

T.R.

GEBZE TECHNICAL UNIVERSITY

GRADUATE SCHOOL

**INVESTIGATION OF THE CAVITY FLOW OVER IMPELLER
BACKFACE OF A CENTRIFUGAL COMPRESSOR**

ESRA SEDEF

A THESIS OF MASTER OF SCIENCE

DEPARTMENT OF ENERGY TECHNOLOGIES

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

ADVISOR: PROF. İLYAS KANDEMİR

JULY 2024

T.R.

GEBZE TECHNICAL UNIVERSITY

GRADUATE SCHOOL

**INVESTIGATION OF THE CAVITY FLOW OVER IMPELLER
BACKFACE OF A CENTRIFUGAL COMPRESSOR**

ESRA SEDEF

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

ADVISOR: PROF. İLYAS KANDEMİR

JULY 2024

T.C.
GEBZE TEKNİK ÜNİVERSİTESİ
LİSANSÜSTÜ EĞİTİM ENSTİTÜSÜ

RADYAL KOMPRESÖRLERDE ÇARK ARKA BOŞLUĞU
AKIŞININ İNCELENMESİ

ESRA SEDEF

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

DANIŞMAN: PROF. DR. İLYAS KANDEMİR

TEMMUZ 2024



MASTER of SCIENCE JURY APPROVAL FORM

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

JURY

MEMBER
(THESIS ADVISOR) : PROF. İLYAS KANDEMİR

MEMBER : DR. ERİNÇ ERDEM

MEMBER : PROF. YAHYA DOĞU

APPROVAL

Gebze Technical University Graduate Administration Board
...../...../..... date and/..... numbered decision.

SIGNATURE/SEAL

*To Mr. Ali Çevik,
who is no longer with us.
May you find peace...*

ABSTRACT

In gas turbine engines, radial compressors are commonly used especially when the space limitations are dictated. Impeller backface cavity flow field becomes an important parameter when the radial compressors are preferred. The flow characteristic in the cavity plays an essential role in the distribution of cooling air, pressurization as well as balancing the axial loads on bearings. In these cavities, the impeller main flow tip leakage typically occurs radially inward and mixes with the seal leakage at the lower radius sections, directing the flow towards the turbine or sump region for cooling and pressurization.

Flow field behavior throughout the backface depends on various parameters which affect the flow distribution. The flow characteristics in rotor-stator cavities are affected by the gap ratio (G) as a geometric parameter, as well as inlet swirl ratio (β_1), rotational Reynolds Number (Re_θ), and turbulent flow parameter (λ_t).

Within the scope of this thesis, the necessary flow parameters that are analytically determined using 1-D Flow Network Solver are utilized as the inlet boundary conditions to the 2-D axisymmetric CFD model of the backface geometry. Then, the CFD results are compared and validated with the engine ground tests in which the static pressures are measured at various radii and under several operating conditions.

The flow structure in CFD model demonstrates similar behavior as in the theory. The flow is pumped radially outward by the rotor side and the required mass flow for the rotor is supplied by the stator over which the flow is radially inward. Just as the swirl ratio attains unity in the cavity, the radial velocity becomes zero over the rotor face at that radius and the flow starts to move radially inward over the rotor as well. Following the flow structure substantiation in the CFD results, the static pressure measurements in the test and pressure field in the model are compared which show a good agreement. At full engine power, the differences between the test data and the CFD model are 0.8%, 4.4% and 1.4%, respectively, for the examined three distinct radii throughout the cavity.

Key Words: Gas Turbine Engines, Secondary Air Systems, Rotor-stator Cavity, Impeller Backface Flow, Radial Inward Flows, Computational Fluid Dynamics.

ÖZET

Gaz türbinli motorlarda, radyal kompresörler bilhassa kısıtlı bir hacim gerekliliği olduğunda yaygın olarak kullanılır. Radyal kompresör ihtiva eden motorlarda çark arkasında bulunan boşluktaki akış alanı da önemli bir etmen haline gelmektedir. Boşluktaki akış karakteristiği, ihtiyaç duyulan bölgelere soğutma havasının dağıtımında, basınçlandırmada ve rulmanlardaki eksenel yükleri dengelemede önemli bir rol oynar. Bu bölgelerde, çark ana akış bölgesinden boşluğa dolan debi radyal olarak içe doğru akar ve alt yarıçapta yer alan keçe debisi ile birleşip, soğutma ve basınçlandırma için türbin veya karter bölgesine yönlendirilir.

Boşluk boyunca akışın davranışı çeşitli parametrelere bağlıdır. Doğru basınç ve hız dağılımının elde edilmesi bu bağlamda önemli bir husustur. Rotor-stator boşluklarındaki akış yapısı, geometrik bir parametre olarak boşluk oranı (G), giriş teğetsel hız oranı (β_1), teğetsel Reynolds sayısı (Re_θ) ve türbülans akış parametresin (λ_t) 'den etkilenir.

Bu tez kapsamında, 1-B akış ağı çözücüsü kullanılarak analitik olarak hesaplanan ilgili akış parametreleri, boşluk geometrisinin 2-B eksenel simetrik olarak kurulmuş HAD modeline girdi yapmıştır. HAD çalışması da statik basınçların farklı yarıçap ve çalışma koşulları altında ölçüldüğü motor yer testleri ile karşılaştırılarak doğrulanmıştır.

Elde edilen sonuçlarda, HAD modeli ile ortaya konulan akış yapısının literatür ile uyumlu olduğu görülmüştür. Akışın rotor yüzeyinden radyal olarak yukarı doğru pompalandığı ve rotor için gerekli kütle debisinin de akışın radyal olarak içe doğru olan stator yüzeyinden sağlandığı görülmüştür. Teğetsel hız oranının bire eşit olduğu çapta ise, rotor yüzeyindeki radyal hızın sıfır olduğu ve akışın rotorun üzerinden de radyal olarak içeri doğru hareket etmeye başladığı ortaya konulmuştur. HAD modeli ile akış yapısının belirlenmesinin ardından, model ve testte görülen statik basınç dağılımları kıyaslanmıştır. Motor tam güçte çalışırken, boşluk geometrisi boyunca incelenmiş üç farklı yarıçap için, test verileri ile HAD modeli arasındaki farkların sırasıyla % 0.8, % 4.4 ve % 1.4 olduğu görülmüştür.

Anahtar Kelimeler: Gaz Türbinleri, İkincil Hava Sistemi, Rotor-stator Boşlukları, Kompresör Çark Arkası Akış, Radyal İçe Doğru Akış, Hesaplamalar Akışkanlar Dinamiği.

ACKNOWLEDGEMENTS

I would like to thank my supervisor Prof. Dr. İlyas Kandemir for his continued support throughout this project. I also would like to express my sincere gratitude and appreciation to Dr. Erinç Erdem whose guidance, support, mentorship, and encouragement has been invaluable throughout this study.

I am deeply indebted to Prof. Dr. Osman Saim Dinç for being the sole individual who believed in my potential during the selection interviews for the master's program.

Special thanks to my colleagues Mr. Abdulkadir Yalçınkaya and Mr. Ahmet Cihat Arıkan whose ongoing support allowed my works to go the extra mile. Also, I would like to thank Mrs. Elif Karakaş, my best friend, for her endless support.

To my dear parents, Serpil & Fikret Çevik, and my loving sister, Elif Çevik, your endless belief in my abilities has been the fuel propelling me forward. Your words of endorsement and motivation have been a source of inspiration during moments of doubt and uncertainty. Thank you for being my lovely family.

Finally, I would like to thank my precious and beloved husband, Mr. Ahmet Yasin Sedef, without whom I would not have been able to complete this research. Your steadfast commitment, sacrifice and perpetual support for my success have been profoundly appreciated. I am going to be eternally grateful for the love and strength you have bestowed upon me.

TABLE OF CONTENTS

	<u>Page</u>
ABSTRACT	iv
ÖZET	v
ACKNOWLEDGEMENTS	vi
TABLE OF CONTENTS	vii
LIST OF SYMBOLS AND ABBREVIATIONS	viii
LIST OF FIGURES	ix
LIST OF TABLES	xi
1. INTRODUCTION	1
1.1. Gas Turbine Engines	1
1.2. Secondary Air Systems	3
1.3. Free Disk Flow	4
1.4. Rotating Flow over a Stationary Disk	7
1.5. Windage	9
1.6. Axial Load on a Disk	11
2. LITERATURE SURVEY	14
2.1. Previous Studies on Rotor-stator Cavity Flow	14
2.1.1. Enclosed Rotor-stator Disk Systems	16
2.1.2. Rotor-stator Disk with Radial Outflow	18
2.1.3. Rotor-stator Disk with Radial Inflow	22
2.2. Aim of the Work	27
3. METHOD	29
3.1. Geometry and Boundary Conditions	29
3.2. Mesh Description	30
3.2.1. Mesh Independency	32
3.3. CFD Model	35
4. RESULTS	37
4.1. Comparison of 1D Solver vs CFD Results	37
4.2. CFD Analysis and Test Results	43
5. CONCLUSION	52
REFERENCES	54
BIOGRAPHY	57

LIST OF SYMBOLS AND ABBREVIATIONS

s	: Axial gap of the cavity (m)
δ	: Boundary layer thickness (m)
ρ	: Density ($\text{kg}\cdot\text{m}^{-3}$)
b	: Disk outer radius (m)
μ	: Dynamic viscosity ($\text{kg}\cdot\text{m}^{-1}\cdot\text{s}^{-1}$)
F	: Force (N)
ν	: Kinematic viscosity ($\text{m}^2\cdot\text{s}^{-1}$)
r	: Local radius (m)
$\dot{m}_o$	: Mass flow rate over rotor ($\text{kg}\cdot\text{s}^{-1}$)
$\dot{m}_s$	: Mass flow rate over stator ($\text{kg}\cdot\text{s}^{-1}$)
C_{Force}	: Non-dimensional axial load
C_W	: Non-dimensional mass flow rate
C_M	: Non-dimensional moment coefficient
Re_θ	: Rotational Reynolds number
Re'_θ	: Rotational Reynolds number (Modified)
ω	: Rotational speed of the fluid ($\text{rad}\cdot\text{s}^{-1}$)
Ω	: Rotational speed of the rotor ($\text{rad}\cdot\text{s}^{-1}$)
τ	: Shear stress ($\text{N}\cdot\text{m}^{-2}$)
c_p	: Specific heat at constant pressure ($\text{J}\cdot\text{kg}^{-1}\cdot\text{K}^{-1}$)
P_s	: Static pressure (kPa)
T_s	: Static temperature (K)
β	: Swirl ratio
G	: The ratio of axial gap between disks and rotor outer radius
x	: The ratio of local radius to axial gap
Γ	: Torque ($\text{N}\cdot\text{m}$)
P_t	: Total pressure (kPa)
T_t	: Total temperature (K)
λ_t	: Turbulent flow parameter
u_z	: Velocity in axial direction ($\text{m}\cdot\text{s}^{-1}$)
u_r	: Velocity in radial direction ($\text{m}\cdot\text{s}^{-1}$)
u_θ	: Velocity in tangential direction ($\text{m}\cdot\text{s}^{-1}$)
$\dot{W}$	: Windage power (W)
BC	: Boundary Condition
CFD	: Computational Fluid Dynamics
ISS	: Inter-Stage Seal
LDA	: Laser Doppler Anemometry
RANS	: Reynolds-Averaged Navier-Stokes
RPM	: Revolutions Per Minute
SAS	: Secondary Air Systems

LIST OF FIGURES

	<u>Page</u>
Figure 1.1: Thermodynamic cycle in an ideal gas turbine engine.	2
Figure 1.2: A typical secondary air system [1].	3
Figure 1.3: A rotating disk in an initially stationary fluid [2].	4
Figure 1.4: A rotating flow above a stationary disk [2].	8
Figure 1.5: The boundary layer associated with rotating flow on stationary disk [2].	8
Figure 1.6: Fluid element under consideration in a rotor-stator system.	11
Figure 1.7: Depiction for a centrifugal compressor cross-section [2].	11
Figure 2.1: Rotor-stator cavity [1].	14
Figure 2.2: Rotor-stator flow configurations (a) the free disk (b) rotating flow over a stationary disk (c) rotating flow over a rotating disk (d) impinging flow on a rotating disk (e-f) rotor-stator cavity with large and small clearance (g) enclosed rotor-stator cavity (h) to (j) rotor-stator cavity with superposed flows [6].	15
Figure 2.3: Velocity profiles for (a) to (c) Batchelor, (d) to (e) Stewartson flow [2].	16
Figure 2.4: Enclosed rotor-stator disk system.	16
Figure 2.5: Flow regimes for enclosed rotor-stator disk system [9].	17
Figure 2.6: The rotor-stator cavity with a superposed supply of air [2].	19
Figure 2.7: The rotor-stator cavity with a superposed radial inflow [2].	22
Figure 2.8: Flow structure in the cavity (a) large inflow rate (b) small inflow rate [2].	23
Figure 3.1: Impeller backface flow domain and applied boundary conditions.	30
Figure 3.2: Mesh structure of the impeller backface domain.	31
Figure 3.3: Non-dimensional static pressure change with mesh size.	33
Figure 3.4: Non-dimensional static pressure change at several radii.	33
Figure 3.5: Non-dimensional total temperature change with mesh size.	34
Figure 3.6: Non-dimensional total temperature change at several radii.	34
Figure 4.1: Non-dimensional static pressure comparison at several radii for Case-1.	38
Figure 4.2: Non-dimensional swirl ratio comparison at several radii for Case-1.	38
Figure 4.3: Non-dimensional temperature comparison at several radii for Case-1.	38
Figure 4.4: Non-dimensional static pressure comparison at several radii for Case-2.	39
Figure 4.5: Non-dimensional swirl ratio comparison at several radii for Case-2.	40
Figure 4.6: Non-dimensional total temperature comparison at given radii for Case-2.	40
Figure 4.7: Non-dimensional static pressure comparison at several radii for Case-3.	41
Figure 4.8: Non-dimensional swirl ratio comparison at several radii for Case-3.	41
Figure 4.9: Non-dimensional total temperature comparison at given radii for Case-3.	41
Figure 4.10: Representative stream function contour of flow domain.	44
Figure 4.11: Reference lines inserted into the fluid domain.	45
Figure 4.12: Swirl ratio vs axial distance at each line for Case-1.	45

Figure 4.13: Swirl ratio vs axial distance at each line for Case-2.	46
Figure 4.14: Swirl ratio vs axial distance at each line for Case-3.	47
Figure 4.15: Swirl ratio distribution in the cavity for three cases.	47
Figure 4.16: Non-dimensional radial velocity distribution in the cavity.	48
Figure 4.17: Non-dimensional total temperature distribution.	49
Figure 4.18: Non-dimensional total pressure distribution.	49
Figure 4.19: Non-dimensional static pressure distribution.	51

LIST OF TABLES

	<u>Page</u>
Table 3.1: Mesh properties of the impeller backface cavity.	32
Table 3.2: CFD modelling inputs.	36
Table 4.1: Properties of operating conditions.	37
Table 4.2: Non-dimensional rotor torques.	42
Table 4.3: Non-dimensional axial loads.	43
Table 4.4: Non-dimensional temperature rise due to windage.	50

1. INTRODUCTION

1.1. Gas Turbine Engines

Gas turbine engines are unquestionably one of the most important inventions of the 20th century, and they have changed our lives in many ways. Power generation in the aviation, defense and energy industries is primarily achieved through the gas turbine engines. For this reason, gas turbines stand out as the most widely used turbomachinery today.

The operation principle of the gas turbine is in fact embodies a Brayton cycle with the air being the working fluid. The main components of gas turbines include a compressor, combustion chamber and turbine. The atmospheric air flows through the compressor that brings it to higher pressure. Thermal energy is then added by ejecting fuel into the air and igniting it in a combustion chamber to generate a high enthalpy flow. The flow enters a turbine, producing a shaft work output which is used to drive the compressor and the remaining energy in the gases can be repurposed for external work, such as directly producing thrust in a turbojet engine, or rotating a second, independent turbine that can be connected to a fan, propeller, or electrical generator. The purpose of the gas turbine determines the design so that the most desirable division of energy between the thrust and the shaft work is achieved.

As shown in Figure 1.1, in an ideal gas turbine, the working fluid undergoes four thermodynamic processes: an isentropic compression, an isobaric (constant pressure) combustion, an isentropic expansion and last, isobaric heat rejection, all together making up the Brayton cycle. The ultimate step of the Brayton cycle, i.e., cooling of the working fluid, is excluded in real-life gas turbines since they are open systems and do not reuse the same air back at the beginning of the cycle.

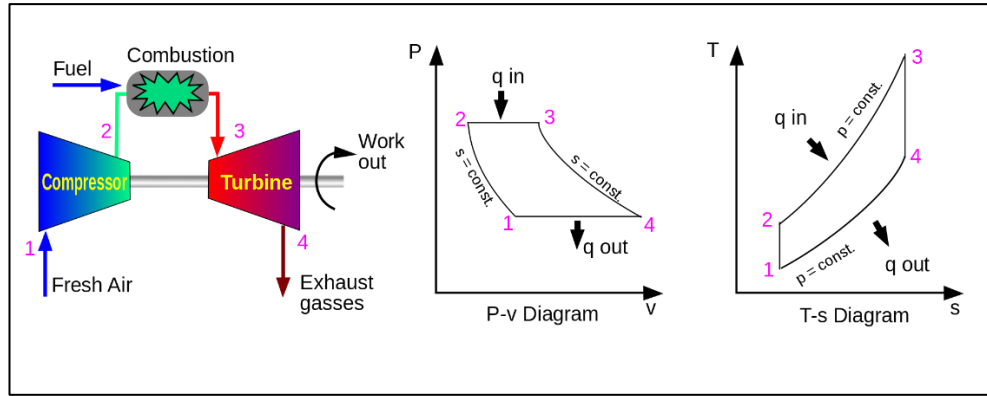


Figure 1.1: Thermodynamic cycle in an ideal gas turbine engine.

Gas turbine engines are widely used to power aircrafts because they are light and compact and have a high power-to-weight ratio. These engines employ diverse types of compressors to carry out effective air compression which is a vital stage in the cycle. The compressor types that can be utilized in these engines are axial and centrifugal compressors. Due to their streamlined design and reduced number of stages, centrifugal compressors have a lighter weight and require less space than axial compressors. This renders them well-suited for installations where space is limited. Their reliability, adaptability, and high-speed operation make them essential components in numerous systems.

In terms of compliance with the content of this thesis, the focus will be given to the radial compressors employed in the aviation industry.

1.2. Secondary Air Systems

To accomplish various tasks in gas turbine engines, a complicated internal flow network, known as the Secondary Air Systems (SAS) is employed. The tasks of the SAS can be categorized into three main headings. These include providing cooling air for necessary components, supplying air required to prevent hot gas ingestion into cavities, ensuring sealing in bearing areas, and managing axial load.

An exemplary SAS arrangement for a turbine section is illustrated in Figure 1.2. As depicted below, the high-pressure cooling air (dark color arrows) guided to first stage vane and blades for both internal and external cooling of that parts since they are directly subjected to the hot main flow air. The remaining stages do not have cooled vanes and blades since the main flow temperatures decrease as advances toward downstream. In addition to these cooled parts, the disks that the blades are mounted on are also cooled by the cooling air to prevent them from overheating during operation. Moreover, the low-pressure cooling air (white color arrows) is thrown overboard for the cooling of low-pressure turbine parts (if necessary) as well.

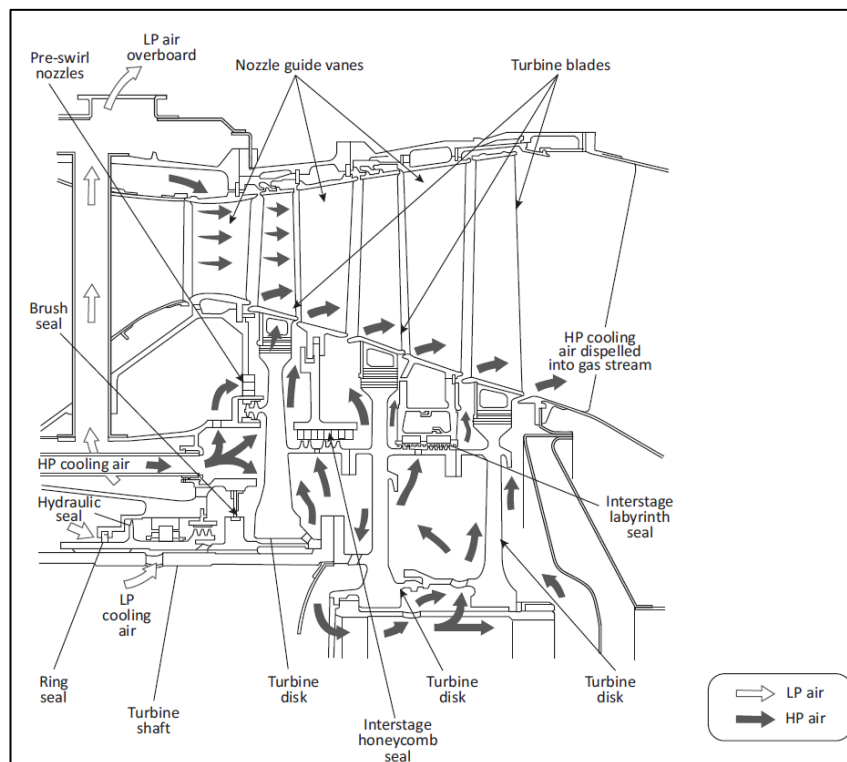


Figure 1.2: A typical secondary air system [1].

A reliable design of the secondary air system is critical for the performance and safety of gas turbine engines. Meeting the increasing demands of gas turbine engine design relies on accurately predicting the behavior of this system under various operating conditions. Secondary air systems should be designed with an understanding of a robustness philosophy that is less susceptible to changes throughout the engine's operating envelope while fulfilling their functions. Determining the effects of parameters influencing the system is necessary to achieve such a robust design.

1.3. Free Disk Flow

The free disk model refers to a disk of radius b rotating at Ω in a fluid of density ρ and dynamic viscosity μ . Figure 1.3 shows the flow associated with a free disk. In such a flow, the most noticeable characteristic is the ‘pumping’ effect of the rotating disk on the fluid such that there occurs a radial outflow over the disk and a corresponding balancing axial inflow. Due to no-slip condition at the disk surface, the shear between the disk and the fluid causes a radial outflow with a velocity u_r in the boundary layer. Considering the conservation of mass, a flow in the axial direction of velocity u_z must be entrained towards the disk surface in order to substitute the radial outflow.

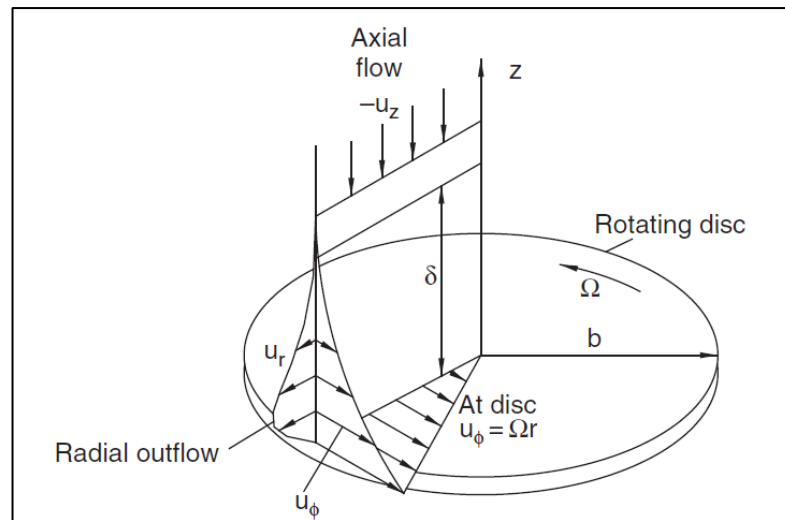


Figure 1.3: A rotating disk in an initially stationary fluid [2].

In order to characterize the flow in the boundary layer, the rotational Reynolds number is used with the definition:

$$Re_\theta = \frac{\rho \Omega b^2}{\mu} \quad (1.1)$$

Since the rotational Reynolds number is a function of radius, it is preferably defined with its local value, such that:

$$Re_{\theta,local} = x^2 Re_\theta = \left(\frac{r}{b}\right)^2 \frac{\rho \Omega b^2}{\mu} = \frac{\rho \Omega r^2}{\mu} \quad (1.2)$$

where x is the ratio of local radius r , to disk outer radius, b .

The flow near the axis of rotation tends to be laminar. However, for large values of angular velocity, the flow can become turbulent even at low radius, and a transition region will exist. Flow over disk considered turbulent for rotational Reynolds number larger than $3 \cdot 10^5$.

In 1921, Von Karman [3] studied Navier-Stokes equations for both laminar and turbulent flow over the free disk. The mass and momentum equations used for the free disk flow are given below in cylindrical coordinates.

$$\frac{\partial u_r}{\partial r} + \frac{u_r}{r} + \frac{1}{r} \frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} = 0 \quad (1.3)$$

$$\rho \left(\frac{\partial u_r}{\partial t} + u_r \frac{\partial u_r}{\partial r} + \frac{u_\theta}{r} \frac{\partial u_r}{\partial \theta} + u_z \frac{\partial u_r}{\partial z} - \frac{u_\theta^2}{r} \right) = -\frac{\partial P}{\partial r} \quad (1.4)$$

$$+ \mu \left(\frac{\partial^2 u_r}{\partial r^2} + \frac{1}{r} \frac{\partial u_r}{\partial r} - \frac{u_r}{r^2} + \frac{1}{r^2} \frac{\partial^2 u_r}{\partial \theta^2} + \frac{\partial^2 u_r}{\partial z^2} - \frac{2}{r^2} \frac{\partial u_\theta}{\partial \theta} \right) + F_r$$

$$\rho \left(\frac{\partial u_\theta}{\partial t} + u_r \frac{\partial u_\theta}{\partial r} + \frac{u_r u_\theta}{r} + \frac{u_\theta}{r} \frac{\partial u_\theta}{\partial \theta} + u_z \frac{\partial u_\theta}{\partial z} \right) = -\frac{1}{r} \frac{\partial P}{\partial \theta} \quad (1.5)$$

$$+ \mu \left(\frac{\partial^2 u_\theta}{\partial r^2} + \frac{1}{r} \frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r^2} + \frac{1}{r^2} \frac{\partial^2 u_\theta}{\partial \theta^2} + \frac{\partial^2 u_\theta}{\partial z^2} + \frac{2}{r^2} \frac{\partial u_r}{\partial \theta} \right) + F_\theta$$

$$\rho \left(\frac{\partial u_z}{\partial t} + u_r \frac{\partial u_z}{\partial r} + \frac{u_\theta}{r} \frac{\partial u_z}{\partial \theta} + u_z \frac{\partial u_z}{\partial z} \right) = - \frac{\partial P}{\partial z} \quad (1.6)$$

$$+ \mu \left(\frac{\partial^2 u_z}{\partial r^2} + \frac{1}{r} \frac{\partial u_z}{\partial r} + \frac{1}{r^2} \frac{\partial^2 u_z}{\partial \theta^2} + \frac{\partial^2 u_z}{\partial z^2} \right) + F_z$$

For these equations, assuming rotational symmetry and steady flow, the terms including $\frac{\partial}{\partial \theta}$ and $\frac{\partial}{\partial t}$ are neglected.

Transforming partial differential equations into ordinary differential equations and solving numerically, following equations for laminar free disk flow are obtained as stated by Childs [2].

$$\delta \approx 5.5 \sqrt{\frac{\nu}{\Omega}} \quad (1.7)$$

which gives the boundary layer thickness and is defined as the axial distance where the tangential velocity of the flow is $0.01\Omega r$. In Eq. 1.7, ν is the kinematic viscosity of the fluid.

The mass flow rate pumped over the rotating disk in a laminar flow is:

$$\dot{m}_o = -2\pi\rho \int_0^r r[u_z]_{z \rightarrow \infty} dr = 2.779\rho b^2 \sqrt{\Omega\nu} \quad (1.8)$$

which can be written in non-dimensional form,

$$C_w = \frac{\dot{m}_o}{\mu b} = 2.779 Re_\theta^{0.5} \quad (1.9)$$

Similarly, the moment on the rotor surface can be expressed in non-dimensional form,

$$C_m = 1.935 Re_\theta^{-0.5} \quad (1.10)$$

It should be noted that, for the laminar flow the boundary layer thickness is not a function of radius whereas both non-dimensional mass and moment coefficients are functions of the rotational Reynolds number.

For the turbulent flow over the disk, applying the similar solution techniques and using 1/7th power law for velocity profile, boundary layer thickness can be evaluated as,

$$\delta = 0.526r \left[\left(\frac{r}{b} \right)^2 Re_\theta \right]^{-0.2} \quad (1.11)$$

which is a function of radius in contrast to its laminar version.

The non-dimensional mass flow rate and moment coefficients take the form of,

$$C_w = 0.219 Re_\theta^{0.8} \quad (1.12)$$

$$C_m = 0.073 Re_\theta^{-0.2} \quad (1.13)$$

1.4. Rotating Flow over a Stationary Disk

The case of a rotating fluid flowing over a stationary disk is also encountered in various engineering systems. As in laminar free disk flow, the mass and momentum equations (Eq.1.3 - Eq.1.6) are applied to flows over stationary disk provided that boundary conditions are properly managed.

A schematic view describing such a flow is given in Figure 1.4.

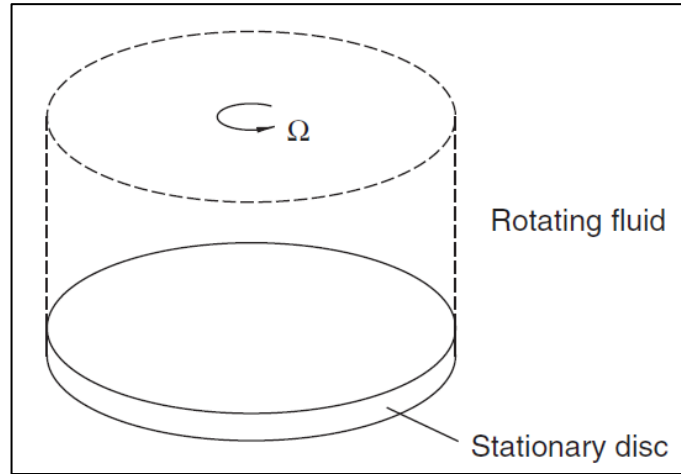


Figure 1.4: A rotating flow above a stationary disk [2].

This type of flow was studied by Bödewadt [4] in 1940, and afterwards by Schlichting [5] in 1979. The fluid over the disk rotates with a constant angular speed of Ω and follows a solid body rotation. Since the fluid just over the surface has less centripetal pressure difference than the surrounding fluid due to no-slip condition, it penetrates towards the core of the surface. Conservation of angular momentum dictates the increase in tangential velocity during this radially inward motion resulting a reversal of flow away back from the core. On the contrary to free disk flow, the fluid moves axially outward from the surface. The depiction of this behavior is given in Figure 1.5.

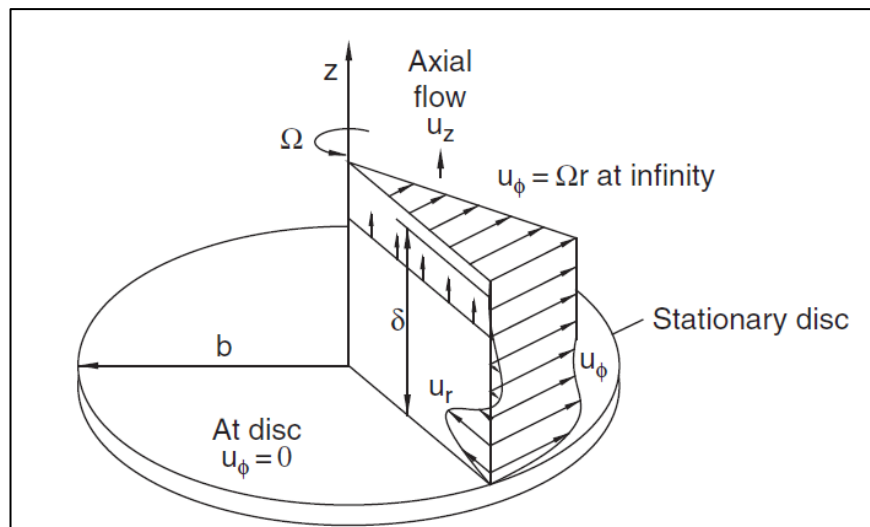


Figure 1.5: The boundary layer associated with rotating flow on stationary disk [2].

If the boundary layer thickness is set to be the axial distance away from the disk where the tangential velocity is 0.99 of the free-stream, then the approximate thickness becomes,

$$\delta \approx 8\sqrt{\frac{\nu}{\Omega}} \quad (1.14)$$

When compared to Eq. 1.7, the boundary layer thickness is almost 50% larger than the one in rotating disk subjected to a stationary fluid. It is apparent that, in the case of a free disk, the axial movement of fluid towards the rotating disk tends to restrain the boundary layer and limit its thickness whereas for stationary disk, the fluid helps to hoist the flow away from the surface resulting in a thicker boundary layer.

In order to evaluate the net radial flow leaving the surface in axially-outward direction, the following non-dimensional equation is executed.

$$C_w = -4.239Re_\theta^{0.5} \quad (1.15)$$

1.5. Windage

A unique term generally addressed in rotating systems is “windage”. The term is also used interchangeably with “heat pick-up”. As the name refers, it is a phenomenon that results in a temperature increase or namely heat pick-up in the fluid. Intuitively, the change in the fluid total temperature inside a control volume is due to heat or work transfer from system boundaries. In a rotor-stator system, however, the heat may be transferred from both rotor and stator surfaces whereas the work transfer is of rotor surface only. If the rotational speed of the rotor is higher than that of fluid, the rotor drags the fluid to revolve or the vice versa provided that the fluid is rotating with a greater tangential velocity than the rotor surface. Besides, the stator surface forces the rotating core fluid to come to a halt by applying torque on it. It is in fact due to no-slip condition on the interface between the fluid and the stator surface. In an adiabatic system, the slowdown of fluid yields the transformation of its external flow energy to internal energy without altering the fluid total temperature.

For the rotor side, the rotational work input from its surface will cause an increase in the energy of the fluid element and thus in the total temperature. Accordingly, this energy transfer due to tangential speed difference, i.e. the rotor power input into the fluid, is called windage. In order to evaluate this power input, the conservation of angular momentum and energy equations shall be invoked. The net torque applied on the fluid element, shown in Figure 1.6, can be found via the angular momentum balance equation, such that,

$$\Gamma_o - \Gamma_s = \dot{m}u_{\theta,2}r_2 - \dot{m}u_{\theta,1}r_1 = \dot{m}(\beta_2r_2^2 - \beta_1r_1^2)\Omega \quad (1.16)$$

where Γ_o and Γ_s are the rotor and stator torques, respectively. They can be determined using empirical or semi-empirical correlations. Note that, β is called swirl ratio denoting the ratio of fluid tangential velocity to the rotor tangential velocity.

Windage power or heat pick-up is then computed as,

$$\dot{W} = \Gamma_o\Omega \quad (1.17)$$

Therefore, using the energy balance equation, the total temperature rise due to windage yields,

$$\Delta T_{t,windage} = T_{t,2} - T_{t,1} = \frac{\dot{W}}{\dot{m}c_p} \quad (1.18)$$

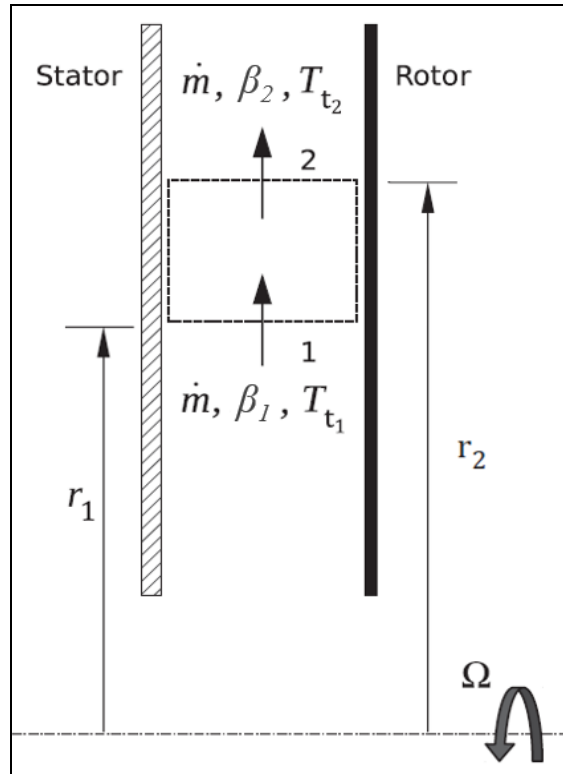


Figure 1.6: Fluid element under consideration in a rotor-stator system.

1.6. Axial Load on a Disk

A regular necessity in rotating machinery systems is the determination of the net axial load, i.e. thrust, on the rotating part, such as an impeller, to introduce a suitable bearing solution. A schematic in Figure 1.7 demonstrates the cross-section of a centrifugal compressor.

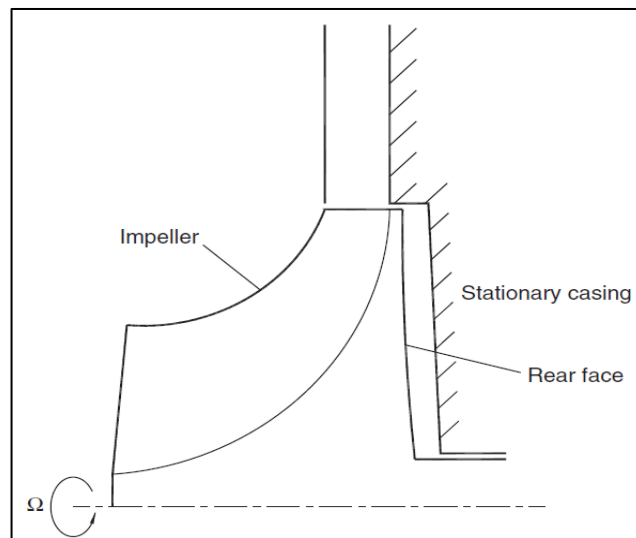


Figure 1.7: Depiction for a centrifugal compressor cross-section [2].

A thin boundary layer is to be formed over both rotor and stator surfaces in such a cavity. The inviscid core region between boundary layers, which rotates at a certain fraction of the impeller speed, Ω , can be treated as axisymmetric steady flow and the static pressure change with respect to radius in this region follows,

$$\frac{\partial P}{\partial r} = \rho \frac{u_\theta^2}{r} - \rho u_r \frac{\partial u_r}{\partial r} - \rho u_z \frac{\partial u_r}{\partial z} + \mu \left(\nabla^2 u_r - \frac{u_r}{r^2} \right) \quad (1.19)$$

Considering that the core rotation drives the radial pressure gradient and the axial velocity is mainly zero in a rotor-stator cavity, both u_r and u_z terms will be omitted from the expression. Moreover, with the negligible viscous terms outside the boundary layers, equation reduces to,

$$\frac{\partial P}{\partial r} = \rho \frac{u_\theta^2}{r} \quad (1.20)$$

or

$$\frac{\partial P}{\partial r} = \rho \omega^2 r \quad (1.21)$$

where ω refers to the angular velocity for the fluid core. Then, the pressure field and hence the thrust over the cavity can be determined by solving Eq. 1.21 as far as the relationship between pressure and density is apparent.

If the fluid is assumed to be incompressible, i.e. the density is constant, then,

$$\int_{r_1}^r dP = \rho \omega^2 \int_{r_1}^r r dr \quad (1.22)$$

The unknown pressure at any radius in the cavity equals to,

$$P = P_1 + \frac{\rho\omega^2}{2}(r^2 - r_1^2) = P_1 + \frac{\rho\beta^2\Omega^2}{2}(r^2 - r_1^2) \quad (1.23)$$

or alternatively,

$$P = P_2 - \frac{\rho\beta^2\Omega^2}{2}(r_2^2 - r^2) \quad (1.24)$$

After determining the static pressure distribution, the overall force, namely thrust can be evaluated for a disk of inner radius a, and outer radius b.

$$F = \int_a^b \left(P_2 - \frac{\rho\beta^2\Omega^2}{2}(b^2 - r^2) \right) 2\pi r dr \quad (1.25)$$

which yields,

$$F = \pi P_2(b^2 - a^2) - \frac{\pi\rho\beta^2\Omega^2 b^2}{2}(b^2 - a^2) + \frac{\pi\rho\beta^2\Omega^2}{4}(b^4 - a^4) \quad (1.26)$$

Notice that the axial load-acting rotor section is considered as rotor's axial plane projection area.

2. LITERATURE SURVEY

2.1. Previous Studies on Rotor-stator Cavity Flow

A rotor-stator system comprises a rotating disk and an adjacent stationary disk, which are coaxially aligned with an axial gap between the two as shown in Figure 2.1. The two disks form a cavity between them that is known as the rotor-stator cavity. A wide range of engineering systems including turbine or compressor disks embody such rotating flow structures.

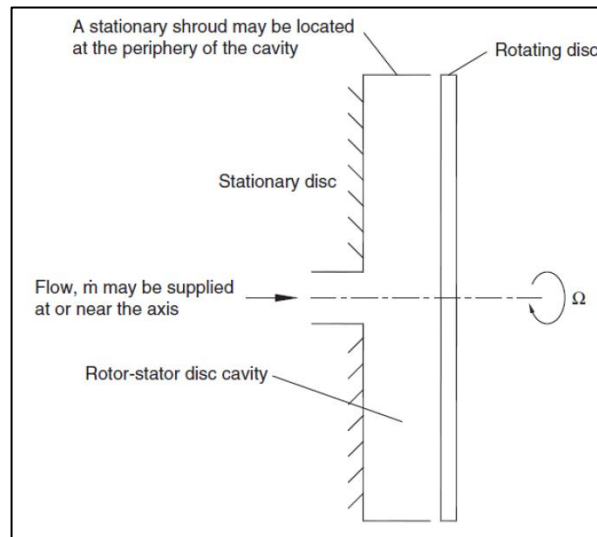


Figure 2.1: Rotor-stator cavity [1].

Rotor-stator disc cavity flow characteristics vary with the proximity of the stator and rotor, the presence of a stationary or rotating shroud, and the introduction of any exterior flow, i.e., imposed flow, to the cavity. Selected configurations for the rotor-stator systems are given in Figure 2.2.

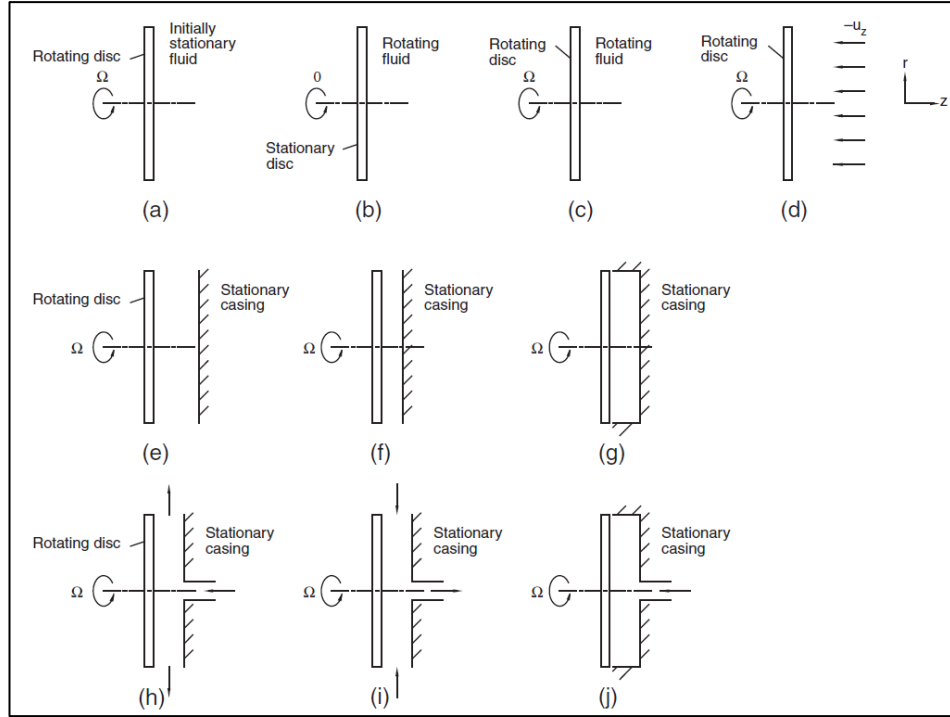


Figure 2.2: Rotor-stator flow configurations (a) the free disk (b) rotating flow over a stationary disk (c) rotating flow over a rotating disk (d) impinging flow on a rotating disk (e-f) rotor-stator cavity with large and small clearance (g) enclosed rotor-stator cavity (h) to (j) rotor-stator cavity with superposed flows [6].

Different approaches were suggested in the literature about the flow structure in rotor-stator cavities before the advent of various experimental data and numerical solutions. Batchelor [7] predicted that there should be a core of fluid rotating at a constant angular velocity between rotor and stator parts. This suggestion dictates separate boundary layers on both rotor and stator surfaces. To supply the pumping need for the rotor, fluid is drawn from the rotating core region and this core region is fed by an efflux of the stator boundary layer. On the other hand, Stewartson [8] proposed that there occurs no rotating core region inside the cavity as in the case of a free disk flow. At the distances away from the rotor boundary layer the fluid attains zero rotational velocity. This dilemma remained a controversial issue at those times. The characteristic tangential, radial, and axial velocity profiles suggested by Batchelor [7] and Stewartson [8] are illustrated in Figure 2.3.

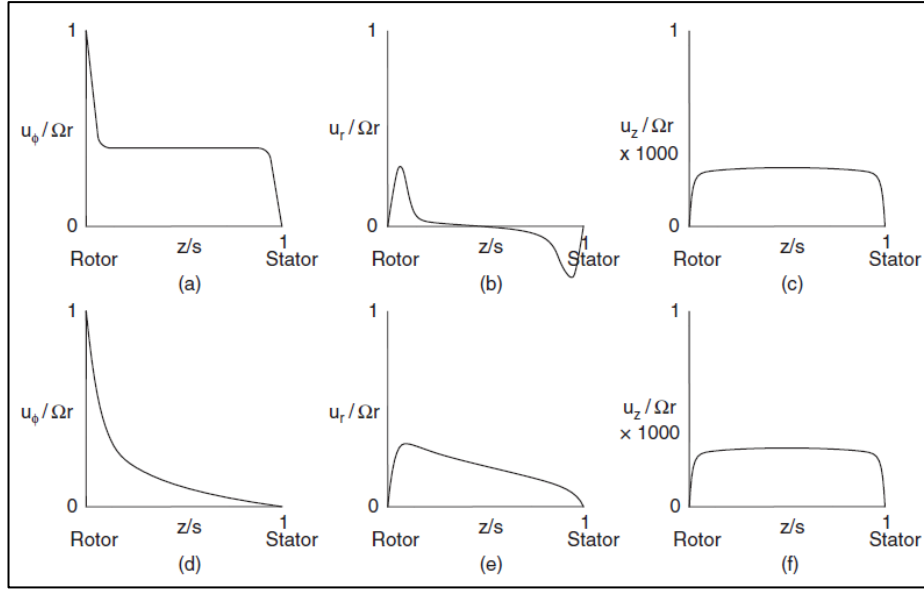


Figure 2.3: Velocity profiles for (a) to (c) Batchelor, (d) to (e) Stewartson flow [2].

It is later understood that both velocity profiles can be encountered in rotor-stator systems depending on the existence of any superposed flow supplied to the cavity.

2.1.1. Enclosed Rotor-stator Disk Systems

In enclosed rotor-stator disk systems, the fluid moves radially outward in the boundary layer of the rotor due to pumping effect and axially towards to stator over the cylindrical outer shroud. Since no flow is imposed to the cavity of enclosed system, there occurs an inviscid fluid core region rotating at a certain percentage of the rotor speed. An example cavity is shown in Figure 2.4.

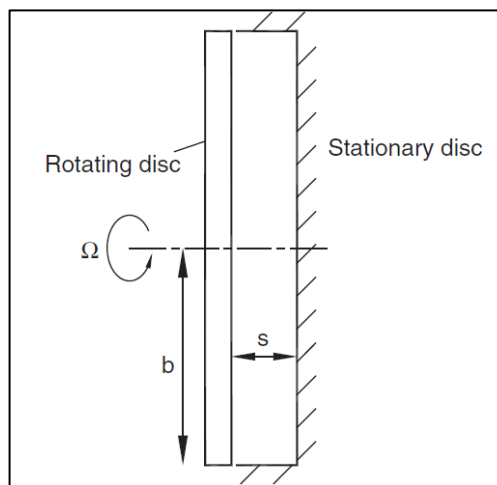


Figure 2.4: Enclosed rotor-stator disk system.

In general, the flow in this type of systems is best described with Batchelor [7] model. The fluid in between the rotor and stator boundary layers rotates at a fraction of rotor angular velocity which can be expressed with a parameter called swirl ratio.

$$\beta = \frac{\omega}{\Omega} \quad (2.1)$$

where ω denotes the angular speed of inviscid core region and Ω is the angular speed of the rotating disk.

The gap between the rotor and stator disks is another important parameter that affects the flow characteristics. Therefore, the gap ratio which is the ratio of axial gap between disks and outer radius of the rotor is given by,

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

In an experimental study performed by Daily and Nece [9], it was found that the flow characteristics depends on the nature of the flow as well as the geometric constrains. In this study, four different regimes were introduced to identify flow. These are explained below and depicted in Figure 2.5.

- Regime 1: small clearance, merged laminar boundary layers.
- Regime 2: large clearance, separate laminar boundary layers
- Regime 3: small clearance, merged turbulent boundary layers.
- Regime 4: large clearance, separate turbulent boundary layers

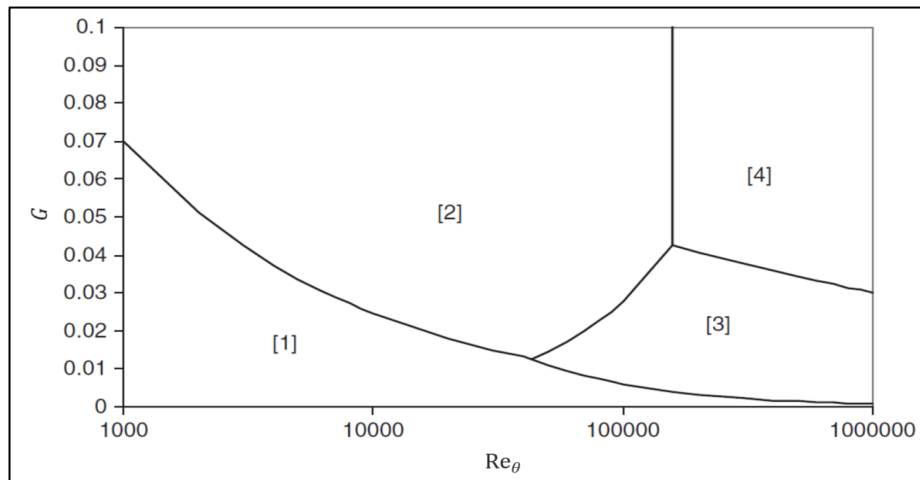


Figure 2.5: Flow regimes for enclosed rotor-stator disk system [9].

They have investigated the moment coefficient of these regimes and established relationships for the rotor disk in each regime as expressed below.

$$C_m = \frac{\pi}{GRe_\theta} \quad (2.3)$$

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

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

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

Eq. 2.3 to Eq. 2.6 correspond to the moment coefficients of Regime 1 to 4, respectively.

The study covers flows having rotational Re numbers between 10^3 and 10^5 with a gap ratio, G of 0.0127 to 0.217.

Note that, in general, Regime 4 suits well to the cavity flows in gas turbine engines, due to inherent turbulent flow structures and high rotational speeds encountered in these systems.

2.1.2. Rotor-stator Disk with Radial Outflow

For the case in which there is an additional flow imposed to the rotor-stator disk space at or near the rotation axis, the flow structure inside the cavity alters. The supply of fluid to the cavity modifies the structure and hence the core rotation between disks, resulting in a net radial outward flow throughout the system. Depending on the amount of flow rate imposed to the cavity, the core rotation is to be totally suppressed, and the nature of the rotating system will tend to attain the Stewartson [8] flow structure. Therefore, the tangential velocity in the boundary layer of the rotor varies from the rotor speed to zero away from the rotating disk. In Figure 2.6, such a system has been illustrated.

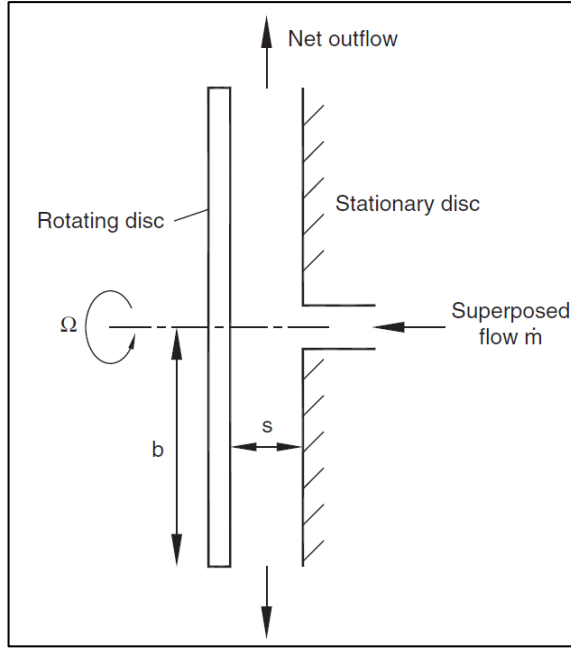


Figure 2.6: The rotor-stator cavity with a superposed supply of air [2].

Generally, the supply of fluid to the cavity increases the moment by reducing the core rotation. In order to prevent the ingress over the shroud, the amount of fluid supplied to the system is important in such a way that the pumping demand of the rotor is satisfied.

In an experimental study by Bayley and Owen [10], the effects of radial outflow in a rotor-stator cavity have been investigated. It is observed that, reducing the amount of imposed flow rate or increasing the rotational Reynold number advances the tangential velocity inside the cavity. Moreover, as the clearance between the disks was increased the inviscid core rotation rate decreased and ultimately disappeared. When the superposed flow rate was excessively increased (more than the pumping need of the rotor) the moment coefficient on the rotor ascended.

Assuming an inviscid core rotating between rotor-stator and using $1/7^{\text{th}}$ power law velocity profile to model boundary layers for both rotor and stator side, the following expressions for boundary layer thicknesses and mass flow rates are obtained by Owen and Rogers [11]. The boundary layer formulation for the rotor and stator surfaces,

$$\left(\frac{\delta}{r}\right)_o = \gamma(x^2 Re_\theta)^{-0.2}(1 - \beta)^{0.6} \quad (2.7)$$

$$\left(\frac{\delta}{r}\right)_s = \gamma(x^2 Re'_\theta)^{-0.2} \quad (2.8)$$

where subscripts o and s denote the rotor and stator sides, respectively. The modified rotational Reynolds number, used for the stator side expressions, is:

$$Re'_\theta = \frac{\rho \omega b^2}{\mu} \quad (2.9)$$

Then, the mass flow rates take the forms of,

$$\frac{\dot{m}_o}{\mu b} = x \frac{49\pi}{60} \alpha_o \gamma_o (x^2 Re_\theta)^{0.8} (1 - \beta)^{1.6} \quad (2.10)$$

$$\frac{\dot{m}_s}{\mu b} = -x \frac{49\pi}{60} \alpha_s \gamma_s (x^2 Re'_\theta)^{0.8} \quad (2.11)$$

where, for the rotor side,

$$\alpha_o \gamma_o = 0.0852(1 - 0.51\beta) \quad (2.12)$$

and β varies between 0 and 0.5.

For the stator side, from Ekman boundary layer solution,

$$\alpha_s = 0.5533 \quad (2.13)$$

$$\gamma_s = 0.0983 \quad (2.14)$$

Ultimately, the mass flow rates for the rotor and stator sides become,

$$\frac{\dot{m}_o}{\mu b} = x \frac{49\pi}{60} [0.0852(1 - 0.51\beta)] (x^2 Re_\theta)^{0.8} (1 - \beta)^{1.6} \quad (2.15)$$

$$\frac{\dot{m}_s}{\mu b} = -x \frac{49\pi}{60} [(0.5533)(0.0983)] (x^2 Re_\theta)^{0.8} (\beta)^{0.8} \quad (2.16)$$

To satisfy the conservation of mass, the superposed flow rate should be the sum of rotor and stator flow rates, therefore,

$$\frac{\dot{m}_o}{\mu b} + \frac{\dot{m}_s}{\mu b} = \frac{\dot{m}_{superposed}}{\mu b} = C_w \quad (2.17)$$

Substituting the mass flow rates into this expression, the superposed flow rate yields,

$$(1 - \beta)^{1.6} (1 - 0.51\beta) - 0.638\beta^{0.8} = 4.57\lambda_t \left(\frac{r}{b}\right)^{2.6} \quad (2.18)$$

where,

$$\lambda_t = C_w Re_\theta^{-0.8} \quad (2.19)$$

and called turbulent flow parameter.

It is seen that, to suppress the rotation of the inviscid core region, i.e., setting $\beta=0$, the above expression transforms into,

$$C_w = 0.219 Re_\theta^{0.8} \left(\frac{r}{b}\right)^{2.6} \quad (2.20)$$

This non-dimensional mass flow rate is nothing but the entrainment flow rate for a free disk in a turbulent flow regime as predicted by Von Karman [3] modelling approach. Moreover, if there is no superposed flow, i.e., C_w and hence λ_t is zero, the solution gives a core rotation value of $\beta=0.4258$ for an enclosed system.

As can be deduced from previous formulations, if less amount of air is imposed into the cavity than the pumping need or alternatively if the rotational speed of the rotor is increased, i.e. the turbulent flow parameter decreases in both cases, more fluid penetrates over the stator to fulfill the disk pumping need. This in fact advances the

tangential velocity of the core and results in an increase in the radial pressure gradient between the core and the external environment.

2.1.3. Rotor-stator Disk with Radial Inflow

When there is somehow generated a large pressure gradient between the outer radius and the axis of the rotor-stator cavity, the fluid will be drawn inward to the space. This flow can therefore suppress the outflow that is inherently driven by disk pumping.

Bayley and Conway [12] investigated the radial inflow case in a rotor-stator system as depicted in Figure 2.7.

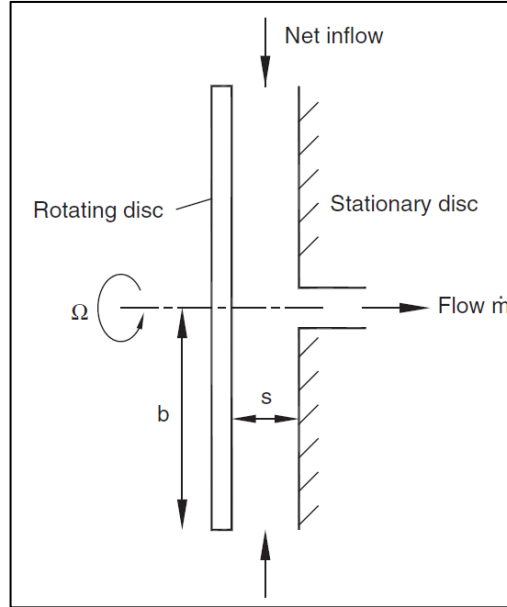


Figure 2.7: The rotor-stator cavity with a superposed radial inflow [2].

They have revealed that increasing the rotational Reynolds number or the radial inflow rate ascended the moment coefficient. However, the gap between rotor and stator had the reverse effect. In order to determine the minimum radial inflow rate to suppress the outflow over the rotor due to pumping was estimated by an integral solution of boundary layer equations.

The solution gives,

$$\frac{C_w}{Re_\theta G} = 2\pi \quad (2.21)$$

Nonetheless, it was later shown that there can be no radial outflow at any point in the cavity when the left-hand side of the equation is appreciably lower than 2π . Therefore, it was suggested to estimate the necessary inflow with,

$$\frac{C_w}{Re_\theta G} = 1 \quad (2.22)$$

In another study performed by Pincombe [13] both for laminar and turbulent inflows flow visualizations revealed following flow structures as shown in Figure 2.8.

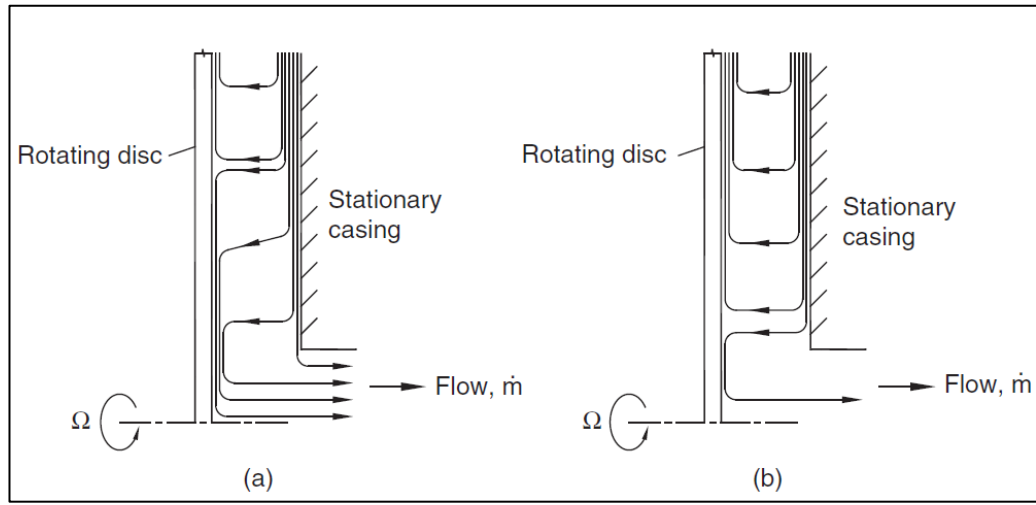


Figure 2.8: Flow structure in the cavity (a) large inflow rate (b) small inflow rate [2].

They have identified a radial stagnation point on the rotor disk and observed that the fluid moves radially inward underneath this radius over the rotating disk with a tangential velocity greater than the local disk speed. They have described the flow field in the cavity via three different regimes.

- Superposed radial inflow dominated regime where the stagnation point was at the rotor tip,
- Rotation dominated regime where the stagnation point was at the rotor hub,
- An intermediate regime where the flow exhibits both of the previously mentioned regimes.

In addition, it was seen that the inlet swirl of the fluid has significant effects around the rotor tip whereas these effects disappeared toward its hub.

The moment coefficient in a rotor-stator cavity with superposed inflow for different gap ratios was studied by Daily et al. [14] and the following correlation was suggested.

$$C_m = 0.51G^{0.1}Re_\theta^{-0.2}(1 + 13.9\beta^*\lambda_t G^{-1/8}) \quad (2.23)$$

It is valid for rotational Reynolds numbers from $2 \cdot 10^6$ to $1 \cdot 10^7$ and for turbulent flow parameters in between 0-0.06.

Apart from aforementioned contents, there are various numerical and/or experimental studies available in literature.

Pincombe [13] executed Laser Doppler Anemometry measurements for the flow visualization of rotor-stator cavity having radially-inward superposed flow. He measured the velocity profiles in the cavity for a broad range of rotational Reynolds number and inlet swirl ratio. It was shown that the value of inlet swirl had the main impact on the swirl ratio of the core region.

Iacovides and Theofanopoulos [15] examined the turbulence modeling approaches for the numerical computation of flow through rotating cavities. The flows in co-rotating disk with radially inward flow and enclosed disk flows with and without throughflow were studied. They concluded that the modelling of near wall regions was the critical factor for the correct prediction of flows in the boundary layer. It is understood that the zonal modelling approach adopted in their study was a cost-effective method of providing a convenient transport-based turbulence model. Results obtained using this method showed good agreement with the experimental data.

In another study performed by Hart and Turner [16], the rotor-stator cavity behind a gas turbine centrifugal compressor was studied. CFD models were set up to simulate test conditions and validate computational methods. The tests covered the rotational Reynolds number ranges of $0.5 \cdot 10^6$ to $1.2 \cdot 10^6$ and turbulent flow parameter ranges of 0 to 0.05 inflow or outflow. The rig was constructed such that it can provide zero or unity inlet swirl ratio. For the inflow cases the obtained test data agreed well with the literature data and constructed CFD models showed good numerical agreement with the experiments excluding the rotor tip region for small gap ratios. Following the validation of CFD models, a CFD code that can be used to estimate pressure and velocity distribution data for typical operation conditions of gas turbines was

introduced in another study by Hart and Turner [17]. A set of data generated from this code was implemented into charts. It is claimed that using these charts was a cost-effective approach and minimized the required time in the design phase of an engine.

Randriamampianina et al. [18] performed numerical studies for flows between rotor-stator shrouded disks including laminar, transitional, and turbulent flow regimes. To calculate turbulent flow regimes, second order transport modelling was used in their numerical code. It is understood that to delineate precisely the turbulent and re-laminarization regions within the cavity, the improved Reynolds stress transport models was decisive over the standard $k-\epsilon$ models. The numerical results suited well with the experimental data.

Debuchy et al. [19] investigated the superimposed radial inflow inside a rotor-stator cavity with a small clearance. It was explained that large rotational Reynolds numbers and small Ekman and Rossby numbers characterized such flows. They performed theoretical analyses and compared them with experimental data. The results showed that the integral relations were approximately valid. It was also noted that to comprehend such complex flows, appropriate turbulence modelling must be included in the calculations.

Poncet et al. [20] studied the rotating fluid structure in a rotor-stator cavity with a radially inward flow both analytically, numerically, and experimentally. The inward throughflow was either pre-rotating or non-rotating. They have seen that when there was a weak throughflow, the structure around the periphery remains the same as in the case of no inflow. However, for a strong throughflow they have revealed that the Ekman boundary layer on the rotor side becomes centripetal as the core rotates faster than the disk. In the study conducted in 2005, Poncet et al. [21] investigated similar flow structure with numerical techniques as well. They have compared the numerical results with the experimental measurements and pointed out that the Reynolds stress model agreed well with the turbulent test data as well as the mean flow results. Later on, Poncet and Schiestel [22] implemented heat transfer considerations into their previous studies. They have compared their numerical results with the literature and suggested an empirical correlation to interpret the average Nusselt number as a function of Reynolds and Prandtl numbers as well as the coolant flow rate.

Haddadi and Poncet [23] have investigated the turbulent flow inside a rotor-stator cavity having through- and no-throughflow. They have compared RSM model solutions with $k-\varepsilon$ model solutions as well as the Laser Doppler Anemometry (LDA) measurements. The presented RSM model provided a good agreement with the velocity measurements.

Da Soghe et al. [24] presented an in-house 1D cavity solver which estimated the inner and outer flow extraction, drags on the rotor-stator as well as leakages and rim seal effects on the cavity. They calibrated their solver by means of extensive CFD analyses. The authors developed a new set of equations for the friction coefficients by means of CFD analyses and implemented them into the in-house code. Even though their results were promising, they have stated that an improvement in the modelling was needed especially to estimate pumping flow rate over the rotor disk. In another study of Da Soghe et al. [25], various turbulence modelling techniques were studied in rotor-stator systems. The standard $k-\varepsilon$ and $k-\omega$ SST as well as the Reynolds stress model were implemented in their analyses. Turbulence models based on low Reynolds approach showed better results for the mean and turbulent fields particularly within the thin boundary layers. They stated that $k-\omega$ SST approach provided the best trade-off between computational cost and accuracy. Moreover, for the case of radially outward flow, the $k-\omega$ SST with a low Reynolds approach and Reynolds stress model established compatible results and predicted the Batchelor to Stewartson transition quite well.

Will and Benra [26] investigated the rotor-stator cavity with radially inward throughflow analytically, numerically, and experimentally. The authors compared their analytical solutions with the CFD data and the experimental results. For the numerical modelling, it was stated that the proper choice of the mesh density and the turbulence model were essential to obtain comparable outcomes.

Shi et al. [27] studied dual inlet rotor-stator disk cavity flow structure. The influence of inflow distributions on the maintaining and exporting characteristics for swirling flow was investigated by numerical simulations. $k-\omega$ SST turbulence model used in the numerical analysis was validated with available experimental data. The focus was given to the interaction between the two inflows. It was revealed that with different inflow distribution ratios the size of the two vortices in the cavity changed.

Erdem et al. [28] investigated the centrifugal compressor impeller cavity flow structure using numerical techniques and 1-D flow network solver. The simulations were conducted with three different rotational speeds of the disk. It was stated that the flow field inside the cavity was mainly driven by the rotor and stator torques whereas at the lower radii the impeller leakage flow gained tangential velocity while mixing with the seal leakage flow.

In a study performed by Sertçakan [29], the impeller backface cavity flow characteristics are examined. 1-D flow network and 2-D CFD analyses are conducted. The numerical results were compared with rig tests in the literature as well as with the engine ground tests performed in a test bench. It is declared that the turbulent flow parameter, λ_t , is the major parameter in cavity analyses. The effects of gap ratio and inlet swirl ratio on the flow characteristics are found to be relatively limited.

2.2. Aim of the Work

In gas turbine engines utilizing radial compressors are commonly encountered in the aviation industry. Embodying a centrifugal compressor in the engine requires the proper management and comprehension of flow structure in the backface cavity of the compressor impeller. This flow structure influences the performance and the efficiency as well as the axial load management of the engine. It is also paramount for the secondary air system in terms of impeller disk metal temperature and parasitic load control like windage.

The rotor-stator field behind the impeller may have various configuration depending on the design. Generally, the mainstream flowing over the impeller blades penetrates the backface cavity at the outer radius which is then may be mixed with another flow at the lower radius and directed to the pertinent parts in the engine. This type of flow structure is named rotor-stator disk cavity with radial inflow.

There are numerous analytical, numerical, and experimental data available in literature for such radial inflow configurations. Even though the flow structure in a rotor stator cavity with radial outflow and inflow has been extensively studied, most of these works concentrated on simple disk geometry. Besides that, the engine configurations used in the industry comprise quite complex geometries.

This study aims to provide a better understanding for the flow behavior in a rotor-stator cavity over the first stage impeller backface of a centrifugal compressor in a gas turbine engine. A comprehensive 2-D axisymmetric CFD analysis has been performed for three different operating conditions of the engine. The factors driving the flow structure inside the cavity have been demonstrated. Cavity pressure distribution, swirl ratio and total temperature are established. The outcomes of CFD model have been compared with the measurements collected during ground tests and numerical approach is validated.

3. METHOD

Numerical analysis by means of CFD is an effective way utilized to characterize flow structure in rotor-stator cavities during the design phase of an engine. By implementing the correct physics and boundary conditions into the analysis together with the appropriately delineated and meshed geometry, CFD analysis can provide results closer to the real case scenarios. This enables low-cost solutions when compared to experimental studies.

In this part of the study, the applied method for the CFD analysis and the pertinent boundary conditions are introduced in detail. The mesh structure which is vital for the correct manifestation of the system is explained.

3.1. Geometry and Boundary Conditions

The CFD model is constructed with the 2-D axisymmetric depiction of the engine geometry. The flow domain between the rotor and the stator takes the form as shown in Figure 3.1.

Both the rotating and the stationary walls are assumed to be adiabatic. At the outer radius, there is a leakage flow coming from the impeller trailing edge and at the lower radius, there is a seal flow discharging through the Inter-Stage Seal (ISS) into the cavity.

The seal configuration is included in the CFD model as well. There is a honeycomb structure located on the stationary side of the ISS in the original configuration which yields a reduction effect on the seal mass flow rate. The CFD model does not comprise this honeycomb structure, yet the effective seal clearance is calculated and reflected to the model considering the effect of honeycomb. The outlet of the cavity which indeed consists of a set of circular holes is positioned at the bottom. To facilitate the CFD model, the total cross-sectional area of the outlet holes is calculated and represented as a circumferential slot on the 2D axisymmetric description.

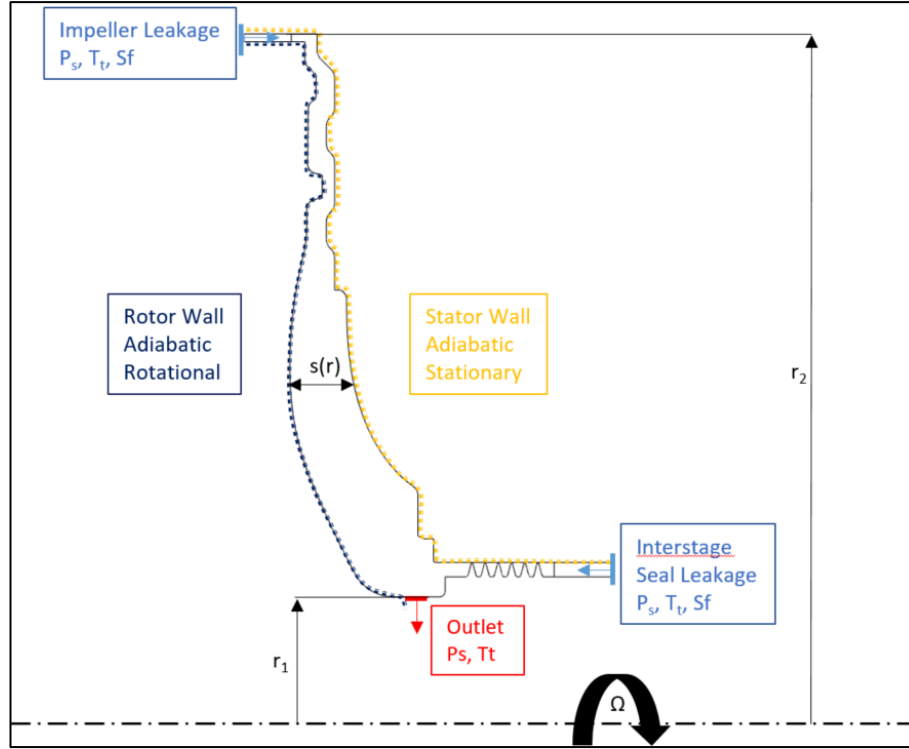


Figure 3.1: Impeller backface flow domain and applied boundary conditions.

For the boundary conditions applied to inlet leakage flows, the static pressure (P_s), the total temperature (T_t) and the fluid rotational speed (ω) are utilized. Since both inlets have pre-swirled fluid flows, the corresponding tangential velocities of them are calculated and applied to the model. For the outlet boundary condition, once again the static pressure and the total temperature of the flow are implemented. The impeller disk which is the rotor-side wall is defined with an angular velocity of Ω , and the stator-side wall is treated as stationary in the CFD model. To achieve physically realistic inlet conditions as in the case of the engine, the inlet domains are intentionally extended further. The additional sections ensure that the inlet flows have the appropriate tangential velocities while penetrating into the flow domain.

3.2. Mesh Description

The impeller backface cavity has a complex and irregular shape. The axial width of the domain changes in the radial direction. Moreover, the inclusion of the ISS structure into the flow domain extends the complexity of the configuration.

To generate a fine (well-established) mesh structure on such a geometry, it is essential to divide domain territory, i.e., walls, into proper subsections. To capture the flow

pattern in both wide and narrow regions of the domain, various body, face, and edge sizing are implemented.

Taking into account all these constitutional factors, the mesh structure is generated for the flow domain as shown in Figure 3.2.

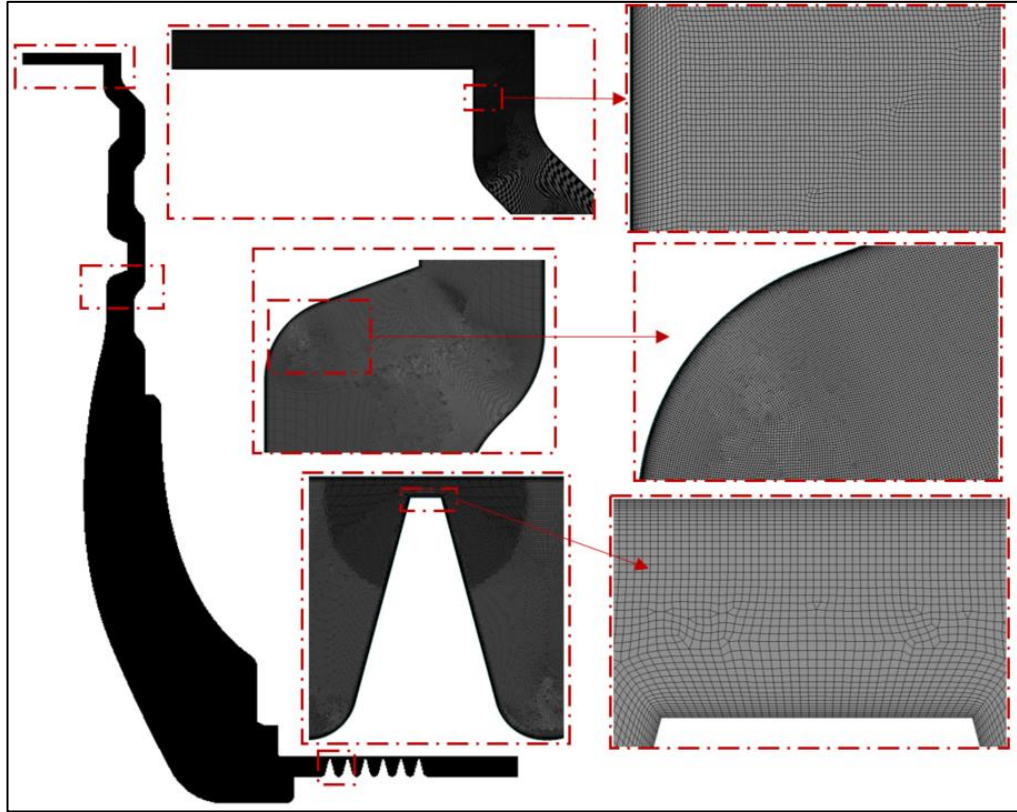


Figure 3.2: Mesh structure of the impeller backface domain.

The domain of interest is discretized using unstructured quadratic (a few of them are inherently triangular) elements. Specifically, 24 layers that grow smoothly are positioned on both rotor and stator wall surfaces to capture flow in the boundary layer. The wall y^+ value of at most 1.0 is ensured. The initial cell height is set to $1 \cdot 10^{-6}$ m. The mesh structure is adjusted to grow gradually from the wall towards the core of the cavity, with a growth rate set to 1.15, and number of maximum layers 24 for cavity region and 18 for the seal teeth region. To catch the high velocity regions for the fluid flowing throughout the seal, 2.0 mm radius of body sizing and element size of 0.015 mm is defined for the teeth region. Eventually, with these inputs, the overall mesh size has become in the order of circa $3 \cdot 10^6$.

Table 3.1 displays the properties obtained in the control volume of the generated mesh structure. When typical values are considered for aspect ratio, skewness, and orthogonal quality parameters in CFD analysis, it is obvious that the numerical mesh is of adequate quality.

Table 3.1: Mesh properties of the impeller backface cavity.

	Min	Avg	Max
Skewness	$1.3 \cdot 10^{-10}$	0.038	0.73
Orthogonal Quality	0.362	0.994	1.00
Aspect Ratio.	1.00	1.66	39.42
Element Quality	0.050	0.907	0.999

3.2.1. Mesh Independency

One of the major and essential issues while constructing mesh structures, to resolve a flow field, is the independency of outcomes with respect to mesh resolution. In order to do that several approaches are applicable. To perform a mesh independency study, one can generate different sizes of meshes and solve the flow field respectively with the predetermined convergence criteria. Once the convergence is established, the obtained flow field properties can be plotted with respect to mesh size. As far as the mesh size modification is properly managed, the relevant fluid properties should not alter after a certain number of mesh.

In this study, following the similar routine explained above, a mesh independency study has been performed. Four different number of mesh sizes are constructed and the pertinent flow field is solved. Fluid properties such as static pressure and total temperature are followed. The plots created for the mesh independency are illustrated in Figure 3.3, Figure 3.4, Figure 3.5, Figure 3.6.

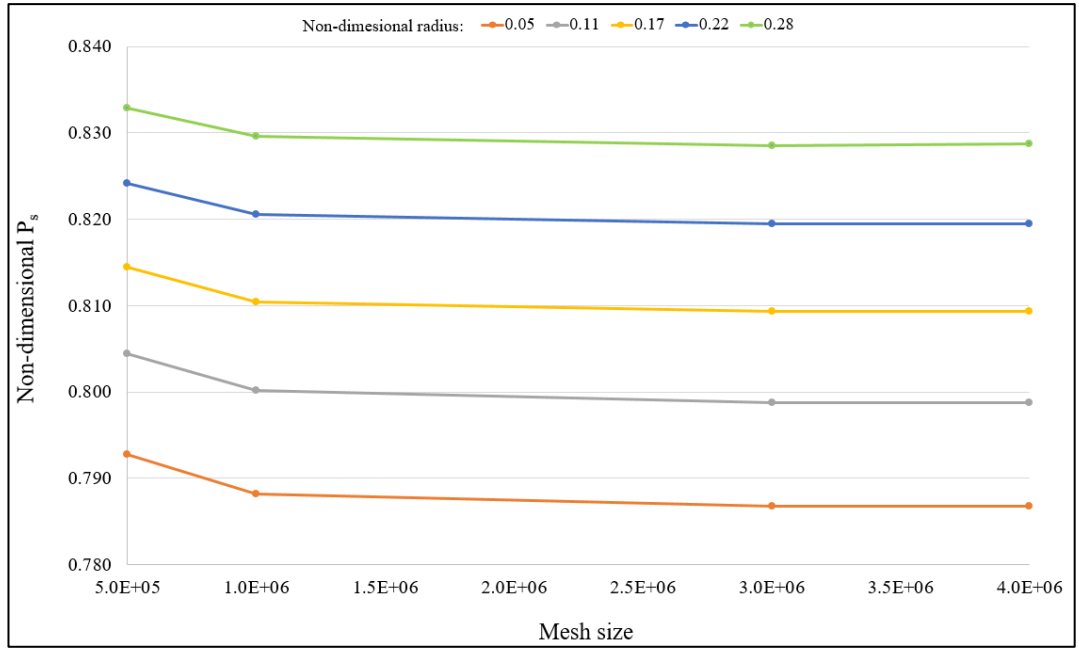


Figure 3.3: Non-dimensional static pressure change with mesh size.

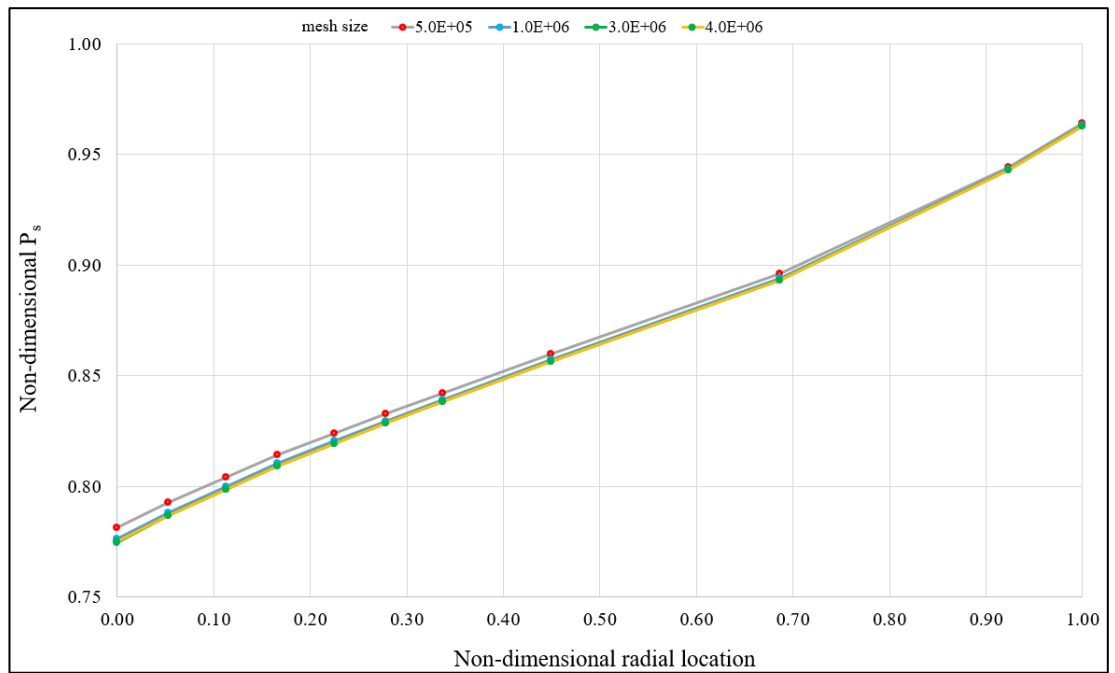


Figure 3.4: Non-dimensional static pressure change at several radii.

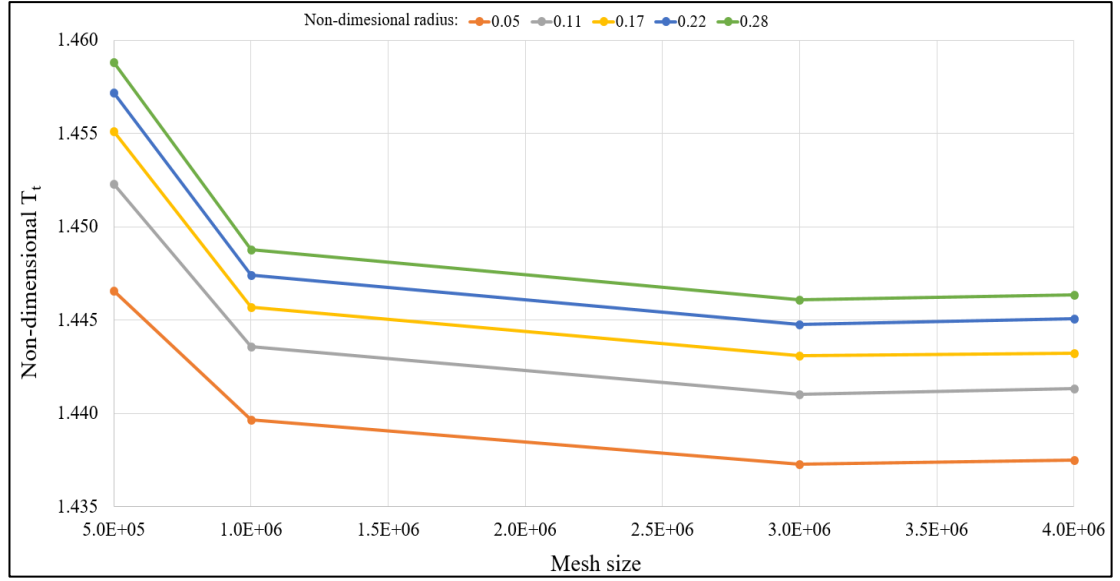


Figure 3.5: Non-dimensional total temperature change with mesh size.

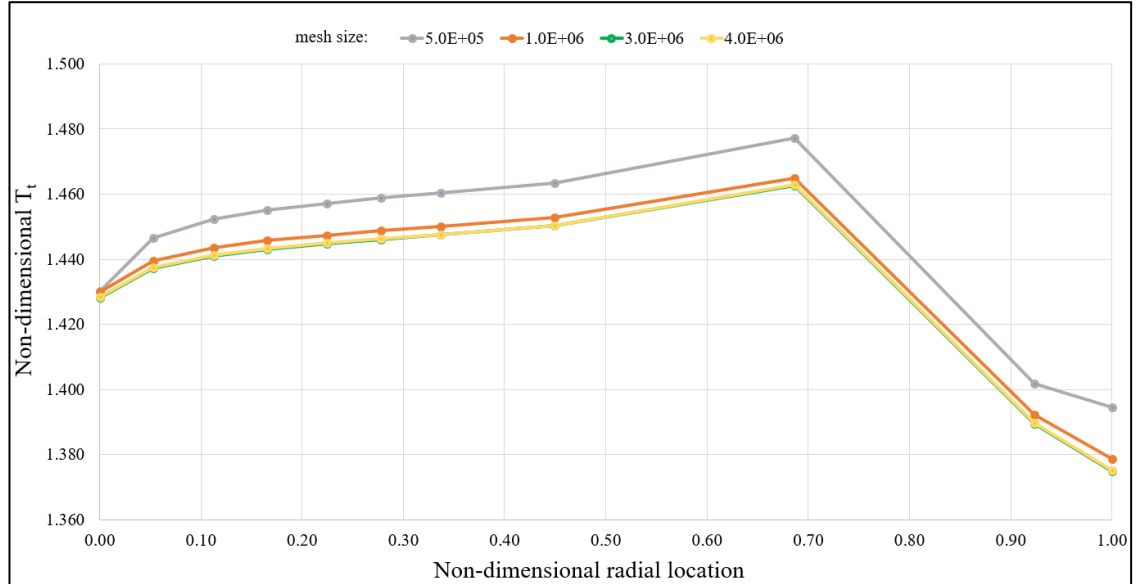


Figure 3.6: Non-dimensional total temperature change at several radii.

As seen in the figures, the static pressure and total temperature results are not alternating and remain stable as the total number of elements exceeds $3 \cdot 10^6$. Therefore, in the study, CFD analysis are performed with that number of elements and the corresponding results are processed in this context.

3.3. CFD Model

To investigate the radially inward flow in the rotor-stator cavity in consideration, inlet and exit boundary conditions are implemented from a flow network solver which employs the conservation of mass and momentum equations in one dimension.

Firstly, the static pressure measurements, obtained from the engine ground tests, are supplied to the 1-D solver and the corresponding inlet and exit conditions to procure those static pressures in the cavity are found iteratively. This provides the flow parameters like static/total pressure, swirl ratio and total temperature at the boundaries of the cavity in one-dimensional form. Then, these calculated inlet and outlet conditions are utilized as inputs to the 2-D axisymmetric CFD model.

For three different rotational speeds representing the three different engine operating conditions are examined with the CFD model. The numerical simulation software, Ansys[®] Fluent 2019R1 is commissioned for the CFD analysis and Reynolds-Averaged Navier-Stokes (RANS) equations are utilized with pressure-based solver. The setup details for the model, methods and controls are given in Table 3.2.

k- ω SST turbulent model is applied with the Low Re Corrections, Viscous Heating, Compressibility Effects and Production Limiter options are on. Turbulent modelling constants are set to default. Velocity formulation is set to absolute, and the operating condition is kept at 0 kPa gage. During the analysis which is performed as steady state, the energy and turbulent equations are activated. The equations “Ideal-gas” for the density, “Piecewise-polynomial” for the specific heat, “Kinetic-theory” for thermal conductivity and “Sutherland” for the viscosity are applied as fluid properties. “Coupled” scheme is applied as solution method and “Green-Gauss Node Based” is selected for the gradient solver. “PRESTO” which is commonly preferred for the rotating flow problems with high speed is opted for the pressure solution. Last, “Second Order Upwind” solution method is chosen for Density, Momentum, Swirl Velocity, Turbulent Kinetic Energy, Specific Dissipation Rate and Energy calculations to achieve higher accuracy results.

Table 3.2: CFD modelling inputs.

Dimensional System, Space	2-D Axisymmetric Coordinate System, Axisymmetric Swirl
Solver Type	Pressure-based
Velocity Formulation	Absolute
Solution Type	Steady State
Operating Condition	0 Pa (gage)
Energy, Turbulent	On
Turbulence Model	k- ω , SST Low Re Corrections, Viscous Heating and Compressibility Effects: On
Fluid Type	Air
Fluid Properties	ρ : Ideal-gas cp: Piecewise-polynomial k: Kinetic-theory μ : Sutherland
Inlet BCs	Pressure inlet (Ps, Tt)
Outlet BC	Pressure outlet (Ps, Tt)
Rotor Wall	Moving Wall, Rotational Motion, No Slip condition, Angular Velocity in rpm, Adiabatic
Stator Wall	Stationary Wall, No Slip condition, Zero velocity, Adiabatic
Solution Methods	Scheme: Coupled, Gradient: Green-Gauss Node Based Pressure: PRESTO! Density: Second Order Upwind Momentum & Swirl Velocity: Second Order Upwind, Turbulent Kinetic Energy & Specific Dissipation Rate: Second Order Upwind.

4. RESULTS

This section is presented in two parts. In the first place, results of 1D flow network solver is compared with 2D CFD analysis to illustrate the affinity of two distinct approaches. In the second part, the comprehensive outcomes of flow domain obtained by CFD analysis is established.

There are three different operating conditions analyzed in the study. For the sake of clarity, in the following paragraphs, these operating conditions of the engine are mentioned as Case 1, Case 2 and Case 3, demonstrating the impeller speeds of 67%, 78% and 100% rpm, respectively. The pertinent details of all cases are summarized in Table 4.1.

Table 4.1: Properties of operating conditions.

	Case-1	Case-2	Case-3
Ω (in percent.)	67%	78%	100%
Re_θ	$2.95 \cdot 10^6$	$4.35 \cdot 10^6$	$7.54 \cdot 10^6$
C_w	2443	3130	3703
λ_t	0.016	0.015	0.012
β_{inlet}	0.63	0.74	0.82

4.1. Comparison of 1D Solver vs CFD Results

In order to compare the results of two approaches, several parameters are investigated. The variation of static pressure, swirl ratio and total temperature throughout the cavity are set forth both from 1D solver and 2D CFD analysis at certain radii. Moreover, the axial load and the torque applied by the rotor on the fluid are demonstrated in similar manner.

In Figure 4.1, Figure 4.2 and Figure 4.3, the distribution of static pressure, swirl ratio and total temperature inside the cavity are illustrated, respectively for Case-1.

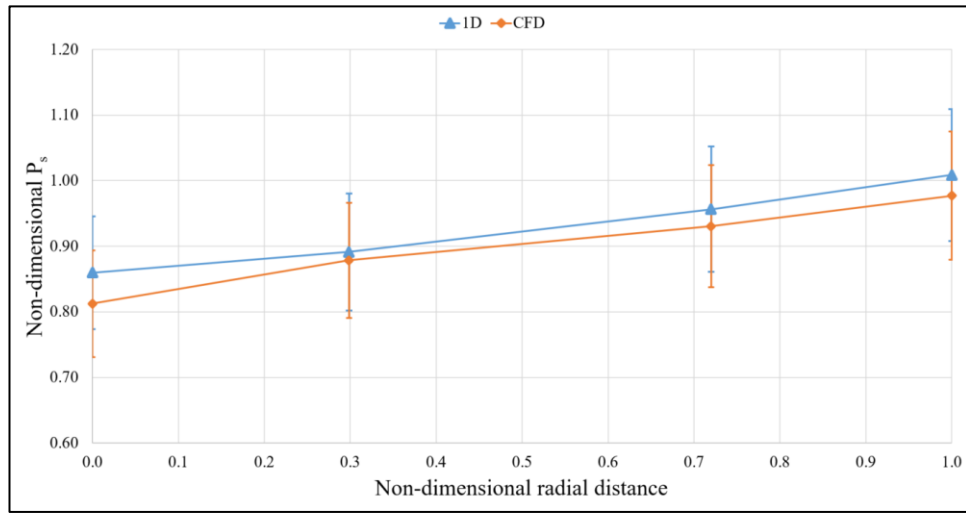


Figure 4.1: Non-dimensional static pressure comparison at several radii for Case-1.

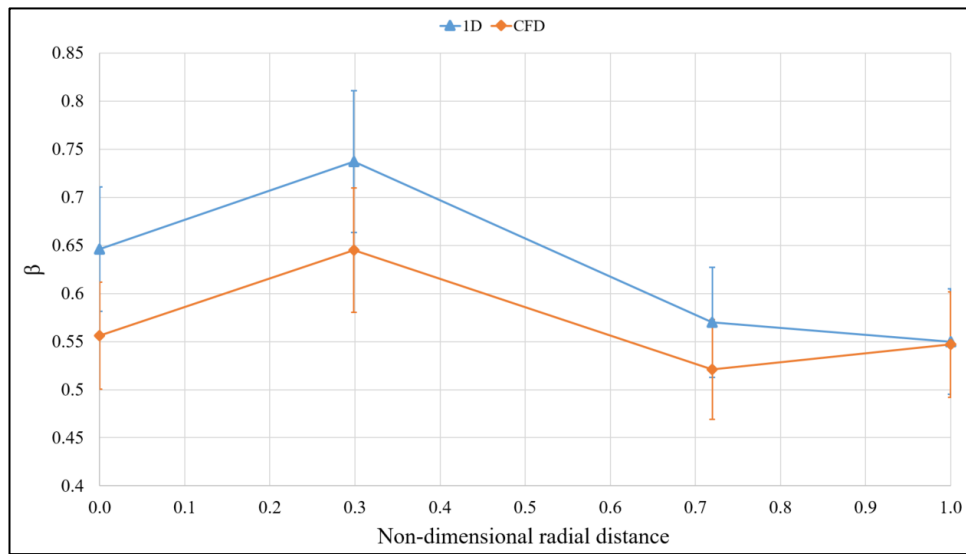


Figure 4.2: Non-dimensional swirl ratio comparison at several radii for Case-1.

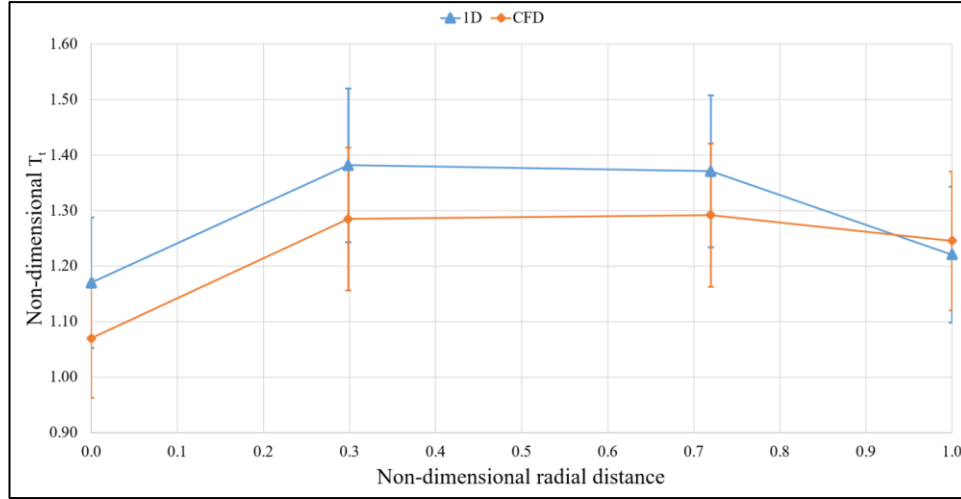


Figure 4.3: Non-dimensional temperature comparison at several radii for Case-1.

The vertical bars plotted in figures illustrate 10% error band. The largest deviation for P_s , β and T_t are spotted in the vicinity of the hub region because of the mixing and recirculation effects due to incoming flow over the seal.

As the radius increases the difference between flow network solver and CFD results diminishes. Moreover, focusing on total temperature plot, which gives hints about the temperature rise due to windage, it is obvious that while the fluid is driven away toward the rotor tip by centrifugal effects, it gains thermal energy. In general, 1-D solver overestimates the rotor torque and thus the total temperature except the rotor tip region. It is considered that the flow structure, especially near the protrusions around the rotor tip, results in larger temperature rise and better captured in 2D CFD model. This effect cannot be registered in 1-D solution.

In Figure 4.4, Figure 4.5 and Figure 4.6, the distribution of static pressure, swirl ratio and total temperature inside the cavity are demonstrated, respectively for Case-2.

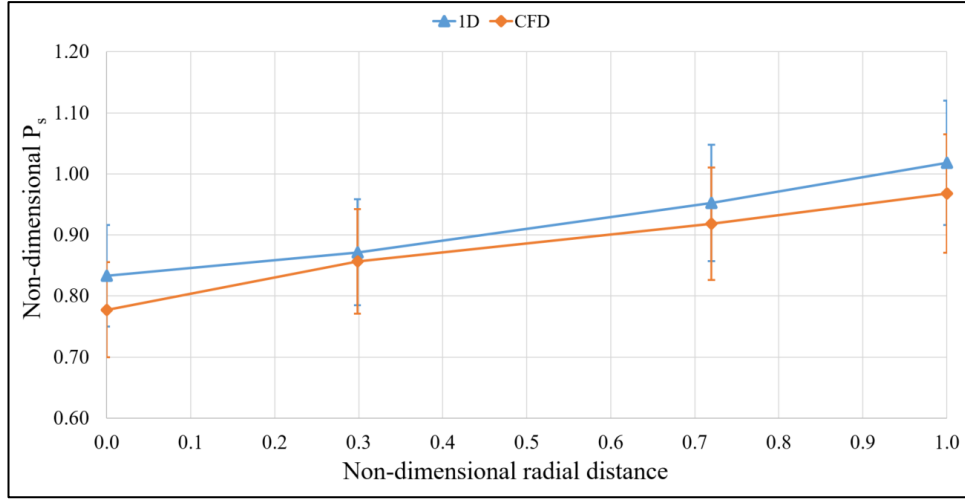


Figure 4.4: Non-dimensional static pressure comparison at several radii for Case-2.

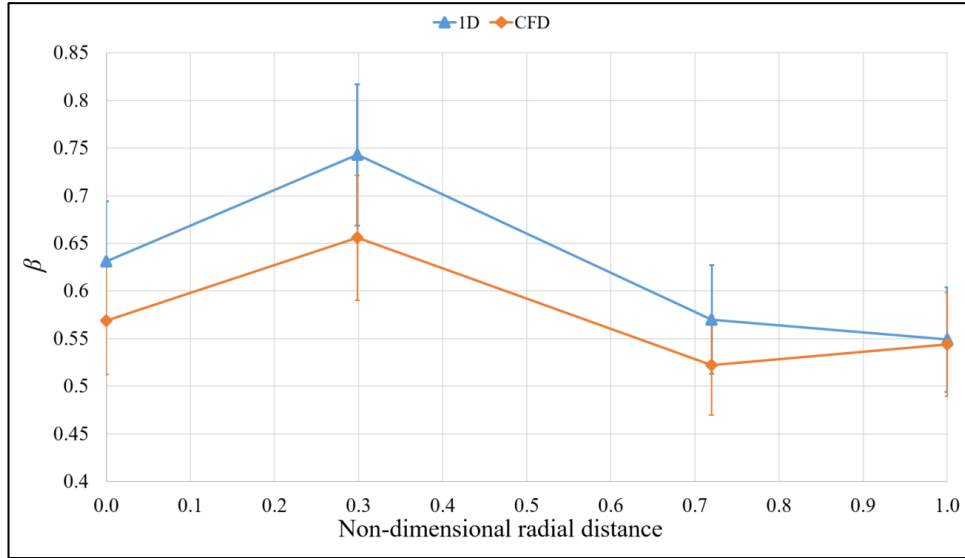


Figure 4.5: Non-dimensional swirl ratio comparison at several radii for Case-2.

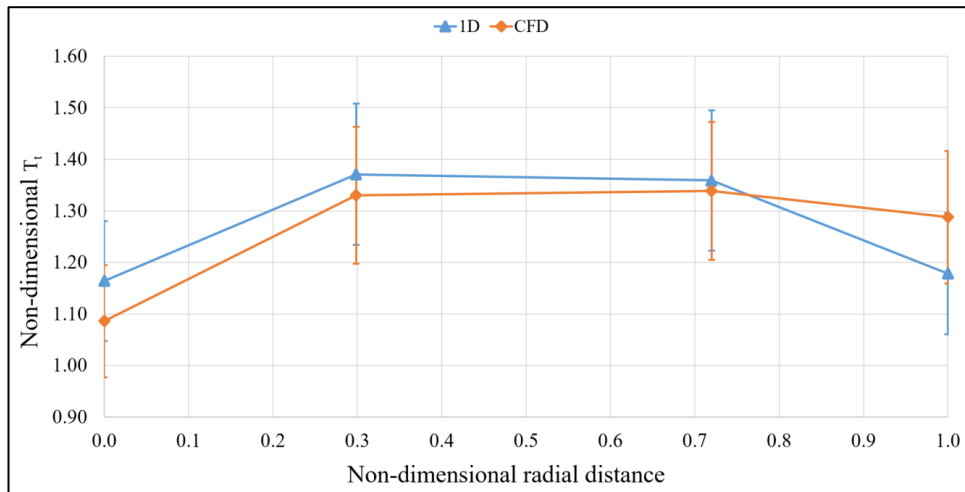


Figure 4.6: Non-dimensional total temperature comparison at given radii for Case-2.

The results for Case-2 is quite similar to previous case. The explanations submitted for the former case is valid as well. The error still exceeds 10% specifically for rotor hub region due to flow through seal and its corresponding effects. Notice that since the rotational speed of the rotor increases, the windage power advances and thus the difference between network solver and CFD model deviates further in the tip region.

In Figure 4.7, Figure 4.8 and Figure 4.9, the distribution of static pressure, swirl ratio and total temperature inside the cavity are plotted, respectively for Case-3.

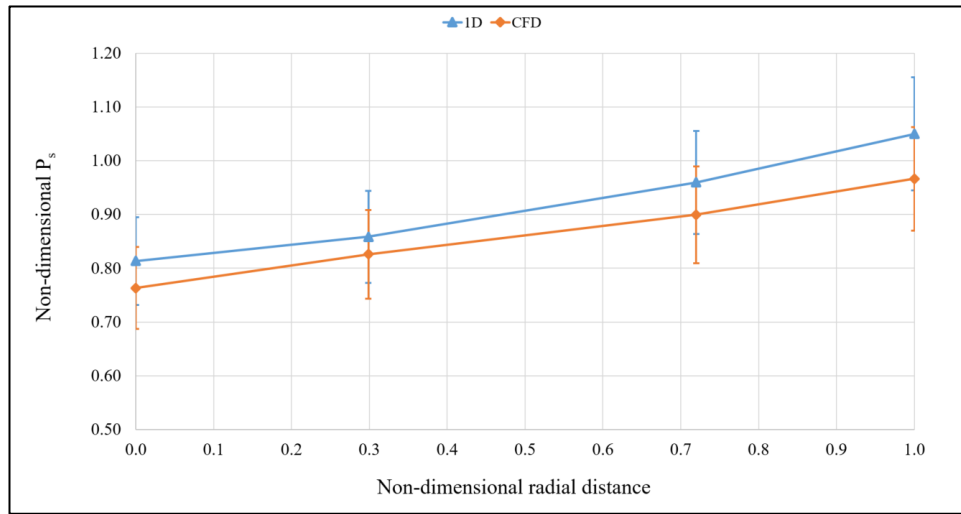


Figure 4.7: Non-dimensional static pressure comparison at several radii for Case-3.

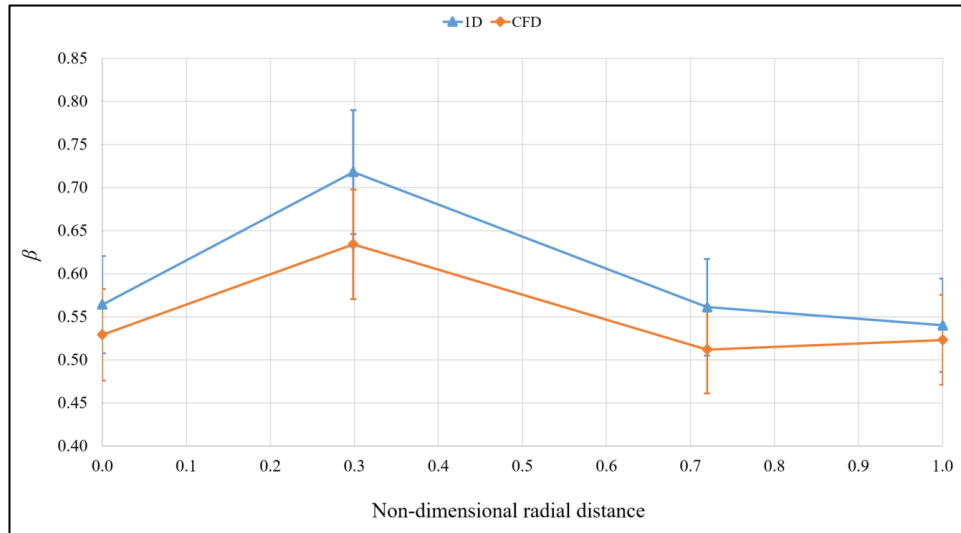


Figure 4.8: Non-dimensional swirl ratio comparison at several radii for Case-3.

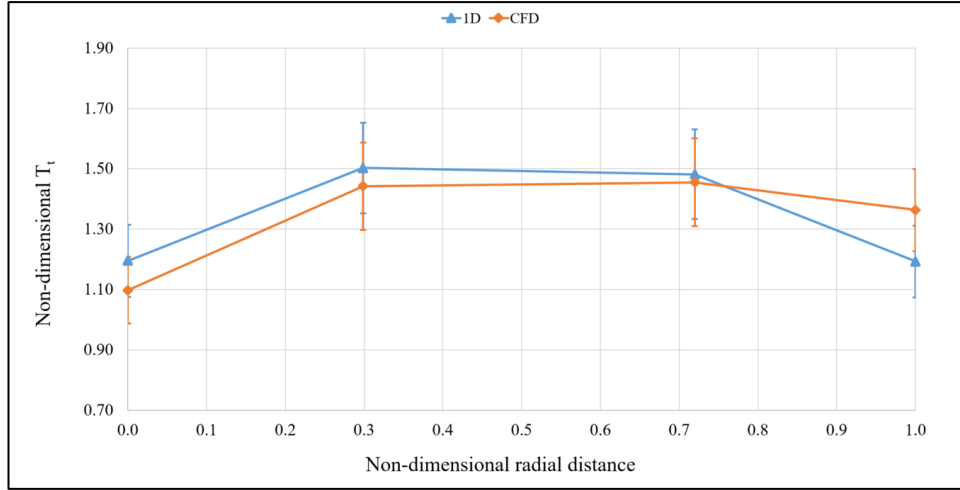


Figure 4.9: Non-dimensional total temperature comparison at given radii for Case-3. The trend is the same for Case-3 as in the first and second cases. The total temperature deviates further following the rotational speed enhancement and thus the largest difference is obtained in Case-3.

As explained in the previous sections of the study, the axial load on the impeller disk and the rotor torque applied by it on the fluid are other two important outcomes of such cavities. Therefore, the comparison between 1-D flow solver and 2D CFD model in terms of these parameters are also invoked.

To accomplish that, the non-dimensional version of the rotor torque is recalled as follows.

$$C_M = \frac{\Gamma}{0.5\rho\Omega^2 b^5} \quad (4.1)$$

Then, the corresponding values of 1-D flow solver and 2D CFD analysis for all cases are given in Table 4.2.

Table 4.2: Non-dimensional rotor torques.

	Case-1	Case-2	Case-3
$(C_M)_{imp,1D}$	$3.79 \cdot 10^{-4}$	$5.48 \cdot 10^{-4}$	$10.3 \cdot 10^{-4}$
$(C_M)_{imp,CFD}$	$3.54 \cdot 10^{-4}$	$4.87 \cdot 10^{-4}$	$9.13 \cdot 10^{-4}$

As previously highlighted, the rotor torque estimation of 1-D solver yields greater results than the CFD model for all cases.

Similarly, the non-dimensional axial load on the impeller disk can be defined as,

$$C_{Force} = \frac{F_{axial}}{0.5\rho\Omega^2 b^4} \quad (4.2)$$

The corresponding axial load estimation of 1-D flow solver and 2D CFD analysis for all cases are given in Table 4.3.

Table 4.3: Non-dimensional axial loads.

	Case-1	Case-2	Case-3
$(C_{Force})_{imp,1D}$	1.187	1.416	1.997
$(C_{Force})_{imp,CFD}$	1.156	1.367	1.869

The force on the impeller is overestimated in 1-D flow solver when compared to the CFD model for all cases.

4.2. CFD Analysis and Test Results

In this part of the study, the outcomes of CFD are to be discussed. The flow structure obtained in the computational domain is presented in terms of various post-processing contours.

When the flow field in the cavity is interpreted with the help of Figure 4.10, the representative stream function contour given for illustration, it is clear that the fluid flows radially inward over the stator side whereas the rotor forces the fluid to move in the radially outward direction. The pumping need of the rotor disk is accommodated with the inward flow over the stator disk. As the radially inward flow penetrates toward the lower radii, the pumping effect is suppressed, causing the fluid to move toward the disk center over the rotor, as well. This flow field agrees well with the radially-inward rotor-stator cavity studies performed by Pincombe [13], Debuchy et al. [19], Poncet et al. [20], Erdem et al. [28] and Sertçakan [29].

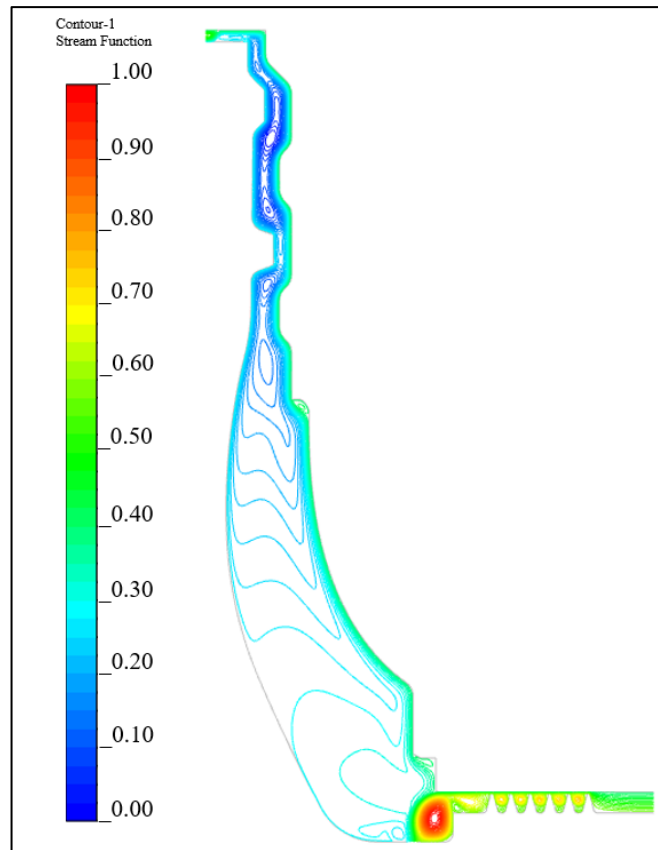


Figure 4.10: Representative stream function contour of flow domain.

In order to delineate the change of fluid rotational speed throughout the cavity, swirl ratios at several intervals of radii are investigated. The reference lines used for this purpose are given in Figure 4.11.

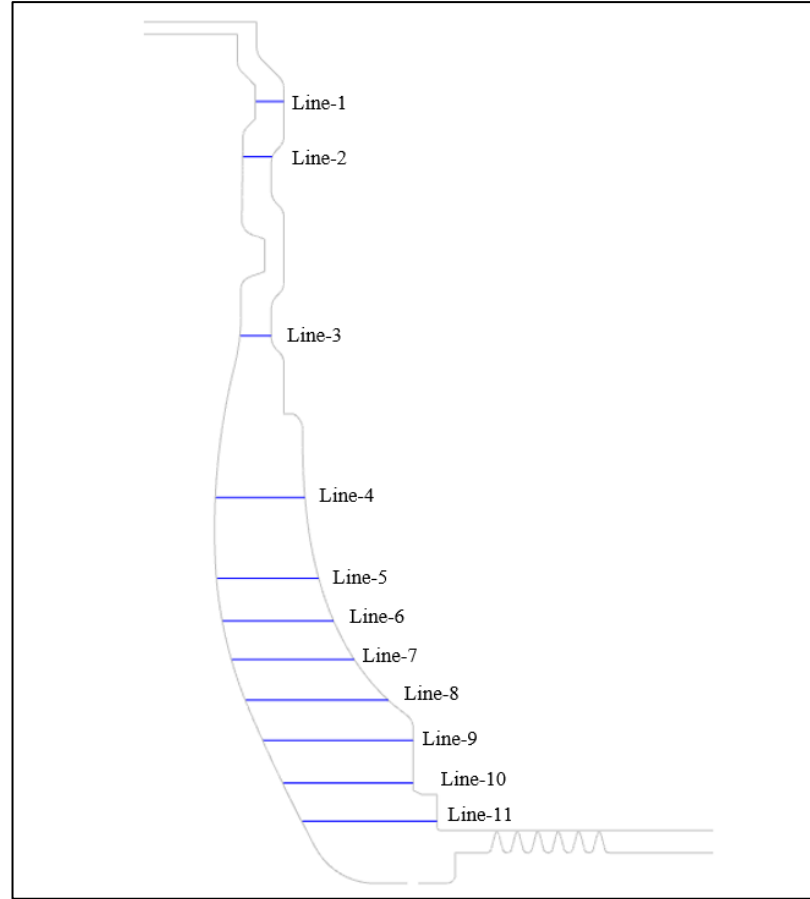


Figure 4.11: Reference lines inserted into the fluid domain.

In Figure 4.12, the swirl ratio distributions along these lines are given for Case-1. The axial distance of lines is converted to non-dimensional form by its own length. As seen in the figure, x-coordinate starts from 0 and ends at 1, demonstrating the rotor and the stator surfaces, respectively.

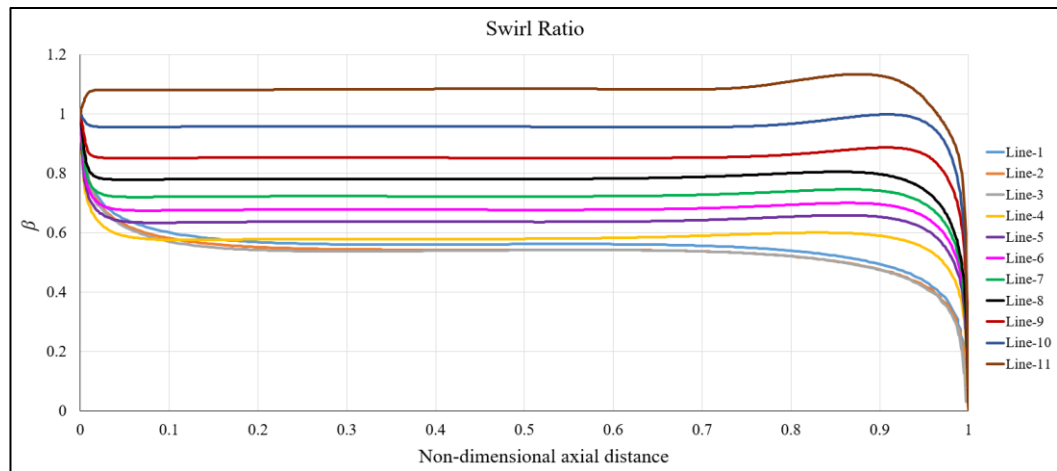


Figure 4.12: Swirl ratio vs axial distance at each line for Case-1.

The fluid has the same tangential velocity as the rotor on its surface and hence attains swirl ratio of unity at that position. Similarly, the stator side velocity is zero resulting in zero swirl ratio over its surface. Between these two ends, there is a plateau (except Line-1 to Line-3) referring to the inviscid core region rotating at a certain rate. This is compatible with the behavior in Batchelor [7] flow. At radii between Lines 1 to 3, however, the shape of the plateau diminishes and the flow structure resembles to Stewartson [8] type because of the small gap between the rotor and the stator. When the flow penetrates to lower radii regions, as in the case for Line-10 and Line-11, the swirl ratio increases dramatically. In particular, the swirl ratio exceeds 1 in the vicinity of Line-11 and in the latter regions, where the mixing with the seal leakage is intensely sensed.

The flow structures in Case-2 and Case-3 are mostly identical with the one in Case-1 as shown in Figure 4.13 and Figure 4.14, respectively.

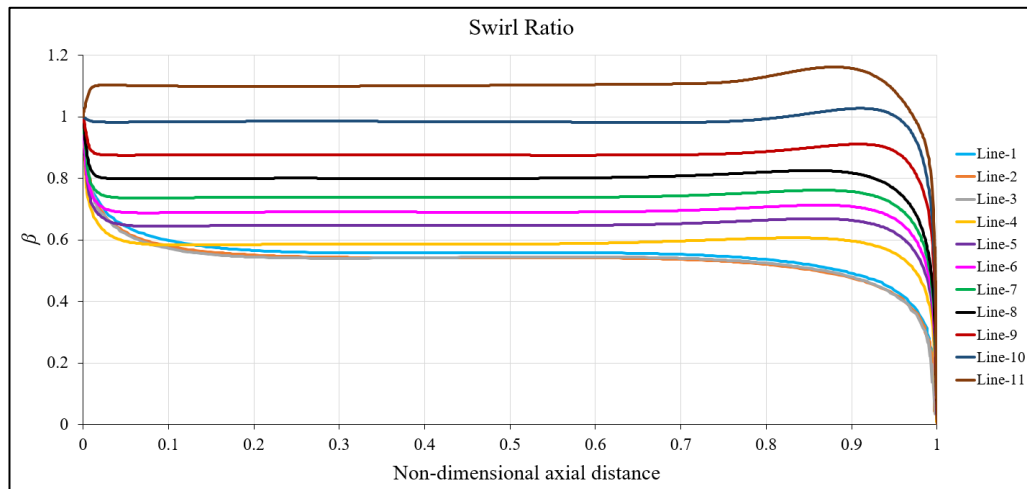


Figure 4.13: Swirl ratio vs axial distance at each line for Case-2.

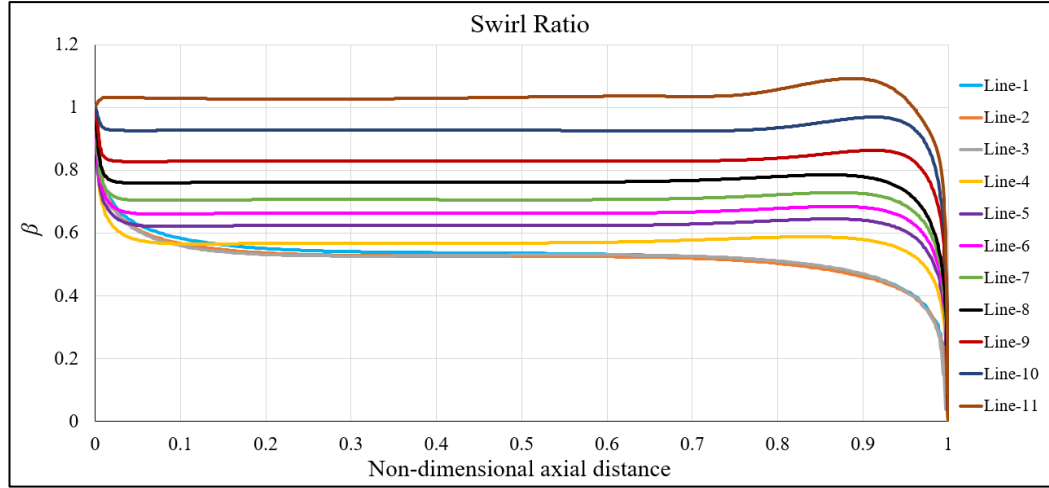


Figure 4.14: Swirl ratio vs axial distance at each line for Case-3.

In addition to previous three figures, the swirl ratio distribution throughout the whole cavity is also demonstrated in Figure 4.15 in a mutual non-dimensional scale.

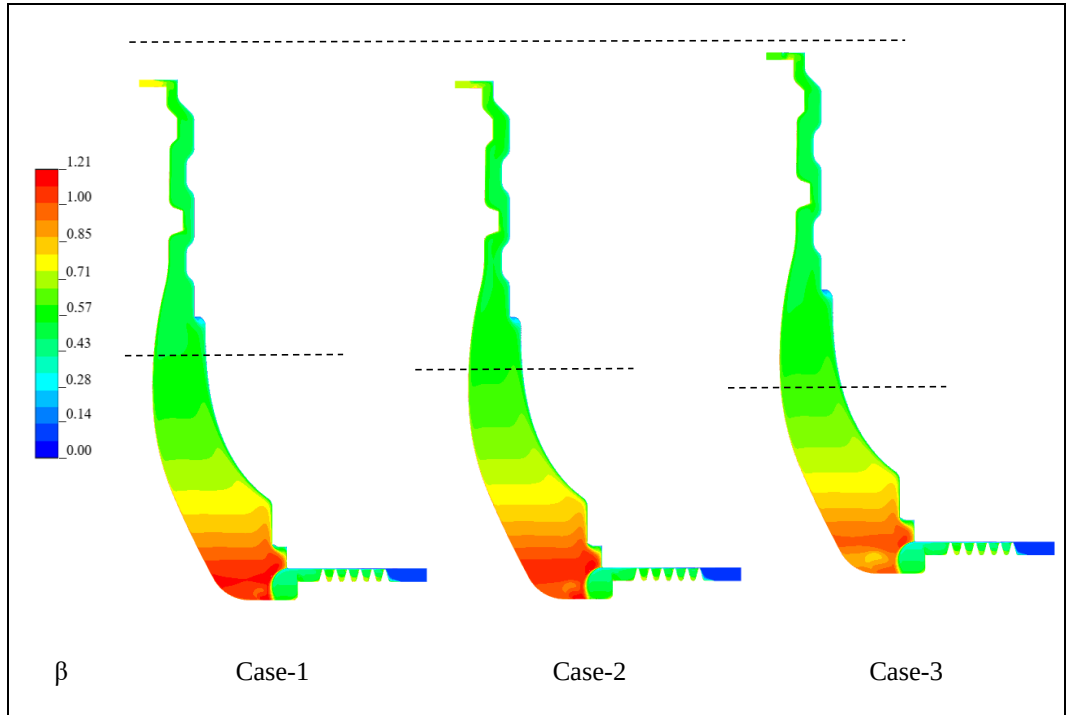


Figure 4.15: Swirl ratio distribution in the cavity for three cases.

There are intentionally-drawn dashed lines over the figure to illustrate the nearly-constant swirl ratio area, i.e., dark green region in the domain, for the three different operating conditions of the engine. Notice that, as the turbulent flow parameter, λ_t decreases, the size of this area enlarges, remarking that the forced vortex-wise behavior prevails in a greater region. In such regions, the static pressure drop is relatively low

compared to the one showing free vortex-wise behavior. Namely, the higher the region for the forced vortex-wise flow the larger the axial load on the impeller. To sustain the axial load management of the engine properly, this outcome canalizes the designer not to hold λ_t pretty low, as underlined by Erdem et al. [28], as well.

As mentioned in the previous sections, the superposed radially inward flow breaks the pumping effect of the rotating disk at a certain radius depending on the order of mass flow rate directed into the cavity across the periphery. In order to delineate this phenomenon, the radial velocity distributions, in non-dimensional and vector form, showing the radial stagnation points for three cases are given in Figure 4.16.

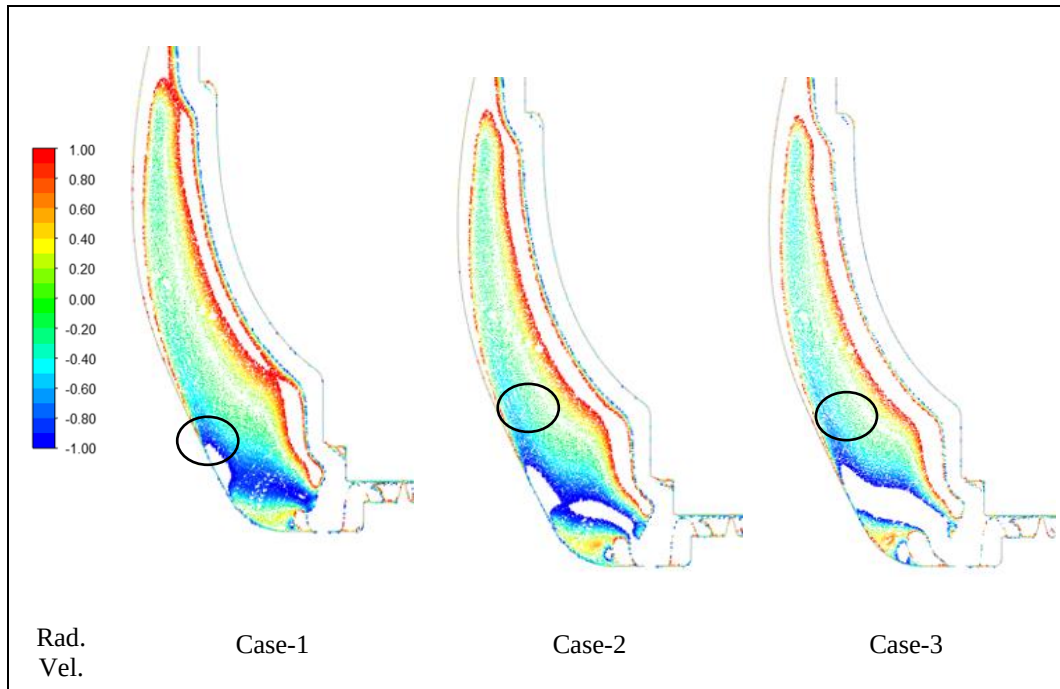


Figure 4.16: Non-dimensional radial velocity distribution in the cavity.

The dotted circles show the zero radial velocity locations in the figure. Since the radial stagnation points are close to rotor hub for all the operating conditions of the engine, the flow perceives the rotation-dominated regime inside the cavity, as Pincombe [13] stated.

In Figure 4.17 and Figure 4.18, non-dimensional total temperature and total pressure distributions in the cavity are given, respectively.

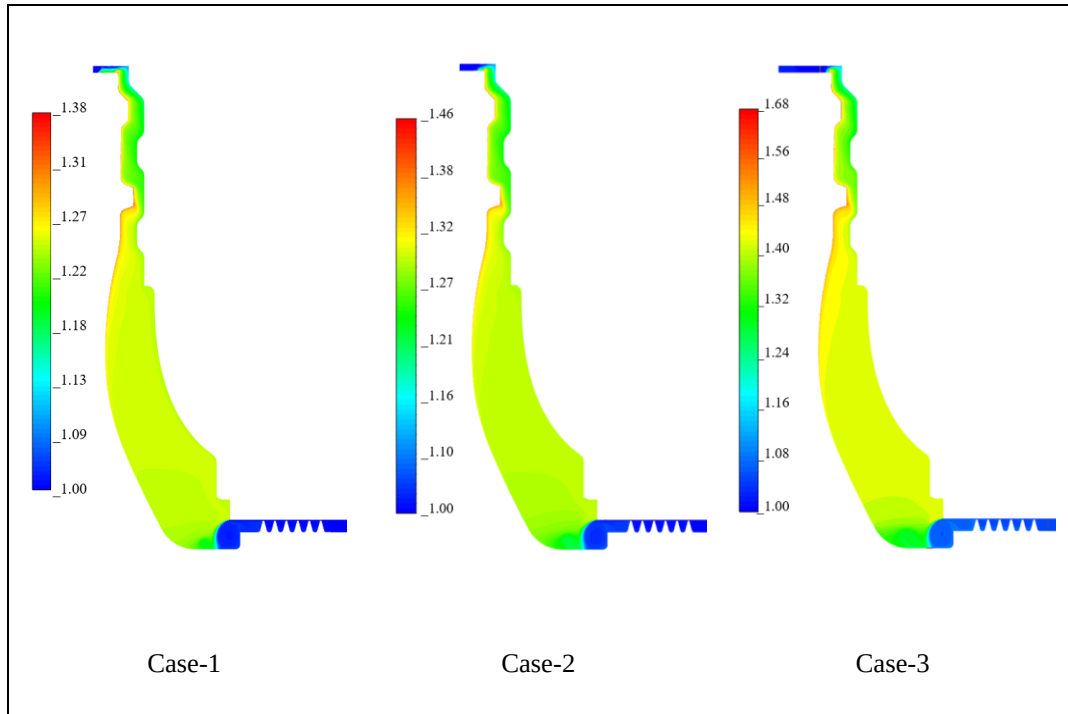


Figure 4.17: Non-dimensional total temperature distribution.

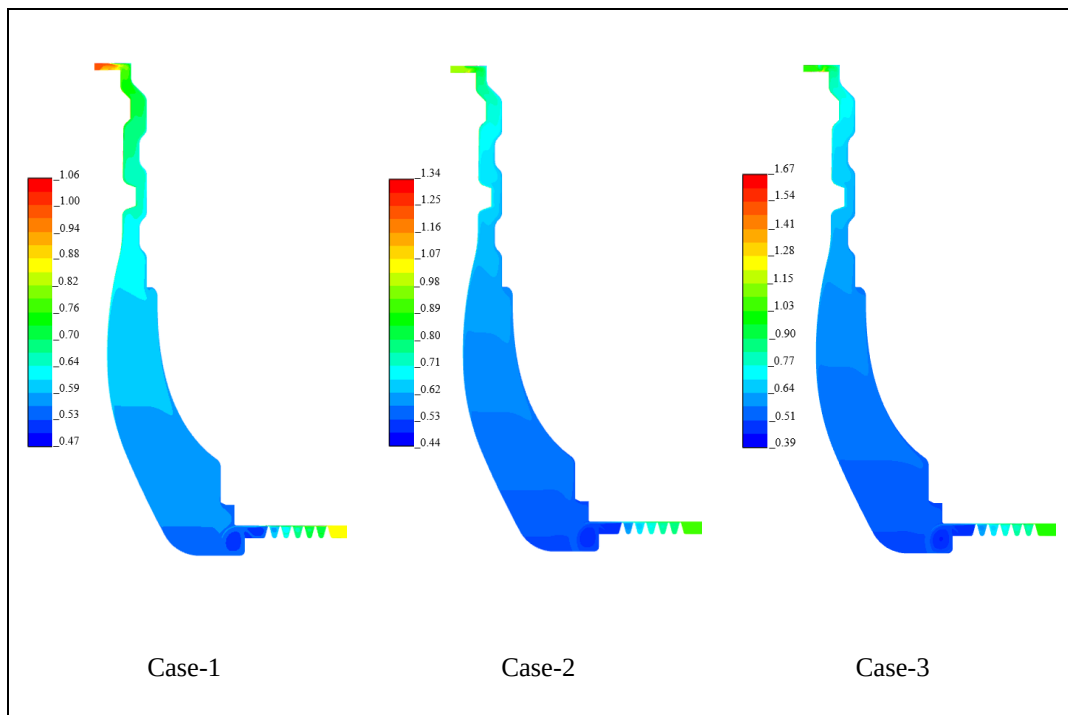


Figure 4.18: Non-dimensional total pressure distribution.

From the total temperature contour, it is obviously shown that, since the rotor applies torque on the fluid, there occurs a total temperature rise in the cavity due to windage in the vicinity of the rotor surface. This situation is especially evident around the protrusions closer to rotor tip over the surface.

The amount of windage due to moment applied by the rotor can be evaluated by writing energy balance for the fluid inside the cavity. Since the model is handled in an adiabatic manner, the sole reason for the total temperature rise at the outlet section of the cavity comes from windage power. In order to that, firstly the exit temperature without windage effect is to be evaluated by enthalpy balance expression,

$$\dot{m}_{in,1}c_{p,in,1}T_{t,in,1} + \dot{m}_{in,2}c_{p,in,2}T_{t,in,2} = \dot{m}_e c_{p,e}T_{t,e} \quad (4.3)$$

where:

$$\dot{m}_{in,1} + \dot{m}_{in,2} = \dot{m}_e \quad (4.4)$$

Then, the total temperature obtained from the numerical model is subtracted from total temperature evaluated by enthalpy balance. It can be expressed in non-dimensional form such that,

$$\frac{\Delta T}{\omega^2 r^2 / 2c_p} = \frac{T_{t,e,wind.} - T_{t,e}}{\omega^2 r^2 / 2c_p} \quad (4.5)$$

By using the equations and processing the total temperature data for three cases via CFD model, the temperature rises for solely inter-stage seal and impeller backface are presented in Table 4.4.

Table 4.4: Non-dimensional temperature rise due to windage.

	Case-1	Case-2	Case-3
$\left(\frac{\Delta T}{\Omega^2 r^2 / 2c_p}\right)_{seal}$	0.102	0.174	0.185
$\left(\frac{\Delta T}{\Omega^2 r^2 / 2c_p}\right)_{cavity}$	0.367	0.759	1.339

For a practical comparison of the results given in the table, the rotational speed, ω , of Case-1 is taken as reference while the datum radius, r , is set to the one in impeller tip inlet for the denominator of the non-dimensional temperature rise expression.

In the total pressure distribution, the pressure loss over the seal is apparent in all operating conditions. Moreover, as the flow penetrates into the lower radii the total pressure drops as expected.

In the following figure, Figure 4.19, the static pressure distributions for the three cases are given together with the pointwise non-dimensional results obtained from the CFD study and the test measurements.

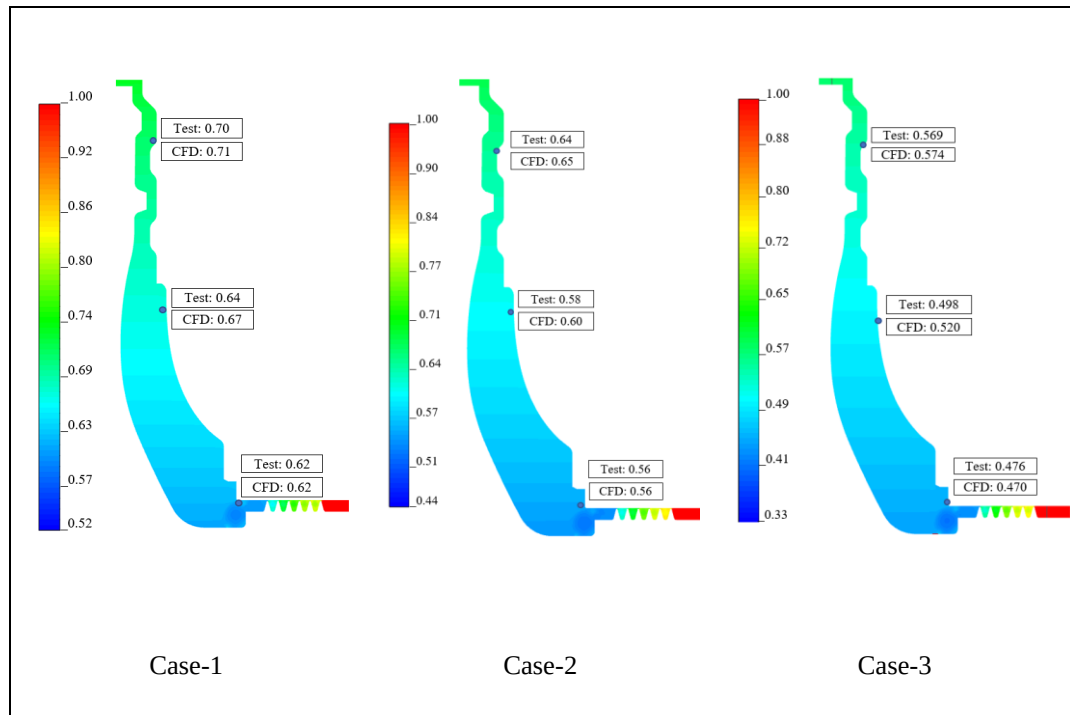


Figure 4.19: Non-dimensional static pressure distribution.

Three different radii test measurements collected from three distinct speeds of engine operation are demonstrated in the figure together with the numerical CFD results in the cavity for the same radii. The comparison of results shows that the differences between numerical study and the tests are quite comparable for all the rotational speeds of the impeller disk. In full power operation of the engine, i.e., 100% rpm, the deviations are 0.8%, 4.4% and 1.4%, respectively from top to bottom radii in consideration. For the intermediate, 78% rpm, and low power, 67% rpm, these deviations are 1.8%, 4.6%, 0.3%, and 1.9%, 4.5%, 0.2%, respectively.

5. CONCLUSION

In this study, the flow characteristics throughout the backface cavity of an impeller in a radial compressor have been studied. The cavity is usually considered as a rotor-stator space having inward radial inlet at the tip and outlet at the hub region. The flow field in such a cavity plays a key role since the distribution of coolant with the desired amount and at the proper temperature are main concerns in engine design phases. Besides, the pressure distribution and thus the axial load on bearings is directly affected by the flow field. The physics behind the rotor-stator cavity with radially inward flow depends on several factors which are examined within the scope of the study.

The rotational Reynolds number, turbulent flow parameter, gap ratio and inlet swirl ratio are the essential modifiers for the flow characteristics in the cavity. In order to examine their effects, a 2D axisymmetric numerical model demonstrating the fluid domain has been established with ANSYS Fluent 2019R1. Reynolds-averaged Navier Stokes equations together with $k-\omega$ SST turbulent model are proposed for the computational domain. For three different cases representing the three distinct operational regimes of the engine are analyzed in steady-state manner.

The obtained flow field has shown that the regime inside the cavity is rotation dominated. In the operational envelope of the engine, the zero-radial velocity, i.e., radial stagnation is experienced over the rotor towards the low radius region of the surface. Having relatively high rotational Reynolds number or low turbulent flow parameter in the flow field results in moderate static pressure drop in that region with a nearly constant swirl ratio. However, as the flow penetrates towards the inner radii, the characteristic of the flow transforms from forced vortex to free vortex and the swirl ratio exceeds unity. Free vortex flow dictates the flow to increase its swirl as the radius descends. The static pressure drop increases substantially in this region and the fluid is subjected to comparably low rotor and stator torques. Since the region for forced vortex-wise field is greater than the free vortex one, the high axial load contribution of the cavity to the engine bearings seems to be apparent. Additionally, just before the exit of the domain, the radially inward flow mixes with the seal leakage which already experienced a large total pressure drop passing through the seal. Being in the vicinity

of the exit region, the seal leakage cannot help itself to pass through the outlet hole entirely, a fraction of it generates a recirculation region.

In the final section of the study, the CFD model has been compared to the engine tests. There are three static pressure probes located on the stator side of the cavity at three distinct radii and the measurements are collected for three rotational speeds of the engine, namely, 67%, 78% and 100%. The test and CFD models show quite good agreement. In full power operation of the engine, i.e., 100% rpm, the deviations are 0.8%, 4.4% and 1.4%, respectively from top to bottom radii in consideration. For the intermediate, 78% rpm, and low power, 67% rpm, these deviations are 1.8%, 4.6%, 0.3%, and 1.9%, 4.5%, 0.2%, respectively.

This study embodies both the numerical and experimental approaches. Being a time-consuming and relatively expensive way of understanding the flow characteristics, the experimental method is not always a good option. In the scope of this work, it is presented that a well-established proper numerical approach has the potential to demonstrate the flow field quite well.

REFERENCES

- [1] Sultanian, B. K., (2018), “Gas Turbines Internal Flow Systems Modeling”, 1st Edition, Cambridge University Press.
- [2] Childs, P. R. N., (2011), “Rotating Flow”, 1st Edition, Butterworth-Heinemann.
- [3] Kármán, Th. von., “Über laminare und turbulente reibung-on laminar and turbulent friction”, *Zeitschrift für angewandte Mathematik Mechanik*, **1921**, 1, 233-252.
- [4] Bödewadt, U., “Die drehströmung über festem grunde”, *Zeitschrift für angewandte Mathematik*, **1940**, 20, 241-253.
- [5] Schlichting, H., (1979), “Boundary-Layer Theory”, 1st Edition, McGraw Hill.
- [6] ESDU, (2007), “Flow in rotating components-discs, cylinders and cavities”, Technical Report No: 07004, Internal Flow Series, ESDU Fluid Mechanics.
- [7] Batchelor, G. K., “Note on the class of solutions of the Navier-Stokes equations representing steady rotationally symmetric flow”, *Quarterly Journal of Applied Mathematics*, **1951**, 4, 29-41.
- [8] Stewartson, K., “On the flow between two rotating coaxial discs”, *Proceedings of the Cambridge Philosophical Society*, **1953**, 49, 333-341.
- [9] Daily, J. W., Nece, R. E., “Chamber dimension effects on induced flow and frictional resistance of enclosed rotating discs”, *Journal of Basic Engineering*, **1960**, 82, 217-232.
- [10] Bayley, F. J., Owen, J. M., “Flow between a rotating and a stationary disc”, *Aeronautical Quarterly*, **1969**, 20, 333-354.
- [11] Owen, J. M., Rogers, R. H., (1989), “Flow and heat transfer in rotating-disc systems”, 1st Edition, Research Studies Press.
- [12] Bayley, F. J., Conway, L., “Fluid friction and leakage between a stationary and rotating disc”, *Journal of Mechanical Engineering Science*, **1964**, 6, 164-172.
- [13] Pincombe, J. R., (1989), “Flow visualisation and velocity measurements in a rotor-stator system with a forced radial inflow”, Technical Report No: 88/TFMRC/TN61, Thermo-Fluid Mechanics Research Centre, University of Sussex, UK.
- [14] Daily, J. W., Ernst, W. D., Asbedian, V. V., (1964), “Enclosed rotating discs with superposed throughflow: Mean steady and periodic unsteady characteristics of induced flow”, Technical Report No: 64, MIT Department of Civil Engineering Hydrodynamics Laboratory, USA.
- [15] Iacovides H., Theofanopoulos I., “Turbulence modelling of axisymmetric flow inside rotating cavities”, *Int. J. Heat and Fluid Flow*, **1991**, 12, 2-11.
- [16] Hart, K., Turner, A. B., (1994), “Influence of radial inflow on rotor stator cavity pressure distributions”, International Gas Turbine and Aeroengine Congress and Exposition, 1-9, The Hague, Netherlands, 13-16 June.
- [17] Hart, K., Turner, A. B., (1995), “Simple Design Methods for the Prediction of Radial Static Pressure Distributions in a Rotor-Stator Cavity With Radial Inflow”, International Gas Turbine and Aeroengine Congress and Exposition, 1-13, Houston, Texas, 5-8 June.

- [18] Randriamampianina, A., Elena, L., Schiestel, R., Fontaine, A. P., “Numerical Prediction of Laminar, Transitional and Turbulent Flows in Shrouded Rotor-Stator Systems”, *Physics of Fluids*, **1997**, 9, 1696-1713.
- [19] Debuchy, R., Dymont, A., Muhe, H., Micheau, P., “Radial inflow between a rotating and a stationary disc”, *European Journal of Mechanics B/Fluids*, **1998**, 17, 791-810.
- [20] Poncet, S., Chauve, M. P., Le Gal, P., “Turbulent rotating disk flow with inward throughflow”, *J. Fluid Mech*, **2005**, 522, 253-262.
- [21] Poncet, S., Chauve, M. P., Schiestel, R., “Batchelor versus Stewartson flow structures in a rotor-stator cavity with throughflow”, *Physics of Fluids*, **2005**, 17, 1-58.
- [22] Poncet, S., Schiestel, R., “Numerical Modeling of Heat Transfer and Fluid Flow in rotor-Stator Cavities with Throughflow”, *International Journal Heat Mass Transfer*, **2007**, 50, 1528-1544.
- [23] Haddadi, S., Poncet, S., “Turbulence Modeling of Torsional Couette Flows”, *International Journal of Rotating Machinery*, **2008**, 1, 1-27.
- [24] DaSoghe, R., Facchini, B., Innocenti, L., Miccio, M., (2009), “Analysis of Gas Turbine Rotating Cavities by a One-Dimensional Model”, Proceedings of ASME Turbo Expo 2009 Power for Land Sea and Air, 355-364, Orlando, Florida, USA, 8-12 June.
- [25] DaSoghe, R., Facchini, B., Innocenti, L., Andreinei A., Poncet S., (2010), “Numerical Benchmark of Turbulence Modeling in Gas Turbine Rotor-Stator System”, Proceedings of ASME Turbo Expo 2010 Power for Land Sea and Air, 1-12, Glasgow, UK, 14-18 June.
- [26] Will, B. C., Benra, F. K., (2009), “Investigation of The Fluid Flow In A Rotor-Stator Cavity With Inward Through-Flow”, Proceedings of the ASME 2009 Fluids Engineering Division Summer Meeting, 207-218, California, USA, 2-6 August.
- [27] Shi, Y., Ding, S., Qui, T., (2020), “Influence of Inflow Distributions on Swirling Flow in Rotor-Stator Cavity Centipetal and Centrifugal Flow”, Proceedings of ASME Turbo Expo 2020 Turbomachinery Technical Conference and Exposition, 205-216, 21-25 September.
- [28] Erdem, E., Sertçakan, M. C., Şal, S., Doğu, Y., Yalçınkaya, A., (2022), “Investigation of Impeller Backface Cavity Flowfield Using CFD and a One-Dimensional Flow Solver”, Proceedings of ASME Turbo Expo 2022 Turbomachinery Technical Conference and Exposition, 1-11, Rotterdam, Netherlands, 13-17 June.
- [29] Sertçakan, M. C., (2024), “Rotor-stator Boşluklarındaki Radyal İçeride Doğru Akışın İncelenmesi”, Doktora Tezi, Kırıkkale Üniversitesi.

BIOGRAPHY

The author, Esra SEDEF, graduated from Ankara Yıldırım Beyazıt University (AYBU), Department of Mechanical Engineering in 2018. She started her career in TUSAS Engine Industry Inc. (TEI) as a Thermal System Design Engineer and has been working in the field of gas turbine design for more than five years. Her technical interests comprise thermo-fluid systems, compressible flow, turbomachinery as well as numerical simulation.