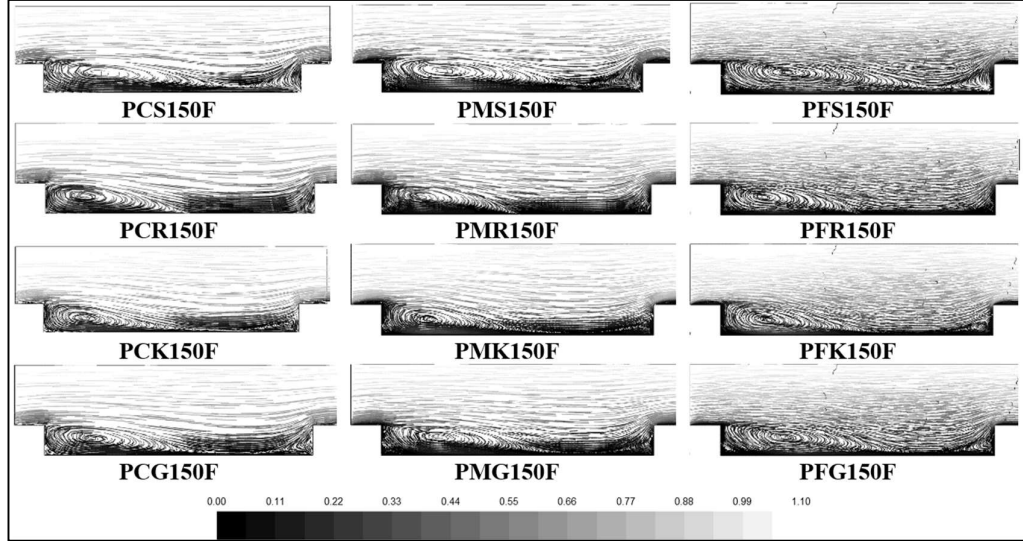


Figure 4.10: Streamlines in mid-plane at Re: 15000 & Ro: 0 for different mesh and turbulence model conditions.

Figure 4.10. shows the streamline contours obtained from CFD under different mesh and turbulence model conditions. Here, the velocity values are normalized as $\sqrt{U^2 + V^2} / U_0$, with U_0 being the channel inlet average velocity. According to the PIV results of the test, it is seen that the reattachment lengths are quite long in all CFD conditions, which seriously affects the heat transfer character. After reattachment, it is seen that the thinnest boundary layer thickness is in the $k-k_l-\omega$ model, and then in the $k-\epsilon$ Realizable model. In **Table 4.8.**, reattachment lengths are quantified and compared with the test data.

Table 4.8: CFD reattachment length comparison with the test.

	<i>k-ω</i> SST		<i>k-ϵ</i> Realizable		<i>k-kl-ω</i>		<i>k-ω</i> GEKO	
Coarse	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	6.75h	75%	7.5h	95%	6.75h	75%	7h	82%
	PCS150F		PCR150F		PCK150F		PCG150F	
Mid	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	6.75h	75%	7h	82%	6.5h	69%	6.75h	75%
	PMS150F		PMR150F		PMK150F		PMG150F	
Fine	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	7h	82%	7h	82%	6.5h	69%	6.75h	75%
	PFS150F		PFR150F		PFK150F		PFG150F	

As can be seen in **Table 4.8.**, all turbulence models fail to accurately predict the reattachment length, regardless of the mesh. While the run with the highest error rate is PCR150F with 95%, the runs with the closest prediction are PMK150F and PFK150F with 69% error rate. As can be remembered from the streamline contours, the thinnest boundary layer and smallest recirculation zone belonged to the *k-kl- ω* model. The table with numbers also gives us the same result. As a reminder to readers, the reattachment length was 3.85h in the specified test condition.

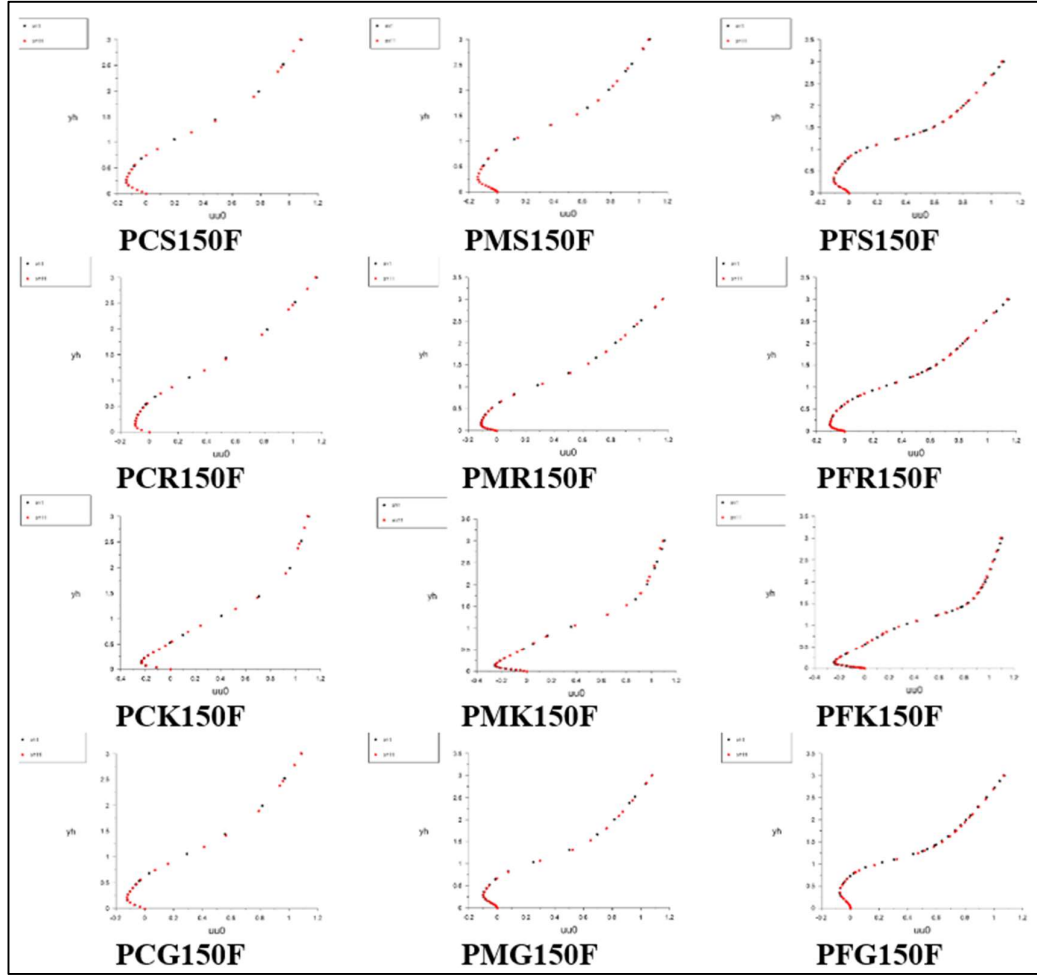


Figure 4.11: Normalized axial velocity at $x/h:1$ (black dots) and $x/h:11$ (red dots) for periodicity check at $Re: 15000$ & $Ro: 0$ for different mesh and turbulence model conditions.

In **Figure 4.11**, it is checked whether the flow reaches periodicity when it reaches the 6th rib. By following the method in the base case, normalized axial velocity graphs were drawn on top of each other at $x/h:1$ and $x/h:11$ between 6-7 ribs. While the red dots represent the 7th rib entry ($x/h:11$), the black dots represent the 6th rib entry ($x/h:1$). The periodicity was captured in all runs, regardless of mesh and turbulence models.

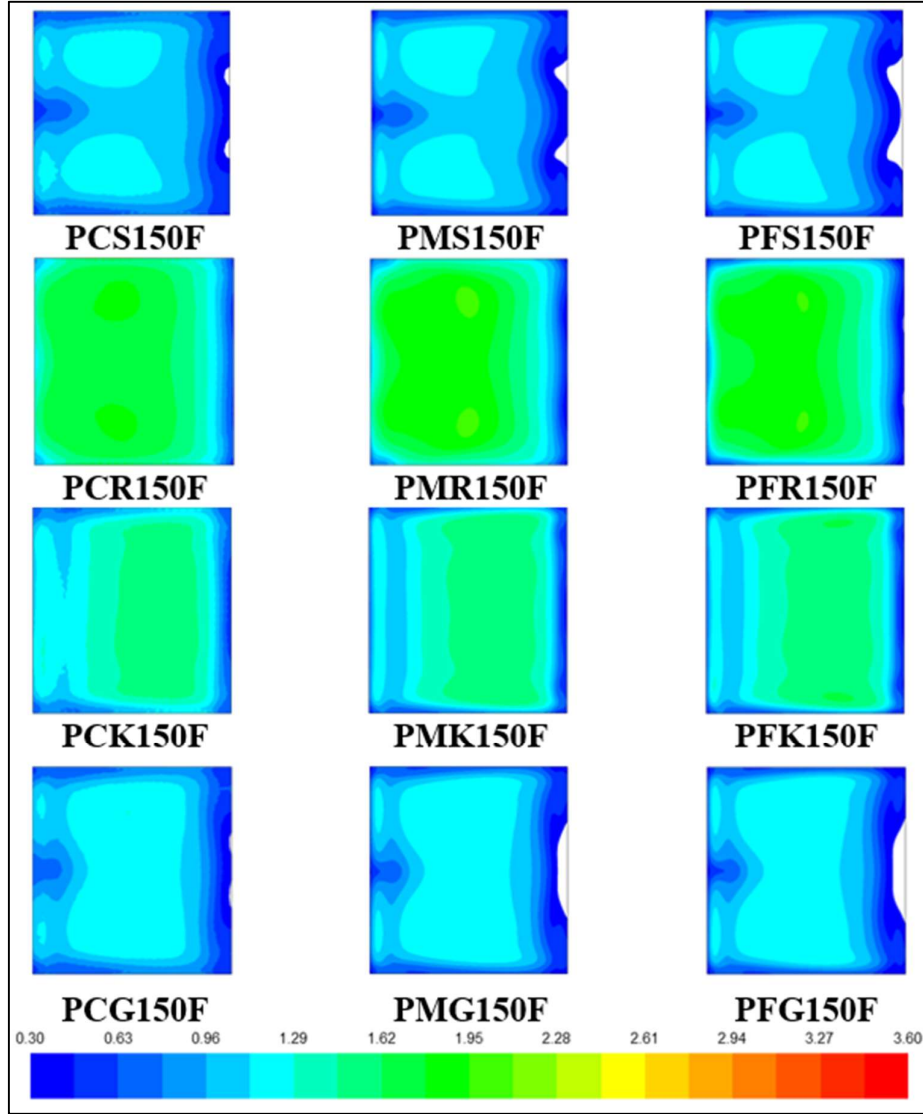


Figure 4.12: HTC EF distribution at $Re: 15000$ & $Ro: 0$ for different mesh and turbulence model conditions.

In **Figure 4.12.**, HTC EF distributions are given for each run. In order to ensure that the contours are compatible with the rotating results in the following sections, one end of the legend is adjusted to 0.30 and the other end is adjusted to 3.6.

Each turbulence model has a different pattern. Since $k-\omega$ SST and $k-\omega$ GEKO have similar turbulence models in terms of structure, they showed a similar distribution and formed two high EF centers. However, $k-\epsilon$ Realizable shows a more diffused EF contour, while $k-kl-\omega$ shows a denser EF contour towards the 7th rib. All turbulence models give consistent results regardless of the mesh.

Table 4.9: HTC EF CFD comparison with the test at Re: 15000&Ro: 0.

	<i>k-ω</i> SST		<i>k-ϵ</i> Realizable		<i>k-kl-ω</i>		<i>k-ω</i> GEKO	
Coarse	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	0.889	66%	1.482	43%	1.232	52%	0.950	63%
	PCS150F		PCR150F		PCK150F		PCG150F	
Mid	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	0.853	67%	1.442	44%	1.220	53%	0.891	65%
	PMS150F		PMR150F		PMK150F		PMG150F	
Fine	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	0.816	68%	1.339	49%	1.217	53%	0.851	53%
	PFS150F		PFR150F		PFK150F		PFG150F	

Table 4.9. shows the area-averaged EF values of the EF contours in **Figure 4.10**. As a reminder to readers, the EF value in the test condition is 2.6. Confusingly, the EF value is less than 1 in the *k- ω* SST and *k- ω* GEKO models. In other words, ribs actually give a lower heat transfer coefficient than a smooth pipe. The main reason for this can be shown as the very thick boundary layer thickness in these models. Thick boundary layer produces lower shear force and creates lower HTC compared to thinner, which means even lower heat transfer coefficient than a smooth pipe flow. Another factor in this situation is the CFD's very long reattachment flow prediction. The two main points of the rib turbulator's HTC increasing mechanism are impingement and high turbulence intensity in recirculation zones. The fact that CFD fails to capture the turbulence intensity in recirculation zones and finds this region longer with an average error of 70% reduces the average HTC value very seriously.

k- ϵ Realizable and *k-kl- ω* turbulence models give higher EF rates compared to ω based models because they capture the reattachment length shorter and create a thinner boundary layer after reattachment. Even in this case, the best error rate is 43% in the PCR150F run, which is a quite high error.

4.8.2. Cases with Rotation & Ro: -0.3

Now let's examine the condition where Ro: -0.3.

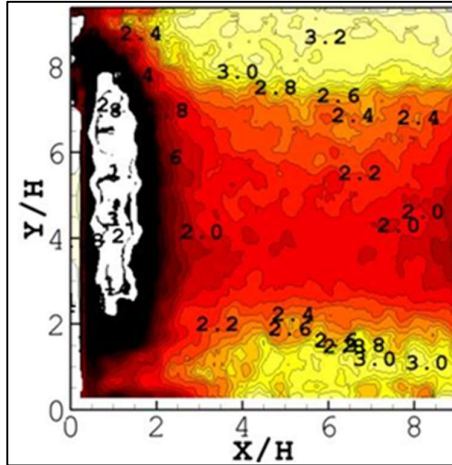


Figure 4.13: HTC EF distribution at Re: 15000 & Ro: -0.3 [28].

The EF distribution on the Ro: -0.3 differs significantly from that in stationary conditions. **Figure 4.13.** illustrates the enlargement of the recirculation bubble on the leading wall, which reduces the EF compared to the stationary case. The EF also decreases in the central region of the leading wall. This reduction is due to a more stable shear layer, lower turbulence intensity, and thickening of the boundary layer on this case. The region after the 6th rib is shown in white because it is below the minimum value of 1.4, which is the legend limit. The average EF value in this case is 2.3.

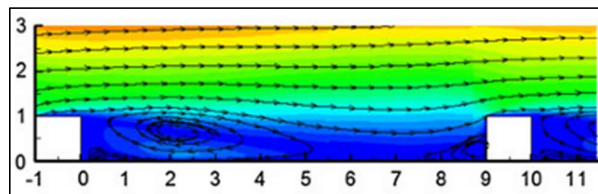


Figure 4.14: Streamlines in mid-plane at Re: 15000 & Ro: -0.3 [29].

The streamline contour at Ro: -0.3 can be seen in **Figure 4.13.** The reattachment length in this condition was reported by the authors as 5.65h.

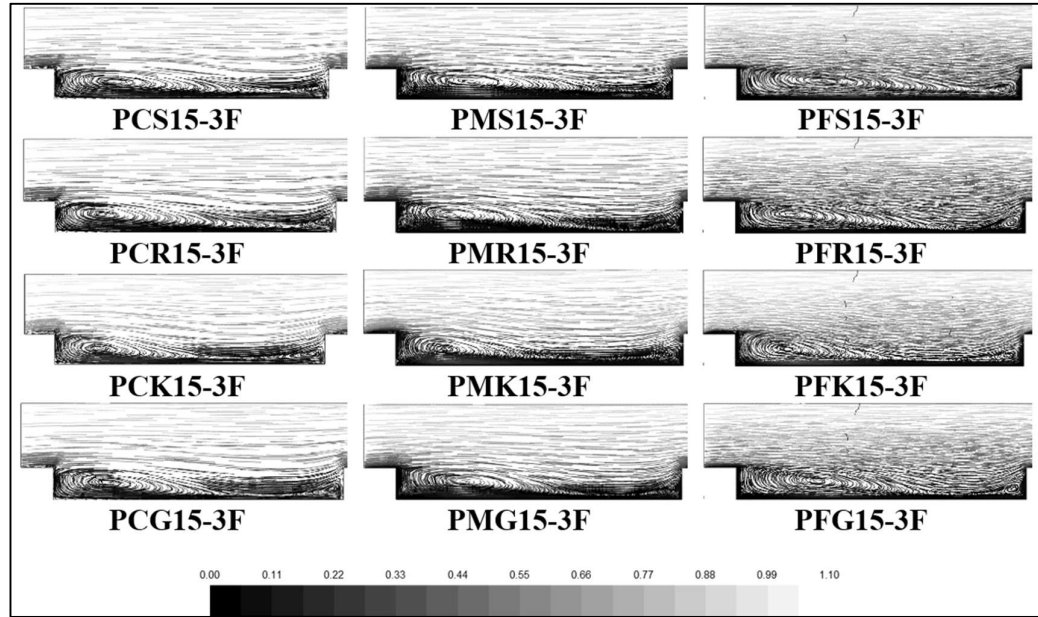


Figure 4.15: Streamlines in mid-plane at Re: 15000 & Ro: -0.3 for different mesh and turbulence model conditions.

The streamline contours are shown in **Figure 4.15**. There are findings very similar to those in the stationary condition. Again, CFD caught the reattachment longer than the test results. Compared to the stationary situation, recirculation zones seem slightly larger and unclear. The boundary layer looks quite thick in all turbulence models, especially in coarse mesh streamlines.

Table 4.10: CFD reattachment length comparison with the test at Re: 15000 & Ro: -0.3.

	<i>k-ω</i> SST		<i>k-ϵ</i> Realizable		<i>k-kl-ω</i>		<i>k-ω</i> GEKO	
Coarse	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	7.5h	33%	7.5h	33%	6.25h	11%	7.25h	28%
	PCS15-3F		PCR15-3F		PCK15-3F		PCG15-3F	
Mid	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	7.75h	37%	7.5h	33%	6.25h	11%	7h	24%
	PMS15-3F		PMR15-3F		PMK15-3F		PMG15-3F	
Fine	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	7.75h	37%	7.5h	33%	6.25h	11%	7.25h	28%
	PFS15-3F		PFR15-3F		PFK15-3F		PFG15-3F	

In **Table 4.10.**, reattachment lengths are given in terms of rib height for runs at Ro: -0.3. It can be seen in the table that in the Ro: -0.3 condition, turbulence models calculate reattachment lengths much better than in the stationary case. The *k-kl- ω*

model did a very good job with an 11% error rate regardless of the mesh. As a reminder to readers, the reattachment length was 5.65h in the specified test condition.

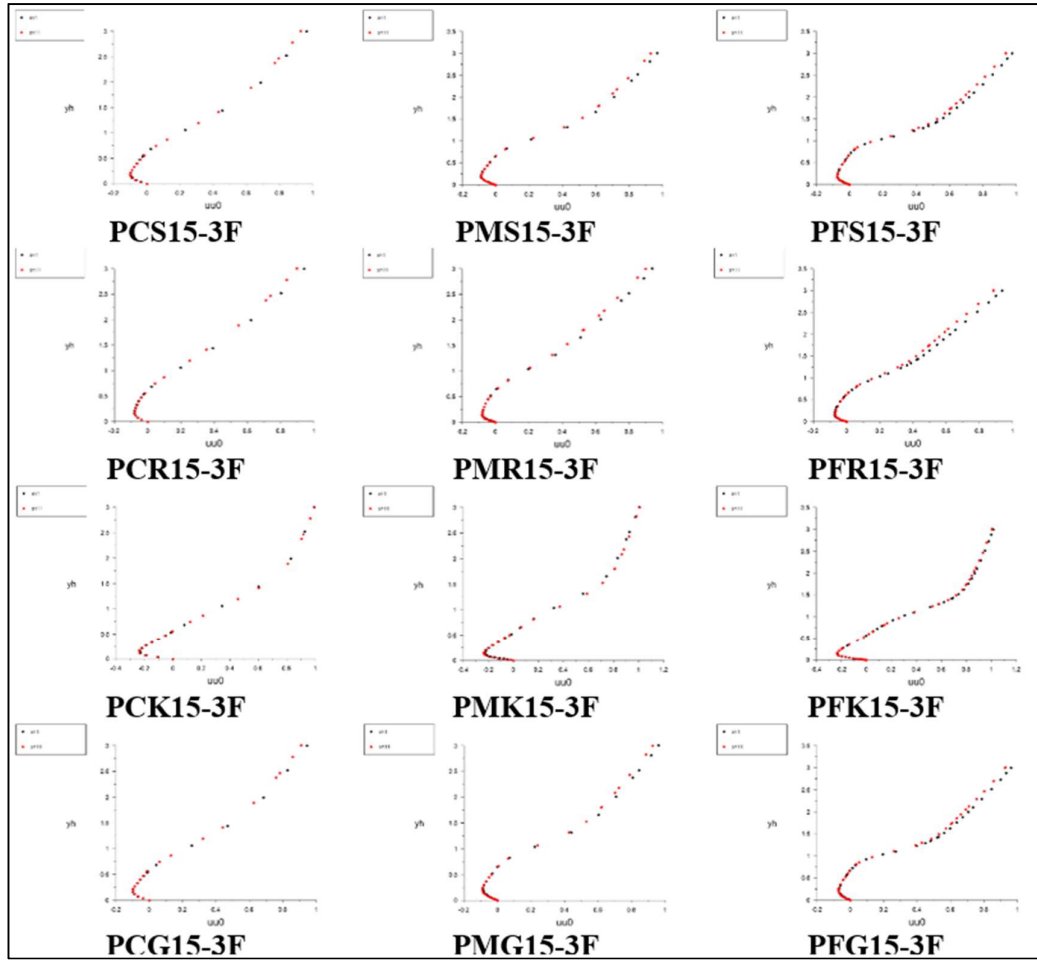


Figure 4.16: Normalized axial velocity at $x/h:1$ (black dots) and $x/h:11$ (red dots) for periodicity check at $Re: 15000$ & $Ro: -0.3$ for different mesh and turbulence model conditions.

Periodicity check was also performed for the $Ro: -0.3$ condition and it was observed that periodicity was achieved regardless of the mesh and turbulence models. Periodicity curves for this condition are shown in **Figure 4.16**.

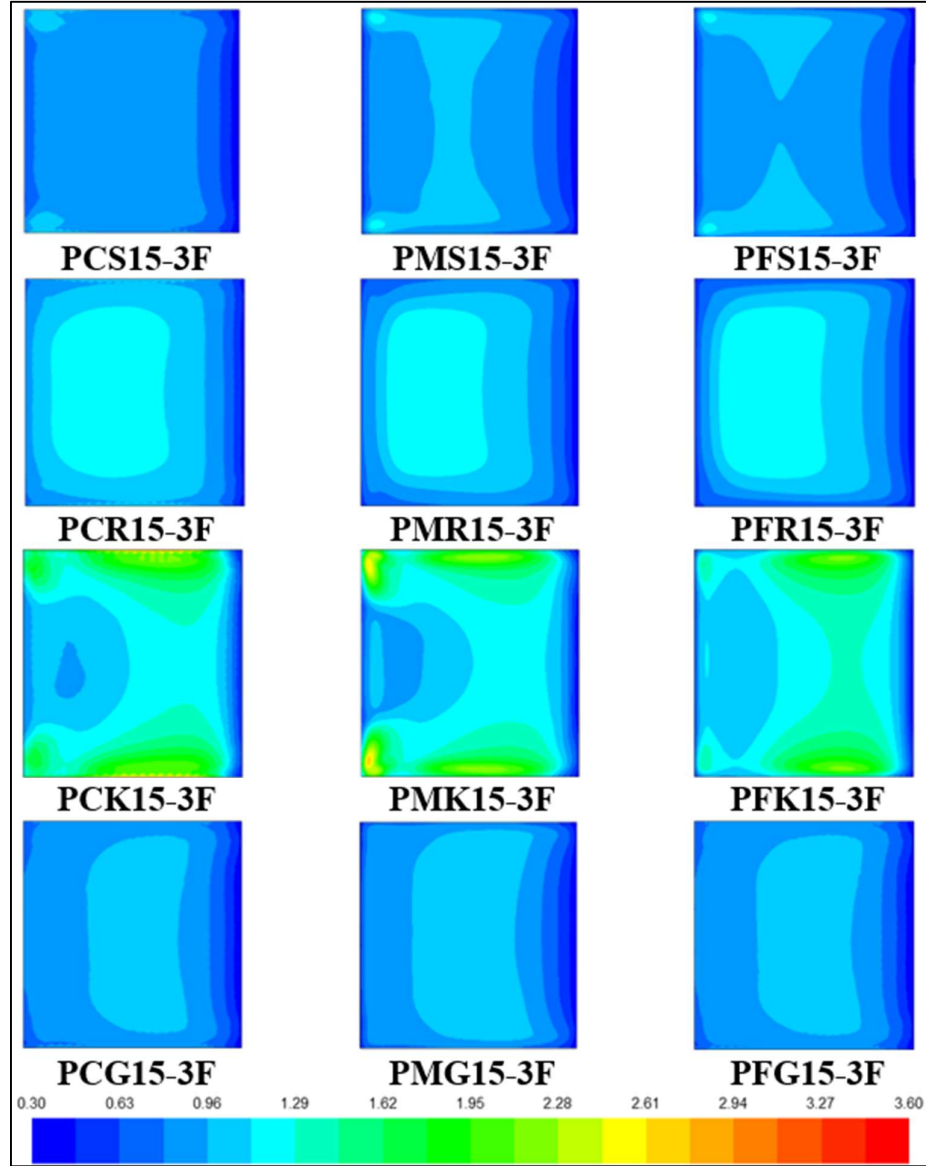


Figure 4.17: HTC EF distribution at Re: 15000&Ro: -0.3 for different mesh and turbulence model conditions.

In **Figure 4.17.**, HTC EF contours are given for the Ro: -0.3 condition. Again, it is seen that each turbulence model has its own unique pattern. This time, $k-\omega$ SST and $k-\omega$ GEKO are not as similar as the stationary situation. While $k-\omega$ models showed a more dispersed distribution, $k-\epsilon$ Realizable showed a slightly more stratified distribution. The $k-kl-\omega$ model showed the most detailed distribution, and with the test, the closest contour distribution came from this model as in the stationary case.

Table 4.11: HTC EF CFD comparison with the test at Re: 15000&Ro: -0.3.

	<i>k-ω</i> SST		<i>k-ϵ</i> Realizable		<i>k-kl-ω</i>		<i>k-ω</i> GEKO	
Coarse	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	0.812	65%	0.989	57%	1.191	48%	0.860	63%
	PCS15-3F		PCR15-3F		PCK15-3F		PCG15-3F	
Mid	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	0.798	65%	0.962	58%	1.168	49%	0.861	63%
	PMS15-3F		PMR15-3F		PMK15-3F		PMG15-3F	
Fine	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	0.800	65%	0.912	60%	1.160	50%	0.849	63%
	PFS15-3F		PFR15-3F		PFK15-3F		PFG15-3F	

Table 4.11. shows the area-averaged EF values of the EF contours in **Figure 4.15**. As a reminder to readers, the EF value in the test condition is 2.3. In this condition, EF values are still less than 1, all the evaluations made in the stationary case are also valid here. The Coriolis forces generate secondary flows that counteract the rib-induced vortices, significantly enhancing heat transfer near the lateral walls by sweeping high momentum fluid towards the edges of the ribbed surface and reducing heat transfer in the center. Because of that, the EF factor is slightly lower as compared to the stationary case. In this condition, the minimum error value belongs to the *k-kl- ω* model, as you may remember, *k- ϵ* Realizable gave the closest results in the stationary case. However, the error value still has not fallen below 48%.

4.8.3. Cases with Rotation & Ro:+0.3

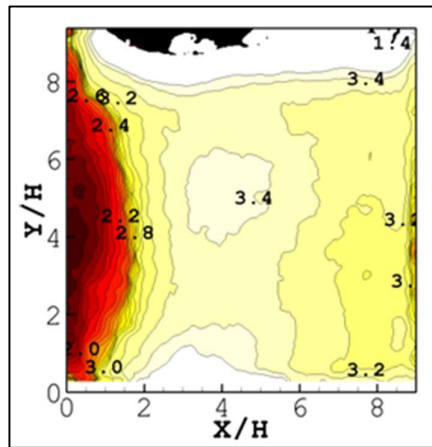


Figure 4.18: HTC EF distribution at Re: 15000&Ro: +0.3 [28].

On the case $Ro: 0.3$, the shear layer downstream of the rib is very unstable, leading to increased turbulence intensity. The Coriolis-driven secondary flows move high momentum fluid from the central region of the channel to the lateral walls. As a result, the boundary layer thickness and the length of the separation bubble decrease. In this case, since the recirculation zones become smaller, the low EF zones are almost non-existent.

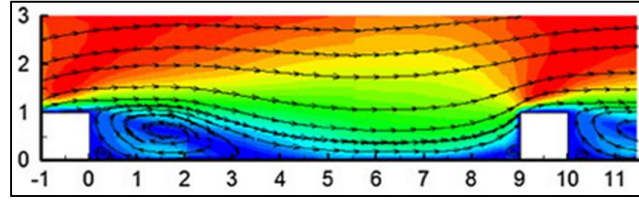


Figure 4.19: Streamlines in mid-plane at $Re: 15000$ & $Ro: 0.3$ [29].

The streamline contour at $Ro: 0.3$ can be seen in **Figure 4.19**. The reattachment length in this condition was reported by the authors as $3.45h$.

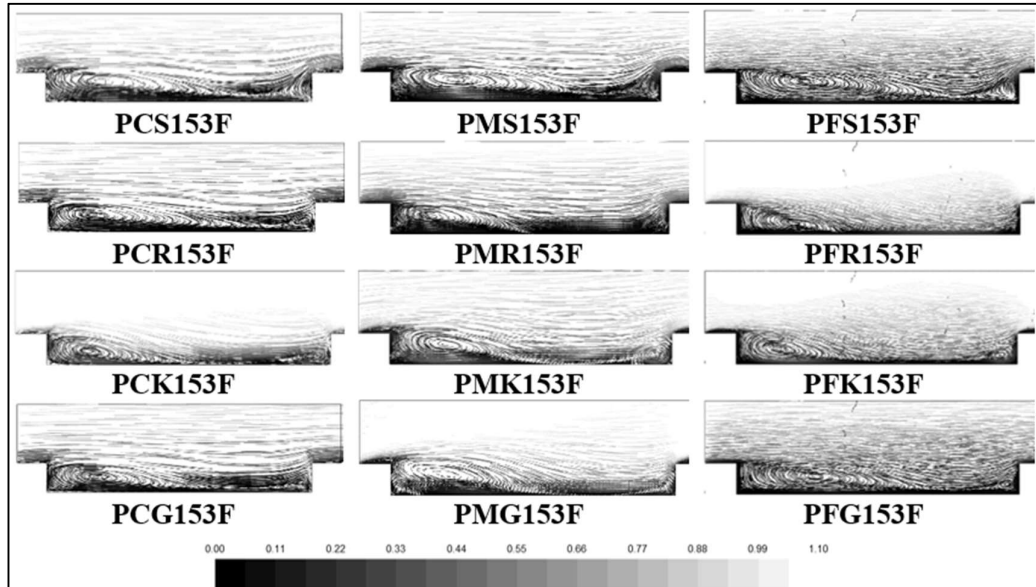


Figure 4.20: Streamlines in mid-plane at $Re: 15000$ & $Ro: 0.3$ for different mesh and turbulence model conditions.

The streamline contours are shown in **Figure 4.20**. In this condition, CFD reattachment length was found to be longer than the test as similar to the previous results. However, the $k-kl-\omega$ model seems to give very good results on streamline contours.

Table 4.12: CFD reattachment length comparison with the test at Re: 15000&Ro: 0.3.

	<i>k-ω</i> SST		<i>k-ϵ</i> Realizable		<i>k-kl-ω</i>		<i>k-ω</i> GEKO	
Coarse	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	7.5	117%	7.5	117%	6.75	96%	7.25	110%
	PCS153F		PCR153F		PCK153F		PCG153F	
Mid	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	7.75	124%	8	132%	6.5	88%	7.5	117%
	PMS153F		PMR153F		PMK153F		PMG153F	
Fine	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	7.75	124%	8	132%	6.5	88%	7.5	117%
	PFS153F		PFR153F		PFK153F		PFG153F	

In **Table 4.12.**, reattachment lengths are given in terms of rib height for runs at Ro: 0.3. As seen in table, CFD is quite unsuccessful in calculating reattachment lengths under the condition where Coriolis forces are dominant. Although the *k-kl- ω* model still made the best prediction, it could not go below the 88% error rate. As a reminder to readers, the reattachment length was reported as 3.45h in the specified test condition.

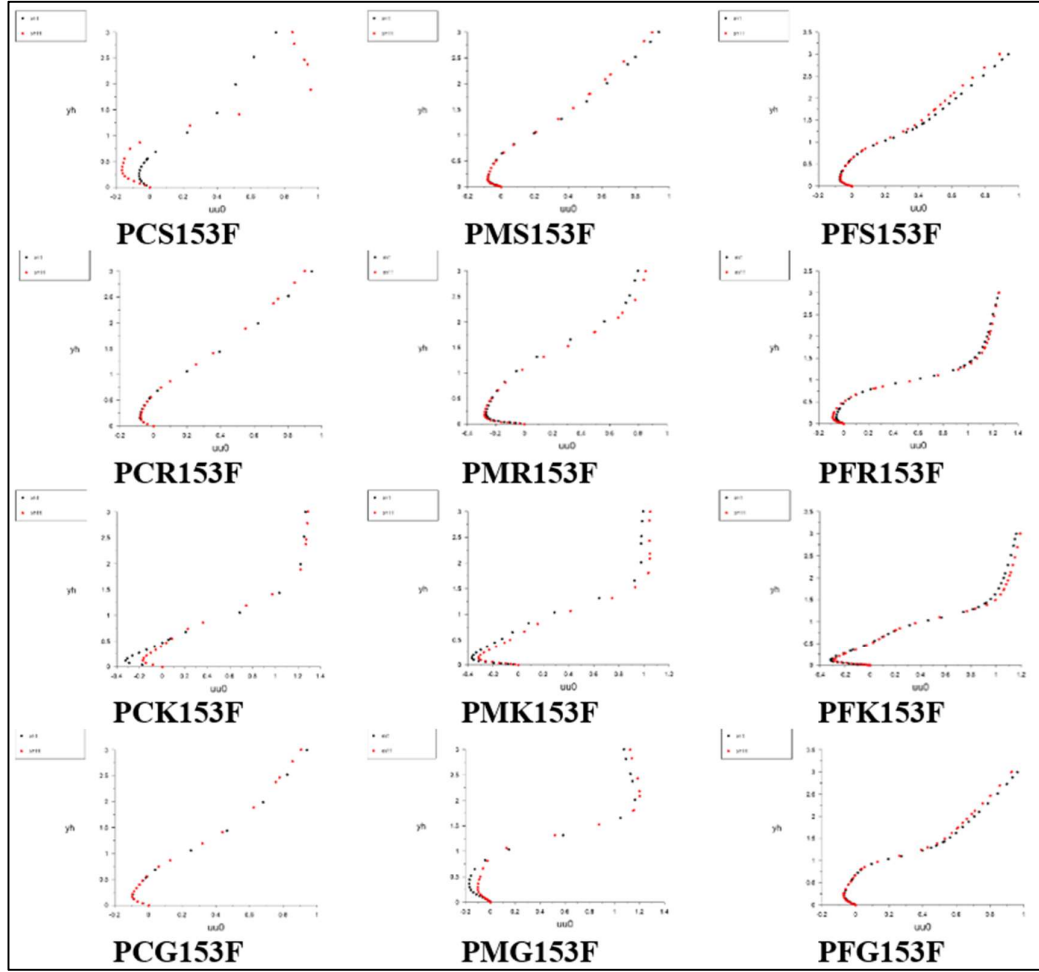


Figure 4.21: Normalized axial velocity at $x/h:1$ (black dots) and $x/h:11$ (red dots) for periodicity check at $Re: 15000$ & $Ro: 0.3$ for different mesh and turbulence model conditions.

Periodicity check was also performed for the case where $Ro: 0.3$. This time, it was observed that periodicity could not be achieved in all cases, unlike the stationary and reverse rotation cases. It can be said that the reason for this is that $Ro: 0.3$ is an unstable rotation direction and turbulence models have difficulty in solving this condition.

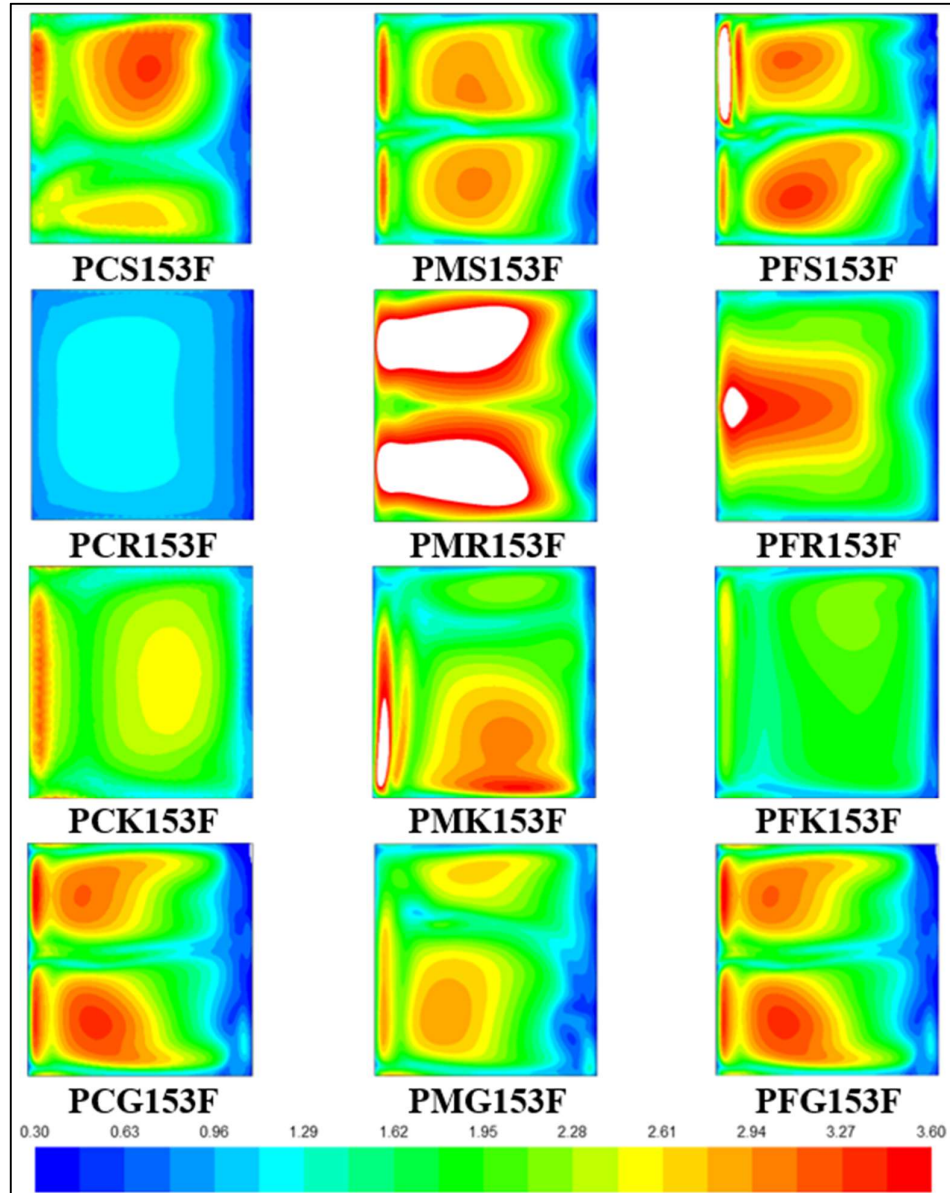


Figure 4.22: HTC EF distribution at Re: 15000&Ro: 0.3 for different mesh and turbulence model conditions.

Figure 4.22. shows the EF contours for the Ro: 0.3 condition. As can be seen, quite inconsistent results are seen in this condition compared to the previous conditions. In base cases, the Ro: 0.3 condition is called an unstable condition due to the high turbulence intensity and the dominance of Coriolis forces. This irregularity in the CFD contours was first interpreted as a convergence problem, however, the convergence of the heat transfer was seen by plotting the HTC value depending on the iteration, and the convergence of the flow field was determined from the input and output pressures and mass flow rate conservation. Since there was no convergence problem in the

analyses, RMS averages were taken and contours were drawn again, but no different results were observed. The conclusion here is that in unstable conditions the results are very dependent on the turbulence models and the mesh.

Table 4.13: HTC EF CFD comparison with the test at Re: 15000&Ro: 0.3.

	<i>k-ω SST</i>		<i>k-ϵ Realizable</i>		<i>k-kl-ω</i>		<i>k-ω GEKO</i>	
Coarse	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	1.652	47%	0.993	68%	2.03	35%	0.860	72%
	PCS153F		PCR153F		PCK153F		PCG153F	
Mid	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	1.744	44%	2.498	19%	1.949	37%	1.765	43%
	PMS153F		PMR153F		PMK153F		PMG153F	
Fine	CFD	Error	CFD	Error	CFD	Error	CFD	Error
	1.602	48%	1.805	41%	1.595	49%	1.580	49%
	PFS153F		PFR153F		PFK153F		PFG153F	

Area averaged EF results are given in **Table 4.13**. As a reminder to readers, the test data in this condition is an average EF value of 3.1. As can be seen from the table, the results are quite different for different options, the deviation is very large. Due to these differences, more analysis should be run and different options with other mesh types should be tried as Ro: 0.3. Since such a large deviation will cause doubt on the temperature distribution in the pressure side region in the analysis of engine operating conditions, it is important to carry out sensitivity studies in the future work.

4.9. Bonus – RSM & Periodic Setup Trials

In this chapter, solution trials of a distinct turbulence and periodic boundary have been shown. Reynolds Stress Model (RSM) also known as the Reynolds Stress Transport Models, has been studied on a mid mesh – stationary configuration and results have been declared. In addition to that, the solution domain has been modified and shortened to a translational periodic domain with a much less mesh load.

4.9.1. RSM Solution

Unlike simpler models such as the $k-\epsilon$ or $k-\omega$ models, which rely on isotropic eddy viscosity assumptions, the RSM directly solves transport equations for the Reynolds stresses. This provides a more detailed and accurate representation of the turbulent flow.

Analyses were taken in stationary and mid-mesh cases, Stress Omega option has been enabled. HTC enhancement contour and streamline contour are given below.

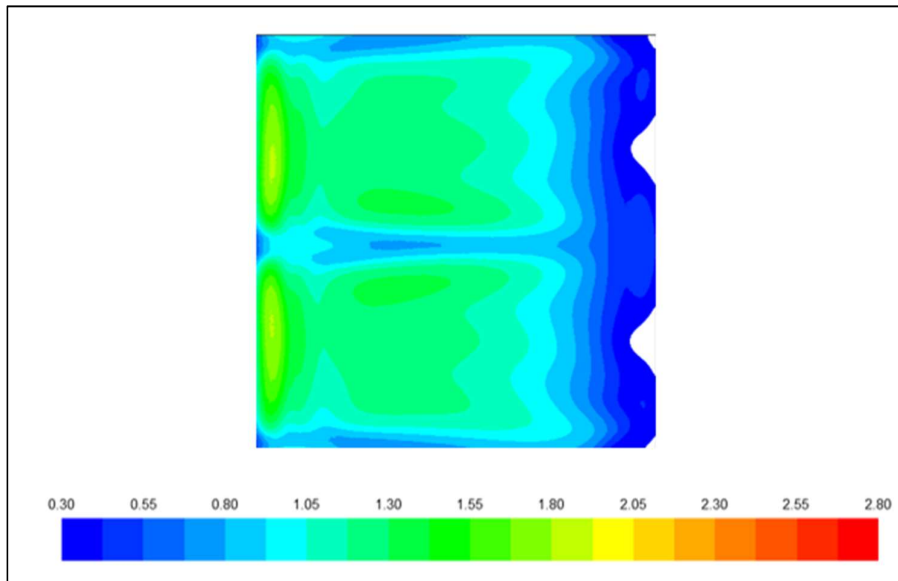


Figure 4.23: HTC enhancement distribution of RSM solution in a stationary condition.

As can be seen in **Figure 4.23.**, HTC enhancement distribution is compatible with other turbulence models. The two eyes distribution similar to the $k-\omega$ SST and GEKO solutions attracts the attention. The area average HTC enhancement value is 0.89, which again gives a result close to the omega based models.

However, when we look at the streamline distribution, we can't see a view as compatible as the HTC enhancement distribution. This can be seen in **Figure 4.24.**

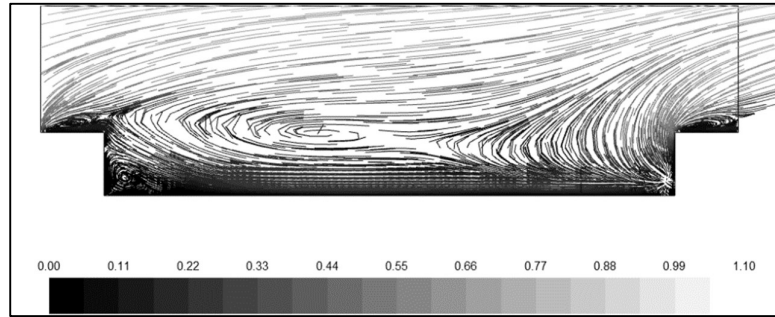


Figure 4.24: Streamline of RSM solution in a stationary condition.

As can be seen from the figure, streamline did not work as expected in RSM solutions. The RSM model captured well the vortex immediately after the rib, however, it was observed that there was flow separation where the flow should reattach.

4.9.2. Periodic Conditions

In the analyses where the periodic boundary conditions were examined, the solution domain was reduced.

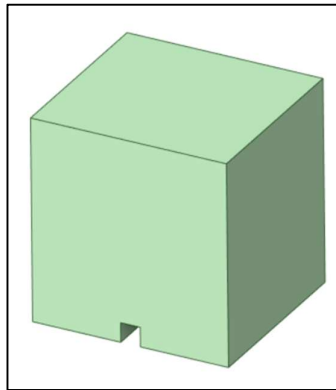


Figure 4.25: Periodic solution domain.

While creating the solution domain, rib was placed in the middle of the domain, like the studies in the literature. All model settings kept same as previous analyses, only inlet and outlet boundary conditions were changed to a translational periodic boundary condition. In Fluent, there are two ways to define translational periodic boundary condition, either pressure gradient or mass flow rate has to be given as input. Since the mass flow rate is known, it was given as boundary condition.

Analysis was taken using the $k-\omega$ SST turbulence model and mid-mesh settings, under stationary condition. HTC enhancement and streamline contours are given below.

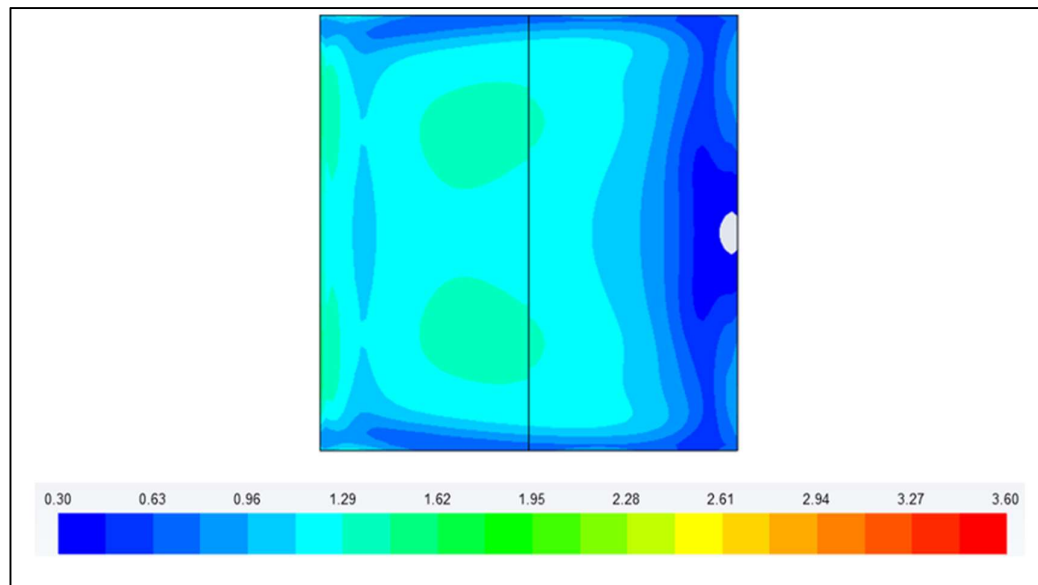


Figure 4.26: HTC enhancement distribution of periodic boundary solution in a stationary condition.

As seen in **Figure 4.26.**, the distribution in the periodic case is quite similar to the solution obtained with the full domain. However, the area average HTC value is 1.06, approximately 24% higher than the full domain results.

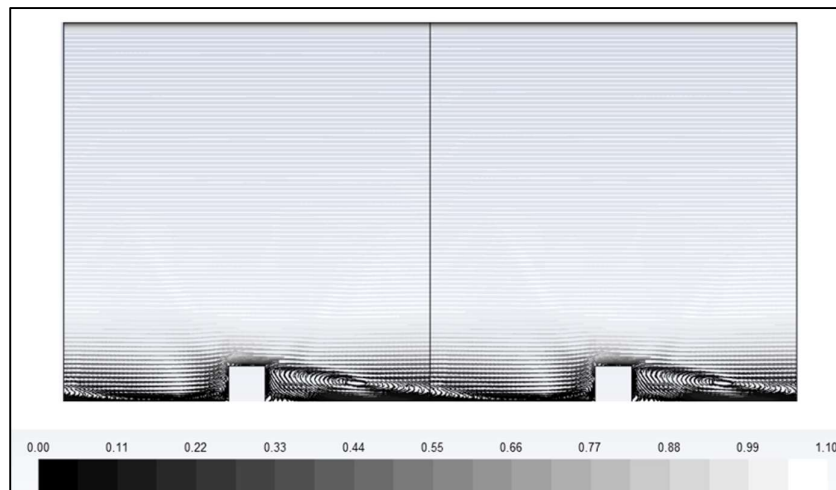


Figure 4.27: Streamline of periodic boundary solution in a stationary condition.

As can be seen in the **Figure 4.27.**, the streamline curves are better than those in the full domain. While in the full domain results, the vortex is a little wider and fluffier, here the vortex structure is a little more crushed and thinner.

When the periodic results were examined, it was seen that the results for the static situation were close to the full domain.

5. SUMMARY AND DISCUSSION

Turbine cooling is one of the key technologies that enable today's engines to produce such large powers. It has been 84 years since the first turbine-cooled engine, Junkers Jumo 004, made its first run. Turbine cooling technologies have evolved over the past 84 years. In the beginning, turbine cooling designs were form of radial smooth channels, than it continued to develop into serpentine structures, and cooling features such as impingement, rib and pin-fin emerged and have been included to turbine cooling designs. In addition, material and coating technologies continued to develop in parallel. As a result of all this, turbines that can operate above melting temperature for thousands of hours have been developed through years.

Studies have been carried out on it for many years due to the high HTC enhancement values brought by rib turbulator cooling structures. In the first years, the ribs were produced with 90 degree angles, then they became angled, and in recent years they have become broken. Nowadays, rib turbulator cooling structures are used in almost all aircraft engines.

In the current study, an extensive literature review of rib turbulator structures was conducted. The effects of different rib structures on heat transfer are explained with physical explanations. The compatibility of RANS CFD models with test results of rib turbulator structures was investigated. In order to avoid any assumptions while establishing analysis models, a study was sought in which all parameters were given transparently and flow field and heat transfer measurements were reported. A test campaign conducted at the Von Karman Institute was found to comply with all the specified requirements, and analysis models were established based on this data set, taking into account the reliability of the institution.

Analyzes were run at a fixed Reynolds number of 15000, in 3 different Rotation numbers. Since 3 different mesh and 4 turbulence models were examined, analyzes were performed and reported under a total of 36 conditions.

The results of the study are listed below;

- k-kl ω is the best turbulence model for correct calculating a reattachment length.
- k- ϵ Realizable is the best for HTC calculation at a stationary condition.
- k-kl ω is the best for HTC calculation at a rotational conditions.
- Periodicity check has been achieved almost all 36 runs, except some runs at a destabilized rotary condition.
- There is no significant impact of y^+ value on calculating HTC and flow field structures at a stationary and stabilized rotary condition.
- In the case of destabilized rotary condition, mesh and turbulence model have a huge effect on results.

Differences of up to 70% were found between analysis models and test conditions. The main reason for this is that RANS models cannot solve recirculation zone structures correctly in reattachment flow problems. RANS models found the reattachment length to be almost 2 times longer than the test. This situation also led to incorrect calculation of the developing boundary layer region that creates high HTC. However, HTC distribution contours, periodicity curves and streamline contours in the analyzes are largely in line with the test. Although absolute values do not match, trends are reported to be similar.

In the unstable condition shown as $Ro: 0.3$, there are problems with trends between the CFD and test results. Under this condition, it was observed that the analyzes were greatly affected by the mesh and turbulence models. Unfortunately, analysis results of this condition was not consistent each other, on the contrary of $Ro: -0.3$ and $Ro: 0$ where the results were compatible. Under the mentioned condition, as a future study, analyzes in different mesh types should be prepared again and a robust analysis setup should be found.

As a result of the study, it was seen that RANS models are not suitable for reattachment flow problems. Cooling designs of turbine blades can be improved with LES studies that provide more accurate solutions. However, the computing power requirement that LES brings is unfortunately not suitable for industrial studies. In this condition, the errors came by RANS models should be accepted and these error rates should be taken into consideration during design.

REFERENCES

- [1] Barber, J. (1791). “Obtaining and applying motive power, & c. a method of rising inflammable air for the purposes of procuring motion, and facilitating metallurgical operations.” UK Patent No. GB1833.
- [2] Web 1, (2024), collection.sciencemuseumgroup.org.uk. Access Date: 27 January 2024.
- [3] Web 2, (2024), <https://www.asme.org/about-asme/engineering-history>. Access Date: 11 June 2024
- [4] Bakken, L. E., Jordal, K., Syverud, E., & Veer, T. (2004). Centenary of the first gas turbine to give net power output: A tribute to ægidius elling. In *Turbo Expo: Power for Land, Sea, and Air* (Vol. 41677, pp. 83-88).
- [5] Eckardt, D. (2013). “Gas turbine powerhouse: the development of the power generation gas turbine at BBC-ABB-Alstom.”, Walter de Gruyter.
- [6] Armstrong, F. (2020). Farnborough and the beginnings of gas turbine propulsion. *J. Aeronaut. Hist. Pap*, 2(2).
- [7] Kreith, Frank, ed. (1998). “The CRC Handbook of Mechanical Engineering (Second ed.).” US: CRC Press. p. 222.
- [8] Web 3, (2012), <https://web.archive.org/web/20120313093034/http://www.ruhr-uni-bochum.de/fem/pdf/in-touch-magazin2005.pdf>. Access Date: 13 August 2012.
- [9] Golley, J., Whittle, F., & Gunston, B. (1996). Genesis of the jet: Frank Whittle and the invention of the jet engine. *Airlife*.
- [10] Eckardt, D., & Rufli, P. (2002). Advanced gas turbine technology: ABB/BCC historical firsts. *J. Eng. Gas Turbines Power*, 124(3), 542-549.
- [11] Rolls Royce, P. (2015). “The jet engine.” John Wiley & Sons.
- [12] Han, J. C., Dutta, S., & Ekkad, S., (2012). “Gas turbine heat transfer and cooling technology.”, CRC press.
- [13] Han, J. C., & Dutta, S. (1995). Internal convection heat transfer and cooling: An experimental approach. *Lecture series-van Kareman Institute for fluid dynamics*, 5, C1-C147.
- [14] Park, J. S., Han, J. C., Huang, Y., Ou, S., & Boyle, R. J. (1992). Heat transfer performance comparisons of five different rectangular channels with parallel angled ribs. *International Journal of heat and mass transfer*, 35(11), 2891-2903.
- [15] Han, J. C., (May 1, 1988). Heat Transfer and Friction Characteristics in Rectangular Channels With Rib Turbulators. ASME. *J. Heat Transfer*, 110(2): 321–328.

- [16] Han, J. C., & Park, J. S. (1988). Developing heat transfer in rectangular channels with rib turbulators. *International journal of heat and mass transfer*, 31(1), 183-195.
- [17] Park, J. S., Han, J. C., Huang, Y., Ou, S., & Boyle, R. J. (1992). Heat transfer performance comparisons of five different rectangular channels with parallel angled ribs. *International Journal of heat and mass transfer*, 35(11), 2891-2903.
- [18] Han, J. C., Zhang, Y. M., and Lee, C. P. (August 1, 1991). Augmented Heat Transfer in Square Channels With Parallel, Crossed, and V-Shaped Angled Ribs. ASME. *J. Heat Transfer*, 113(3): 590–596.
- [19] Han, J. C., & Zhang, Y. M. (1992). High performance heat transfer ducts with parallel broken and V-shaped broken ribs. *International Journal of heat and mass transfer*, 35(2), 513-523.
- [20] Metzger, D. E., Chyu, M. K., Bunker, R. S., & Kim, J. H. (1988). The contribution of on-rib heat transfer coefficients to total heat transfer from rib-roughened surfaces. *Transport Phenomena in Rotating Machinery*.
- [21] Taslim, ME, & Lengkong, A. (1998) "45° Round-Corner Rib Heat Transfer Coefficient Measurements in a Square Channel." Proceedings of the ASME 1998 International Gas Turbine and Aeroengine Congress and Exhibition. Volume 4: Heat Transfer; Electric Power; Industrial and Cogeneration. Stockholm, Sweden. June 2–5, 1998. V004T09A038. ASME.
- [22] Chandra, P. R., & Cook, M. M. (1994). Effect of the number of channel ribbed walls on heat transfer and friction characteristics of turbulent flows. *ASME-PUBLICATIONS-HTD*, 271, 201-201.
- [23] Keshmiri, A., Osman, K., Benhamadouche, S., & Shokri, N. (2016). Assessment of advanced RANS models against large eddy simulation and experimental data in the investigation of ribbed passages with passive heat transfer. *Numerical Heat Transfer, PartB:Fundamentals*, 69(2),96–110.
- [24] Han, J., Zhang, Y., and Kalkuehler, K. (1993). Uneven Wall Temperature Effect on Local Heat Transfer in a Rotating Two-Pass Square Channel With Smooth Walls. ASME. *J. Heat Transfer*, 115(4): 912–920.
- [25] Johnston JP, Halleent RM, Lezius DK. (1972) Effects of spanwise rotation on the structure of two-dimensional fully developed turbulent channel flow. *Journal of Fluid Mechanics*, 56(3):533-557.
- [26] Han, J., Zhang, Y., and Kalkuehler, K. (1993). Uneven Wall Temperature Effect on Local Heat Transfer in a Rotating Two-Pass Square Channel With Smooth Walls. ASME. *J. Heat Transfer*, 115(4): 912–920.
- [27] Wagner, J. H., Johnson, B. V., and Kopper, F. C. (1991). Heat Transfer in Rotating Serpentine Passages With Smooth Walls. ASME. *J. Turbomach*, 113(3): 321–330.

- [28] Mayo, I., Arts, T., El-Habib, A., and Parres, B. (2014). Two-Dimensional Heat Transfer Distribution of a Rotating Ribbed Channel at Different Reynolds Numbers. ASME. *J. Turbomach*, 137(3): 031002.
- [29] Coletti, F., Maurer, T., Arts, T., & Di Sante, A. (2012). Flow field investigation in rotating rib-roughened channel by means of particle image velocimetry. *Experiments in fluids*, 52, 1043-1061.
- [30] Web 4, (2024)
<https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node1.htm>. Access Date: 14 July 2024.

BIOGRAPHY

The author, He graduated from Istanbul Technical University, Department of Mechanical Engineering in 2021. He has started his career in the gas turbine industry as a Thermal System Design and Component Cooling Engineer and is still working in this field.