

**T.C.
GEBZE TECHNICAL UNIVERSITY
GRADUATE SCHOOL**

**NUMERICAL INVESTIGATION OF COMPRESSOR STATOR
WELL**

AHMET CÜNEYT KAHRAMAN

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

ADVISOR: PROF. DR. ALİ ATA

JULY 2024

T.R.
GEBZE TECHNICAL UNIVERSITY
GRADUATE SCHOOL

NUMERICAL INVESTIGATION OF COMPRESSOR
STATOR WELL

AHMET CÜNEYT KAHRAMAN

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

ADVISOR: PROF. DR. ALİ ATA

JULY 2024

T.C.
GEBZE TEKNİK ÜNİVERSİTESİ
LİSANSÜSTÜ EĞİTİM ENSTİTÜSÜ

KOMPRESÖR STATÖR KUYUSUNUN NUMERİK
ANALİZİ

AHMET CÜNEYT KAHRAMAN

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

DANIŞMAN: PROF. DR. ALİ ATA

TEMMUZ 2024



MASTER of SCIENCE JURY APPROVAL FORM

A thesis submitted by Ahmet Cüneyt Kahraman, defended on 12/07/2024 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) : Prof. Dr. Ali ATA

MEMBER

: Dr. Erinç ERDEM

MEMBER

: Prof. Dr. Hasan KARABAY

APPROVAL

Gebze Technical University Graduate Administration Board

...../...../..... date and/..... numbered decision.

SIGNATURE/SEAL

ABSTRACT

This thesis investigates the flow characteristics within the compressor well cavity, a region located beneath the shrouded compressor stator. The cavity significantly influences both aerodynamic performance and the operation of the thermal and secondary air systems. Excessive leakage flow into the cavity degrades aerodynamic performance and increases windage power loss. Conversely, insufficient flow can result in elevated air temperatures exceeding material limits. Consequently, precise control of leakage flow rate is crucial for optimizing gas turbine engine efficiency.

First, experimental test data from the literature is utilized. The test case geometry is employed to understand flow behaviour within the compressor well cavity with varying inlet swirl ratio and turbulent flow parameter. The entire cavity is divided into three subcavities: the upstream cavity, the seal cavity, and the downstream cavity. Swirl ratio values are calculated for each cavity by implementing angular momentum conservation. It is observed that majority of the rotor torque originates from the upstream cavity. Moreover, for low inlet swirl ratio cases, which realistically represent stator exit swirl ratios, swirl ratio decreases with increasing turbulent flow parameter due to low momentum of the leakage air and rotor torques are predominant with $(1 - Xk)^{1.35}$.

Also, test data is used to validate Computational Fluid Dynamics method by comparing it with experimental results. Alongside the CFD results, the results of Flow Simulator, a 1D solver program similar to that described in Chapter 2, are presented. Overall, both CFD and Flow Simulator effectively capture flow characteristics consistent with experimental data. A mesh dependency study is conducted for the case with the highest rotational speed and pressure ratio.

Finally, a generic stator well case is selected to analyze flow parameters at three different power settings of the engine. Additionally, three different clearances for the highest power/thrust condition is examined to investigate clearance effect on the flow parameters within stator well. It is observed that upstream cavity is the main responsible for the windage heating, approximately %45. Rotor torques dominates within the upstream cavity. The radial distribution of the rotor torque sheds light into the possibility of reducing windage power by enhancing swirl ratio beyond the nominal geometrical features.

Keywords: Compressor Well Cavity, Rotor-Stator Cavity, Windage Heating, Labyrinth Seal, Computational Fluid Dynamics (CFD)

ÖZET

Bu tez, örtülü (shrouded) kompresör statorunun altında bulunan bir bölge olan kompresör kuyusu boşluğu içindeki akış özelliklerini araştırmaktadır. Boşluk hem aerodinamik performansı hem de termal ve ikincil hava sistemlerinin çalışmasını önemli ölçüde etkilemektedir. Boşluğa aşırı sızıntı akışı aerodinamik performansı düşürür ve rüzgar gücü kaybını (windage power loss) artırır. Tersine, yetersiz akış malzeme sınırlarını aşan yüksek hava sıcaklıklarına neden olabilir. Sonuç olarak, gaz türbini motor verimliliğini optimize etmek için kaçak akış oranının hassas kontrolü çok önemlidir.

İlk olarak, literatürdeki deneysel test verilerinden yararlanılmıştır. Test geometrisi, değişen giriş girdap oranı (swirl ratio) ve türbülanslı akış parametresi ile kompresör kuyusu boşluğu içindeki akış davranışını anlamak için kullanılır. Tüm boşluk üç alt boşluğa bölünmüştür: yukarı (upstream) akış boşluğu, sızdırmazlık (seal) boşluğu ve aşağı (downstream) akış boşluğu. Her bir boşluk için girdap oranı değerleri açısal momentum korunumu uygulanarak hesaplanmıştır. Rotor torkunun büyük kısmının yukarı akış boşluğundan kaynaklandığı gözlemlenmiştir. Ayrıca, stator çıkış girdap oranlarını gerçekçi bir şekilde temsil eden düşük giriş girdap oranı durumları için, türbülanslı akış parametresinin artmasıyla girdap oranı azalır, çünkü kaçak havanın momentumu düşüktür ve rotor torkları $(1 - Xk)^{1.35}$ ile baskındır.

Ayrıca, Hesaplamalı Akışkanlar Dinamiği (HAD) yöntemini deney sonuçlarıyla karşılaştırarak doğrulamak için deneysel bir test vakası seçilmiştir. CFD sonuçlarının yanı sıra, Bölüm 2'de açıklanana benzer bir 1B çözücü programı olan Flow Simulator'ın sonuçları da sunulmuştur. Genel olarak, hem HAD hem de Flow Simulator'ı deneysel verilerle tutarlı sonuçlar vermektedir. En yüksek dönme hızı ve basınç oranına sahip durum için bir ağ bağımlılığı çalışması yapılmıştır.

Son olarak, motorun üç farklı güç ayarında akış parametrelerini analiz etmek için genel bir stator kuyusu seçilmiştir. Ayrıca, stator kuyusu içindeki akış parametreleri üzerindeki conta boşluğu etkisini araştırmak için en yüksek güç/itki koşulu için üç farklı boşluk incelenmiştir. Yukarı akış boşluğunun, yaklaşık %45'lik bir oranla rüzgarlama etkisinden ana sorumlu olduğu gözlemlenmiştir. Rotor torkları yukarı akış boşluğu içinde baskındır. Rotor torkunun radyal dağılımı, nominal geometrik özelliklerin ötesinde girdap oranını artırarak rüzgar gücünü azaltma olasılığına ışık tutmaktadır.

Anahtar Kelimeler: Kompresör Stator Kuyusu, Rotor-Stator Boşluğu, Rüzgarlama Etkisi ile Isınma, Labirent Contası, Hesaplamalı Akışkanlar Dinamiği (HAD)

ACKNOWLEDGEMENTS

I extend my sincere gratitude to my instructor and chief engineer, Erinç Erdem, for his invaluable guidance and support throughout my TEI career. His mentorship was instrumental in my technical development, and their encouragement propelled me to the successful completion of this study.

I would like to express my appreciation to my colleague, Onur Garip, for his collaborative spirit and generous sharing of technical expertise. His contributions were essential to the advancement of this research.

My sincere appreciation goes to my family, whose steadfast support has been a constant source of strength throughout my working and studying life. Their encouragement has been instrumental in helping me reach this point.

Finally, I dedicate this work to my beloved wife and best friend, İpek Su Yılmaz Kahraman. Words cannot adequately express my gratitude for her unconditional love and unwavering support.

TABLE OF CONTENTS

	<u>Page</u>
ABSTRACT	v
ÖZET	vi
ACKNOWLEDGEMENTS	vii
TABLE OF CONTENTS	viii
LIST OF SYMBOLS AND ABBREVIATIONS	ix
LIST OF FIGURES	xi
LIST OF TABLES	xiii
1. INTRODUCTION	14
1.1. Problem Definition	14
1.2. Aerodynamic Perspective	15
1.2. Thermo-Mechanical and Secondary Air System (SAS) Perspective	22
1.3. Research Objective	26
2. THEORETICAL BACKGROUND	27
2.1. Rotor – Stator Cavity Flows	27
2.1.1. Enclosed Rotor-Stator System	28
2.1.2. Rotor-Stator Cavity with Radial Outflow	31
2.1.3. Rotor-Stator Cavity with Radial Inflow	31
2.2. Windage Heating	32
2.3. Labyrinth Seal Flows	37
3. MATHEMATICAL FORMULATION	31
3.1. Navier-Stokes Equations	31
3.2. RANS Equations	33
4. ONE DIMENSIONAL CAVITY SOLVER FORMULATION	36
5. VALIDATION TEST CASE	45
5.1. Test Facility and Rig Instrumentation	45
5.2. The Numerical Simulation Setup, Grid Generation and BC's	47
5.3. The Results of Numerical Simulations	51
6. GENERIC STATOR WELL CAVITY	57
6.1. The Numerical Simulation Setup, Grid Generation and BC's	57
6.2. The Results of Numerical Simulations	60
7. SUMMARY AND DISCUSSION	75
REFERENCES	77
BIOGRAPHY	80

LIST OF SYMBOLS AND ABBREVIATIONS

A	: Area
C_d	: Discharge coefficient
C_f	: Skin friction coefficient
C_m	: Moment Coefficient
cl, C_r	: Clearance
C_{m_1}	: Moment coefficient for regime I
C_{m_2}	: Moment coefficient for regime II
C_{m_3}	: Moment coefficient for regime III
C_{m_4}	: Moment coefficient for regime IV
c_p	: Specific heat capacity
C_w	: Non-dimensional flow rate
G	: Gap ratio
H	: Total enthalpy per unit mass
k	: Thermal conductivity coefficient
L	: Seal pitch distance
$\dot{m}$	: Mass flow rate
N	: Shaft speed
P, P_s, p	: Static pressure
P_t	: Total pressure
PR	: Pressure ratio
r, R	: Disc radius
Re_θ	: Rotational Reynolds number
Re'_θ	: Modified rotational Reynolds number
s	: Axial gap of the cavity, clearance
t	: Seal teeth thickness, time
T_T	: Total temperature
u, v, w	: Velocity components
u_τ	: Friction velocity
V_θ	: Tangential velocity
$\dot{W}_{windage}$	: Windage power
XK	: Swirl ratio
y^+	: Non-dimensional wall distance
α	: Carry-over factor
λ_T	: Turbulent flow parameter
μ	: Dynamic viscosity
ν	: Kinematic viscosity
ρ	: Density
ω, Ω	: Rotational speed
Γ	: Torque
τ_θ	: Tangential component of the shear stress
Φ	: Dissipation
θ	: Windage number
DNS	: Direct Numerical Solution
LES	: Large Eddy Simulation

RANS	:	Reynolds-Averaged Navier-Stokes
rpm	:	Revolution per minute
SAS	:	Secondary air system
SST	:	Shear Stress Transport Model

LIST OF FIGURES

	<u>Page</u>
Figure 1.1: Generic Stator Well Cavity Flow Field Features.	17
Figure 1.2: Typical cavity parameters.	19
Figure 2.1: Characteristic velocity profiles in a rotor-stator cavity (a) to (c) Batchelor type of flow structure; (d) to (f) Stewartson type of flow structure [19].	28
Figure 2.2: Rotating disk in an enclosed cavity [16].	29
Figure 2.3: Regimes of flow for an enclosed rotor-stator cavity [20].	30
Figure 2.4: Schematic of a rotor-stator cavity with superimposed radial outflow: (a) small outflow rate and (b) large outflow rate [16].	31
Figure 2.5: Schematic of a rotor-stator cavity with superimposed radial inflow: (a) small inflow rate and (b) large inflow rate [16].	32
Figure 2.6: Typical Rotor - Stator Cavity [16].	34
Figure 2.7: Representation of kinetic energy carry-over coefficient [23]	39
Figure 2.8: CD graph for Vermes equation [25]	41
Figure 4.1: Simplified Compressor Well Cavity.	36
Figure 4.2: Cavity Swirl Distributions Based on a Simplified Bulk Model with $X_{Kinlet} = 0.05$.	40
Figure 4.3: Cavity Swirl Distributions Based on a Simplified Bulk Model with $X_{Kinlet} = 0.15$.	42
Figure 4.4: Cavity Swirl Distributions Based on a Simplified Bulk Model with $X_{Kinlet} = 0.3$.	42
Figure 4.5: Cavity Swirl Distributions Based on a Simplified Bulk Model with $X_{Kinlet} = 0.5$.	43
Figure 5.1: The Experimental Model [13].	45
Figure 5.2: The schematic of the test rig [13].	46
Figure 5.3: The Locations of Measurements [13].	47
Figure 5.4: Computational Grid for Test Case Geometry.	50
Figure 5.5: Computational Grid Dependency Study with, $Re\theta = 5.99E + 6$ and $\lambda T = 0.191$, for Case 10, in terms of Exit Swirl Ratio, Mass Flow Rate, Discharge Coefficient and Windage Number.	51
Figure 5.6: Swirl Ratio at R=263 mm, Mass Flow Rate and Exit Temperature for PR=1.1.	52
Figure 5.7: Swirl Ratio at R=263mm, Mass Flow Rate and Exit Temperature for PR=1.3.	53
Figure 5.8: Swirl Ratio Contours for PR=1.1, 5100 rpm (top) and 8100 rpm (bottom).	55
Figure 5.9: Swirl Ratio Contours for PR=1.3, 5100 rpm (top) and 8100 rpm (bottom).	56
Figure 6.1: Computational Domain and Boundary Conditions.	57
Figure 6.2: Computational Grid Dependency Study for Case 3 Based on C_w , XK at $x = 0.96$ and Total Temperature Rise.	59
Figure 6.3: Swirl Ratio Contours, Top: Case1, Middle: Case2, Bottom: Case3.	61
Figure 6.4: Non-Dimensional Stream Function Contours, Top: Case1, Middle: Case2, Bottom: Case3.	62
Figure 6.5: Swirl Ratio Contours, Top: Case5, Middle: Case3, Bottom: Case4.	63
Figure 6.6: Non-Dimensional Stream Function Contours, Top: Case5, Middle: Case3, Bottom: Case4.	64

Figure 6.7: Variation of Swirl Ratio, Non-Dimensional Static Pressure and Total Temperature Rise within Stator Well: Case1.	65
Figure 6.8: Variation of Swirl Ratio, Non-Dimensional Static Pressure and Total Temperature Rise within Stator Well: Case2.	66
Figure 6.9: Variation of Swirl Ratio, Non-Dimensional Static Pressure and Total Temperature Rise within Stator Well: Case3.	67
Figure 6.10: Variation of Swirl Ratio, Non-Dimensional Static Pressure and Total Temperature Rise within Stator Well: Case4.	68
Figure 6.11: Variation of Swirl Ratio, Non-Dimensional Static Pressure and Total Temperature Rise within Stator Well: Case5.	69
Figure 6.12: Total Temperature Rise Contributions from Each Sub Cavity, Top: Cases 1-3, Bottom: Cases 3-5.	70
Figure 6.13: Torque Contributions from Each Sub Cavity, Top: Rotor Torque, Bottom: Stator Torque.	71
Figure 6.14: Non-Dimensionalized Torque Distributions in Radial Direction for Upstream Cavity (round symbols) and Downstream Cavity (diamond symbols).	72
Figure 6.15: Variation of Swirl Ratio and Total Temperature Rise within Stator Well for Different Surface Roughness Values: Case3.	73

LIST OF TABLES

	<u>Page</u>
Table 1.1: Typical Dimensions and Parameter Ranges [4].	18
Table 5.1: CFD model setup.	48
Table 5.2: Boundary Conditions for CFD Simulations.	49
Table 5.3: Calculated Parameters	49
Table 6.1: Boundary Conditions for CFD Simulations.	58
Table 6.2: The Calculated Parameters.	59

1. INTRODUCTION

Gas turbine engines have been developed through centuries and remarkable increase in the efficiencies of compressors and turbines have been achieved. In today's multistage core compressors are designed to have high efficiency values reaching beyond %92 polytropic efficiency. However, because of the fact that increasing cost of the fuel and increasing of the environmental concerns, engines with low specific fuel consumption is highly preferred by airplane companies, which necessitates further advancements in gas turbine technology. As further advancements in compressor aerodynamic design encounter escalating complexity, comprehensive understanding and mitigation of secondary losses such as rotor blade tip leakages and seal leakages assume paramount importance.

1.1. Problem Definition

Effects of the blade tip leakages on the compressor efficiency, stall margin and the pressure rise have been investigated by numerous researchers up to date. In general, interested tip clearance ratios to span height is around %3. For each 0.01 increase in the clearance-to-span ratio, compressor efficiency typically experiences a 1.5 percentage point reduction [1]. Moore and Obsorn [2] investigated various tip clearance ranged from 0.061 to 0.178 centimeter. It was found that stall margin decreased from 15 percent to 3 percent, as tip clearance increased to 0.102 centimeter from 0.061. Further increase in tip clearance to 0.178 centimeter resulted in a complete stall margin loss (0%). Wisler [3] investigated experimentally two different ratio of the blade tip clearance to span height, %1.36 and %2.8. It ended up with a decrease in peak efficiency by 1.5 points, a decrease in stalling flow coefficient by %11, indicating a smaller flow range before the compressor stalls, and a decrease in peak pressure rise by 9.7%.

The design of compressor stator vanes hinges upon two distinct configurations: cantilevered and shrouded. Cantilevered vanes comes into prominence in terms of manufacturing efficiency, facilitated assembly, and minimized maintenance burdens. Additionally, their continuous inner annulus line offers aerodynamic benefits. However, the necessity of substantial clearances between the vanes and the casing to

preclude contact leads to a precipitous decline in efficiency as tip clearance expands. Conversely, shrouded vanes demonstrate superior vibrational damping and are generally recognized to achieve superior aerodynamic efficiency. This advantage arises from the inherent challenges associated with effectively mitigating tip leakage in cantilevered designs [4].

Furthermore, when employing a shrouded stator design, a dedicated cavity forms beneath the stator vane, designated as the stator well. This cavity necessitates the implementation of labyrinth seals, incorporating either abradable linings or honeycomb structures. These seals function to impede the migration of air from the downstream side of the vane to the upstream side, driven by the existing static pressure differential. The level of leakage flow is directly correlated with the labyrinth seal's clearance value.

The accounting of leakage flows during the early stages of compressor design is paramount. Neglecting this crucial aspect leads to an overestimation of the compressor's actual efficiency, potentially resulting in significant performance discrepancies and downstream operational shortcomings. Integrating leakage flow analysis early in the design process allows for the fine-tuning of key parameters like clearance tolerances, seal configurations, and flow path geometry. This ultimately promotes the attainment of a more realistic and achievable efficiency target.

1.2. Aerodynamic Perspective

Research on stator well flows and temperatures can be broadly categorized into two main areas based on their primary objectives. The first area, aerodynamic objectives, delves into the impact of the stator well on the flow behavior within the compressor annulus. This research is particularly interested in how the presence of the stator well influences the overall efficiency of the compressor. A key area of focus within this domain is the interaction between the airflow exiting the front cavity of the stator well and the main annulus flow. Researchers in this field aim to understand how this interaction disrupts or "spoils" the overall flow pattern within the compressor, potentially leading to performance losses. The second major research area lies within the domain of air system and thermo-mechanical studies. This domain focuses on the air temperature rise in the cavity due to windage effect, local thermal conditions within

the stator well, specifically the temperatures experienced by the component. Researchers here are particularly interested in the heat transfer that occurs at the surface between the stator well and the disc rims. Understanding these two parameters is crucial for optimizing both material selection and stress analysis of components located near the stator wells. This optimization ultimately aims to ensure the structural integrity and longevity of these components under demanding operating conditions.

Axial compressors utilize shrouded stator vanes that are interposed between a dual-ring assemblies. The outer ring serves as a fixed attachment point to the compressor housing. Conversely, the inner ring provides the necessary anchoring points for the stator vanes. Due to the rotor blades residing within designated grooves on the rotor disks, the rotor shaft must incorporate a cavity between adjacent disks to house the inner ring. This cavity, particularly critical in high-speed compressors, fulfills a vital function: it establishes a precise clearance between the high-speed rotating rotor and the stationary inner ring, also known as the landing. This clearance plays a crucial role in ensuring the operational stability of the stator vanes. Typical stator well cavity is shown in Figure 1.1. leakage flow travels radially inward, adhering to stator wall. Some portion is pumped radially outward on rotor surface. Before the labyrinth seal, another small circulation zone occurs due to abrupt turning of the flow, which increases sealing performance. As travelling the labyrinth seal, flow experiences pressure loss, minimizing leakage flow. Flow is pumped radially outward on rotor surface at downstream cavity and some portion of air moves stator surface, generating another circulating flow pattern.

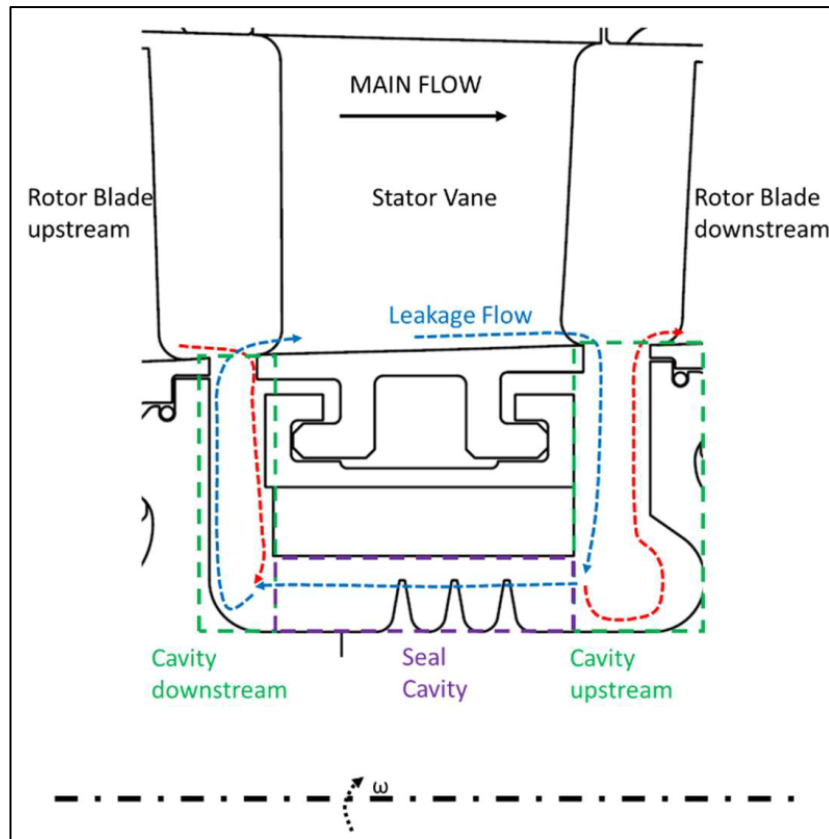


Figure 1.1: Generic Stator Well Cavity Flow Field Features.

Although frictional losses within a stator cause a decrease in total pressure, static pressure actually experiences an increase. This phenomenon occurs as the stator acts to decelerate the high-swirl airflow exiting the rotor, thereby reducing its rotational component (swirl). This rise in static pressure establishes a pressure-driven flow path within the cavity of the rotor shaft. This secondary flow path operates in the opposite direction of the main airflow through the compressor.

Minimizing leakage flow through the rotor cavity is paramount for the aerodynamic research community due to its detrimental effects on compressor performance. Leakage flow represents a departure from the ideal flow path within the compressor, essentially spoiling the stator hub flow. This "spoiling" disrupts the organized flow pattern established by the stator vanes in two key ways. Firstly, the leakage flow typically possesses a lower angular momentum compared to the main flowpath stream. As these flows merge, the leakage flow induces negative incidences on the stator leading edge. This negative incidence disrupts the smooth flow transition over the vane, resulting in a thickening of the boundary layer on the pressure side of the vane. Secondly, the presence of leakage flow can exacerbate an existing flow phenomenon

known as the horseshoe vortex. This vortex naturally forms on the pressure side of the stator vane regardless of leakage flow. However, the "spoiling effect" of the leakage flow contributes to the growth of this horseshoe vortex. Additionally, the low pressure on the suction side of the vane can cause this secondary flow structure to migrate towards that side. The combined effects of a thickened boundary layer and a larger horseshoe vortex lead to an increase in both stator losses and annulus blockage. These phenomena have a significant negative impact on the overall aerodynamic performance of the compressor, reducing efficiency and potentially leading to performance degradation.

In a 2002 study, typical dimensions and parameter ranges associated with compressor stator well are categorized by Lewis [4], and it is shown in Table 1. While these values are specific to the data presented in the study, they nonetheless offer valuable insights into the critical parameters governing the compressor stator well domain. These dimensions are defined in Figure 1.2.

Table 1.1: Typical Dimensions and Parameter Ranges [4].

Parameter	Typical Range
Annulus Passage Height	25 to 60 mm
Hub Radius (from centerline)	250 to 330 mm
Annulus flow	100 to 150 kg/sec
Seal Flow (as proportion of annulus flow)	%0.1 to %0.4
Seal Clearance (as proportion of span height)	%0.3 to %2.0
Stator Well Depth	20 to 25 mm
Rotational speed of the rotor	10000 to 13000 rpm
Hub Static pressure in annulus	7 to 27 Mpa
Total air temperature in annulus	550 to 900K
Hub swirl velocity fraction at stator inlet	0.50 to 0.60
Hub swirl velocity fraction at stator exit	0.05 to 0.08

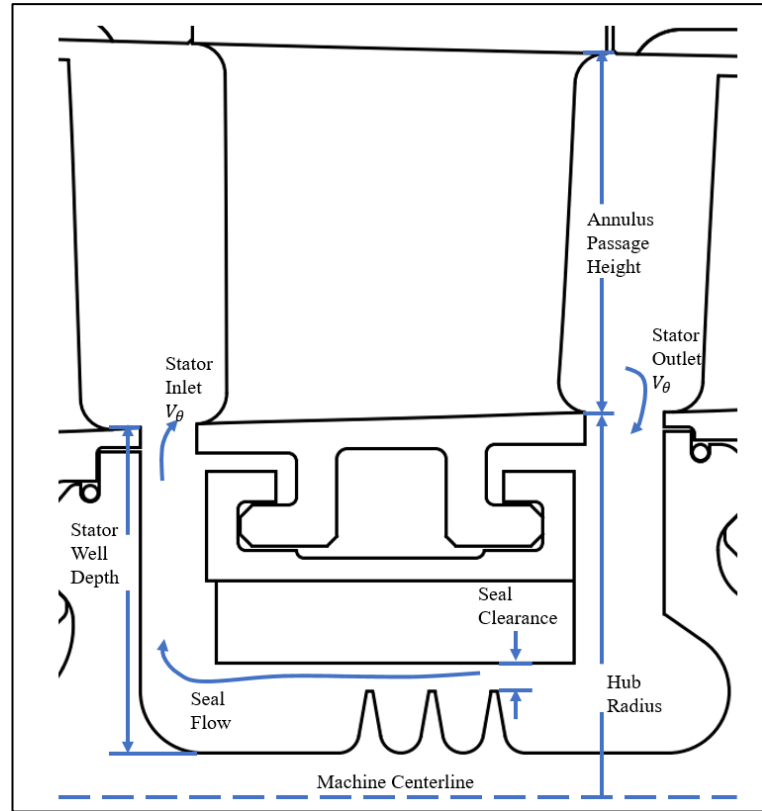


Figure 1.2: Typical cavity parameters.

As mentioned before, one of the central focus of this investigation is the influence of cavities on the characteristics of the primary airflow and the overall aerodynamic efficiency of the compressor. Driven by competitive pressures within the gas turbine industry, axial compressors are continuously pushed towards higher pressure ratios and rotor drum speeds. This pursuit, however, leads to increased leakage flow rates and the emergence of unfavorable flow conditions. As such, a profound understanding of cavity behavior becomes essential to minimize their detrimental effects on the aerodynamic performance of the axial compressor.

An experimental investigation by Wellborn and Okiishi [5] conducted a low-speed multistage axial compressor to evaluate the impact of shrouded stator cavity flows on compressor aerodynamic performance. According to authors, increasing the seal-tooth clearance-to-span ratio (seal clearance to annulus passage height in Figure 1.2) by 1% has a demonstrated detrimental effect on compressor performance. Their data indicates a 1 point reduction in performance and a 3% decrease in the pressure rise. Wellborn and Okiishi [5] explained that performance degradation of the compressor due to increased leakage flow with two distinct ways. Firstly, a rise in seal-tooth leakage directly degrades the near-hub performance of the stator row where it occurs. This

additional working fluid generates increased flow blockage, deviation, and total pressure loss near the stator hub endwall. This additional blockage forces the airflow in a radial direction towards the shroud, thereby reducing the aerodynamic loading on a significant spanwise portion of the stator. Secondly, the altered flow conditions exiting the stator due to increased leakage impair the performance of the next downstream stage. This occurs by decreasing the work input of the downstream rotor and increasing the total pressure loss of the downstream stator. These effects contribute to a cumulative performance degradation for each subsequent stage in the compressor.

Also, Kamdar et al. [6] showed that increasing seal clearance within the stator well caused degradation of the compressor performance and total pressure ratio. It is stated 0.5 point increase in efficiency penalty and 0.6 percent increase in total pressure rise increase are observed for every 1-percent increase in ratio of labyrinth seal clearance to stator span.

Demargne and Longley [7] performed experiments on a linear cascade in which leakage mass flow and tangential velocity are controlled independently in order to investigate the interaction of the stator shroud leakage with mainstream flow. Their experimental results confirm the adverse influence of increasing leakage mass flow rate, but also unequivocally demonstrate that increasing cavity tangential velocity enhances the aerodynamic performance of the blade row, which leads to performance of the compressor largely insensitive to the amount of the leakage flow. The research also identified rotational speed as another parameter influencing the circumferential velocity of the leakage flow. An increase in rotational speed resulted in a corresponding increase in circumferential velocity, suggesting that cavity flows at part-speed might differ from those near design speed. More importantly, the data indicated that the change in circumferential velocity decreased as the discharge flow angles increased. Therefore, the main parameters concluded to be dictating the interaction of cavity leakage and primary flows are the leakage mass flow rate and circumferential velocity of the leakage flow.

Heidegger et al. [8] simulated the seal cavity flow. Their study showed that leakage flow rate through a compressor seal cavity significantly impacts the flow profile at the stator blade. The study found that increasing the size of the gap or removing the seal entirely led to a rise in leakage flow (up to 3.56% of the main passage mass flow). This leakage flow accelerated rapidly within the cavity, reaching over two-thirds of its

final exit tangential velocity by -15% stator span in the downstream cavity trench. This rapid acceleration resulted in a final tangential velocity of approximately 75% of the hub wheel speed. The high tangential velocity of the leakage flow exiting the cavity caused increased windage and a higher total temperature. Additionally, the leakage flow's high tangential momentum resulted in a 20-degree increase in flow incidence near the hub compared to the mid-span, leading to flow separation on the stator blade's suction surface. The authors emphasize that the injection of high tangential momentum leakage flow must be considered during compressor design, with corrective measures focused on the stator blade design rather than the seal cavity itself. Moreover, a study by Kamdar et al. [6] found that increasing the ratio of the seal clearance to span leads an increase near-hub incidence, which causes the increase in the possibility of the hub corner separation. Yet, they found that hub leakage flow has a minor effect on the stator diffusion factor. It is stated that whilst there is an increase in diffusion factor within the %10 span, a decrease in the diffusion factor are observed for the rest of the passage.

Understanding the flow structure of the seal cavity is also crucial in order to minimize negative effects of the leakage flow on the stator performance at near the hub. Research consistently shows swirling flow patterns (vortices) exist within compressor seal cavities. A study by Wellborn [9] found these vortical flow structures near the cavity's entrance and exit, with most fluid movement happening close to the downstream sides of these areas. Another investigation by Heidegger et al. [8] showed that cavity vortical flow structures cause blockage near the hub, pushing the flow profiles higher, especially at the leading edge for high leakage scenarios. Demargne and Longley [7] observed similar swirling flow patterns in their experiment on a linear cascade at low leakage rates. Interestingly, these vortical flow structures vanished at leakage rates exceeding 0.66%, and the researchers suggested they play a crucial role in how fluids and flow properties exchange between the cavity and the main flow path when leakage is low. Also, in their simulation, Kamdar et al. [6] found that even the blockage area near the hub of the stator increases as the leakage flow rate increases, they observed minimum hub corner separation for two cases which the leakage flows are not minimum. So, they concluded that there is an optimum clearance of the labyrinth seal within the compressor well to reduce and optimize flow blockage, which may not be the case of zero leakage flow rate.

1.2. Thermo-Mechanical and Secondary Air System (SAS) Perspective

However, the air system community emphasizes the potential for excessive temperature rise within the cavity due to windage. To reconcile these competing objectives, a labyrinth seal is strategically positioned between the rotating shaft and the stationary inner ring (landing) of the compressor. This seal plays a crucial role in achieving a delicate balance. It aims to minimize leakage flow, as desired by the aerodynamic community, while simultaneously permitting a controlled amount of airflow to mitigate windage heating, as emphasized by the air system community.

The labyrinth seal features radially protruding teeth that extend outwards from the rotor shaft towards the landing. However, to prevent catastrophic failure or damage, these teeth must maintain a specific clearance from the landing. This clearance allows for a certain degree of recirculation within the cavity, opposing the primary flow. A distinguishing characteristic of compressor stator labyrinth seals compared to conventional designs is the presence of inlet and outlet cavities (wells) formed by the rotor and stator disks, along with extended rotating walls. The flow structure, swirl patterns, and windage heating within these stator wells play a significant role in influencing the leakage characteristics of the labyrinth seal.

Leakage flow originating from the downstream region of the stator vane exhibits a low swirl characteristic. As this flow travels radially inwards, it adheres to the stator wall. This phenomenon, combined with the rotating disk on the rotor side, establishes a disk-pumping flow pattern, which imparts additional angular momentum to the leakage flow. It is noteworthy that only a portion of the circulating flow within the rear section of the stator well actually penetrates the labyrinth seal. As the leakage flow traverses the labyrinth seal, it experiences a further increase in angular momentum due to the frictional interaction within seal passages. Upon exiting the seal, the flow path undergoes a reversal, with the leakage flow travelling outwards on the rotor side. This outward flow ultimately merges with the main flow path upstream of the stator vane. The confluence of the leakage flow with the main flow path introduces an additional layer of complexity. On the stator side, the leakage flow induces entrainment of fluid from the main flow, creating a distinct circulating flow pattern. This interaction between the flows also leads to a reduction in the swirl ratio within the downstream

cavity, as the leakage flow rejoins the main flow at a higher radial position. Furthermore, throughout its entire circuitous path, the leakage flow experiences "windage heating". This phenomenon refers to the transfer of energy from the high-speed rotating components, such as the upstream and downstream cavities and the labyrinth seal itself, due to frictional interaction. This windage transfer represents a parasitic loss, as the energy extracted from the rotating components is not utilized for the core function of the compressor, which is to increase the pressure of the working fluid. The cumulative effect of these phenomena associated with stator leakage flow manifests as a reduction in overall compressor efficiency. The energy imparted to the leakage flow through disk-pumping and windage transfer, along with the flow disruptions caused by entrainment, all contribute to a decrease in the usable work done by the compressor on the working fluid.

A distinguishing characteristic of compressor stator labyrinth seals, compared to conventional labyrinth seals, lies in the presence of dedicated inlet and outlet rotor-stator disc cavities (wells) and extended rotating walls. The intricate flow structure, including the development of swirl patterns and the generation of windage heating within these stator wells, exerts a significant influence on the leakage characteristics of the labyrinth seal. The interaction of the compressor well cavity with the primary flow, the cavity's internal flow structure, windage heating phenomena, and the re-ingestion of hot fluid back into the mainstream airflow constitute a critical area of investigation.

In addition to the importance of the effects of the compressor seal cavity on the aerodynamic performance of the compressor, the rising temperature of leakage flow within the cavity presents significant consequences for the secondary air system and thermal-mechanical aspects of the compressor. The tangential velocity of the flow inside the cavity reaches a fraction of the rotor wheel speed. Because of this difference, shear work done on the leakage flow that causes the temperature increase, known as windage heating, of the flow inside the cavity. The details of this phenomena will be presented in the next chapters. Temperature increase should be limited in order not to decrease the life of the material and to use economic materials for the disc of the rotor and shroud of the stator. To mitigate the effects of windage heating and maintain suitable materials for the rotor disc and stator shroud, controlled leakage of mainstream air into the cavity should be allowed.

In the second part of their study Kamdar et al. [10] showed that total temperature rise and tangential velocity of the leakage air affected by the seal clearance, the size of vortical flow structures, and air flowpath (along the stator surface and pumping in rotor surface), where these parameters are also interdependent. Moreover, they found that even the total temperature rise within the stator well increases as the flow rate decreases with decreasing seal clearance, overall windage heating decreases, which highlight that mass flow rate dominates the windage heating since the circumferential velocity of the air increases (decrease in velocity difference between air and rotor surface) as flow rate increases and this cause reduction on the shear work done by the rotor surface to leakage air.

Ozturk et al. [11] made 3D computational studies to investigate the leakage flow and windage heating inside the compressor stator well cavity. Their investigation focused on temperature rise at various radial and axial locations. The study revealed that after ingestion of the flow into the cavity, flow starts to be heated. Moreover, the highest temperature were observed at rotor surface. Also for all regions, downstream cavity, labyrinth seal and upstream cavity, the highest temperatures on the rotor surface were seen on the inner radius of the rotor, which indicates that the most critical component of the stator well is the inner radius of the rotor. One other findings of their study is that temperature of the flow increases as passes through the throttling regions of the seal, and temperature decreases afterwards, which indicates that tip of the labyrinth seal teeth are subjected to heating and thermal stresses. Moreover, authors modified the cavity in order to restrict the flowpath and reduce the leakage mass flow without changing the seal clearance. A circumferential ring added to the downstream rotor disc within the compressor stator well cavity. The results showed a %9 decrease in both mass flow through the labyrinth seal and windage heating within the system. The authors explain the mechanism behind the reduced flow and heating. The ring placement lowers the pressure at the labyrinth seal inlet compared to the baseline case. This weaker pressure gradient translates to less flow being driven from the downstream well to the upstream well, resulting in a %9 flow reduction. Furthermore, even the mass flow rate decreases %9, the difference of the flow temperature leaves the upstream well between base and modified case insignificant, which implies a reduction in the shear stress responsible for windage heating, likely by around %9.

Building upon their previous work, Ozturk et al. [12] then studied seven different stator well configurations in order to investigate the effect of incorporation of fins on the rotor surface in order to reduce leakage mass flow rate. The aim was that by increasing the tangential speed of the leaked air at the inlet of the labyrinth seal, pressure of the air would decrease, and causes the reduction in the mass flow rate without changing the labyrinth seal clearance. The authors observed that incorporating radial fins on the rotor surface led to a significant reduction in leakage flow. Notably, this reduction was only slightly greater than that achieved with the baseline model featuring a halved clearance.

Kong et al. [13] employed a combined experimental and numerical approach to investigate total temperature variations within a generic compressor stator well cavity. The study focused on the influence of labyrinth seal clearances on windage heating. By implementing labyrinth seal rings with varying tip clearances, the researchers observed a near-linear relationship between increasing rotor speed and windage heating. Additionally, the study revealed an inverse relationship between tip clearance and windage heating within the labyrinth seal.

Also Kong et al. [14] observed in both their numerical and experimental studies that most dominant part for the windage heating is inlet cavity contributing around %50 of the total temperature rise, whereas the least portion of the windage heating occurred at outlet cavity with %20. Moreover, these proportion of the windage temperature rise for each cavity within the stator cavity are in line with the findings of the study presented by Kamdar et al. [10].

Ingested air flow into the cavity comes from the stator hub region, and it mixes with mainstream flow at the near hub region when it emerges from the compressor stator well cavity. Since ingested air flow is heated inside the cavity due to shear work done from the rotor surfaces, this flow raises the near hub mainstream temperature. This phenomena brings out the possibility of re-ingestion of the hot air that passed through the stator blade row into the downstream stator well cavity for further windage heating. In Ozturk et al. [11] numerical studies, the researchers investigated whether the hot layer near the stator hub could migrate through the stator passage and reach the outlet, where the air flows into the cavity, or whether passage vortex and turbulence dissipate and diffuse the hot layer and leads to spanwise mixing of the stator flow. To achieve this, the authors defined a temperature boundary condition at the inlet. This condition

specified a temperature of 500 K for a thin layer near the hub, encompassing only 5% of the total blade height. The remaining spanwise region was assigned a temperature of 443 K. Their results showed that while the temperature near the hub exhibits a reduction across most of the circumferential span as the flow traverses the blade row, the exit temperature remains comparatively elevated relative to the bulk flow, which empowers the possibility of the repetitive windage heating of ingested flow. Also, Bayley and Childs [15] observed that lower circulation rates in the tight clearance configuration resulted in increased peak temperature rises, reaching up to 30 °C. They further hypothesized that re-ingestion of hot air might occur, potentially exacerbating windage and stator cavity well heating. These findings are also supported by the study of Kamdar et al. [10] since the highest temperatures are observed at the cases of re-ingestion of the hot fluid. Consequently, continuous re-ingestion of the hot air into the stator well cavity could lead to a progressive rise in both air and surfaces temperatures, which can cause the necessity of high temperature resistance material usage in the cavity or decrease of the material life.

1.3. Research Objective

In today's multistage core compressors, achieving further advancements poses increased challenges because they have already surpassed 92% polytropic efficiency. Consequently, comprehending secondary losses becomes even more crucial.

The objective of this study is to understand the flow characteristics within the compressor well cavity using 1D numerical method, validating the 2D axisymmetric CFD model with experimental data from existing literature. Subsequently, this study aims to examine the generic stator well cavity encountered in the gas turbine engines. Various parameters including mass flow rate leaked to the cavity, total temperature rise of the leaked air, swirl distribution within the cavity, torques of both stator and rotor surfaces, along with their proportion for main separated regions, are investigated by non-dimensionalizing to establish comprehensive understanding of the flow behaviour within the compressor stator well cavity.

2. THEORETICAL BACKGROUND

Prior to 1D and 2D numerical investigations, it is essential to establish a strong theoretical foundation. This foundation should encompass the principles governing main flow parameters within the compressor stator well, including rotor-stator cavity flows, windage heating mechanisms, and the labyrinth seal flows.

2.1. Rotor – Stator Cavity Flows

Within the core flow path of gas turbine engines, rotor-stator cavities and rotor-rotor cavities are frequently encountered design features. A thorough comprehension of the flow parameters governing these cavities is paramount for the development of accurate one-dimensional models. However, it is crucial to acknowledge that only rotor-stator cavities are present within the compressor well domain. Therefore, the following discussion will exclusively focus on defining and characterizing rotor-stator cavities. These cavities are characterized by while radial inflow occurs at the upstream cavity, radial outflow occurs at the downstream cavity.

In its most general form, as Sultanian [16] stated rotor surface tends to pump flow radially outward and serves as a bladeless compressor or turbine, depending on whether the adjacent fluid core rotates slower or faster than the disk. In the former scenario, energy transfers from the rotor disk to the fluid, while in the latter, it moves from the fluid to the disk. Tangential velocity on the disk varies linearly with radius. Stator torque reduces the flow's angular momentum, regardless of its direction or rotation, but isn't directly involved in energy transfer with the fluid.

Two main model had proposed about how fluid flows within the space between a rotating disc and stationary wall before experimental data and computational simulations were available.

The first model, proposed by Batchelor [17] in the 1950s, assumes that the angular velocity of the core fluid lies between that of the rotor and stator. This model suggests the presence of boundary layers on both rotor and stator surfaces. Whilst the fluid that is entrained from the core is pumped radially outwards on the rotor to supply its boundary layer, radial inflow occurs at the stator surface and air moves from the stator's boundary layer to core region.

Conversely, Stewartson [18] proposed another model that assumes the tangential velocity of the fluid decreases from the rotor's boundary layer to the value of zero, resulting in no core rotation, similar to “free discs” flow. Velocity profiles for these two models are presented in the Figure 2.1. To sum up, both of these flow models can be encountered in the gas turbine engines. Since the gap ratios is generally large enough, Batchelor model is more common flow model.

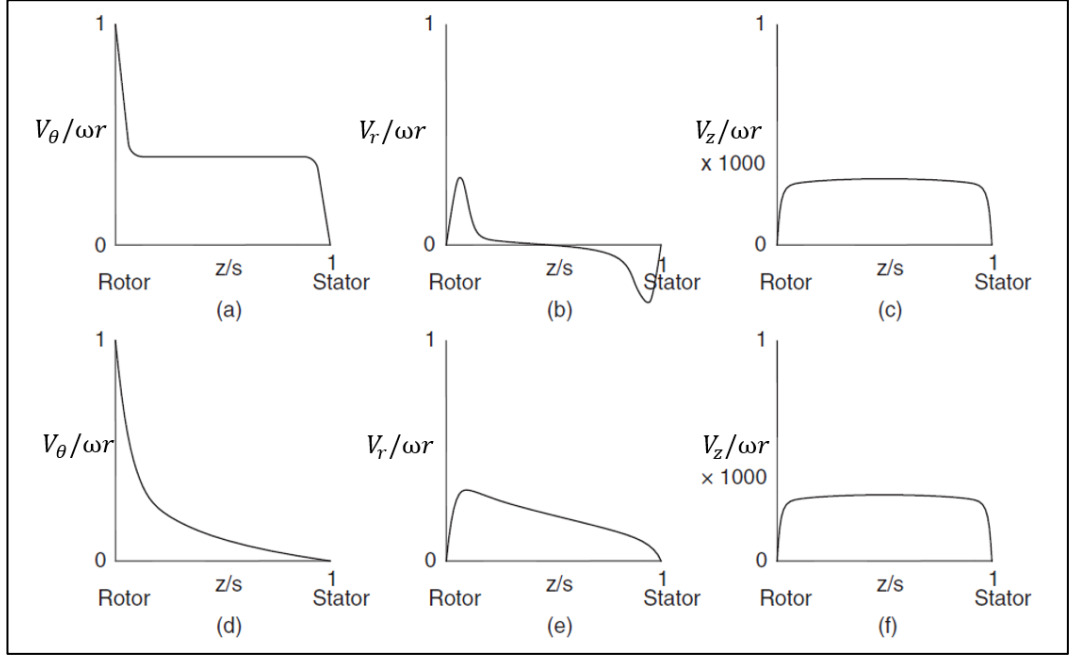


Figure 2.1: Characteristic velocity profiles in a rotor-stator cavity (a) to (c) Batchelor type of flow structure; (d) to (f) Stewartson type of flow structure [19].

2.1.1. Enclosed Rotor-Stator System

Batchelor flow model is best to describe the flow behavior for the enclosed rotor-stator system where the no throughflow supplied or extracted as shown in the Figure 2.2. The core flow occurs between two boundary layers rotating with some fraction of the rotor wheelspeed. Typically, the fluid in the core region rotates at a tangential velocity equal to 40% of the rotor's speed. This fraction is characterized as “swirl ratio” in the gas turbine applications,

$$XK = \frac{\omega_{fluid}}{\omega_{rotor}} = \frac{V_{\theta fluid}}{\omega_{rotor} r} \quad (2.1)$$

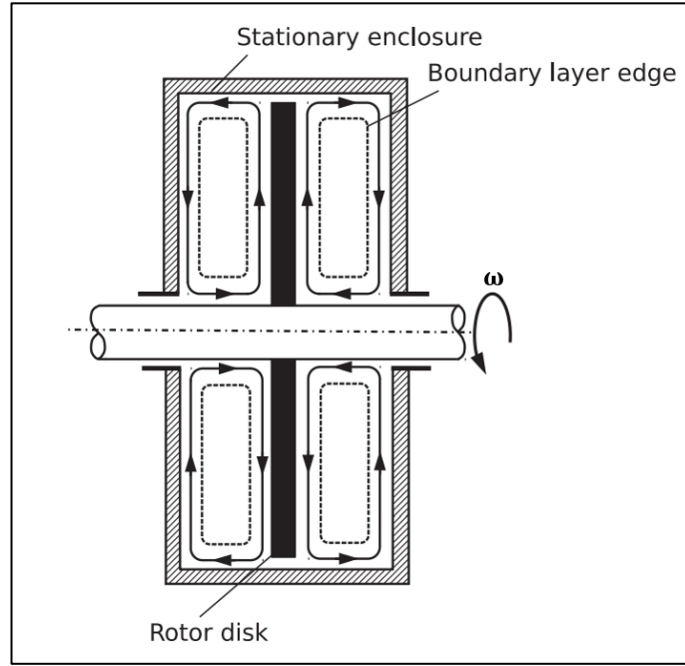


Figure 2.2: Rotating disk in an enclosed cavity [16].

Also gap ratio which is the ratio of axial clearance, s , between the rotor and stator surfaces to disc radius, R , is defined in Equation 2.2. Daily and Nece [20] reported that as the gap ratio decreases, swirl ratio increases to value of $XK \approx 0.5$.

$$G = \frac{s}{R} \quad (2.2)$$

For an enclosed disc cavity, Daily and Nece [20] conducted comprehensive experimental and theoretical investigations. Through experiments, conducted with various disc speeds and gap ratios, velocity, pressure, and torque measurements were made. While rotational Reynolds number is ranged from $Re_\theta = 10^3$ to $Re_\theta = 10^7$, gap ratio is ranged from $G = 0.0127$ to $G = 0.217$ with a constant tip clearance. Consequently, the researchers identified four distinct regions, illustrated in Figure 2.3, can occur within the enclosed cavity depending on the rotational Reynolds number and gap ratio.

Specification of the flow regimes with specific moment coefficient are defined below for each regimes.

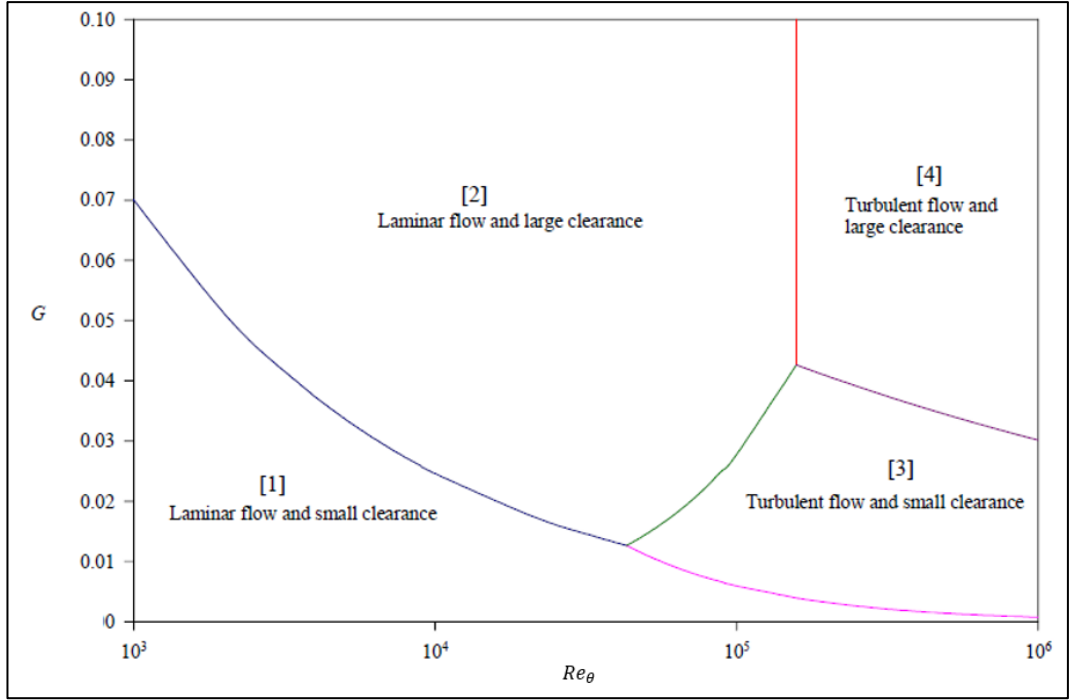


Figure 2.3: Regimes of flow for an enclosed rotor-stator cavity [20].

- Regime I : Laminar flow, small gap ratio, merged boundary layers,

$$C_{m_1} = \frac{\pi}{G \cdot Re_\theta} \quad (2.3)$$

- Regime II : Laminar flow, large gap ratio, separate boundary layers,

$$C_{m_2} = 1.85 \cdot G^{0.1} Re_\theta^{-0.5} \quad (2.4)$$

- Regime III : Turbulent flow, small gap ratio, merged boundary layers,

$$C_{m_3} = 0.04 \cdot G^{-0.167} Re_\theta^{-0.25} \quad (2.5)$$

- Regime IV : Turbulent flow, large gap ratio, separate boundary layers,

$$C_{m_4} = 0.051 \cdot G^{0.1} Re_\theta^{-0.2} \quad (2.6)$$

In gas turbines, the Reynolds numbers reach high levels during the high power/thrust conditions, resulting in predominantly turbulent flow. Additionally, the gaps between

the rotating disk and stationary components are sufficiently wide, leading to Regime IV becoming the most probable occurrence.

2.1.2. Rotor-Stator Cavity with Radial Outflow

For rotor-stator cavity with radial outflow, Sultanian [16] states that in cases where the superposed flow rates is insufficient to meet the pumping requirements of the rotor disc, external fluid can be ingested. This fluid flows radially inward on the stator surface until it reaches a radius where the disc's pumping flow rate exceeds the superimposed flow rate to make up for the difference, which is known as “rotationally-induced ingress”. Contrary, if the superimposed flow is enough to meet the demand of the rotor disc pumping, no radially inward flow occurs on the stator surface. These cases are shown in Figure 2.4.

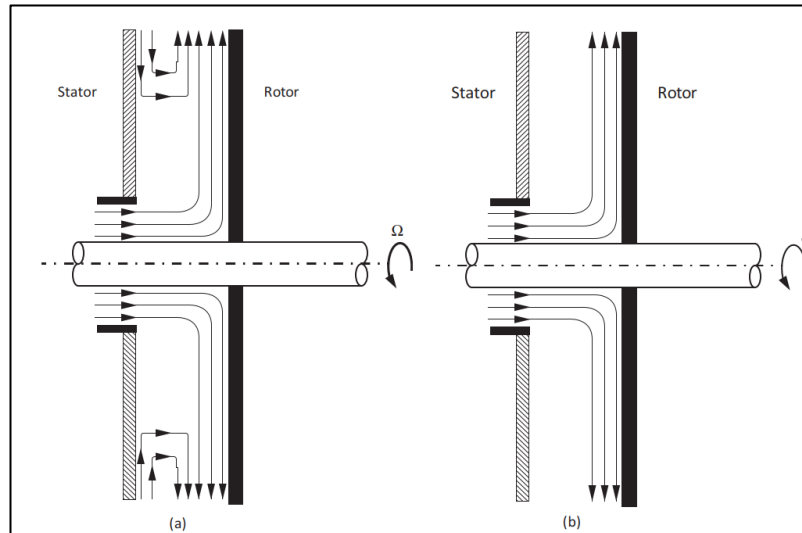


Figure 2.4: Schematic of a rotor-stator cavity with superimposed radial outflow: (a) small outflow rate and (b) large outflow rate [16].

2.1.3. Rotor-Stator Cavity with Radial Inflow

A rotor-stator cavity featuring a superimposed radial inflow is illustrated in Figure 2.5. Sultanian [16] states that for a small inflow rate cases with $XK < 0.5$, flow enters the cavity along stator surface. Some portion of this air moves to the rotor surface in order to meet its pumping requirement, causing a reversal of flow over a section of the cavity near the rotor surface. As the flow reaches a lower radius, it accelerates similarly to a free vortex and tends to move downward along the rotor surface to minimize wall shear

force. This region of the rotor disk, experiencing radial inward flow, functions as a bladeless turbine. Conversely, in the case of a large radial inflow shown in Figure 2.5b, the flow behaves more like a free vortex and predominantly descends along the rotor surface to minimize shear force. In this scenario, the entire rotor disk operates as a bladeless radial turbine, fluid rotates faster than the rotor surface ($XK > 1$), with energy transferred from the fluid to the rotor.

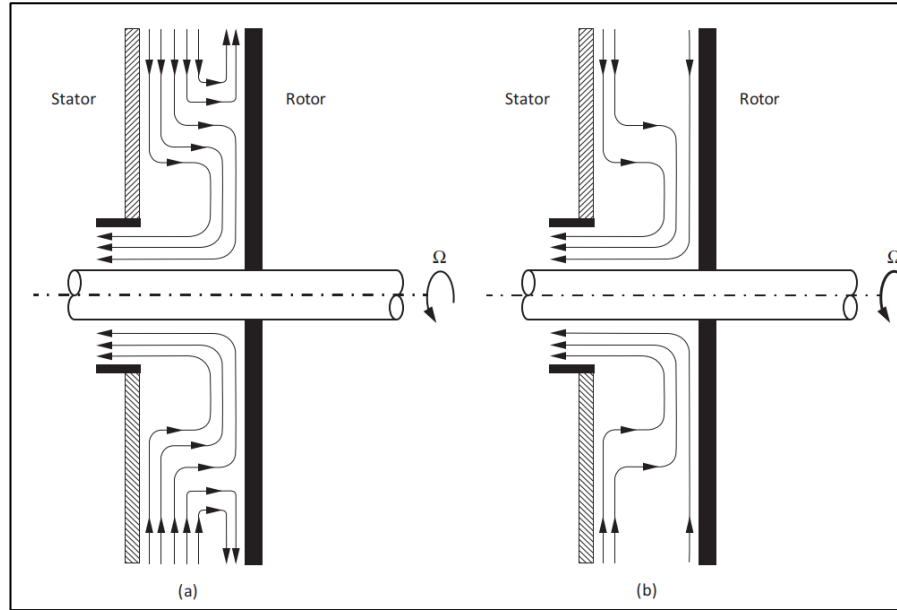


Figure 2.5: Schematic of a rotor-stator cavity with superimposed radial inflow: (a) small inflow rate and (b) large inflow rate [16].

2.2. Windage Heating

Within a gas turbine engine, the total air temperature experiences a modification due to either work transfer, heat transfer, or a synergistic effect of both processes. Work transfer occurs exclusively within rotating components, arising from the interaction between the fluid and the moving walls. This interaction establishes a velocity gradient across a boundary layer adjacent to the wall. However, the no-slip boundary condition dictates zero relative velocity between the fluid and the stationary wall. Consequently, wall shear stress acting on the stationary surfaces cannot contribute to any net work transfer between the fluid and the wall. In essence, windage heating is precisely defined as the rotor's power input into the surrounding fluid. It's crucial to differentiate this phenomenon from viscous dissipation, which represents the process by which the

fluid's viscosity transforms external flow energy into internal energy, with no net change in the fluid's total temperature under adiabatic conditions. While stators themselves cannot directly impart work to the fluid, stationary surfaces can indirectly influence windage by establishing velocity gradients within the fluid. This occurs because the fluid experiences a deceleration due to the negative torques exerted by the stationary walls [16].

As illustrated in Figure 2.6, a typical rotor – stator cavity configuration is presented. Under adiabatic conditions, in order to find temperature change of the fluid, work power transferred from the rotating component should be calculated. So, windage power ($\dot{W}_{windage}$) is defined as:

$$\dot{W}_{windage} = \Gamma_{rotor}\Omega \quad (2.7)$$

Where Γ_{rotor} is the rotor torque and Ω is the rotational speed of the rotor disc.

When considering the balance of the angular momentum between two sections, we obtain,

$$\Gamma_{rotor} - \Gamma_{stator} = \dot{m}V_{\theta_2}r_2 - \dot{m}V_{\theta_1}r_1 = \dot{m}(XK_2r_2^2 - XK_1r_1^2)\Omega \quad (2.8)$$

$$\Gamma_{rotor} = \Gamma_{stator} + \dot{m}(XK_2r_2^2 - XK_1r_1^2)\Omega \quad (2.9)$$

Where Γ_{rotor} and Γ_{stator} , respectively, are the rotor torque and stator torque, V_{θ} is tangential velocity of the air, XK is the swirl ratio and Ω is the rotational speed of the rotor. When these two equations are combined,

$$\dot{W}_{windage} = \Gamma_{stator}\Omega + \dot{m}(XK_2r_2^2 - XK_1r_1^2)\Omega^2 \quad (2.10)$$

Where first term on right-hand in Equation 2.10 denotes that work done by stator to fluid, which cannot be true because of the fact that there is no relative motion of the stator. Hence, one should utilize Equation 2.7 and Equation 2.9 separately to prevent nonphysical representation. Then, temperature change of the air because of the rotor work done can be calculates as

$$\Delta T_T = T_{T_2} - T_{T_1} = \frac{\dot{W}_{windage}}{\dot{m}c_p} \quad (2.11)$$

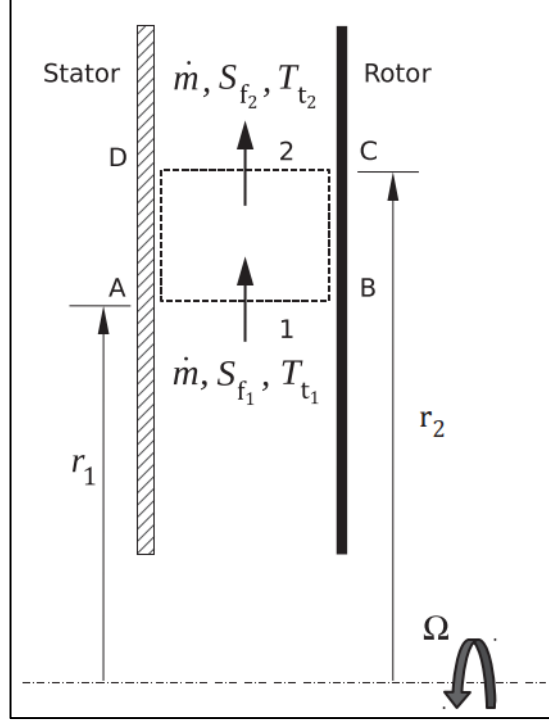


Figure 2.6: Typical Rotor - Stator Cavity [16].

The calculation of rotor and stator torques relies on empirical correlations derived from experimental data. Since the rotor torque stems from the shear stress in the tangential direction over the area, it can be calculated as

$$\Gamma_{rotor} = \int_0^R \tau_\theta r dA \quad (2.12)$$

Where $dA = 2\pi r dr$

$$\Gamma_{rotor} = \int_0^R \tau_\theta r (2\pi r dr) = 2\pi \int_0^R \tau_\theta r^2 dr \quad (2.13)$$

Shear stress in the tangential direction can be calculated as,

$$\tau_{\theta} = \frac{1}{2} C_f \rho (wr)^2 \quad (2.14)$$

Assuming constant skin friction throughout the radius,

$$\Gamma_{rotor} = C_f \rho \omega^2 \pi \int_0^R r^4 dr \quad (2.15)$$

$$\Gamma_{rotor} = \frac{C_f \rho \omega^2 \pi R^5}{5} \quad (2.16)$$

And in terms of moment coefficient, C_m ,

$$\Gamma_{rotor} = \frac{C_f \rho \omega^2 \pi R^5}{5} = \frac{1}{2} C_m \rho \omega^2 R^5 \quad (2.17)$$

Which gives the relation between skin friction coefficient and moment coefficient as

$$C_m = \frac{2\pi}{5} C_f \quad (2.18)$$

Based on the actual gas turbine test experience, and by using the experimental data of Daily and Nece [20], the empirical correlation for the skin friction coefficient on one side of the rotor proposed by Haaser et al. [21] as

$$C_{f_R} = 0.042(1 - XK)^{1.35} Re_{\theta}'^{-0.2} \quad (2.19)$$

And for the one side of the stator disk as

$$C_{f_S} = 0.063XK^{1.87} Re_{\theta}'^{-0.2} \quad (2.20)$$

Where the full disk Reynolds number ($Re_{\theta} = \rho \omega R^2 / \mu$) has been modified to,

$$Re_{\theta}' = \frac{\rho \omega R_o (R_o - R_i)}{\mu} \quad (2.21)$$

However, Sultanian [16] has found that if these C_f values are used for the calculation of the partial disk torques where $R_i/R_o < 0.5$, partial disk torques leads to higher values than full disks, which is physically impossible. In order to prevent this misleading, instead of using constant shear stress as assumed in Haaser et al. [21], Sultanian [16] assumed that skin friction coefficient of the tangential stress is constant. Using relative tangential air velocity to evaluate the dynamic pressure, dynamic pressure changes to, respectively, $0.5\rho(1 - XK)^2\omega^2r^2$ and $0.5\rho XK^2\omega^2r^2$ for rotor and stator. Then skin friction coefficients are re-derived for stator and rotor disks as,

$$C_{f_R} = 0.070 (1 - XK)^{-0.65} Re_\theta^{-0.2} \quad (2.22)$$

$$C_{f_S} = 0.105 XK^{-0.13} Re_\theta^{-0.2} \quad (2.23)$$

Then, rotor torque equation for partial disk becomes,

$$\Gamma_{rotor} = C_{f_R} \rho (1 - XK)^2 \omega^2 \pi \int_{R_i}^{R_o} r^4 dr \quad (2.24)$$

$$\Gamma_{rotor} = C_{f_R} \rho (1 - XK)^2 \omega^2 \pi \left(\frac{R_o^5}{5} - \frac{R_i^5}{5} \right) \quad (2.25)$$

$$\Gamma_{rotor} = 0.044 \rho (1 - XK)^{1.35} \omega^2 (R_o^5 - R_i^5) Re_\theta^{-0.2} \quad (2.26)$$

As mentioned earlier, windage power is calculated as,

$$\dot{W}_{windage} = \Gamma_{rotor} \omega \quad (2.27)$$

$$\dot{W}_{windage} = 0.044 \rho (1 - XK)^{1.35} \omega^3 (R_o^5 - R_i^5) Re_\theta^{-0.2} \quad (2.28)$$

Then, finally windage power equation becomes by expanding Rotational Reynolds number into windage power equation,

$$\dot{W}_{windage} = 0.044\rho^{0.8}(1 - XK)^{1.35}\omega^{2.8}\mu^{0.2}R_o^{4.6}\left(1 - \frac{R_i^5}{R_o^5}\right) \quad (2.29)$$

Windage power is affected most by R_o with a power of 4.6, which is followed by rotational speed of the disk ω with a power of 2.8. Third, the ratio of the difference of the air tangential velocity and rotor tangential velocity to the rotor tangential velocity, $(1 - XK)$, effects the windage power with a power of 1.35. Finally, air density and viscosity affect the windage power with a power of 0.8 and 0.2, respectively.

2.3. Labyrinth Seal Flows

Labyrinth seals find extensive application in gas turbine engines for multiple reasons. As they can be used to minimize the leakage flows, also they are used to meter the cooling flows as required for the downstream components. Labyrinth seals function by creating complex flow passages. These passages generate pressure loss as air flows through them, thereby reducing the overall airflow. The widespread preference for labyrinth seals stems from their advantageous combination of low cost, contactless operation, and the ability of broad operating range for temperatures, pressures, and rotor speeds [16].

In order to find mass flow rate passes through the labyrinth seals, analytical, experimental and numerical methods are used. For analytical approach, one method is very similar to orifice meter applications used in practice for flow measurement according to the cross-sectional constriction method. In orifice meters, 1-Dimensional flow, where the entire flow is considered as a streamline approach, the leakage flow rate can be calculated using the continuity and the Bernoulli equations. Then for compressible isentropic flow, flow rate can be calculated with,

$$\dot{m}_{theoric} = \frac{P_1 A_2}{\sqrt{kRT_1}} \frac{\sqrt{\frac{2k^2}{k-1} \left[\left(\frac{P_2}{P_1}\right)^{\frac{2}{k}} - \left(\frac{P_2}{P_1}\right)^{\frac{k+1}{k}} \right]}}{\sqrt{1 - \left(\frac{P_2}{P_1}\right)^{\frac{2}{k}} \left(\frac{A_2}{A_1}\right)^2}} \quad (2.30)$$

Venant developed an equation in 1871 derived from Bernoulli equation. Since Venant equation is developed from the Bernoulli equation, the narrow section where the

throttling occurs is taken as the low pressure region. The flow velocity at upstream cavity is negligible compared the flow velocity where throttling occurs. The final version of the Venant equation yields as,

$$\dot{m}_{theoric} = \frac{P_1 A_2}{\sqrt{k R T_1}} \cdot \sqrt{\frac{2k^2}{k-1} \left[\left(\frac{P_2}{P_1} \right)^{\frac{2}{k}} - \left(\frac{P_2}{P_1} \right)^{\frac{k+1}{k}} \right]} \quad (2.31)$$

Martin [22] developed an equation to estimate the leakage flow rate in labyrinth seals. This equation, shown below, is derived to estimate the required number of teeth to achieve the desired pressure drop. This equation is based on a purely analytical approach and it is in a good agreement with experimental data. The subsequent equations are based on Martin's equation and are focused on reducing the errors due to assumptions in the Martin's equation, where the developed convection or flow coefficients were added as multipliers to the Martin's equation. Also, while Venant's equation can be applied for a single tooth, the Martin equation can give the leakage flow for multiple teeth in a single calculation.

$$\dot{m} = \frac{A P_{t1}}{\sqrt{R T_1}} \cdot \sqrt{\frac{1 - \left(\frac{P_{s2}}{P_{t1}} \right)^2}{n - \ln \left(\frac{P_{s2}}{P_{t1}} \right)}} \quad (2.32)$$

Egli [23] add a kinetic energy carry-over coefficient, which is obtained from experimental data, to Martin's Equation.

$$\dot{m} = \mu^{emperical} \cdot \frac{A P_{t1}}{\sqrt{R T_1}} \cdot \sqrt{\frac{1 - \left(\frac{P_{s2}}{P_{t1}} \right)^2}{n - \ln \left(\frac{P_{s2}}{P_{t1}} \right)}} \quad (2.33)$$

In labyrinth seals, the fluid accelerates through each tooth. And in ideal cases, it is assumed that the flow expands completely in the cavity between two teeth and that all of the jet kinetic energy is absorbed and that this is repeated for each cavity. However, this is not the real case. As shown in the Figure 2.7, some of the fluid passing through the tooth will join the circulation inside the cavity, while some of it is directed directly

towards the other tooth as a jet. The added coefficient takes into account the portion of kinetic energy carried over from one cavity to subsequent cavity. Hence this multiplier is greater than 1 and has the effect of increasing the flow rate. Egli found that kinetic energy carry over factor is increased with increasing seal clearance or decreasing axial distance between seal teeth. The proportionality of the leakage flow rate generated using Egli's method is shown below.

$$\dot{m} \approx \frac{1}{\sqrt{n - \ln\left(\frac{P_2}{P_1}\right)}} \quad (2.34)$$

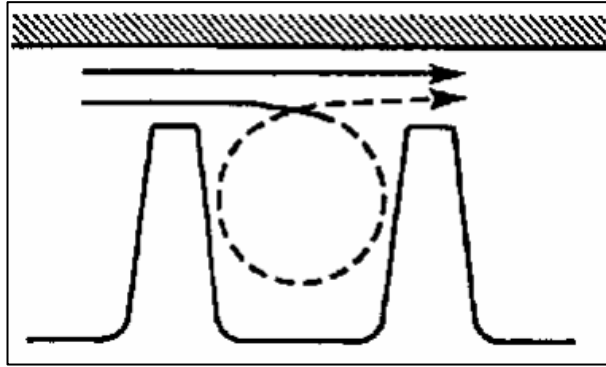


Figure 2.7: Representation of kinetic energy carry-over coefficient [23].

Hodkinson [24] made a modification to Egli's Equation. While Egli was using empirical coefficient to account the fact that all of the fluid kinetic energy is not dissipated between two teeth, Hodkinson changed the empirical correlation with semi empirical coefficient which uses geometric features of the labyrinth seals. His assumption was the conical expansion of the fluid jet at a small angle from the tip of the inlet tooth and undistorted entry of a portion into the next cavity.

Hodkinson found that kinetic energy carry over factor is increased with increasing of number of teeth and seal clearance and with decreasing of axial distance between two teeth. However, this increase is not infinite. After certain point, not much change is observed. Also, he observed that at the high pressure drop cases, carry-over factor becomes insignificant, seal clearance becomes the most dominant factor defining the flow rate. Carry-over factor becomes important at critical pressure ratios and if the liquid is used as a working fluid. Hodkinson's equation is stated below.

$$\dot{m} = \mu \cdot \frac{AP_1}{\sqrt{RT_1}} \cdot \sqrt{\frac{1 - \left(\frac{P_2}{P_1}\right)^2}{n - \ln\left(\frac{P_2}{P_1}\right)}} \quad (2.35)$$

$$\mu = \sqrt{\frac{1}{1 - \left(\frac{n-1}{n}\right) \cdot \left(\frac{\frac{Cr}{L}}{\left(\frac{Cr}{L}\right) + 0.02}\right)}} \quad (2.36)$$

Vermes [25] added his own kinetic energy carry-over factor coefficient, which is for flat and and stepped labyrinth seals and compressible fluids, to Martin's equation. This equation is developed from boundary layer theory, and to calculate the carry-over factor, following coefficient, α , is needed. His equation is defined as,

$$\dot{m} = Cd \cdot \mu \cdot \frac{AP_1}{\sqrt{RT_1}} \cdot \sqrt{\frac{1 - \left(\frac{P_2}{P_1}\right)^2}{n - \ln\left(\frac{P_2}{P_1}\right)}} \quad (2.37)$$

$$\mu = \sqrt{\frac{1}{1 - \alpha}} \quad (2.38)$$

$$\alpha = \frac{8.52}{\frac{s-t}{Cr} + 7.23} \quad (2.39)$$

On the Figure 2.8 below, the change of Cd value with Reynolds and the ratio of thickness of tooth to seal clearance is shown.

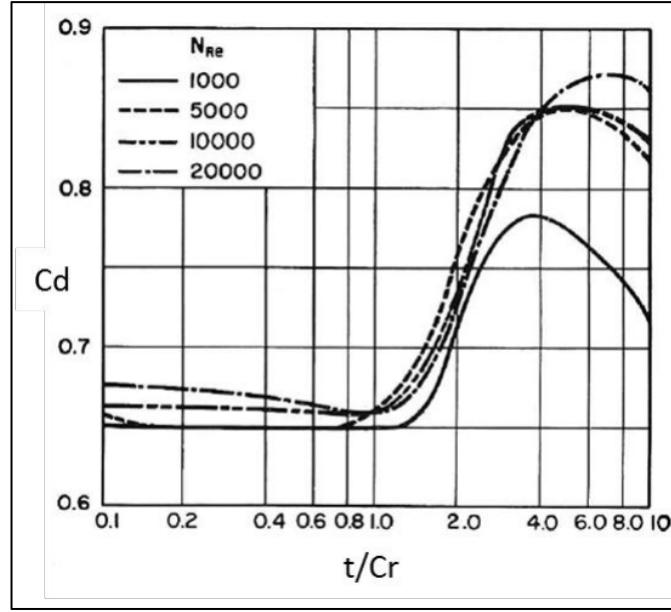


Figure 2.8: C_D graph for Vermes equation [25].

Neumann developed an equation which is a function of high and low pressures of the teeth (Childs [26]). Instead of single equation, it generates set of equations for the number of the teeth (n) that gives the pressure distribution and leakage mass flow rate. Neumann's equation includes a semi-empirical flow coefficient (C_f) and a kinetic energy transport coefficient (μ). To calculate the semi-empirical flow coefficient, the coefficient β is needed as a function of low and high pressures, and to calculate the kinetic energy transport coefficient, the coefficient α is needed as a function of seal clearance and pitch. These coefficients, calculated using the Chaplygin formula defined by Gurevich [27], which is used to calculate the Neumann equation, are given below.

$$\dot{m} = C_{f_i} \cdot \mu_i \cdot A_i \cdot \sqrt{\frac{P_i^2 - P_{i+1}^2}{RT}} \quad (2.40)$$

$$C_{f_i} = \frac{\pi}{\pi + 2 - 5\beta_i + 2\beta_i^2} \quad (2.41)$$

$$\beta_i = \left(\frac{P_i}{P_{i+1}} \right)^{\frac{k-1}{k}} - 1 \quad (2.42)$$

$$\mu_i = \sqrt{\frac{n}{n \cdot (1 - \alpha_i) + \alpha_i}} \quad (2.43)$$

$$\alpha_i = 1 - \frac{1}{\left(1 + 16.6 \cdot \frac{Cr_i}{s_i}\right)^2} \quad (2.44)$$

Zimmerman and Wolf [28] developed an equation for straight-through labyrinth seals and compressible flows that is in good agreement with the test results they presented. They proposed treating the first tooth different from the other teeth because of the fact that carry-over effect is not presented and that has better performance for reducing flow rate. They used the Venant's Equation for the first tooth and for the rest of the teeth Martin's Equation with the kinetic energy carry-over coefficient is used. The kinetic energy transport coefficient is similar to the Hodgkinson model and is used in the same way.

$$\dot{m} = \frac{P_1 A_2}{\sqrt{kRT_1}} \cdot \sqrt{\frac{2k^2}{k-1} \left[\left(\frac{P_2}{P_1}\right)^{\frac{2}{k}} - \left(\frac{P_2}{P_1}\right)^{\frac{k+1}{k}} \right]}, n = 1 \quad (2.45)$$

$$\dot{m} = \mu \cdot \frac{AP_1}{\sqrt{RT_1}} \cdot \sqrt{\frac{1 - \left(\frac{P_2}{P_1}\right)^2}{n - \ln\left(\frac{P_2}{P_1}\right)}}, n > 1 \quad (2.46)$$

3. MATHEMATICAL FORMULATION

Nowadays, computational fluid dynamics (CFD) programs, also known as flow solvers, are widely used in order to investigate flow characteristics within the compressor stator well since setting up a test setup is challenging and costly. These flow solver programs provide fast results and are supported by previous test results. So in terms of both time and cost, the use of such programs is quite functional and widespread. Flow solver programs utilizes basic conservation equations to provide results, which the most familiar of these are continuity and momentum equations. Yet, depending of the physics of the problem, heat transfer and compressible flow effects are involved, energy equations are also solved.

3.1. Navier-Stokes Equations

The continuity equation states that entering the mass flow into the defined control volume and leaving mass flow from that control volume are equal for steady state conditions. The continuity equation is defined in Equation 3.1;

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \quad (3.1)$$

If the velocity vector is expended in three dimension;

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (3.2)$$

Equation 3.2 represents the most general form of the continuity equation. However, for steady and incompressible problems, simplifications can be made. If the problem is steady, time dependent derivative of the density is zero. Also, since the density remains constant for incompressible flow, density term in the velocity component can be eliminated. So, the equation is simplified as;

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (3.3)$$

The momentum equation is based on Newton's 2nd law of motion, which states that the rate of the change of momentum of a system is equal to the net forces acting on the system. The forces acting on the system can be mass forces such as gravity or surface forces such as pressure, viscosity and reaction forces at the contact points. In its most closed form, the momentum equation is as in Equation 3.4.

$$\rho \left[\frac{\partial \vec{V}}{\partial t} + \vec{V}(\vec{\nabla} \vec{V}) \right] = -\vec{\nabla} p + \mu \nabla^2 \vec{V} + \rho \vec{g} \quad (3.4)$$

The momentum equation can be written for three dimension as

$$\rho \left(\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = -\frac{\partial P}{\partial x} + \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) + \rho g_x \quad (3.5)$$

$$\rho \left(\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = -\frac{\partial P}{\partial y} + \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) + \rho g_y \quad (3.6)$$

$$\rho \left(\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = -\frac{\partial P}{\partial z} + \mu \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) + \rho g_z \quad (3.7)$$

The conservation of the energy equation is derived from 1st law of the thermodynamics. It states that the rate of the change of energy on the fluid particle is equal to sum of the amount of net work done on the fluid particle and net heat transferred to fluid particle. Pressure and velocity distribution are taken from continuity and momentum equations in order to solve energy equation. The energy equation is written below in Equation 3.8. While the velocity term in the left represents the convection terms, second derivative of the temperature on the right represents the advective terms. Also, Φ term, dissipation term, represents heat generation originating from viscosity.

$$\rho c_p \left[\frac{\partial T}{\partial t} + (\vec{V} \cdot \vec{\nabla}) T \right] = k \nabla^2 T + \Phi \quad (3.8)$$

3.2. RANS Equations

Most of the flows in nature are turbulent and in order to simulate turbulent flows within computation domain, three primary turbulence modelling approaches are employed: Direct Numerical Solution (DNS), Large Eddy Simulation (LES) and Reynolds-Averaged Navier-Stokes (RANS). RANS method assumes a steady state and solves turbulence equations using time-averaged values. This method is the cheapest in terms of computational cost and the mostly used in the industrial applications.

So, applying this approach to continuity and momentum conservation equations with steady and incompressible assumption yields;

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (3.9)$$

$$\bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \overline{u'_i u'_j}}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j^2} \quad (3.10)$$

where ν is the kinematic viscosity. Equation 3.11 can be rearranged as,

$$\bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\nu \frac{\partial \bar{u}_i}{\partial x_j} - \overline{u'_i u'_j} \right] \quad (3.11)$$

Expressing viscous terms as shear stresses, Equation 3.12 can be written as,

$$\bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{1}{\rho} \frac{\partial}{\partial x_j} \left[\tau_{ij} - \rho \overline{u'_i u'_j} \right] \quad (3.12)$$

where τ_{ij} is the viscous stress tensor. The term $\rho \overline{u'_i u'_j}$ is defined as Reynolds-stress tensor and needs to be modelled.

Shear Stress Transport Model (SST) was introduced by Menter [29]. Whilst it utilizes Wilcox $k - \omega$ model [30] in the near wall region, standard $k - \epsilon$ model [31] is used outside of the boundary layer in order to benefit from its free stream independence feature. It is achieved by transforming $k - \epsilon$ model into $k - \omega$. The difference

between $k - \omega$ and the new one is that there is a additional cross-diffusion term in the ω –equation and uses different modelling constants.

Original $k - \omega$ model;

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho u_i k}{\partial x_i} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_{k1} \mu_t) \frac{\partial k}{\partial x_j} \right] \quad (3.13)$$

$$\frac{\partial \rho \omega}{\partial t} + \frac{\partial \rho u_i \omega}{\partial x_i} = \frac{\gamma_1}{\nu_t} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta_1 \rho \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_{\omega 1} \mu_t) \frac{\partial \omega}{\partial x_j} \right] \quad (3.14)$$

Transformed $k - \epsilon$ model;

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho u_i k}{\partial x_i} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_{k2} \mu_t) \frac{\partial k}{\partial x_j} \right] \quad (3.15)$$

$$\frac{\partial \rho \omega}{\partial t} + \frac{\partial \rho u_i \omega}{\partial x_i} = \frac{\gamma_2}{\nu_t} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta_2 \rho \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_{\omega 2} \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2\rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (3.16)$$

After original $k - \omega$ and transformed $k - \epsilon$ model equations are multiplied by a function of F_1 and $(1 - F_1)$ respectively, corresponding equations are added together and formed new model as,

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho u_i k}{\partial x_i} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right] \quad (3.17)$$

$$\frac{\partial \rho \omega}{\partial t} + \frac{\partial \rho u_i \omega}{\partial x_i} = \frac{\gamma}{\nu_t} \tau_{ij} \frac{\partial u_i}{\partial x_j} - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2\rho(1 - F_1) \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (3.18)$$

where k is the turbulent kinetic energy and ω is the specific turbulence dissipation.

Let ϕ_1 and ϕ_2 represent any constant in the original model $(\sigma_{k1}, \dots)$, the transformed $k - \epsilon$ model $(\sigma_{k2}, \dots)$, respectively. So ϕ represents corresponding constant of the new model $(\sigma_k, \dots)$ with a relation,

$$\phi = F_1 \phi_1 + (1 - F_1) \phi_2 \quad (3.19)$$

where constants for set 1 corresponds to ϕ_1 (Wilcox),

$$\sigma_{k1} = 0.85, \sigma_{\omega_1} = 0.65, \beta_1 = 0.0750 \quad (3.20)$$

$$\beta^* = 0.09, \kappa = 0.41, \gamma_1 = \beta_1 / \beta^* - \sigma_{\omega_1} \kappa^2 / \sqrt{\beta^*}$$

and set 2 corresponds to ϕ_2 (standard $k - \epsilon$),

$$\sigma_{k2} = 1, \sigma_{\omega_2} = 0.65, \beta_2 = 0.0750 \quad (3.21)$$

$$\beta^* = 0.09, \kappa = 0.41, \gamma_2 = \beta_2 / \beta^* - \sigma_{\omega_2} \kappa^2 / \sqrt{\beta^*}$$

The turbulent-viscosity definition supplement the model as,

$$\nu_t = \frac{a_1 k}{\max(a_1 \omega; \Omega F_2)} \quad (3.22)$$

where F_2 is the second blending function. The turbulent stress tensor, $\tau_{ij} = -\rho \overline{u'_i u'_j}$, is then given by,

$$\tau_{ij} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} \rho k \delta_{ij} \quad (3.23)$$

4. ONE DIMENSIONAL CAVITY SOLVER FORMULATION

In order to get pressure and swirl ratio distributions in the compressor well cavity, one-dimensional cavity solver tool is generated. The details of the tool have been discussed in Ref [32,33]. The tool is implemented to compressor well cavity shown below in Figure 4.1. The geometry which three cavities are forming the cavity is based on the work presented by Kong et al. [13]. The inlet is located on the right and the outlet on the left, which is typical for compressors as the static pressure increases when passing through the stator. In the test setup, the inlet is situated on the left, where there is no throughflow of the stator. Since these variations do not affect the results and the results are not compared with test data in this chapter, a realistic visual representation is provided here.

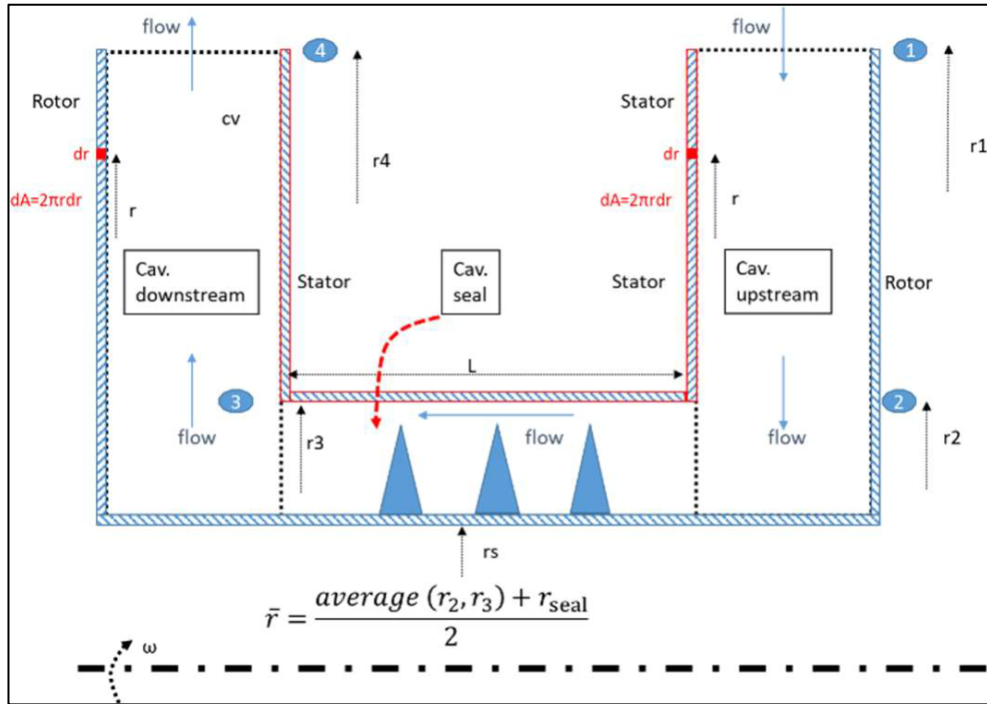


Figure 4.1: Simplified Compressor Well Cavity.

The inlet of the cavity is designated as point one. The end of the upstream cavity, which also serves as the inlet of the seal cavity, is designated as point two. The outlet of the

seal cavity, which also serves as the inlet of the downstream cavity, is designated as point three. Finally, the outlet of the cavity is designated as point four.

It is started with continuity and conservation of angular momentum equations. Continuity equation reveals,

$$\dot{m} = \dot{m}_1 = \dot{m}_2 = \dot{m}_3 = \dot{m}_4 \quad (4.1)$$

Non-dimensional flow rate with turbulent flow parameter is expressed below, where non-dimensional flow rate and rotational Reynolds number are calculated with static properties at the inlet of the upstream cavity.

$$C_w = \frac{\dot{m}}{\mu r_1} \quad (4.2)$$

$$\lambda_T = C_w Re_\theta^{-0.8} \quad (4.3)$$

$$Re_\theta = \frac{\rho \omega r_1^2}{\mu} \quad (4.4)$$

The angular momentum conservation for the upstream cavity is written below. At the nodes specified in the figure below, connections between cavities are constructed.

$$\dot{m}(r_2 V_{\theta 2} - r_1 V_{\theta 1}) = \Sigma \Gamma_{rotor} - \Sigma \Gamma_{stator} \quad (4.5)$$

Tangential velocity of the air is written with respect to shaft speed and local radius by inserting swirl ratio parameter $XK = V_{theta} / \omega r$, and writing torque expressions for the disc surfaces reveals,

$$\Gamma_{rotor} = \int_{r_2}^{r_1} r C_{f_{rotor}} 0.5 \rho (1 - XK_2)^2 \omega^2 r^2 2\pi r dr \quad (4.6)$$

$$\Gamma_{stator} = \int_{r_2}^{r_1} r C_{f_{stator}} 0.5 \rho XK_2^2 \omega^2 r^2 2\pi r dr \quad (4.7)$$

Then conservation of angular momentum equation is written as,

$$\dot{m}\omega(r_2^2 XK_2 - r_1^2 XK_1) = \int_{r_2}^{r_1} C_{f_{rotor}} \rho (1 - XK_2)^2 \omega^2 \pi r^4 dr - \int_{r_2}^{r_1} C_{f_{stator}} \rho XK_2^2 \omega^2 \pi r^4 dr \quad (4.8)$$

Which can be solved iteratively in order to find out swirl ratio XK . Exit swirl ratio is assumed that equal to cavity bulk swirl ratio ($XK = XK_2$). Skin friction coefficients can be written for rotor and stator surfaces. Also, these coefficients differ based on the surface type, cylinder-like or disk-like. Seal cavity is treated as cylinder-like surface located at an average radii for the rotor surface and at a radius of r_3 for the stator surface. The skin friction coefficients for rotor and stator surfaces are defined as Sultanian [16].

For rotating surfaces:

$$C_{f_{disk/cone}} = \frac{0.07}{Re_{\theta}^{0.2} (1 - XK)^{0.65}} \quad (4.9)$$

$$C_{f_{cylinder}} = \frac{0.042}{Re_{\theta}^{0.2} (1 - XK)^{0.65}} \quad (4.10)$$

For static surfaces:

$$C_{f_{disk/cone}} = \frac{0.105}{Re_{\theta}^{0.2} XK^{0.13}} \quad (4.11)$$

$$C_{f_{cylinder}} = \frac{0.042}{Re_{\theta}^{0.2} XK^{0.13}} \quad (4.12)$$

By using respective swirl ratio and representative radii for disk-like or cylindrical surfaces, rotational Reynolds number can be formulated. Static pressure drop can be determined subsequent to swirl ratio calculation by using the radial pressure equilibrium equation coupled with incompressible flow approach. This approach incorporates the average density between the two relevant radii, as detailed in [32].

In the context of a stator well cavity-disk cavity system with upstream, downstream, and seal cavities, the previously derived equations (Eq. 4.8-4.12) can be algebraically summed to yield a new set of equations. These new equations written below directly determine the bulk swirl ratio for each individual cavity, namely XK_2, XK_3, XK_4 .

$$\frac{(\bar{r}^2 XK_2 - r_1^2 XK_1)}{0.07(1 - XK_2)^{1.35} - 0.105XK_2^{1.87}} = \frac{\pi(r_1^5 - \bar{r}^5)}{5\lambda_T r_1^3} \quad (4.13)$$

$$\frac{(XK_3 - XK_2)}{0.042(1 - XK_3)^{1.35} - 0.063XK_3^{1.87}} = \frac{\pi L}{\lambda_T \left(\frac{r_1}{\bar{r}}\right)^{2.6} \bar{r}} \quad (4.14)$$

$$\frac{(r_4^2 XK_4 - \bar{r}^2 XK_3)}{0.07(1 - XK_4)^{1.35} - 0.105XK_4^{1.87}} = \frac{\pi(r_4^5 - \bar{r}^5)}{5\lambda_T \left(\frac{r_1}{r_4}\right)^{2.6} r_4^3} \quad (4.15)$$

Where,

$$\bar{r} = \frac{\text{average}(r_2, r_3) + r_{\text{seal}}}{2} \quad (4.16)$$

The calculation of rotor torque from the seal surface leverages the mean radius as a representative location. To facilitate this calculation, the density and thermophysical properties of air within the stator well are assumed to be constant. This simplification is justified by the low tangential Mach numbers encountered in the system.

Four different inlet swirl ratios (0.05, 0.15, 0.3, and 0.5) are investigated. For each inlet swirl ratio, the inlet turbulent flow parameters are varied between 0.01 and 0.3. Thus, the effects of both the inlet swirl ratio and the inlet turbulent flow parameter are studied.

Figure 4.2 illustrates the distribution of the bulk cavity swirl ratios for an inlet swirl ratio of 0.05 at varying λ_T values. For a fixed geometry and shaft speed, it is a clear fact that swirl ratio decreases as λ_T increases. The selected inlet swirl ratio (0.05) corresponds to a high power operating condition. This condition falls slightly above the design point and resides on the choking side of the compressor's characteristic curve. In such a scenario, negative incidences can occur on both the rotor blades and stator vanes. The observed reduction in cavity swirl with increasing λ_T contradicts expectations based on previous findings in [32]. This discrepancy can be attributed to the specific geometry of the upstream cavity. Because the upstream cavity is shallow and situated at a large radius, the torques acting on the cavity flow are considerably

stronger than the angular momentum introduced by the low-swirl leakage flow injected into the cavity. Consequently, within the upstream cavity, the rotor torque, proportional to $(1 - XK)^2$, dominates the stator torque, proportional to XK^2 . This dominance dictates the distribution of the swirl ratio. In the seal cavity, the larger rotor surface area compared to the stator surface leads to a slight increase in swirl. However, real-world applications often incorporate honeycombs, which would enhance the stator torque and mitigate the rise in swirl along the seal cavity. The downstream cavity exhibits a similar behavior, with dominant rotor torque leading to an increased swirl ratio. Even with the highest λ_T value of 0.3, representing a significant introduction of angular momentum at the inlet, overcoming the dominant rotor torque remains a considerable challenge. As the parameter λ_T approaches infinity, the cavity swirl ratio would theoretically reach a free vortex state, with a value around 0.06. However, this scenario is not attainable in real-world compressor operation, even if a seal failure were to occur.

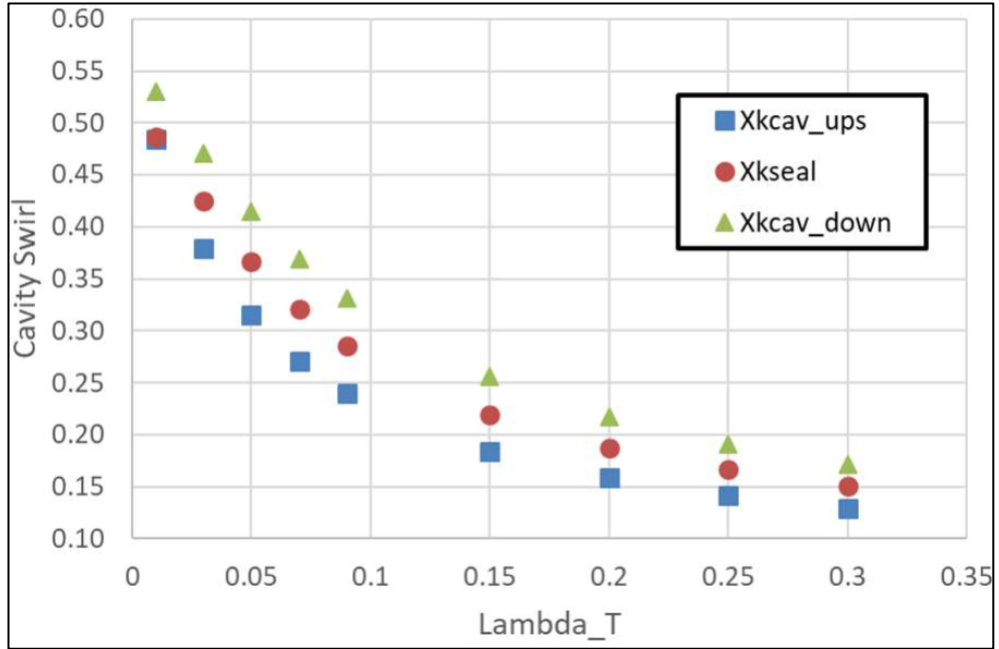


Figure 4.2: Cavity Swirl Distributions Based on a Simplified Bulk Model with $XK_{inlet} = 0.05$.

Figures 4.3-4.5 illustrate the distributions of bulk cavity swirl ratios for various inlet swirl ratios (0.15, 0.3, and 0.5). The case of the typical design value (0.15), presented in Figure 4.3, exhibits trends similar to the previous case. However, a notable difference arises for moderate to high λ_T values. Here, the downstream cavity swirl

and seal cavity swirl begin to converge towards each other. Considering the high radii and shallow size of the stator well cavities, ideal free vortex swirl ratio in the upstream cavity is slightly higher compared to the inlet value.

In Figure 4.4 the bulk cavity swirl ratio distributions for an inlet swirl ratio of 0.3 shows that from low to moderate λ_T all the swirl ratios overlap whereas downstream cavity swirl ratio starts to lagging behind for the high λ_T values. This phenomenon can be explained by that dominance of rotor torques on the angular momentum of the swirled flow decreases, which can be seen at low speed off-design condition of a compressor, especially for front stages.

As mentioned in Denecke [34], swirl ratio downstream of the seal with infinitely many teeth converges to a value around 0.55 (stator to rotor area ratio of 0.7). For seals with a limited number of teeth, the change of the swirl ratio depends on the incoming swirl ratio, where swirl ratio is reduced by seal if the swirl ratio of the incoming flow has high value; and vice versa. In this case, the incoming flow has a swirl ratio of about 0.4. Therefore, the seal counteracts this slightly, and the final swirl ratio reaches a value just above 0.4 for high λ_T values.

Figure 4.5 illustrates the behavior for an inlet swirl ratio of 0.5. It is important to note that this value is uncommon in real-world compressor operation and typically only could be encountered in situations with substantial flow disruptions downstream of the stator, such as flow breakdown. The observed trend in swirl ratio aligns with what we expect for a standard rotor-stator cavity that receives radial inflow: as λ_T increases, the upstream cavity swirl ratio also increases. However, the labyrinth seal counteracts this increase, reducing the swirl because the incoming flow has a value exceeding 0.55 (a point where labyrinth seals tend to stabilize swirl). Interestingly, as the amount of leakage flow increases, this dampening effect by the seal weakens. This is likely a consequence of the high angular momentum carried by the leakage flow. For this specific inlet swirl ratio and with such a significant momentum injection, the swirl ratio distributions for all cavities eventually reach a plateau, signifying a constant value at high λ_T values.

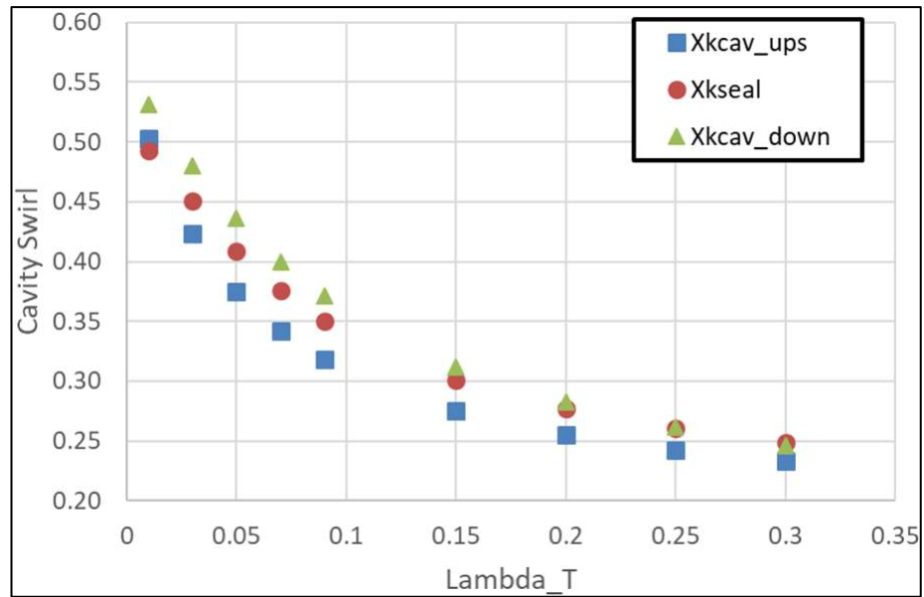


Figure 4.3: Cavity Swirl Distributions Based on a Simplified Bulk Model with $XK_{inlet} = 0.15$.

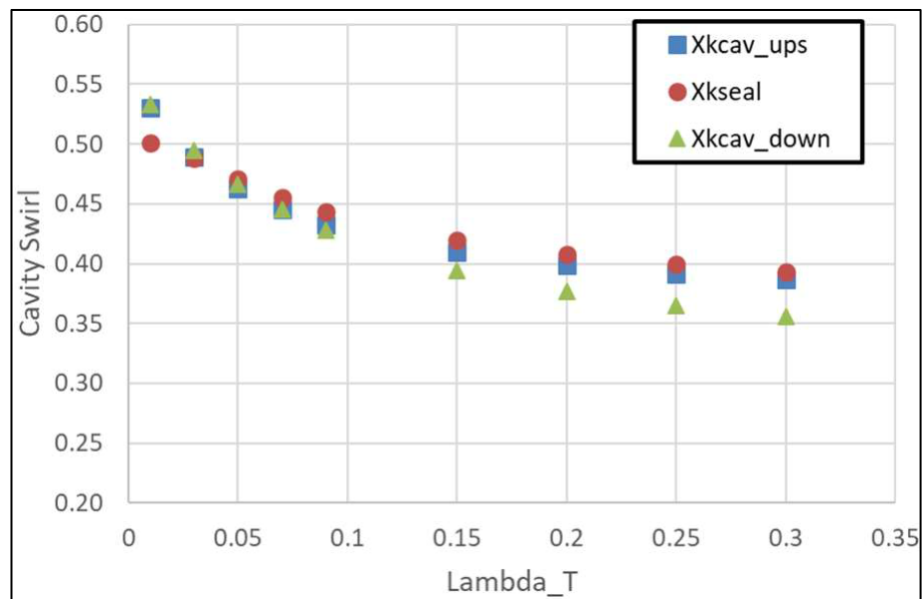


Figure 4.4: Cavity Swirl Distributions Based on a Simplified Bulk Model with $XK_{inlet} = 0.3$.

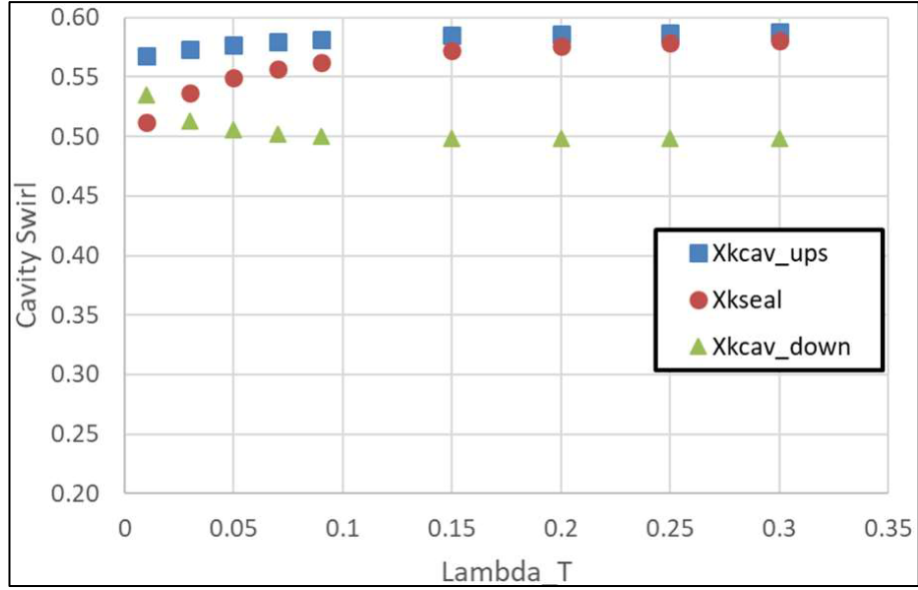


Figure 4.5: Cavity Swirl Distributions Based on a Simplified Bulk Model with $XK_{inlet} = 0.5$.

This simplified one-dimensional model offers valuable insights into the trends and behavior within the cavities. Notably, for a realistic inlet swirl ratio of 0.15, the results demonstrate that the upstream cavity, with its lowest swirl distribution, contributes most significantly to the rotor torque. With restriction of the seal increases it has been seen that the effect of rotor torque increases dramatically to some point and then starts to level down, so is windage power. Finally, it is understood that both aspects of compressor aerodynamic performance and windage power loss, the flow rate injected into the cavity should be kept low, which is counterintuitive finding from familiar rotor-stator cavity flows.

The windage temperature rise within each cavity is directly related to the rotor torque acting on the leakage flow in that cavity. This relationship can be understood by equating the product of rotor torque and rotational speed to the stagnation enthalpy rise (a measure of temperature increase due to work done on the fluid). Consequently, the cavity experiencing the highest rotor torque will exhibit the most significant windage temperature rise.

$$\dot{m}(H_2 - H_1) = \sum \Gamma_{rotor} \omega \quad (4.17)$$

$$\theta = \frac{\Delta T_T}{\frac{\omega^2 r^2}{2c_p}} \quad (4.18)$$

5. VALIDATION TEST CASE

Test rig data from Kong et al. [13] is used in order to validate numerical method, which will be used to investigate the generic compressor stator well cavity flowfield.

5.1. Test Facility and Rig Instrumentation

The test rig includes upstream and downstream cavities that are connected with labyrinth seal in the middle and the size of the cavity is presented in Figure 5.1. The inlet of the test rig is at the left and outlet is at the right.

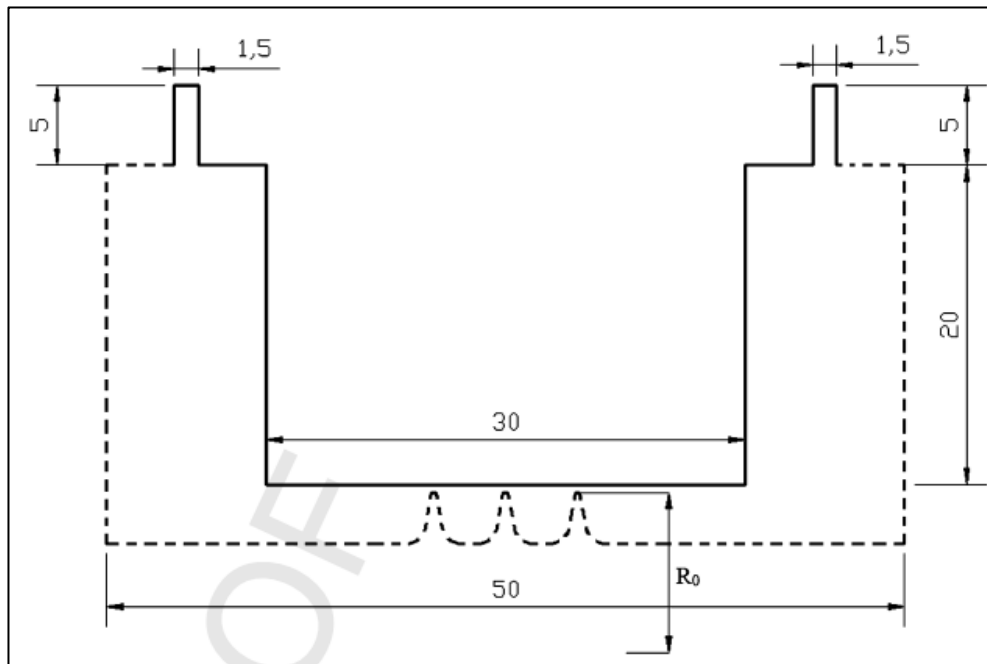


Figure 5.1: The Experimental Model [13].

Moreover, the test rig is shown in Figure 5.2. Kong et al. [13] set up this test rig, designing the rotor disc with left and right discs for easy installation and replacement of labyrinth rings with different geometries. Five labyrinth rings, each with the same tooth shape but different initial rotor tip radii, were tested to evaluate the influence of tip clearance on inter-stage leakage flow and the effects of rotational speed. The discs and labyrinth rings were tightly assembled with seals to prevent leakage. Airflow entered the supply chamber, passed through axial orifices into the central test section, and exited through the exhaust cavity.

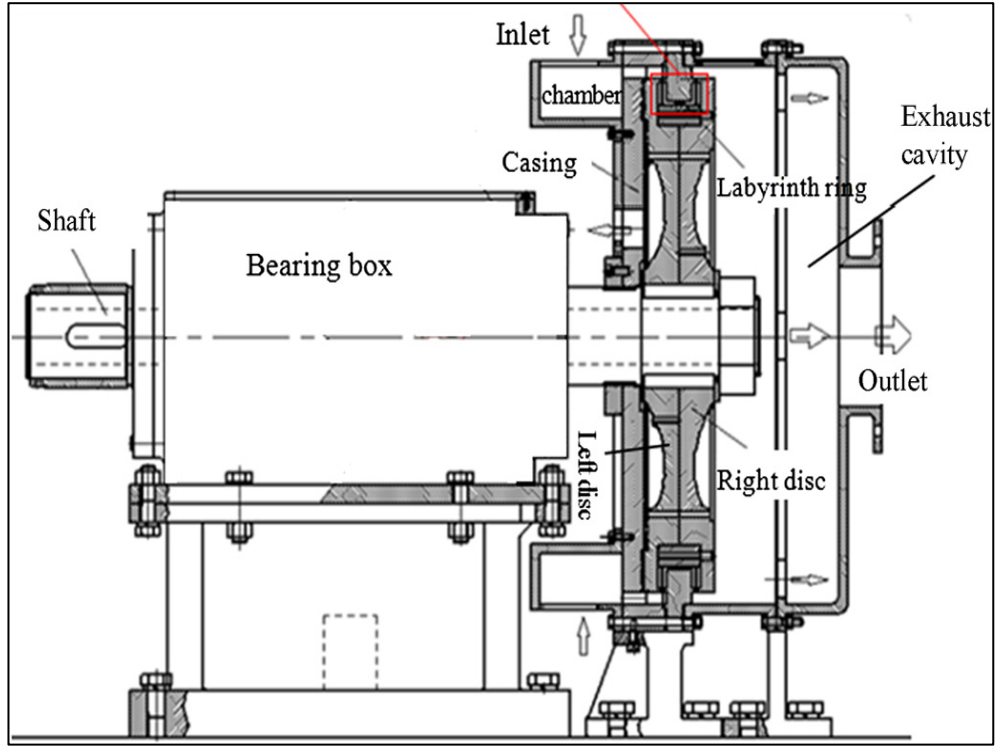


Figure 5.2: The schematic of the test rig [13].

Kong et al. [13] measured, as illustrated in Figure 5.3, static pressure and total temperature at four distinct locations within the test model with an uncertainties of $\pm 0.05\%$ and $\pm 1\text{K}$, respectively. At location 1, due to the negligible inlet velocities, the static pressure was assumed to be equal to the total pressure for pressure ratio calculations throughout the experiments. At location 5, swirl velocity was determined. Here, the static pressure was measured at the wall, while the total pressure was measured at the center. Based on the assumption of constant static pressure at a given radial position and negligible radial velocity, the tangential velocity component was subsequently calculated.

Also they measured the tip clearance of the labyrinth seal with two laser distance sensors with an uncertainty of $\pm 0.005\text{ mm}$. One sensor measured the radial displacement of the labyrinth ring, while the other measured the radial displacement of the stator casing. The flowrate was measured with four high-precision orifice meters, characterized by a maximum uncertainty of $\pm 1\%$.

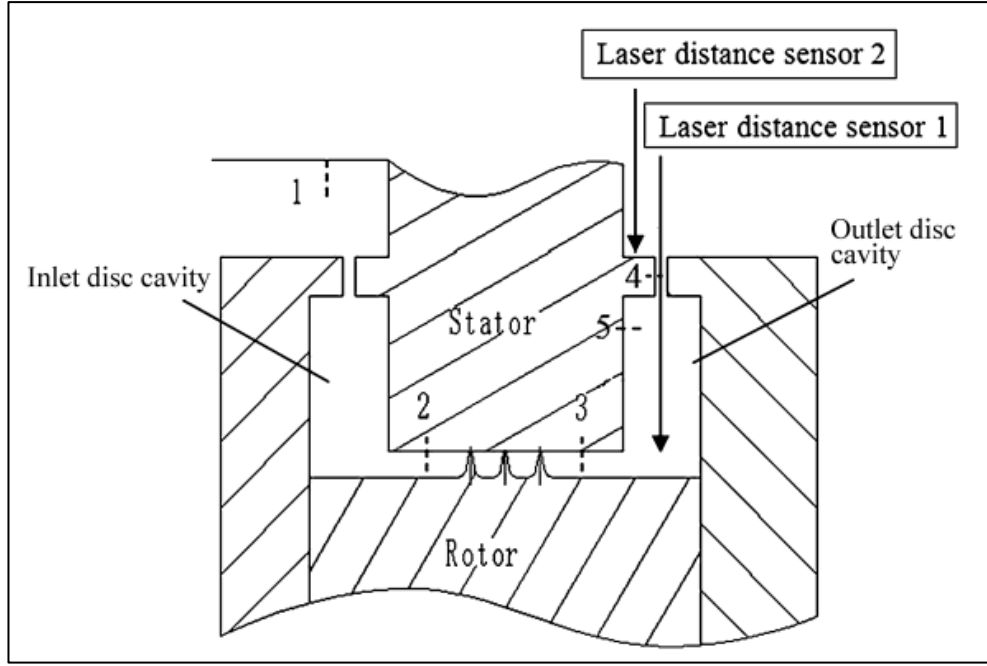


Figure 5.3: The Locations of Measurements [13].

5.2. The Numerical Simulation Setup, Grid Generation and BC's

In this study, the 2D axisymmetric RANS equations with swirl are solved using the pressure-based double precision solver in Ansys Fluent 2022 R1. The pressure-based coupled algorithm addresses the combined system of momentum equations and the pressure-based continuity equation, while the other equations are solved separately as in the segregated algorithm. A second-order upwind scheme is applied to all variables except pressure, for which the PRESTO! scheme is used. The computational domain is defined between adiabatic rotating and stationary walls with a no-slip shear condition. Total pressure and total temperature are specified at the inlet, while static pressure is specified at the outlet. Xk values at the location of 5, at the inlet and exit of the seal region, and the total temperature at the outlet was monitored in to ensure simulation convergence. A numerical setup is given in Table 5.1.

Table 5.1: CFD model setup.

Coordinate System	2D Axisymmetric swirl
Fluid - Air	Density : Ideal gas Specific Heat (Cp) : Piecewise-polynomial Viscosity : Sutherland
Reference Pressure	0 Pa
Turbulence Model	k- ω SST
Discretization Schemes	Pressure – Velocity Coupling : Coupled Gradient : Least-Squares Cell Based Pressure: PRESTO! 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
Wall BC's	Rotating and stationary adiabatic walls with no slip shear condition
Inlet BC's	Total Pressure and Total Temperature
Exit BC's	Static Pressure
Monitoring	Xk value at a radius of 265 mm on downstream cavity Xk values at inlet and exit seal region Exit total temperature

Boundary conditions for the numerical simulation are detailed in Table 5.2, while the calculated parameters are presented in Table 5.3.

During the experiments, rotational speed were varied from 5100 rpm to 8100 rpm, pressure ratio range is 1.1 and 1.3 based on inlet to outlet conditions.

For the validation of the numerical model, two clearance levels are selected, 0.75 mm for a pressure ratio of 1.1 and 0.85 mm for a pressure ratio of 1.3. Inlet swirl ratio is zero due to very low velocity at the inlet as mentioned before. These parameters along with $\dot{m}$, Re_θ , C_w and λ_T are tabulated in Table 4.1. Re_θ ranges from 3 to 6 million with turbulent λ_T conditions ranging from 0.1 to 0.3.

An examination of Table 5.3 reveals a decrease in mass flow rate with increasing rotational speed at constant pressure ratio and clearance value. This observation supports the hypothesis that increasing the swirl ratio at the labyrinth seal inlet (which reduces static pressure) can lead to a decrease in leakage mass flow rate. Yet, this effect is less pronounced for PR=1.3 cases due to higher-pressure ratio and higher λ_T values, which can be also understood from Figure 4.2.

Table 5.2: Boundary Conditions for CFD Simulations.

<u>Case #</u>	<u>1</u>	<u>2</u>	<u>3</u>	<u>4</u>	<u>5</u>
N (rpm)	5100	6000	6900	7500	8100
PR	1.1				
$cl.$ (mm)	0.75				
$P_{t_{inlet}}$ (kPa)	132				
$T_{t_{inlet}}$ (K)	300				
$P_{s_{outlet}}$ (kPa)	120				
<u>Case #</u>	<u>6</u>	<u>7</u>	<u>8</u>	<u>9</u>	<u>10</u>
N (rpm)	5100	6000	6900	7500	8100
PR	1.3				
$cl.$ (mm)	0.85				
$P_{t_{inlet}}$ (kPa)	156				
$T_{t_{inlet}}$ (K)	300				
$P_{s_{outlet}}$ (kPa)	120				

Table 5.3: Calculated Parameters

<u>Case #</u>	<u>1</u>	<u>2</u>	<u>3</u>	<u>4</u>	<u>5</u>
$\dot{m}$ (kg/s)	0.138	0.135	0.131	0.131	0.126
Re_{θ}	3.19E+06	3.75E+06	4.32E+06	4.69E+06	5.07E+06
C_w	2.74E+04	2.67E+04	2.60E+04	2.59E+04	2.50E+04
λ_T	0.171	0.147	0.128	0.119	0.108
<u>Case #</u>	<u>6</u>	<u>7</u>	<u>8</u>	<u>9</u>	<u>10</u>
$\dot{m}$ (kg/s)	0.272	0.268	0.261	0.26	0.255
Re_{θ}	3.77E+06	4.44E+06	5.10E+06	5.55E+06	5.99E+06
C_w	5.39E+04	5.31E+04	5.17E+04	5.15E+04	5.05E+04
λ_T	0.295	0.255	0.222	0.207	0.191

Figure 5.4 illustrates the domain geometry. The computational domain is discretized using quadrilateral elements focusing on the regions with high gradients. Also, quality criterias such as aspect ratio of less than 50, skewness of less than 0.6, orthogonality of higher than 0.6 and grow ratio of 1.2 are satisfied. While adhering to the previously mentioned quality criteria, the mesh generation process creates multiple computational grids that further optimize for $y^+ = yu_{\tau}/\nu \sim 1$.

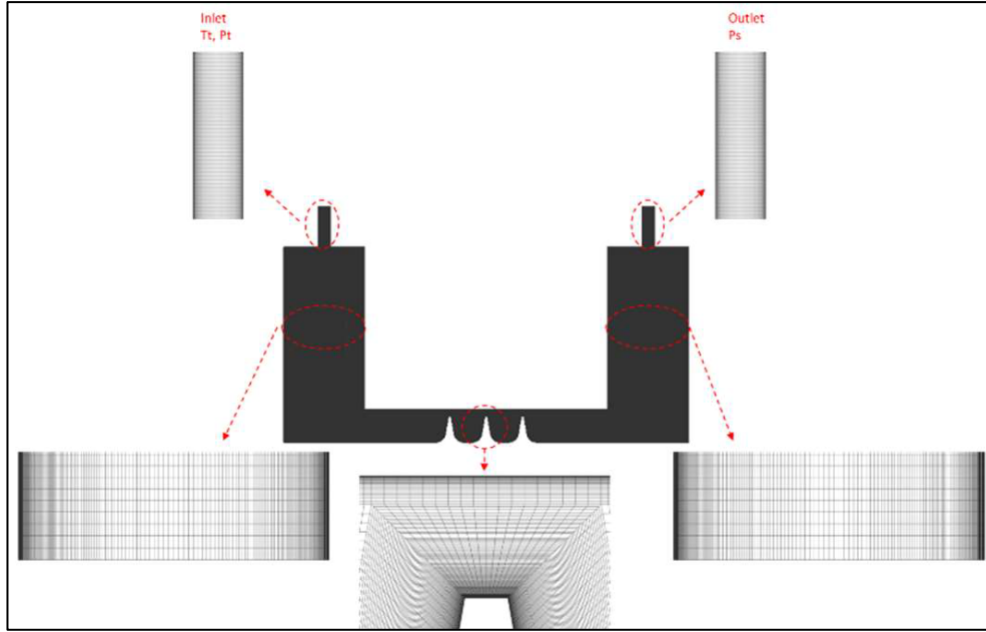


Figure 5.4: Computational Grid for Test Case Geometry.

Case 10, as presented in the Table 5.3, is selected for mesh independency study due to it is the case where the Reynolds number and pressure ratio is highest. To assess mesh independence, a series of two-dimensional axisymmetric swirl simulations were performed using varying mesh densities. The mesh levels ranged from 175,000 to 765,000 quadrilateral elements. Key parameters including swirl ratio at a radius of 265 mm (location 5 in the Figure 5.3), mass flow rate, labyrinth seal discharge coefficient (C_d), and windage number were monitored and compared across different mesh densities as shown in Figure 5.5. Mesh level 4, consisting of approximately 520,000 elements, demonstrated sufficient resolution to capture the bulk flow dynamics within the cavity. Consequently, this mesh level was chosen for further investigations.

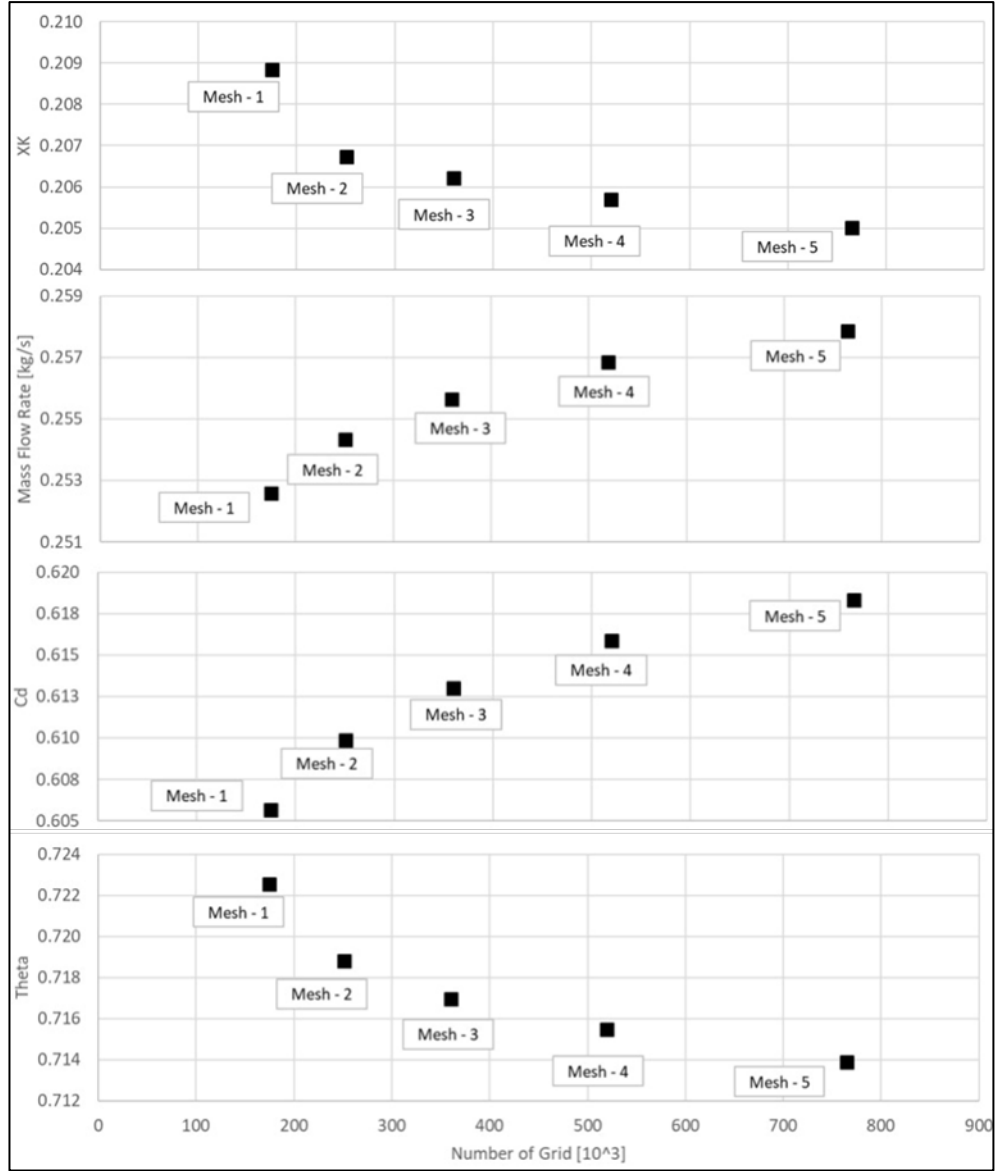


Figure 5.5: Computational Grid Dependency Study with, $Re_\theta = 5.99E + 6$ and $\lambda_T = 0.191$, for Case 10, in terms of Exit Swirl Ratio, Mass Flow Rate, Discharge Coefficient and Windage Number.

5.3. The Results of Numerical Simulations

Numerical results against experimental data are presented in Figure 5.6 and Figure 5.7 showing the swirl ratio value at a radius of 263 mm at the downstream cavity, mass flow rate and exit temperature for both pressure ratios of 1.1 and 1.3. Additionally, results obtained using a commercial software, Flow Simulator©, are incorporated for reference. As mentioned in Chapter 4, Flow Simulator© employs a similar approach to the in-house cavity solver. This approach involves dividing the flow domain into

sub-cavities and utilizes the angular momentum conservation equation along with correlations for the skin friction coefficient within each sub-cavity.

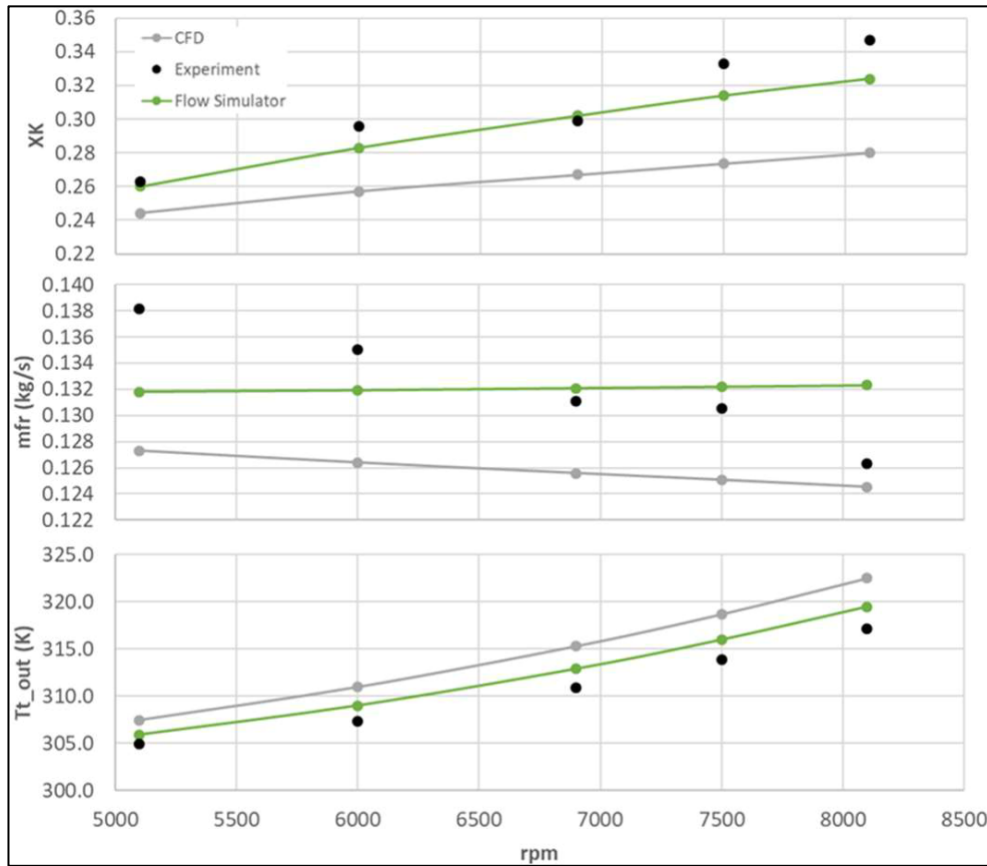


Figure 5.6: Swirl Ratio at R=263 mm, Mass Flow Rate and Exit Temperature for PR=1.1.

The CFD results exhibit good agreement with the experimental data for both rotational speeds, demonstrating the suitability of the chosen numerical method for practical applications. The predicted trends in mass flow rate with increasing rotational speed align well with the experimental observations. However, Flow Simulator© shows a nearly constant value for the mass flow rate in the PR=1.1 case. This suggests an unrealistic scenario where the upstream pressure of the seal remains constant despite increasing rotation. Apart from this discrepancy, the trends and values obtained using CFD are consistent with the experimental data.

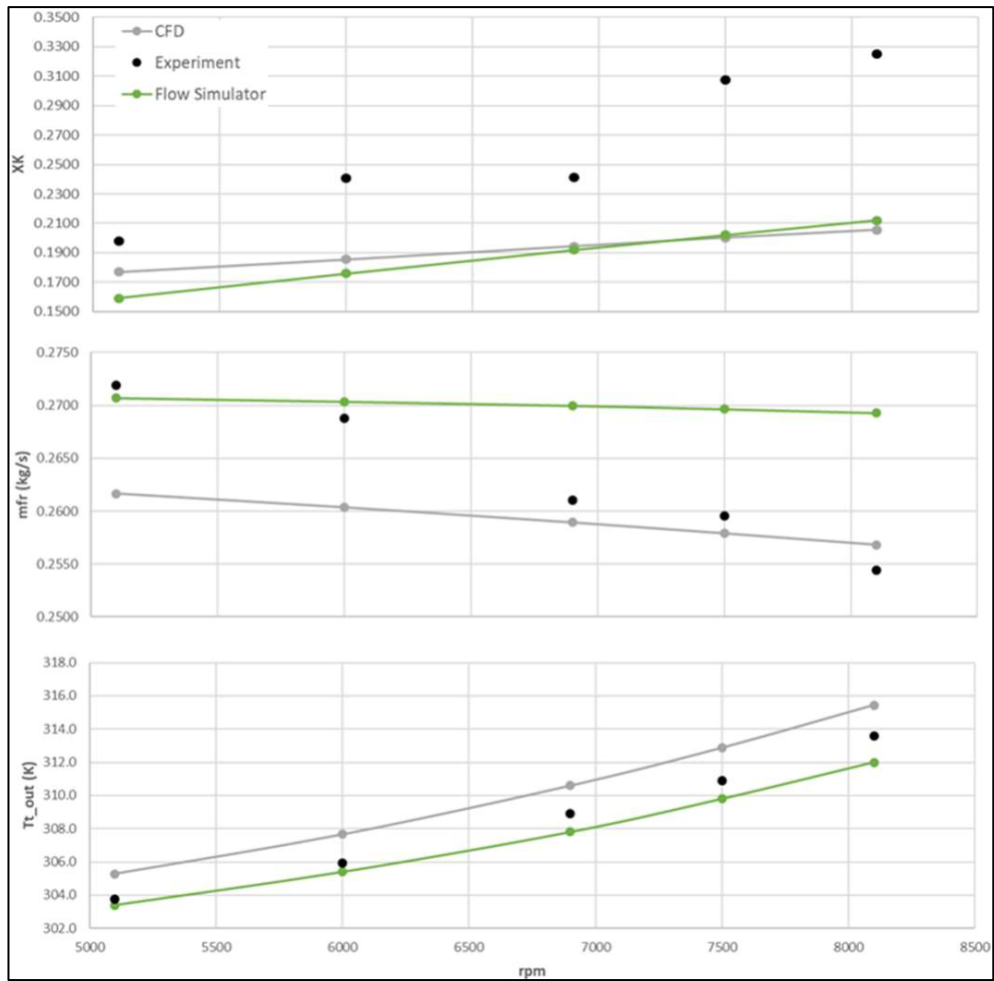


Figure 5.7: Swirl Ratio at R=263mm, Mass Flow Rate and Exit Temperature for PR=1.3.

In contrast to the PR=1.1 case, both CFD and Flow Simulator© capture the correct trend for the PR=1.3 case. However, both methods underpredict the magnitude of the variation. Additionally, the predicted swirl ratio exhibits an underestimation after the labyrinth seal. Encouragingly, the overall windage temperature rise is captured well for both pressure ratios within a small margin. This discrepancy between predicted and measured windage temperature rise can likely be attributed to heat transfer from the test rig to the surrounding environment. This heat transfer would result in a lower measured outlet flow temperature compared to the adiabatic conditions assumed in the simulations. While the agreement between Flow Simulator© results and experimental data for windage temperature may seem unrealistic, it can potentially be explained by the difference in predicted flow rates between the simulations and the experiment.

These figures consistently reveal several key trends. Firstly, as expected, increasing rotational speed leads to both a rise in cavity swirl values and an increase in windage

temperature. Secondly, the static pressure drop observed in the upstream cavity is directly influenced by the swirl ratio, which in turn determines the leakage flow through the seal. Finally, with decreasing λ_T due to higher rotational speeds, a clear increase in upstream cavity swirl is observed. This enhanced swirl results in a pressure reduction and consequently, lower flow rates.

The observed discrepancies in flow rate between experimental data and numerical simulations can potentially be attributed to two key factors: differences in upstream cavity swirl predictions and discharge coefficient calculations. Figure 4.2 (which depicts a very low inlet swirl value) showcases a clear trend across the cavities. The downstream cavity consistently exhibits a higher swirl ratio compared to the upstream cavity. Since the CFD simulations predict lower overall swirl ratio values compared to the experiments, it is likely that the upstream cavity swirl is also underestimated. While the discharge coefficient values for Case 10 are similar in both the tests (around 0.61) and the CFD results, their numerical prediction might still contribute to the flow rate differences. It is well-documented that limitations exist in applying Reynolds-Averaged Navier-Stokes (RANS) simulations for predicting torque, which is the integral of tangential shear stress acting on rotor and stator walls. Consequently, CFD may overpredict the swirl ratio in the downstream cavity while underpredicting it in the upstream cavity. When comparing different pressure ratios (PR), cases with PR=1.3 have higher λ_T values. This signifies that angular momentum plays a relatively more significant role compared to PR=1.1 cases. As a result, these cases exhibit less variation in mass flow rate with increasing rotational speed (as evident from the smaller variation in swirl ratio observed for higher λ_T values in Figure 4.2).

To investigate this behavior in more detail, Figures 5.8 and 5.9 present swirl ratio contours for Cases 1 & 5 (PR=1.1) and Cases 6 & 10 (PR=1.3). For both PR=1.1 and PR=1.3 cases, flow enter the cavity at the left. It moves radially inward on the stator surface while some portion of the air moves to the rotor surface, generating circulating zone. At the seal pockets swirl ratios increases, and after the seal cavity flow moves radially outward in the rotor surface while some portion of the air moves to the stator surface, generating another circulating zone.

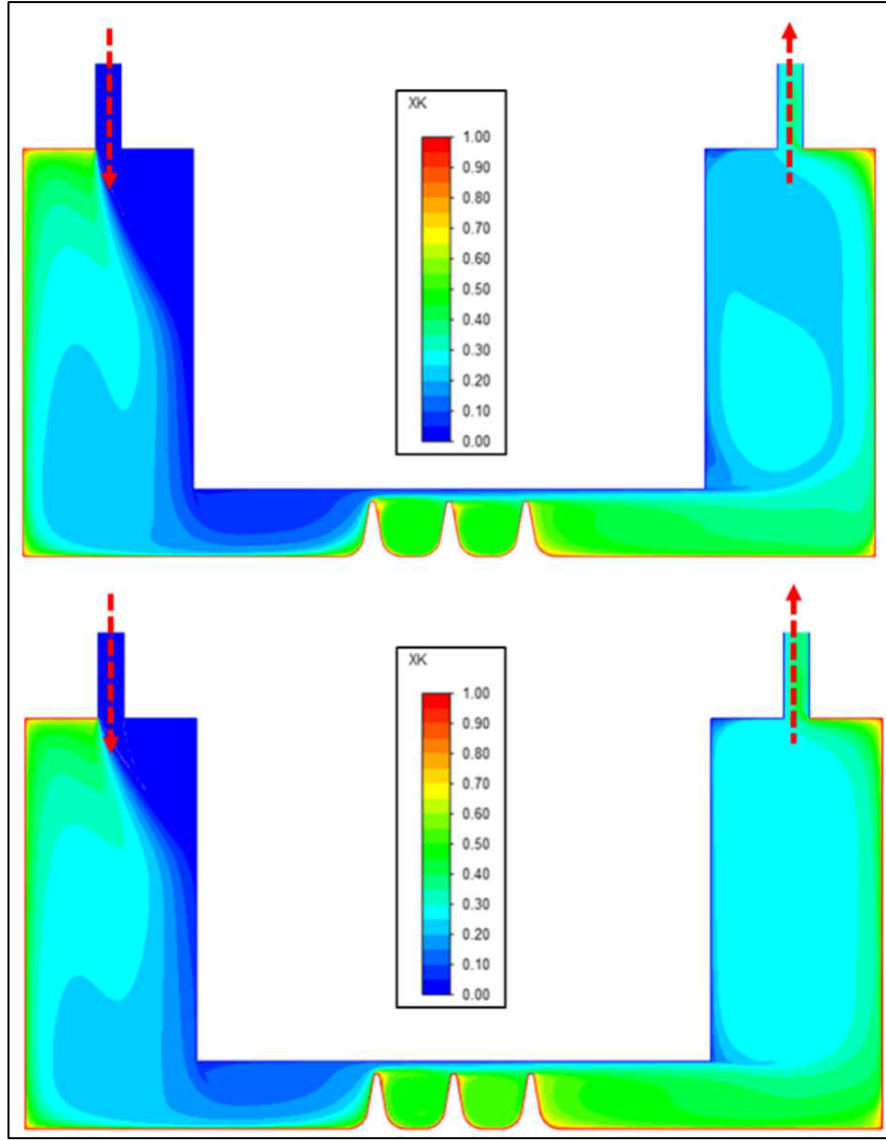


Figure 5.8: Swirl Ratio Contours for PR=1.1, 5100 rpm (top) and 8100 rpm (bottom).

These cases were chosen due to the significant difference in their rotational speeds. While significant similarities exist in the flow fields for both pressure ratios, increasing rotational speed (also means reducing λ_T) leads to elevated swirl levels in both the upstream (left) and downstream (right) cavities.

This phenomenon can be attributed to the well-established fact that rotor torque is proportional to $\omega^2 r^5 (1 - XK)^{1.35} Re_\theta^{-0.2}$. Notably, the dependence on rotational speed becomes $\omega^{1.8}$ within this range. Consequently, the change in swirl ratio between 5100 rpm and 8100 rpm is less dramatic than the speed increase itself. This translates to a significant amplification of rotor torque, ultimately leading to intensified swirl distributions within both cavities.

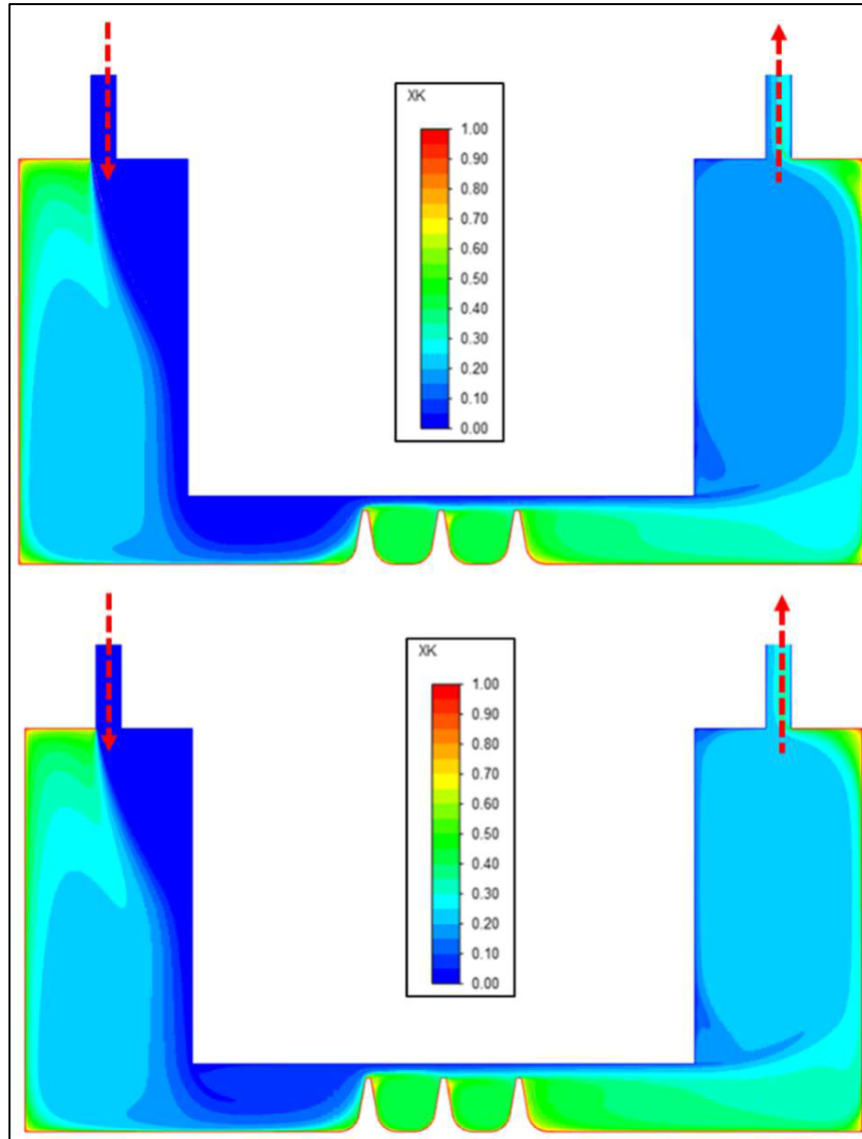


Figure 5.9: Swirl Ratio Contours for PR=1.3, 5100 rpm (top) and 8100 rpm (bottom).

With identical upstream and downstream boundary conditions for both PR=1.1 and PR=1.3 cases, increasing speed is expected to reduce static pressure at both locations upstream and downstream of the labyrinth seal. However, the reduction will likely be more pronounced upstream due to the minimal density decrease across the seal.

6. GENERIC STATOR WELL CAVITY

In the current study, a generic stator well cavity with axisymmetric geometry in two dimensions is investigated.

6.1. The Numerical Simulation Setup, Grid Generation and BC's

The setup for the numerical simulation and the convergence criteria are consistent with those defined in Chapter 5.2. The only additional parameter is the definition of the swirl ratio at the inlet, which reflects the actual downstream conditions of the stator.

As shown in the Figure 6.1, the computational domain is formed by an upstream (on the right) and downstream (on the left) cavity that are connected with a three teeth labyrinth seal, as encountered in real engine configurations. All the walls are rotating except stator walls. In Figure 6.1, implemented boundary conditions are shown. At the inlet static pressure and total temperature are specified with XX_{inlet} value. At the outlet static pressure is applied as a boundary condition. In terms of turbulence boundary conditions, intensity values with hydraulic diameters are designated at the inlet and the outlet.

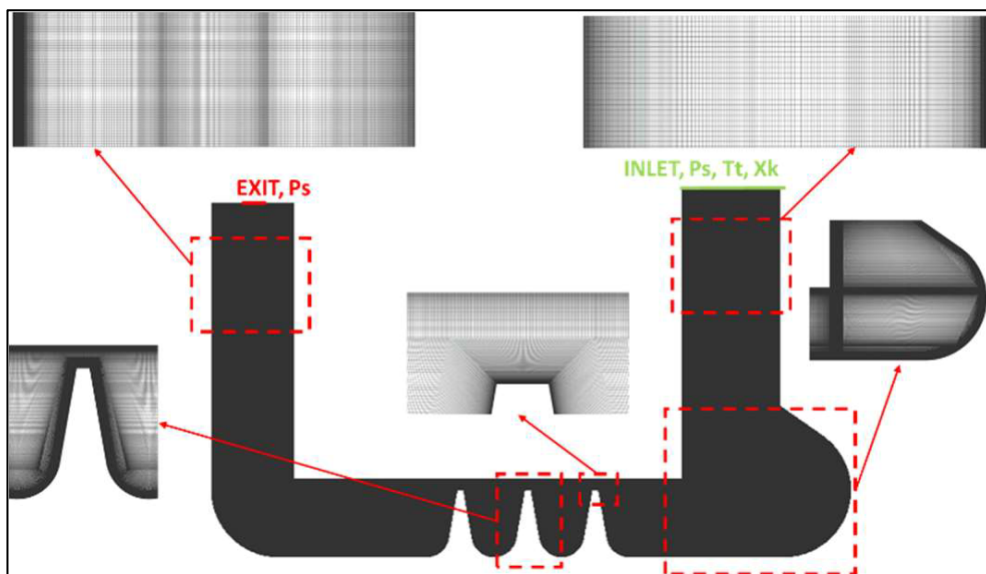


Figure 6.1: Computational Domain and Boundary Conditions.

Fully structured regular quadratic elements are used to discretize fluid domain. A non-uniform grid spacing is employed in order to ensure that values of y^+ for the grid points nearest the wall are less than one. Additionally, fine mesh at core region for the labyrinth seal is used to resolve high-velocity flow and ensure realistic prediction of the exit swirl ratio. For all regions, the mesh structure is set to gradually coarsen from wall to cavity core while maintaining acceptable mesh quality criteria. Specifically, the aspect ratio and skewness are limited to a maximum of 50 and 0.6, respectively, as shown in Figure 6.1.

Boundary conditions that are used in CFD simulations are tabulated in Table 6.1 and the calculated parameters are presented in Table 6.2. $\%N$ denotes the relative high-pressure compressor rotor speed compared to the nominal condition, while $\%c$ represents the relative seal clearance compared to the nominal condition. Rotational Reynolds number is ranged between 4 and 10.7 million based on inlet condition. Values of λ_T are varied between 0.09 and 0.18 based on inlet flow conditions. Finally XK_{inlet} is around 0.11 for all cases. The investigation encompasses three distinct operating regimes: low thrust, cruise, and high power/thrust. Case 1 represents the low thrust scenario, Case 2 corresponds to typical cruise conditions, and Case 3 simulates high power/thrust operation. Cases 4 and 5 investigate impact of the clearance on the stator well cavity dynamics under high power/thrust condition.

Table 6.1: Boundary Conditions for CFD Simulations.

	<u>Case 1</u>	<u>Case 2</u>	<u>Case 3</u>	<u>Case 4</u>	<u>Case5</u>
$\%N$	74.5	92.6	100	100	100
$\%cl$	201	128	100	128	68
XK_{inlet}	0.115	0.105	0.116	0.116	0.116

Table 6.2: The Calculated Parameters.

	<u>Case 1</u>	<u>Case 2</u>	<u>Case 3</u>	<u>Case 4</u>	<u>Case5</u>
Re_θ	4.04E+6	8.87E+6	10.7E+6	10.7E+6	10.7E+6
C_w	34870	57110	56940	71510	37800
λ_T	0.181	0.158	0.135	0.169	0.090
XK_{inlet}	0.115	0.105	0.116	0.116	0.116

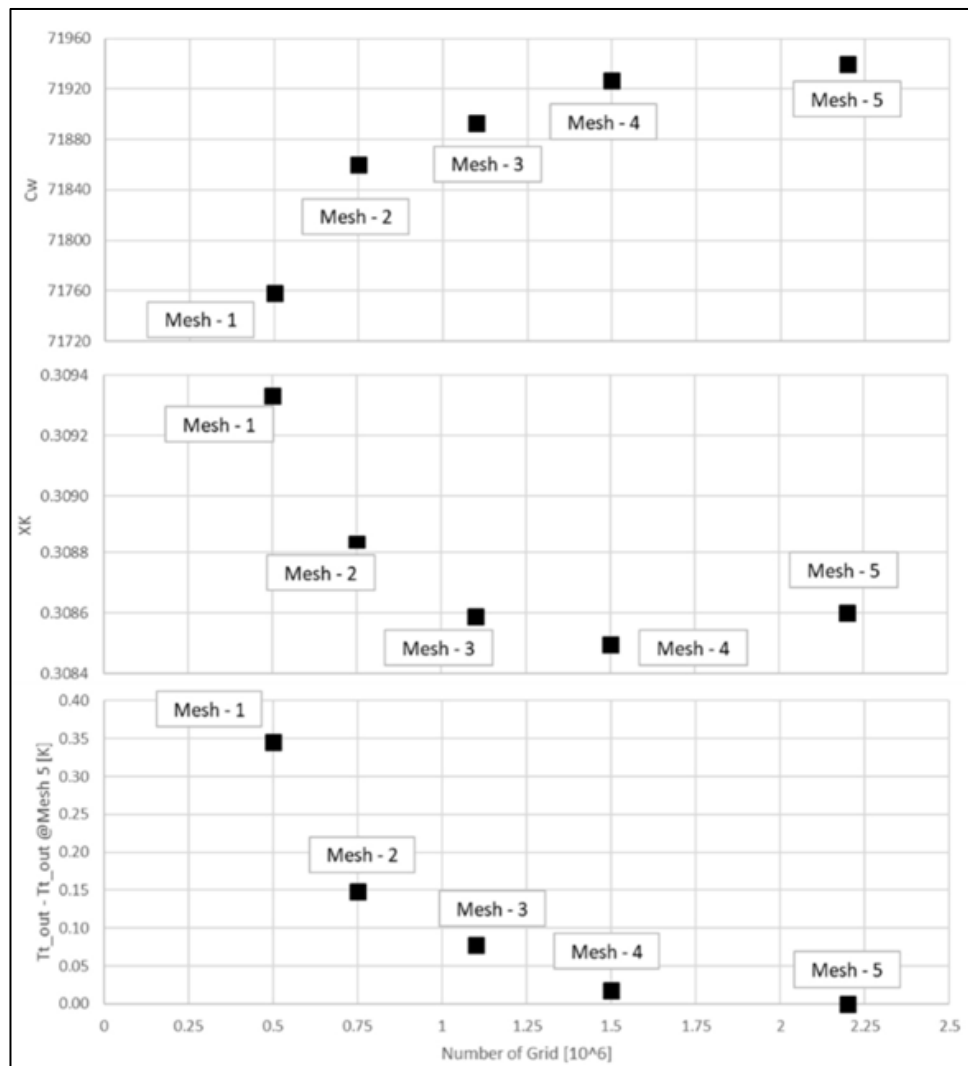


Figure 6.2: Computational Grid Dependency Study for Case 3 Based on C_w , XK at $x = 0.96$ and Total Temperature Rise.

As conducted for validation test case, mesh independency studies are conducted for generic stator well cavity. Case 3, as presented in Table 6.1, is selected since it is the case where rotational Reynolds number and pressure ratio is highest. Five different mesh levels are investigated, and non-dimensional mass flow rate, C_w , swirl ratio at $x = 0.96$ and total temperature rise throughout the cavity are shown with respect to mesh levels in Figure 6.2. Total temperature rise throughout the cavity is normalized by the difference from the results obtained with the highest grid level (around 2.2M). Mesh level 4, about 1.5M number of elements, is found to be adequate for a mesh independent two-dimensional axisymmetric swirl solutions.

6.2. The Results of Numerical Simulations

Swirl ratio, stream function contours are shown in Figure 6.3-6.6 for the conditions tabulated at Table 6.1. First three cases are plotted together in order to see the effect of running condition whereas Cases 3-5 are shown separated in order to investigate seal clearance effect on flow structures inside the stator well cavity at the high power/thrust setting. Behavior of swirl ratio and stream function are in line as expected, and flow structures inside the cavity are clearly demonstrated.

The specified range of λ_T values (0.181 - 0.135) is expected to have a minimal impact on the flow structures. Case 3 exhibits the highest upstream cavity swirl due to its tightest clearance (highest rpm) and consequently, the lowest λ_T value, which is in line with finding in Chapter 4. This trend extends to the downstream cavity swirl, where Case 3 also exhibits the highest value. The labyrinth seal accelerates the flow tangentially due to having a small upstream swirl ratio, less than 0.55 based on Denecke et.al [34] for all the cases. Through three teeth labyrinth seal circulating seal pocket structures are visible.

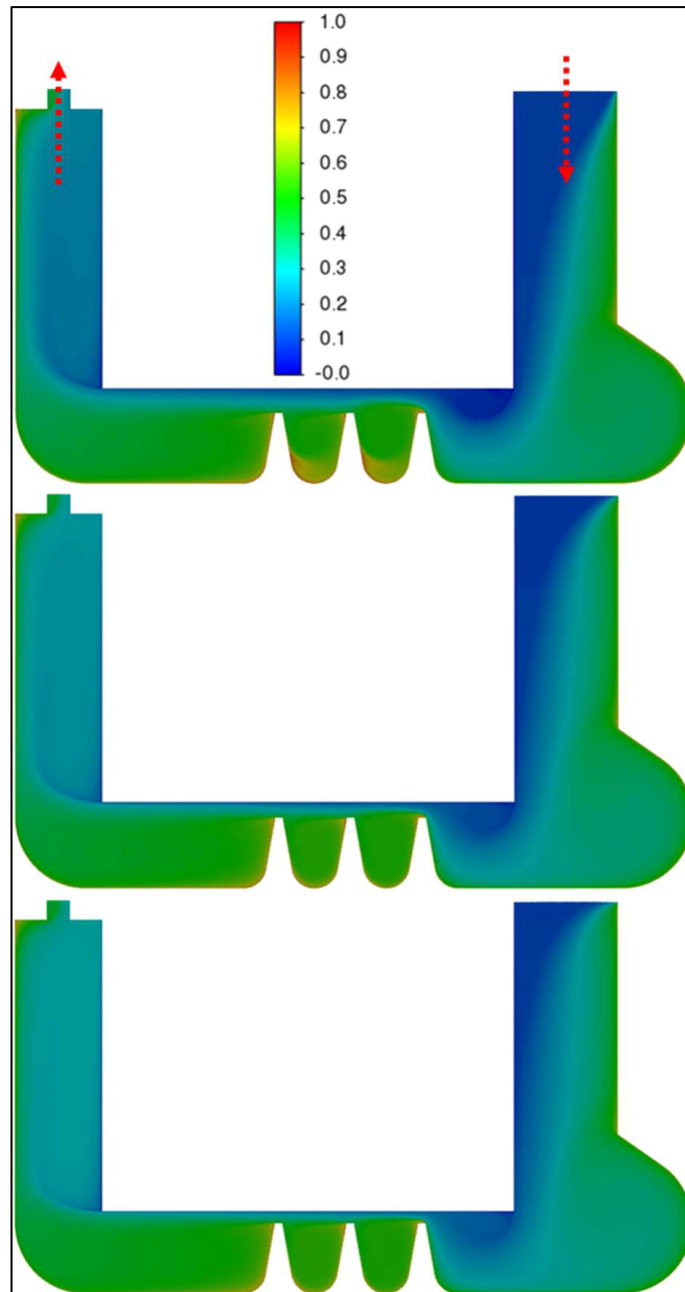


Figure 6.3: Swirl Ratio Contours, Top: Case1, Middle: Case2, Bottom: Case3.

Upon close examination of the Figures 6.3 and 6.4, minimal change in the flow field across different engine running conditions is observed. First, within the upstream cavity leakage flow travels radially inward on the stator surface, then rotor pumps the flow radially outward, which generates big circulating structure that extends toward the base of the seal. A secondary circulating zone arises near the labyrinth seal inlet due to abrupt turning of the flow, which typically enhances the sealing effectiveness.

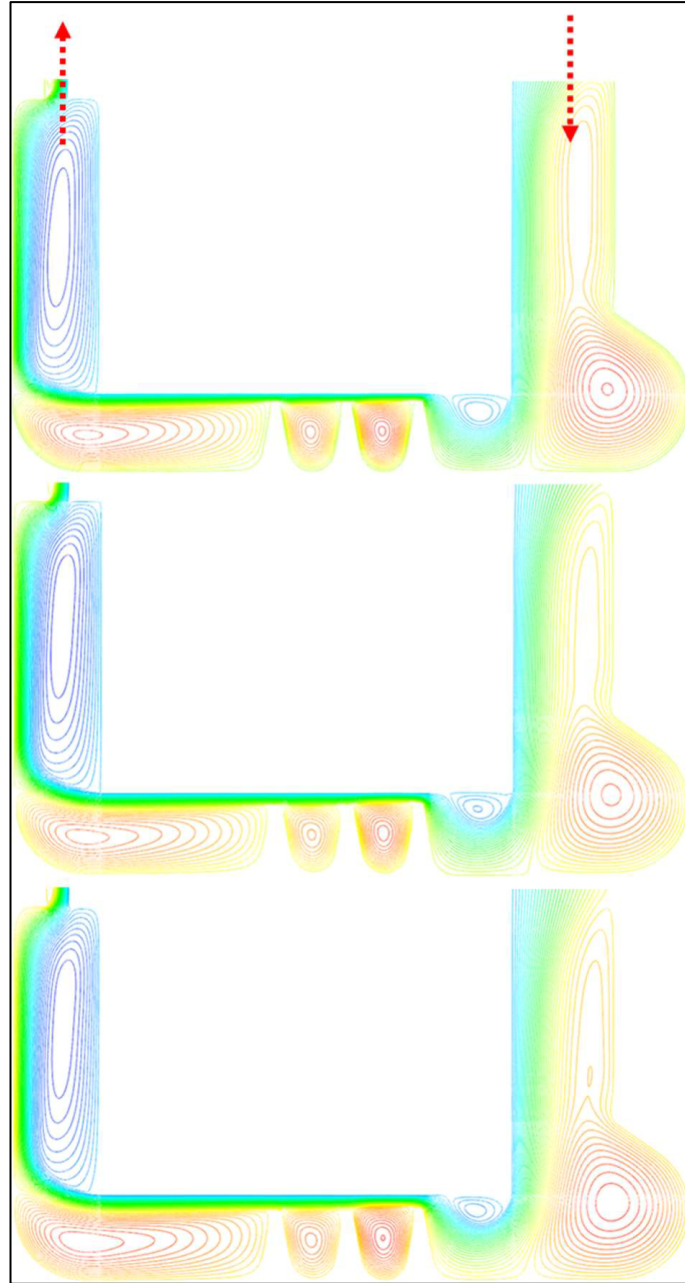


Figure 6.4: Non-Dimensional Stream Function Contours, Top: Case1, Middle: Case2, Bottom: Case3.

After leakage flow passes through the labyrinth seal, it undergoes a slight expansion and impinges on the rotor surface within the downstream cavity, which pumps the flow radially outward. Notably, two circulation zones emerge in downstream cavity: the first arises immediately downstream of the seal, while the second, a larger zone, forms near the outlet as the pumped flow transitions from the rotor to the stator surface, moving radially inward.

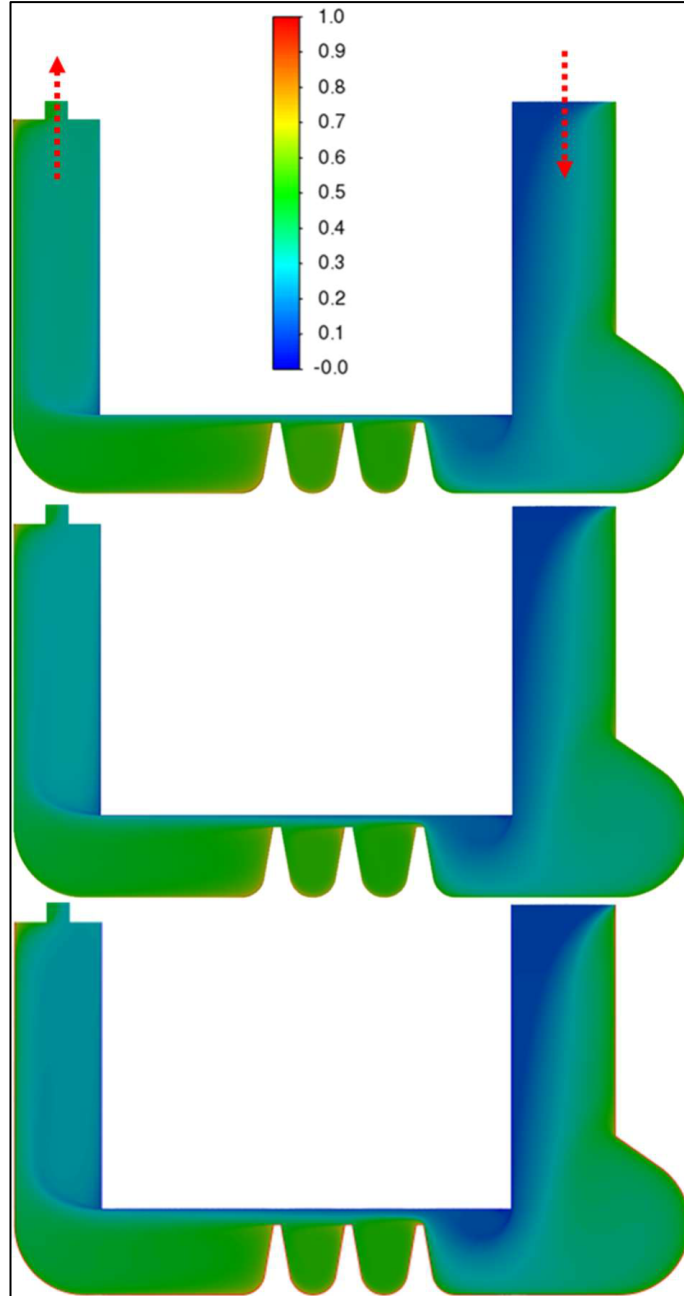


Figure 6.5: Swirl Ratio Contours, Top: Case5, Middle: Case3, Bottom: Case4.

Upon close examination of the Figures 6.5 and 6.6, all cases reveals highest power/thrust condition with different labyrinth seal clearances, resulting with different λ_T values for each cases. Case 5, due to its tightest clearance, exhibits the highest λ_T value, which consequently leads to the highest swirl distribution throughout all cavities.

Case 5 exhibits the weakest disk pumping flow within the upstream cavity, consequently positioning the separation line between this flow and the seal flow furthest to the right compared to other cases. This trend continues in the downstream

cavity, where Case 5 demonstrates the separation point located at the smallest radius. Notably, Case 4, due to its widest clearance, displays a significantly more pronounced expansion of the seal leakage jet.

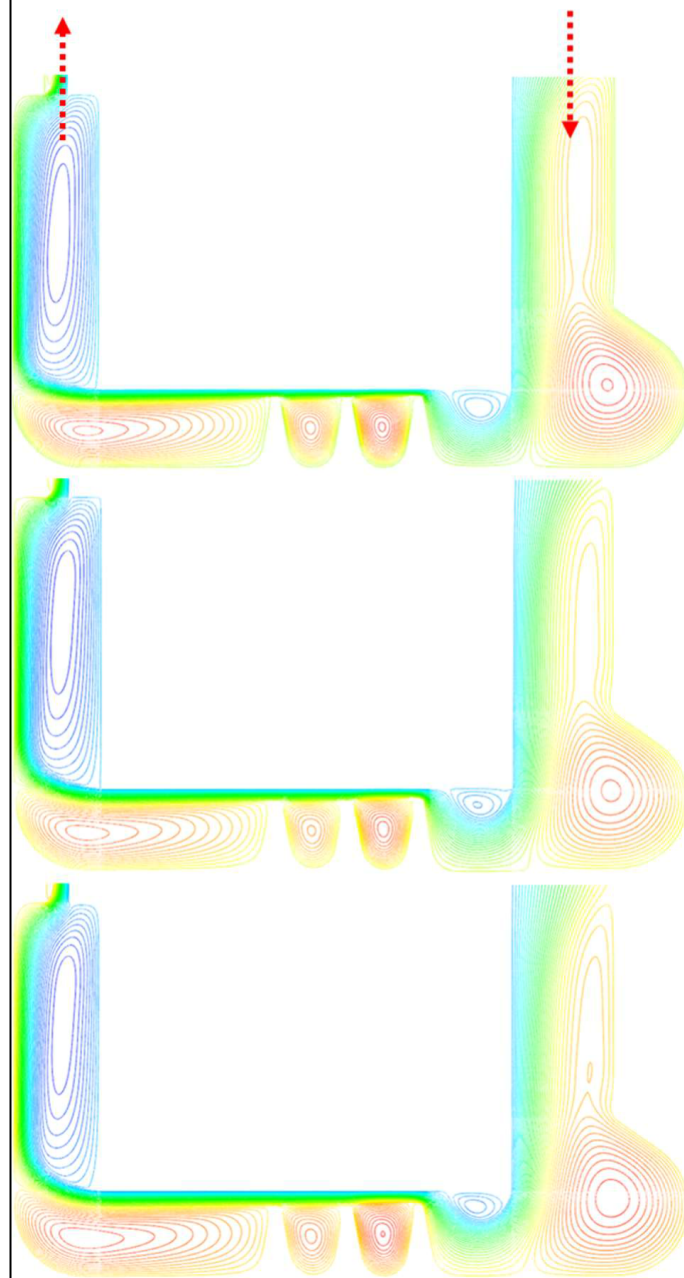


Figure 6.6: Non-Dimensional Stream Function Contours, Top: Case5, Middle: Case3, Bottom: Case4.

Figures 6.7 to 6.11 illustrate the distributions of swirl ratio, static pressure non-dimensionalized by inlet conditions, and total temperature rise throughout the cavity at various stations within the stator well for all Cases 1-5 as tabulated at Table 6.1. CFD results are shown along with Flow Simulator results, which is a one dimensional

solution that each value is mass weighted averaged of the line at defined location is a representative value of that element.

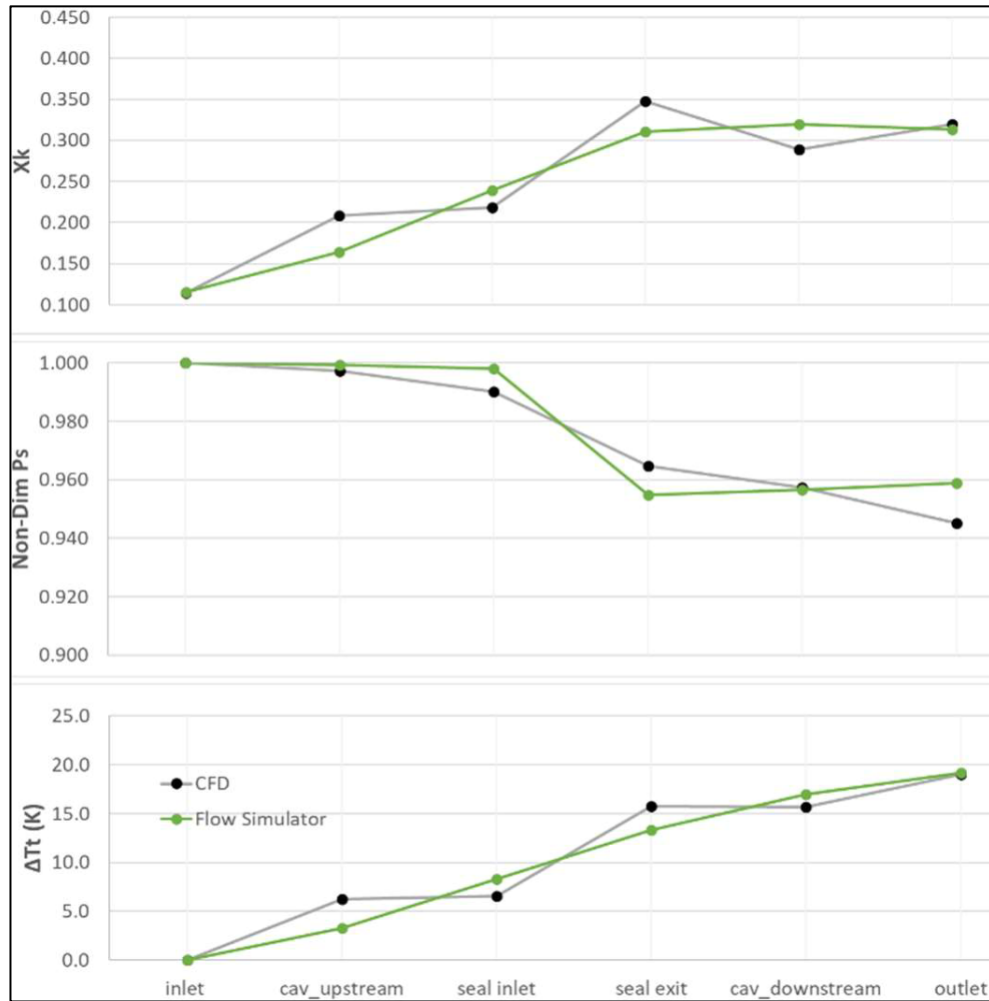


Figure 6.7: Variation of Swirl Ratio, Non-Dimensional Static Pressure and Total Temperature Rise within Stator Well: Case1.

An examination of Cases 1-3 (Figures 6.7-6.9) demonstrates a high degree of similarity in the flowfields, with minimal change in swirl ratio values at each station. As expected with increasing rotational speed and consequent decrease in λ_T , a slight rise in station values is observed across all cases. Flow Simulator exhibits same behavior, yet it underpredicts upstream cavity swirl and makes up for it just before the labyrinth seal inlet for all cases, which could stem from the under-prediction of the rotor torque or over-prediction of the stator torque at the seal, resulting in smaller swirl ratio value at the upstream cavity. However, on the downstream cavity Flow Simulator over-predicts swirl ratio yet eventually swirl ratio result at the exit reach very close value for CFD and Flow Simulator.

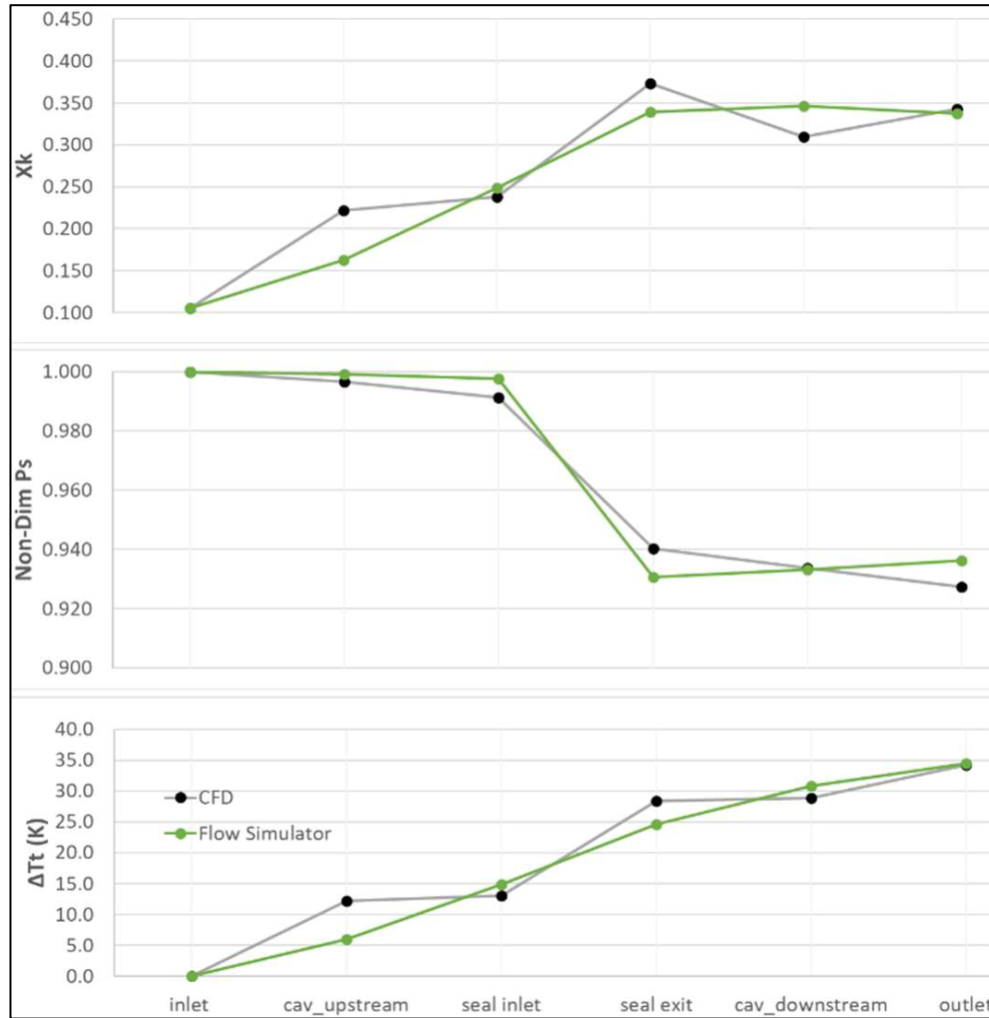


Figure 6.8: Variation of Swirl Ratio, Non-Dimensional Static Pressure and Total Temperature Rise within Stator Well: Case2.

A comparison of non-dimensional static pressure reveals a clear trend of monotonically decreasing pressure with increasing rotational speed. Notably, while Cases 1-3 exhibit similar swirl ratio distributions, it's important to remember that these cases represent distinct engine operating conditions characterized by variations in both rotational speed and density. As anticipated, static pressure reduction is amplified with increasing engine running power condition. The Flow Simulator results further support this observation.

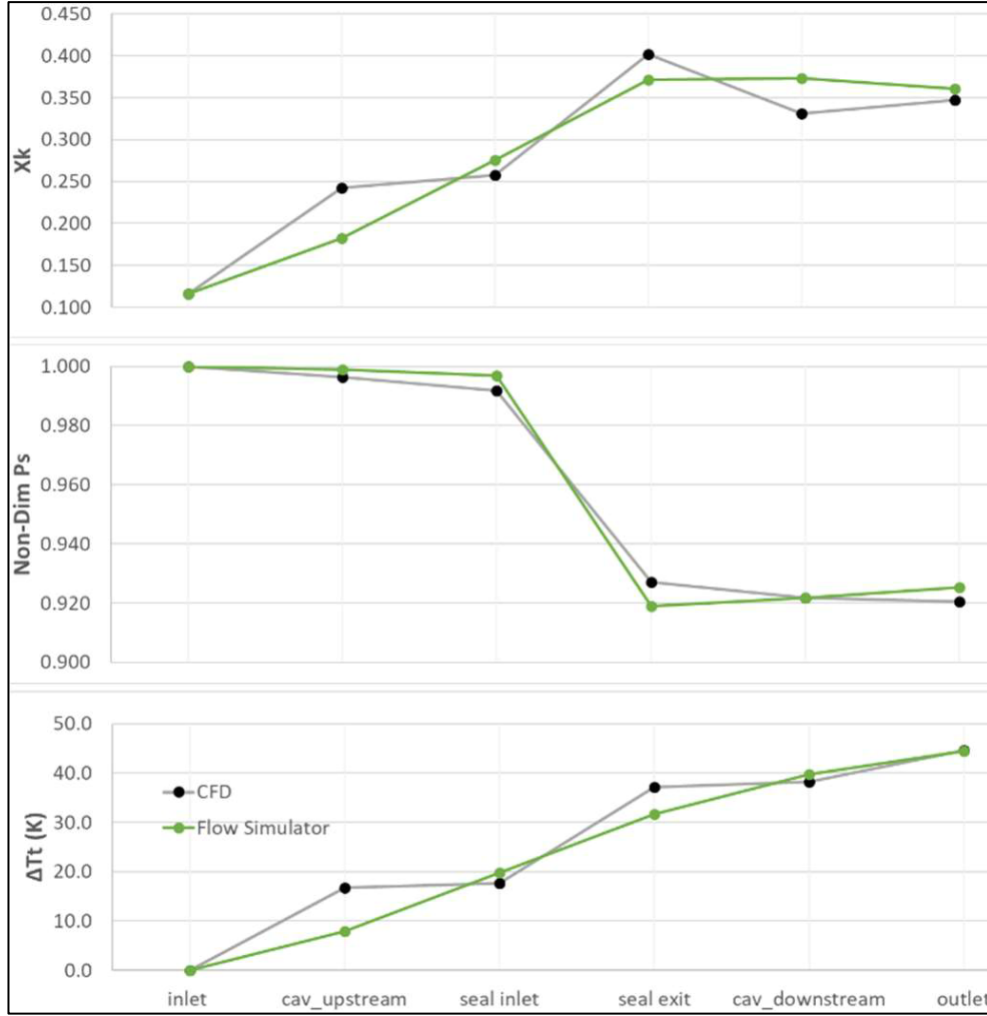


Figure 6.9: Variation of Swirl Ratio, Non-Dimensional Static Pressure and Total Temperature Rise within Stator Well: Case3.

Windage temperature rise plays a key role, with rotor torque primarily driven by the cubic dependence on rotational speed (ω^3) and secondarily by the increase in density. Leakage mass flow rate increase, of course, with increasing engine running power conditions even for the tighter labyrinth seal clearances. However, this rise is not as significant as the increase in rotational speed, leading to a reduction in λ_T .

Moreover, a distinct rise in temperature throughout the stator well is observed with increasing engine rpm. Flow Simulator follows similar trends with CFD in terms of overall change. However, discrepancies exist in the specific distribution of the temperature rise due to the inherent differences in the selection of sub-cavities for analysis.

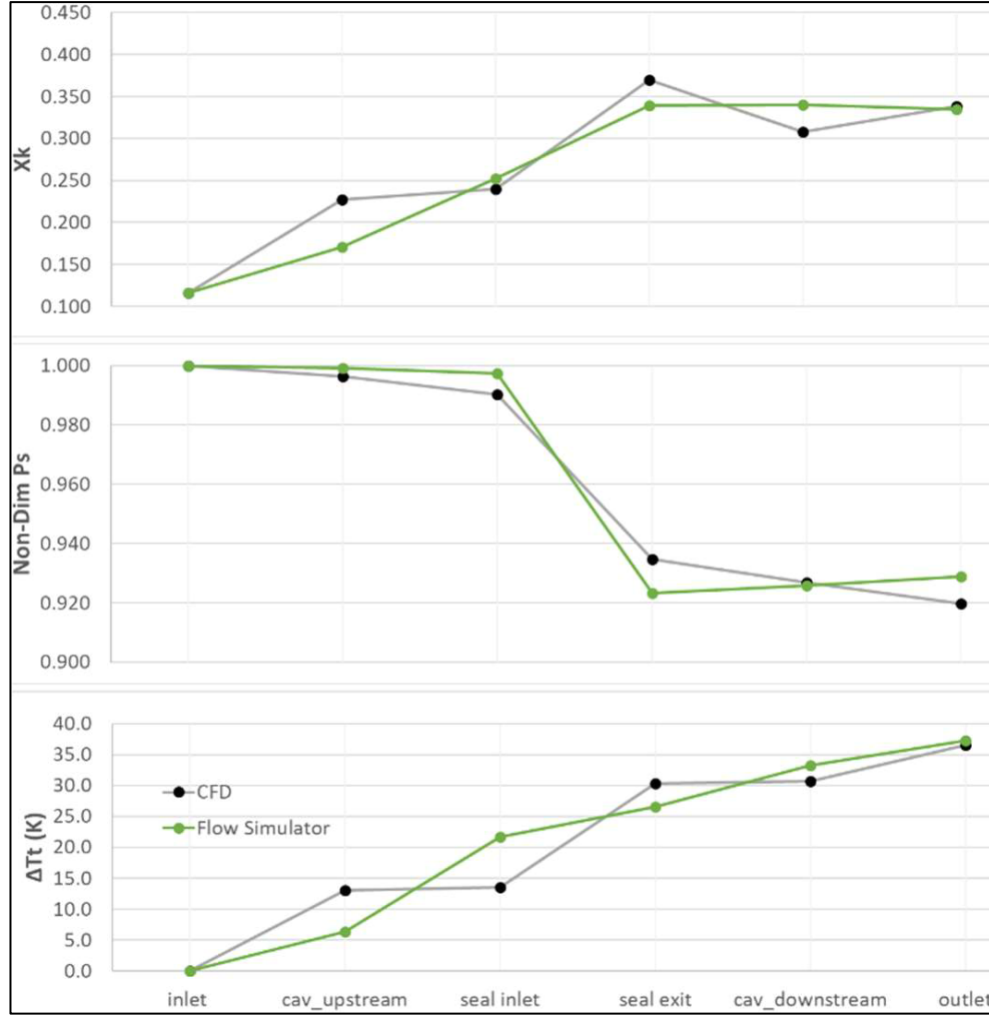


Figure 6.10: Variation of Swirl Ratio, Non-Dimensional Static Pressure and Total Temperature Rise within Stator Well: Case4.

Upon close examination of the Figures 6.9 and 6.11 reveals diminishing of swirl ratio rise throughout cavity with increasing seal clearance due to increase of λ_T . This trend aligns with the finding in Chapter 4, as higher λ_T values (associated with larger seal clearances for these cases) lead to lower swirl ratios. Boundary conditions for these three cases are identical, and the seal clearance is the only different parameter. Consequently, the non-dimensional pressure drop across the labyrinth seal is naturally highest for the case with the smallest leakage flow. In contrast, the change in the pressure on upstream and downstream cavities is rather small since it is dependent on the radial pressure distribution equation. So, the majority of the pressure drop comes from the seal region.

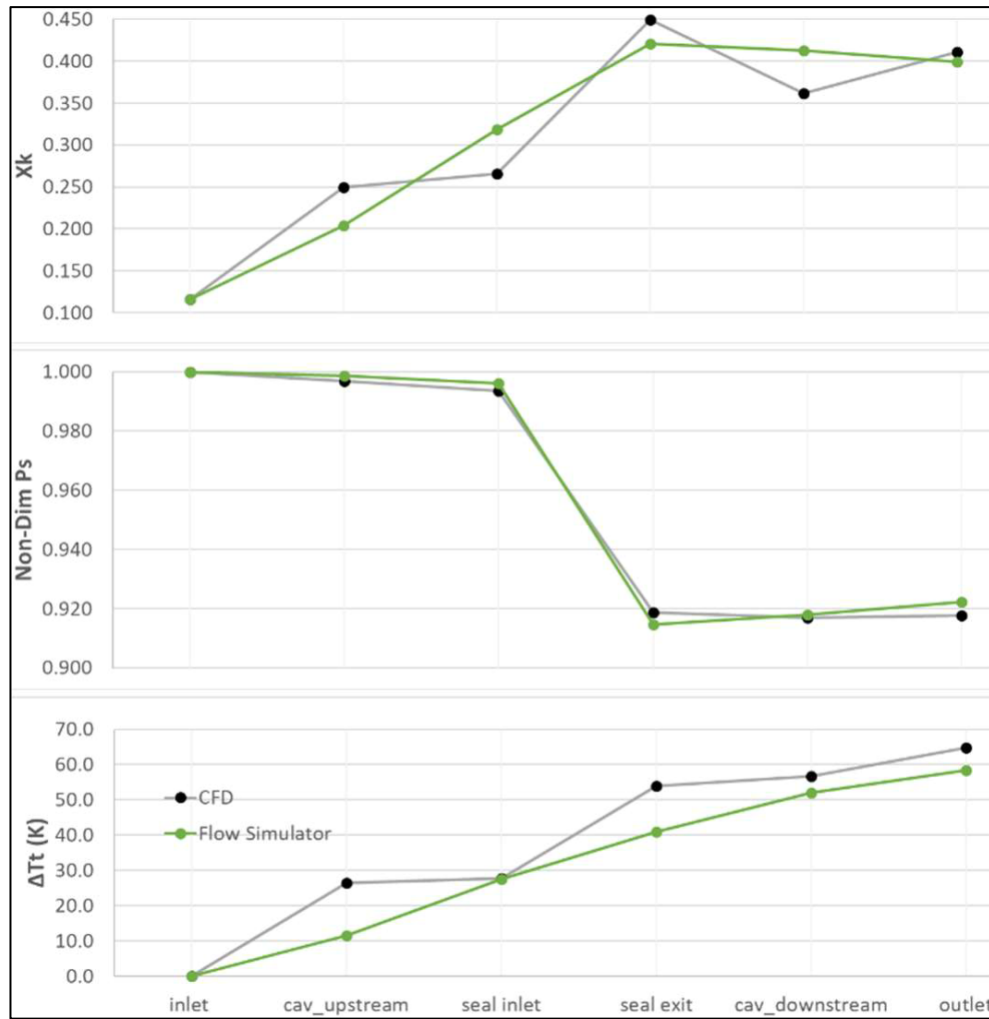


Figure 6.11: Variation of Swirl Ratio, Non-Dimensional Static Pressure and Total Temperature Rise within Stator Well: Case5.

Encouragingly, CFD and Flow Simulator results exhibit good agreement for windage temperature rise. Both methods predict very similar values at the exit, despite minor variations in the distributions of change throughout the domain. Notably, only in Case 5 does the Flow Simulator predict a slightly lower value at the outlet compared to CFD.

Figure 6.12 presents the relative contribution of windage power loss, divided into three regions (upstream cavity, seal cavity, downstream cavity) for Cases 1-3 and Cases 3-5. The CFD results are shown only. The top chart reveals that upstream cavity is the primary contributor for the temperature rise, responsible nearly %45. The remaining portion is distributed between the seal cavity and the downstream cavity. Moreover, as rotational speed increases the contribution comes from the seal region increases and

eventually surpasses that of the upstream cavity, reaching approximately %43. This observation suggests a non-linear relationship between rotor torque and seal geometry.

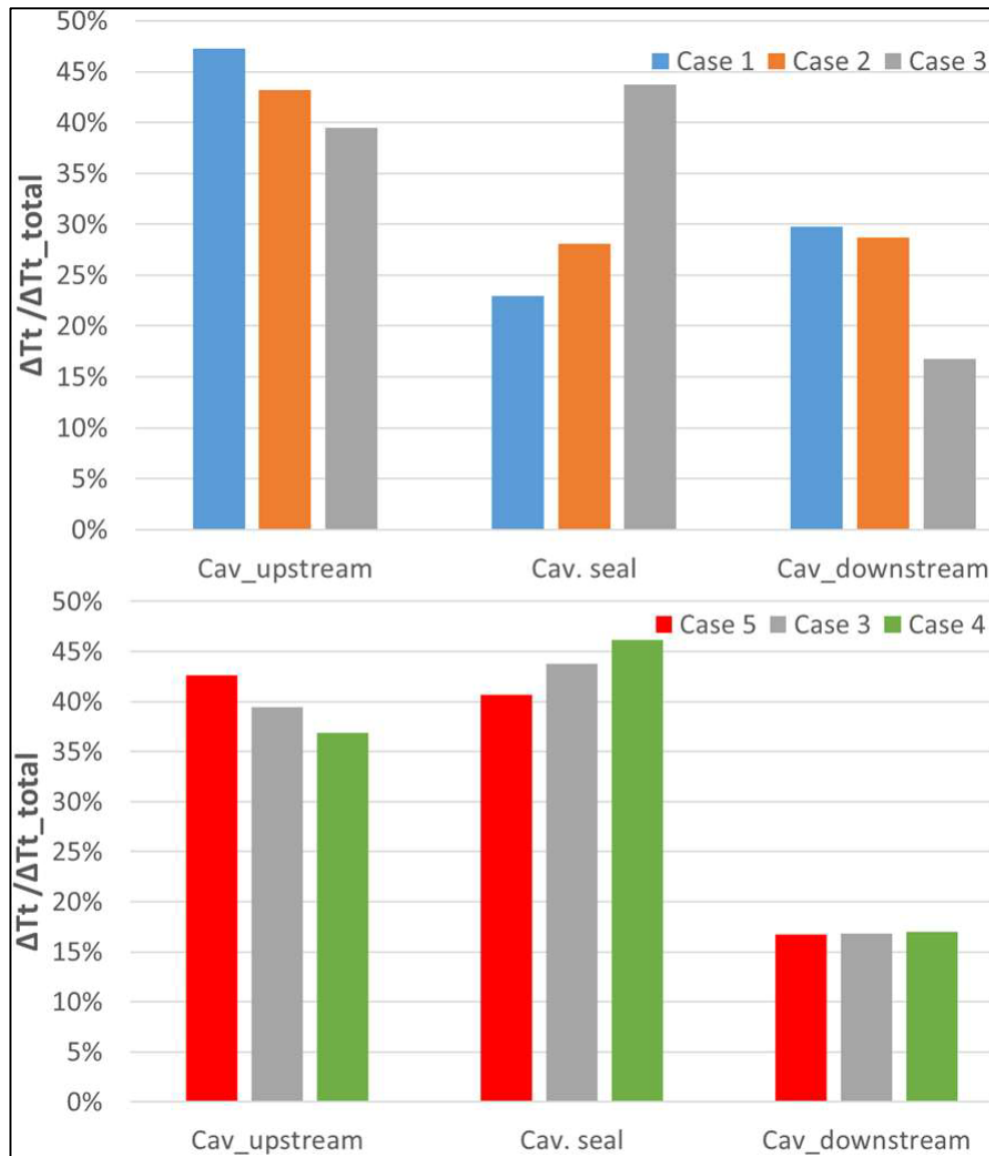


Figure 6.12: Total Temperature Rise Contributions from Each Sub Cavity, Top: Cases 1-3, Bottom: Cases 3-5.

Additionally, the downstream cavity exhibits a similar impact to the seal cavity under low power conditions (Cases 1-3).

The bottom chart presents high power/thrust setting results across varying seal clearance conditions. It emphasizes the relatively insignificant role of the downstream cavity in windage temperature rise. Notably, the upstream and seal cavities emerge as the primary contributors, with their relative proportions influenced by leakage flow.

Cases with lower leakage flow exhibit a higher contribution from the upstream cavity and a correspondingly smaller contribution from the seal cavity.

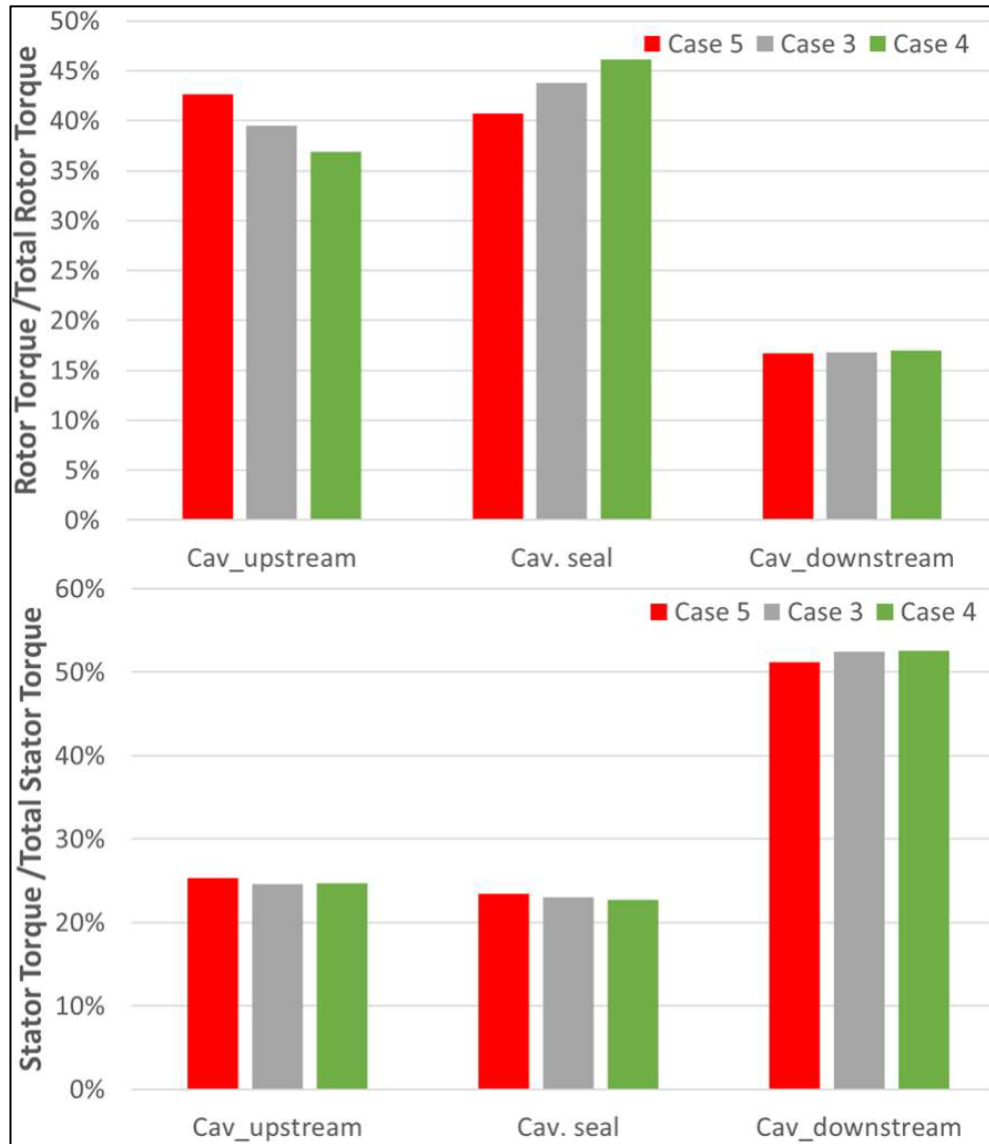


Figure 6.13: Torque Contributions from Each Sub Cavity, Top: Rotor Torque, Bottom: Stator Torque.

Figure 6.13, based solely on CFD data, presents the impact of the seal clearance on the relative contributions of rotor and stator torques for Cases 3-5. Rotor torques exerted within the upstream and seal cavities remain dominant throughout the stator well cavity compared to the downstream cavity, which is in line with the findings in Figure 6.12. Interestingly, the stator torque generated in the downstream cavity, although lower than the rotor torque in that region, contributes to roughly 50% of the total stator torque exerted.

Figure 6.14 presents the radial distributions of non-dimensionalized rotor torques, normalized by their respective maximum local values, for both the upstream cavity (represented by round symbols) and downstream cavity (represented by diamond symbols). As anticipated, increasing clearance values lead to a slight amplification of rotor torques in both cavities. This amplification can be attributed to the reduction in swirl ratios observed across all cavities. However, it's important to note that the relative contribution of the upstream cavity's rotor torque to the total rotor torque decreases with increasing clearance.

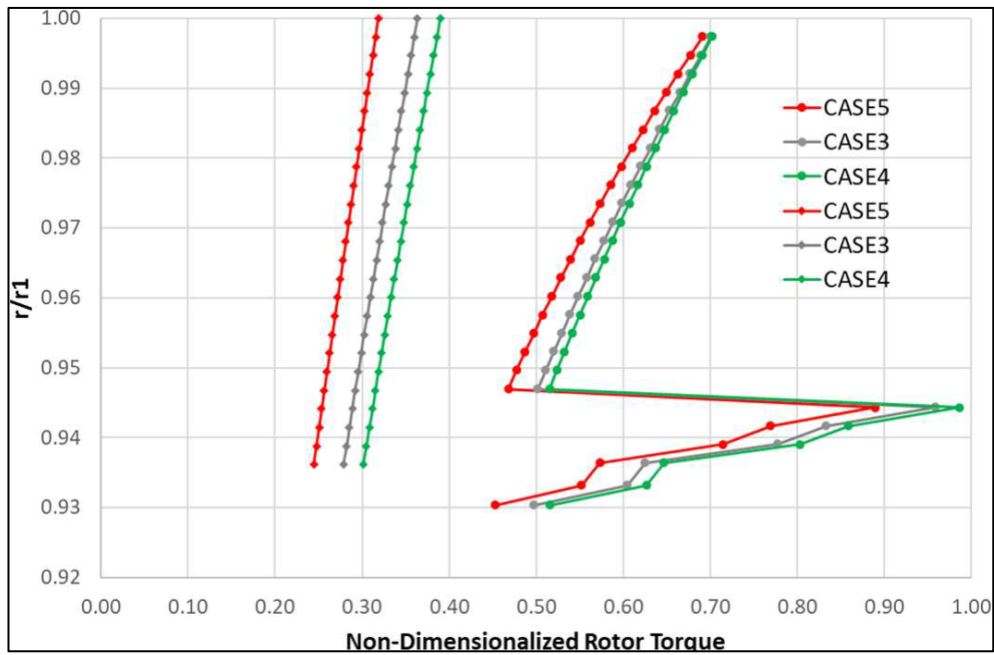


Figure 6.14: Non-Dimensionalized Torque Distributions in Radial Direction for Upstream Cavity (round symbols) and Downstream Cavity (diamond symbols).

These results strongly suggest that mechanisms enhancing upstream cavity swirl beyond current geometric designs, coupled with a more significant increase in downstream cavity rotor torque compared to stator torque, could lead to a further reduction in overall windage power loss and consequent temperature rise. Such advancements would demonstrably benefit gas turbine engine performance. This aligns with the findings reported by Kong et al. [13-14].

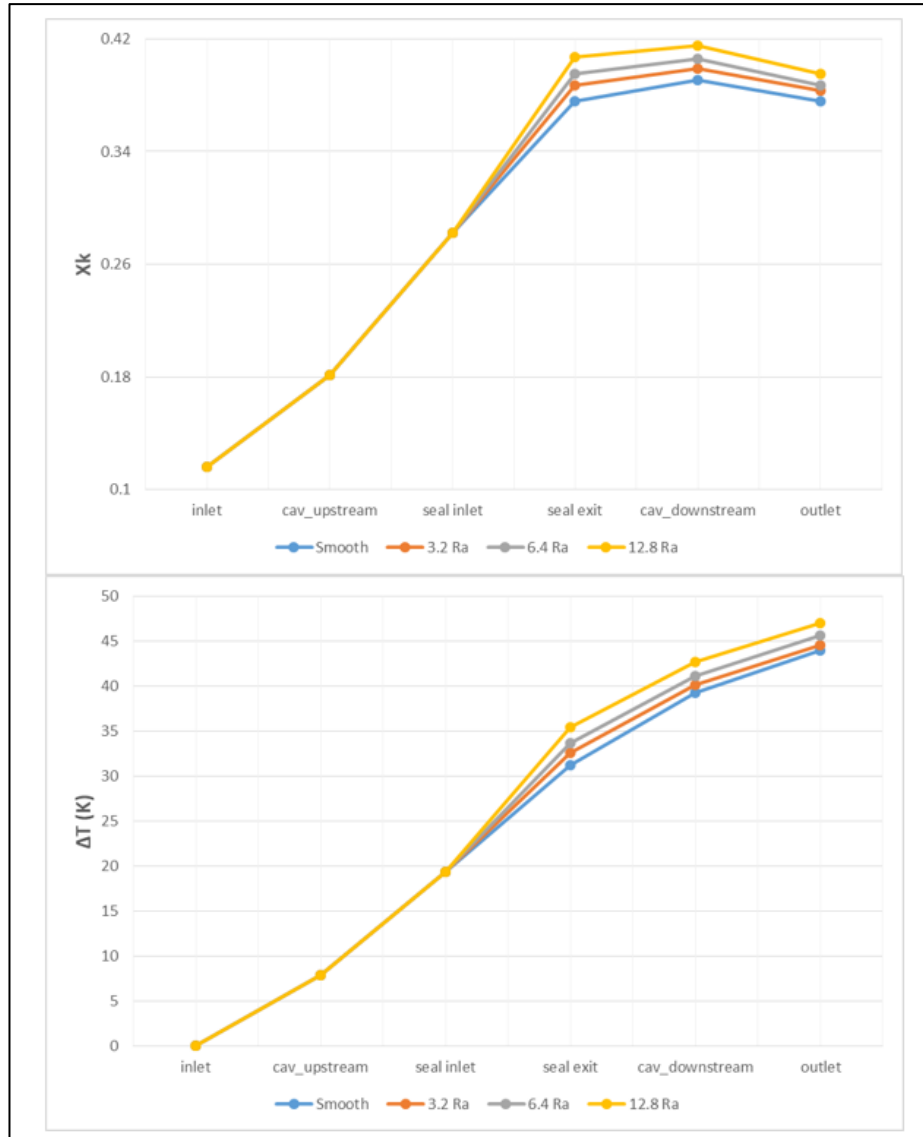


Figure 6.15: Variation of Swirl Ratio and Total Temperature Rise within Stator Well for Different Surface Roughness Values: Case3.

Furthermore, labyrinth seals within the compressor well cavity generally feature honeycomb surfaces on the stator and abradable linings on the rotor teeth to maintain lower clearances and permit the rubbing of seal teeth against the honeycomb. Consequently, both the honeycomb and seal teeth surfaces are made of relatively rough materials. Up to this point, all analyses were conducted assuming smooth surfaces. Therefore, the effect of surface roughness on the seal cavity was investigated using the Flow Simulator flow solver program.

In the Flow Simulator, to simulate the effect of roughness, a friction multiplier is added to the surfaces to increase the skin friction coefficient. To calculate the friction multiplier, the tips of the seal teeth are considered as pipes, neglecting the pocket

regions. Using this pipe analogy, the Moody Chart is applied to calculate the friction multiplier for three different surface roughness values: 3.2, 6.4, and 12.8 μm . The ratio of the friction factor for each rough surface to the friction factor for a smooth surface is used as the friction multiplier.

The effect of surface roughness on swirl ratio and temperature increase is presented in Figure 6.15. Since the rotor area is greater than the stator area in the seal cavity, the swirl ratio increases with increasing surface roughness. Additionally, as expected, the rotor torques increase with increasing surface roughness, leading to a higher temperature increase in the seal cavity. These findings underscore the importance of the compressor well cavity in real-gas turbine applications.

7. SUMMARY AND DISCUSSION

In this thesis, the flow characteristics within the compressor well cavity, a region located beneath the shrouded compressor stator are investigated. First, a comprehensive review of literature is compiled, covering the effects of the compressor stator well from both aerodynamic performance and thermal and secondary air system perspectives. Then, theoretical knowledge is summarized to provide a foundational understanding of the flow characteristics within the compressor well cavity. This includes covering fundamental flow parameters such as the rotor-stator cavity, windage heating, and labyrinth seal flow.

Furthermore, an experimental test case is used. First, test case geometry is utilized to understand flow behaviour by generating 1D numerical solution. The compressor well cavity is divided into three subcategories: the upstream cavity, the seal cavity, and the downstream cavity. Angular momentum conservation is employed to the cavity in order to calculate bulk swirl ratio for each cavities. The analysis reveals that the upstream cavity is the primary source of rotor torque. Additionally, for low inlet swirl conditions, typical of stator exit flows, the swirl ratio diminishes as the turbulent flow parameter increases because of the fact that leakage flow has low momentum and rotor torques are predominant with $(1 - Xk)^{1.35}$. After understanding the basics of the flow characteristics inside the cavity by using test case geometry, 2D axisymmetric CFD method is validated by using its experimental data. Also, alongside the CFD results, the results of Flow Simulator, which is a 1D flow solver program similar to described in Chapter 2, are presented. Both numerical approaches effectively captured flow characteristics consistent with experimental observations. A mesh sensitivity analysis was performed for the case exhibiting the highest rotational speed and pressure ratio.

A generic stator well case was selected for analyzing flow parameters at three distinct engine power settings. Furthermore, three different clearances were examined under the maximum power/thrust condition to assess their impact on stator well flow characteristics. Results indicate that the upstream cavity is primarily responsible for approximately 45% of windage heating which is inline with finding of Kamdar et al. [10]. Rotor torques dominate within this cavity. The radial distribution of rotor torque suggests potential for windage power reduction through swirl ratio enhancement beyond nominal geometric configurations.

In conclusion, this study is noteworthy for being the first in the literature to explain the compressor well cavity flow behavior using the turbulent flow parameter, a generic parameter employed to describe flow behaviors in cavities. The findings reveal that the stator well cavity flow exhibits a reduction in swirl ratio throughout the stator well and an increase in windage power loss with higher λ_T , indicating more leakage flow. Maintaining low leakage flow is advantageous for engine performance by enhancing compressor efficiency and reducing windage power loss. Additionally, it is crucial to monitor the air temperature increase to ensure it remains within material limits.

REFERENCES

- [1] Ludwig L., (1978), "Gas Path Sealing in Turbine Engines", Seal Technology in Gas Turbine Engines, AGARD Publication.
- [2] Moore R. D., & Osborn, W. M., (1977), "Effects of tip clearance on overall performance of transonic fan stage with and without casing treatment", (No. E-8945).
- [3] Wisler D. C., (1985), "Loss reduction in axial-flow compressors through low-speed model testing", ASME.
- [4] Lewis L. V. (2002), "In-engine measurements of temperature rises in axial compressor shrouded stator cavities", In: Turbo Expo: Power for Land, Sea, and Air (Vol. 36088, pp. 781-794), American Society of Mechanical Engineers.
- [5] Wellborn S. R., & Okiishi T. H., (1999), "The influence of shrouded stator cavity flows on multistage compressor performance", In: Turbo Expo: Power for Land, Sea, and Air (Vol. 78620, p. V001T01A005), American Society of Mechanical Engineers.
- [6] Kamdar N., Lou F., & Key N. L., (2021), "Details of shrouded stator hub cavity flow in a multi-stage axial compressor part 1: interactions with the primary flow", In Turbo Expo: Power for Land, Sea, and Air (Vol. 84904, p. V02AT31A041), American Society of Mechanical Engineers.
- [7] Demargne A. A. J., & Longley J. P., (2000), "The aerodynamic interaction of stator shroud leakage and mainstream flows in compressors", In: Turbo Expo: Power for Land, Sea, and Air (Vol. 78545, p. V001T03A095), American Society of Mechanical Engineers.
- [8] Heidegger N., Hall E., & Delaney R., (1996), "Parameterized study of high-speed compressor seal cavity flow", In 32nd Joint Propulsion Conference and Exhibit (p. 2807).
- [9] Wellborn S. R., (2001), "Details of axial-compressor shrouded stator cavity flows", In Turbo Expo: Power for Land, Sea, and Air (Vol. 78507, p. V001T03A078), American Society of Mechanical Engineers.
- [10] Kamdar N., Lou F., & Key N. L., (2022), "Details of shrouded stator hub cavity flow in a multistage axial compressor part 2: leakage flow characteristics in stator wells", Journal of Engineering for Gas Turbines and Power, 144(1), 011027.
- [11] Ozturk H. K., Childs P. R. N., Turner A. B., Hannis J. M., & Turner J. R., (1998), "A Three Dimensional Computational Study of Windage Heating Within an Axial Compressor Stator Well", (Vol. 78651, p. V004T09A022), American Society of Mechanical Engineers.
- [12] Öztürk H. K., Turner A.B., Childs P. R. N., & Bayley F.J., (2000), "Stator well flows in axial compressors", International journal of heat and fluid flow, 21(6), 710-716.
- [13] Kong X., Liu G., Liu Y., & Zheng L., (2017), "Experimental testing for the influences of rotation and tip clearance on the labyrinth seal in a compressor stator well", Aerospace Science and Technology, 71, 556-567.

- [14] Kong X., Liu G., Liu Y., & Feng Q., (2016), "Investigation on the leakage flow, windage heating and swirl development of rotating labyrinth seal in a compressor stator well", In: Turbo Expo: Power for Land, Sea, and Air (Vol. 49781, p. V05AT15A014), American Society of Mechanical Engineers.
- [15] Bayley F. J., & Childs, P. R. N., (1994), "Air temperature rises in compressor and turbine stator wells (Vol. 78866, p. V004T09A034), American Society of Mechanical Engineers.
- [16] Sultanian B., (2018), "Gas turbines: internal flow systems modeling" (Vol. 44). Cambridge University Press.
- [17] Batchelor G. K., (1951), "Note on a class of solutions of the Navier-stokes equations representing steady rotationally symmetric flow" Quart. J. Mech. App. Math., (4):29–41.
- [18] Stewartson K., (1953), "On the flow between two rotating coaxial disks", In: "Mathematical Proceedings of the Cambridge Philosophical Society" (Vol. 49, No. 2, pp. 333-341), Cambridge University Press.
- [19] Child P. R. N., (2010), "Rotating Flow", Elsevier.
- [20] Daily J. W., Nece R. E., (1960), "Chamber dimension effects on induced flow and frictional resistance of enclosed rotating disks", Journal of Basic Engineering 82, p. 217-232.
- [21] Haaser F., Jack J., & McGreehan W., (1988), "Windage rise and flowpath gas ingestion in turbine rim cavities".
- [22] Martin H. M., (1908), "Labyrinth Packings," Engineering, pp. 35-36
- [23] Egli A., (1935), "The Leakage of Steam through Labyrinth Seals", Transactions of the ASME, 57, pp. 115-122.
- [24] Hodkinson B., (1939), "Estimation of the Leakage through a Labyrinth Gland", Proceedings of the Institution of Mechanical Engineers, 141, pp. 283–288.
- [25] Vermes G., (1961), "A Fluid Mechanics Approach to the Labyrinth Seal Leakage Problem", ASME Transactions -Journal of Engineering for Power, 83 (2), pp. 161-169.
- [26] Childs D. W., (1993), "Turbomachinery Rotordynamics – Phenomena, Modeling, and Analysis", John Wiley & Sons, New York.
- [27] Gurevich M. I., (1965), "Theory of Jets in an Ideal Fluid", Translated by Street R. L. & Zagustin K., Academic Press, New York.
- [28] Zimmerman H. & Wolff K. H., (1987), "Comparison between Empirical and Numerical Labyrinth Flow Correlations", ASME 87-GT-86.

- [29] Menter F., (1993), “Zonal Two Equation k- ω Turbulence Models for Aerodynamic Flows”, 23rd Fluid Dynamics, Plasmadynamics, and Lasers Conference, doi:10.2514/6.1993-2906.
- [30] Wilcox D. C., (1988), “Reassessment of the Scale-determining Equation for Advanced Turbulence Models”, AIAA Journal, 26(11), 1299-1310, doi:10.2514/3.10041.
- [31] Jones W., & Launder B. (1972), “The Prediction of Laminarization with a Two-equation Model of Turbulence”, International Journal of Heat and Mass Transfer, 15(2), 301-314. doi:10.1016/0017-9310(72)90076-2
- [32] Erdem E., Sertcan M.C., Sal S., Dogu Y., & Yalcinkaya A., (2022), “Investigation of Impeller Backface Cavity Flowfield Using Cfd and a One-Dimensional Flow Solver”, GT2022-84036, 2022.
- [33] Yazan A., Karasu Z.T., Yilmaz S., Erdem E., & Sal S. (2023), “Investigation of Rotor Stator Cavity Flowfield at Downstream of a High Pressure Turbine Disk” GT2023-103713.
- [34] Denecke J., Farber J., Dullenkopf K., & Bauer H. J., (2008), “Interdependence of Discharge Behavior, Swirl Development and Total Temperature Increase in Rotating Labyrinth Seals”, GT2008-51429.

BIOGRAPHY

Ahmet Cüneyt KAHRAMAN graduated from the Department of Aerospace Engineering at Istanbul Technical University (ITU) in 2020, ranking second in his department. He began his career in the gas turbine industry as a Thermal System Design Engineer, specializing in oil system design, bearing pressurization and ventilation. He continues to work in this field.