CFD Investigation of Flow Structures in Rotor-Stator Disc Cavity Systems

Minoo Arzpeima
Abstract

Ingestion of hot gas from the main annulus to the rotor-stator disc cavity is a major concern related to industrial turbines. Fluctuating pressure structures which are previously shown to form inside the disc cavity can influence the gas ingestion from the main flow stream. These rotating low pressure regions can reduce the lifetime of the turbine by overheating the components, induced by ingestion of hot gas, and influencing the mechanical integrity of the device. The hot gas ingestion is usually reduced by supplying a cooling air stream to the cavity and a proper rim seal design. The amount of the cooling air however needs to be minimized so as not to adversely affect the turbine performance and efficiency. A preliminary numerical study was conducted using computational fluid dynamics (CFD) on a one-stage test turbine in order to investigate the flow structure and pressure distribution in the turbine disc cavity. The test turbine consists of a newly designed blisk which is soon to be installed at Energy department, Royal Institute of Technology (KTH). The cavity model was simplified including the front cavity and only a section of the main annulus with no blades. The geometry of the rim seal was modified in the model by using a larger clearance in order to increase the probability of capturing the unsteady pressure structures. In order to obtain the boundary conditions to the cavity model, the main annulus was simulated separately using a sector model. The influence of the purge air mass flow rate on the flow structure and gas ingestion was further investigated. The results showed recirculation of the flow inside the cavity. Alternating ingress and egress through the rim seal was observed for the low purge case, while the cavity was totally sealed at high purge flow. No unsteady pressure structures were detected by the results of this study. The model needs however to be improved in order to obtain more reliable results.
Acknowledgement

I would like to thank my supervisor, Johan Dahlqvist, for his guidance and support throughout this work, and Tobias Gezork for his enormous help with CFD, and him always being ready to answer my questions. I am also thankful to Dr. Jens Fridh for his support and feedbacks during this work. The engineers at Siemens and GKN involved in the Turbo Power program are thanked for the meetings and discussions we had on several occasions. Finally I would also like to thank the gas turbine research unit at Bath University for their valuable input.
# Table of Contents

Abstract ........................................................................................................................................................................... ii  
1 Introduction .......................................................................................................................................................... 1  
2 Objectives .............................................................................................................................................................. 2  
3 Method ................................................................................................................................................................... 2  
4 Background ........................................................................................................................................................... 3  
4.1 Flow inside a rotor-stator disc cavity ....................................................................................................... 3  
4.1.1 The influence of the gap ratio .......................................................................................................... 4  
4.1.2 The influence of the superimposed flow ....................................................................................... 5  
4.1.3 The influence of the shroud ............................................................................................................. 5  
4.2 Axial turbine cavity flow ............................................................................................................................ 6  
4.2.1 Pressure fluctuation inside the disc cavity...................................................................................... 7  
4.3 Test turbine .................................................................................................................................................. 8  
4.4 Governing equations ................................................................................................................................10  
4.4.1 Turbulent flow ..................................................................................................................................11  
4.4.2 Turbulence models ..........................................................................................................................12  
4.4.3 Finite volume method .....................................................................................................................14  
5 CFD modeling ....................................................................................................................................................15  
5.1 Main annulus model .................................................................................................................................15  
5.1.1 Computational domain ...................................................................................................................15  
5.1.2 Grid structure ...................................................................................................................................16  
5.1.3 Operating and boundary conditions .............................................................................................16  
5.1.4 Interfaces ...........................................................................................................................................17  
5.1.5 CFD calculations .............................................................................................................................17  
5.1.6 Grid sensitivity study .......................................................................................................................20  
5.1.7 Main annulus solution .....................................................................................................................22  
5.2 Disc cavity model ......................................................................................................................................23  
5.2.1 Geometry creation ...........................................................................................................................23  
5.2.2 Grid structure ...................................................................................................................................23  
5.2.3 Boundary conditions ........................................................................................................................25  
5.2.4 CFD calculation ................................................................................................................................26  
6 Results and Discussion ......................................................................................................................................28  
7 Conclusion and future work .............................................................................................................................36  
References .....................................................................................................................................................................37
List of Figures

Figure 1. An industrial gas turbine; the magnified picture shows the flow passage of the turbine section and the disc cooling system [1].................................................................................................................................... 1

Figure 2. Schematic of a rotor-stator system with superimposed flow and stationary shroud; adopted from [2]........................................................................................................................................... 3

Figure 3. The schematic of the flow structure in a shrouded disc cavity with a supply of a superimposed flow [9] ............................................................................................................................................. 4

Figure 4. Flow regimes for a shrouded rotor-stator disc cavity, adopted from [2] ........................................................................................................................................................................ 5

Figure 5. An overview of one stage in an axial turbine [8] ...................................................................................................................... 6

Figure 6. Schematic of the one-stage turbine rig .................................................................................................................................... 8

Figure 7. The 3D view of the main annulus computational domain ......................................................................................................... 15

Figure 8. The main annulus grid structure ................................................................................................................................................ 16

Figure 9. The boundary conditions to the computational domain ...................................................................................................... 17

Figure 10. Total to total efficiency compared for three meshes ........................................................................................................ 20

Figure 11. Mass flow rate compared for three meshes ...................................................................................................................... 21

Figure 12. Static pressure profile 1 mm downstream of the vane trailing edge for three meshes ........................................................................ 21

Figure 13. Instantaneous static pressure distribution along the passage at mid-span ................................................................................ 22

Figure 14. The instantaneous velocity and pressure contour downstream of the vane trailing edge and upstream of the blade leading edge, respectively ........................................................................................................ 22

Figure 15. The lateral cross section view of the computational domain with (a) the original design of the rim seal and (b) the larger rim seal gap ........................................................................................................ 23

Figure 16. The 2D mesh of the disc cavity with larger rim seal gap ...................................................................................................... 24

Figure 17. The 3D mesh of the disc cavity, obtained by rotation of the 2D mesh ..................................................................................... 24

Figure 18. The disc cavity domain boundaries ........................................................................................................................................ 25

Figure 19. The main inlet boundary condition exported from the main annulus modeling results, and expanded to the whole cavity model (the plotted contours show only the U velocity component) ........................................................................................................ 26

Figure 20. The instantaneous pressure structure at cavity mid-plane for the main annulus and cavity models ...................................................................................................................... 28

Figure 21. Instantaneous pressure distribution at mid-plane for low purge flow case (left) in the main annulus and the disc cavity, (right) inside the disc cavity only ........................................................................................................................................ 28

Figure 22. Instantaneous pressure distribution at mid-plane for high purge flow case (left) in the main annulus and the disc cavity, (right) inside the disc cavity only ................................................................................................................ 29

Figure 23. Streamlines inside the disc cavity for the case of (a) low purge flow and (b) high purge flow .............................................................................................................................. 30

Figure 24. Circumferential variation in time-averaged pressure coefficient, 0.5 mm above the hub at mid-plane and (a) x=2.7 mm (b) x=3.1 mm (c) x=4.7 mm downstream of the vane trailing edge ........................................................................................................................................ 32

Figure 25. The pressure contours in the main annulus at mid-plan, and the streamlines on two planes at two different pitch locations, for low purge case ........................................................................................................ 33

Figure 26. Velocity vectors near the rim seal on the same planes shown in Figure 24 ........................................................................................................................................................................... 33

Figure 27. The pressure contours in the main annulus at mid-plan, and the streamlines on two planes at two different pitch locations, for high purge case ........................................................................................................................................... 34

Figure 28. Velocity vectors near the rim seal on the same planes shown in Figure 26 ........................................................................................................................................................................... 34

Figure 29. The contours for concentration of the scalar variable for (a) low purge flow and (b) high purge flow .................................................................................................................................................... 35
List of Tables

Table 1. Geometrical characteristics of the turbine.................................................................9
Table 2. Operating condition of the turbine............................................................................9
Table 3. The mesh statistics for three meshes.......................................................................16
Table 4. The main annulus model setting............................................................................19
Table 5. The disc cavity model setting................................................................................27
Nomenclature

Greek symbols

$\beta_1$  k- $\omega$ model constant
$\beta^*$ k- $\omega$ model constant
$\gamma_1$  k- $\omega$ model constant
$\varepsilon$  rate of dissipation of turbulent kinetic energy
$\eta_1$  sealing effectiveness
$\eta_{is}$  isentropic efficiency
$\Lambda_p$  degree of reaction, pressure based
$\mu$  dynamic viscosity
$\mu_t$  eddy viscosity
$\nu$  kinematic viscosity
$\Pi$  pressure ratio
$\sigma_k$  Prandtl number (turbulent model constant)
$\sigma_\varepsilon$  Prandtl number (k-$\varepsilon$ model constant)
$\sigma_{\omega}$  Prandtl number (k- $\omega$ model constant)
$\rho$  density
$\Phi$  dissipation function
$\varphi$  circumferential coordinate
$\phi$  general flow property
$\Omega$  angular velocity
$\omega$  turbulence frequency in k-$\omega$ model

Latin Symbols

$b$  outer radius of disc cavity
$C$  local concentration of the additional variable
$C_{in}$  concentration of the additional variable at the inlet
$C_w$  dimensionless cooling flow rate
$C_p$  dimensionless pressure coefficient
$C_{1\varepsilon}$  k-$\varepsilon$ model constant
$C_{2\varepsilon}$  k-$\varepsilon$ model constant
$C_\mu$  k-$\varepsilon$ model constant
$E$  total energy
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$G$</td>
<td>gap ratio</td>
</tr>
<tr>
<td>$G_c$</td>
<td>shroud clearance ratio</td>
</tr>
<tr>
<td>$i$</td>
<td>internal energy</td>
</tr>
<tr>
<td>$k$</td>
<td>thermal conductivity, turbulent kinetic energy</td>
</tr>
<tr>
<td>$M_{\omega}$</td>
<td>Mach number related to rotor rotational speed</td>
</tr>
<tr>
<td>$M_{\theta}$</td>
<td>tangential Mach number</td>
</tr>
<tr>
<td>$m_{\text{purge}}$</td>
<td>mass flow rate of the purge flow</td>
</tr>
<tr>
<td>$n$</td>
<td>rotor disc rotational speed (rpm)</td>
</tr>
<tr>
<td>$p$</td>
<td>static pressure</td>
</tr>
<tr>
<td>$p_{\text{avg}}$</td>
<td>average static pressure across pitch</td>
</tr>
<tr>
<td>$P_k$</td>
<td>rate of production of turbulent energy</td>
</tr>
<tr>
<td>$P_R$</td>
<td>rotor pitch</td>
</tr>
<tr>
<td>$P_S$</td>
<td>stator pitch</td>
</tr>
<tr>
<td>$q$</td>
<td>dynamic pressure of fluid at hub speed</td>
</tr>
<tr>
<td>$Q$</td>
<td>heat transfer</td>
</tr>
<tr>
<td>$R_{\text{hub}}$</td>
<td>hub radius</td>
</tr>
<tr>
<td>$R_{e_w}$</td>
<td>external flow Reynolds number</td>
</tr>
<tr>
<td>$R_{e_p}$</td>
<td>rotational Reynolds number</td>
</tr>
<tr>
<td>$s$</td>
<td>axial distance between the rotor and stator discs</td>
</tr>
<tr>
<td>$s_c$</td>
<td>axial distance between the shroud and the rotor</td>
</tr>
<tr>
<td>$s_{ij}$</td>
<td>deformation rate</td>
</tr>
<tr>
<td>$S_M$</td>
<td>momentum source term</td>
</tr>
<tr>
<td>$t$</td>
<td>time</td>
</tr>
<tr>
<td>$t^*$</td>
<td>vane pitch</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature</td>
</tr>
<tr>
<td>$U$</td>
<td>hub speed</td>
</tr>
<tr>
<td>$u$</td>
<td>velocity in x direction</td>
</tr>
<tr>
<td>$v$</td>
<td>velocity in y direction</td>
</tr>
<tr>
<td>$w$</td>
<td>velocity in z direction</td>
</tr>
</tbody>
</table>


1 Introduction

Gas turbines are today one of the most essential means of power generation. Although use of gas turbines in power generation from renewable sources is growing today, fossil fuel-based operating turbines are still dominant. Increase of efficiency of the turbines can however lead to reduction in fuel consumption and thereby contribute to sustainability.

Gas turbine consists of three major parts: compressor, combustion chamber and turbine which are usually mounted on the same shaft. The air pressure is increased by passing through the compressor stages. The compressed air enters the combustor where it is mixed with the added fuel and combustion occurs. The hot gases which are the combustion products enter the turbine, where they are expanded by passing through several turbine stages and rotate the shaft and produce power. Part of the shaft power is supplied to the compressor and the rest is used to drive the generator to produce electricity.

The gas temperature at turbine inlet has increased significantly (up to 1800K) during the past few decades, resulting in increased thermal efficiency of turbines. This has been partly due to development of new materials which can withstand such high temperatures, but perhaps more than that, owing to the new cooling techniques. A lot of focus has been devoted to cooling techniques to protect the blades, using methods such as film cooling. However, cooling the cavity between the rotating and stationary discs has become equally important in order to protect the discs and blade roots from damages caused by the hot gas ingested from the main annulus.

Figure 1 shows a picture of an industrial gas turbine, where the turbine section is magnified, showing the flow across the turbine stages. The blue arrows show the turbine cooling system which uses air to purge the gaps between the discs.

Figure 1. An industrial gas turbine; the magnified picture shows the flow passage of the turbine section and the disc cooling system [1]

The lower disc cavity will be the focus of this study. The thermal load is usually reduced by the cooling air which is extracted from the compressor and supplied near the shaft. This air stream is radially fed to the cavity in order to pressurize the wheel-space and hence reduce the ingestion of hot gases. As shown in Figure 1, this radial outflow of air then leaves through the rim seal and joins the mainstream gas. The interaction with this secondary air flow can however disturb the main flow. Besides, the compressed air used for cooling should be considered as a loss. The amount of seal air supplied into the cavity needs therefore to be minimized so as not to adversely affect the turbine performance and efficiency [2], [3]. The
design of the rim seal is also important in limiting the flow exchange between the main flow and the cavity flow. Instabilities in the flow structure inside the cavity have been previously detected. These structures are shown to influence the gas ingestion. Besides, the pressure fluctuations inside the disc cavity can affect the mechanical integrity and lifetime of the turbine.

Numerical studies of the disc cavity systems give complementary insight to the experiments about the flow structure and ingestion phenomenon. Better understanding of the flow structure can aid to improve the turbine efficiency.

2 Objectives

The purpose of this work is to, based on the previous studies, investigate the flow structures and pressure distributions inside the rotor-stator disc cavity of a test turbine using Computational Fluid Dynamics (CFD). The influence of the purge air flow rate and the flow ingestion from the main annulus into the cavity is as well studied. The test rig consists of a newly designed blisk which is going to be installed at the turbomachinery laboratory at energy department, KTH. This preliminary study is meant therefore to be used to help to predict the performance of the new stage, and enable comparison with future experimental studies on the turbine.

3 Method

A numerical study is performed on a one-stage test turbine, available at Energy Department, Heat and Power Division, Royal Institute of Technology (KTH). The main annulus of the turbine is first modeled separately, by moving from steady-state to transient simulation. The main flow boundary conditions to the cavity model will then be obtained from the main annulus model’s solution. The cavity model consists of a 360 degree, 3D model of the rotor-stator disc cavity together with a section of the main annulus of the turbine. For the sake of simplification, no vane or blade is included in the cavity model.

Since previous studies [4] have shown an increase in the scale of pressure instabilities inside the disc cavity with increase in the rim seal clearance, a larger clearance is investigated by modifying the original (rotor hub model. The flow problems are solved using a commercially available CFD code and for different cooling air flow rates.
4 Background

4.1 Flow inside a rotor-stator disc cavity

A schematic of rotor-stator disc cavity with a stationary shroud and superimposed flow is shown in Figure 2. The gap ratio and the shroud clearance ratio are defined as:

$$G = \frac{s}{b}$$  \hspace{1cm} (4.1)

and

$$G_c = \frac{s_c}{b}$$  \hspace{1cm} (4.2)

where $s$ is the axial distance between the two discs, $s_c$ is the axial distance between the shroud and the rotor (the shroud clearance), and $b$ is the outer radius of the cavity.

![Schematic of a rotor-stator system with superimposed flow and stationary shroud](image)

Figure 2. Schematic of a rotor-stator system with superimposed flow and stationary shroud; adopted from [2]

The complicated case of confined flow between rotating-disc systems has been of great interest for decades, especially due to its practical importance in industrial devices such as turbomachines. Batchelor [5] was one of the first who studied the case of flow between two discs. His model proposed development of boundary layers on both the stator and the rotor, with radial outflow on the rotor side, and radial inflow on the stator side. The model further suggested the existence of an inviscid core flow, confined between the two boundary layers, which rotates with a fraction of the rotor angular velocity ($\Omega$). This fraction was later found to be around 30%$\Omega$ for laminar flow and 40%$\Omega$ if the flow is turbulent [6]. Shortly after, Stewartson [7] model questioned the existence of a rotating core, suggesting that the tangential velocity tends to zero outside the rotor boundary layer [2].

Further numerical and experimental studies showed that both Batchelor and Stewartson models could be valid depending on the flow condition. The shape of the cavity, the distance between the rotor and stator, the shroud clearance, the supply of any superposed flow to the cavity and the nature of the flow; i.e., whether it is laminar or turbulent, are among the important parameters in determining the flow structure.
However, for most rotor-stator systems in engineering practice, the Batchelor model often best describes the flow [3].

A schematic of the flow structure in a shrouded rotor-stator cavity with superimposed flow is shown in Figure 3. This structure is what Batchelor model describes. As can be seen the fluid moves radially outward on the rotor boundary layer and inward in the stator boundary layer. In between the two boundary layers a rotating core of fluid is formed. Egress occurs due to the so called disc-pumping effect, i.e., the centrifugal forces at the rotor side driving the cavity fluid radially outward. The rotating core that is formed inside the disc cavity causes a positive radial pressure gradient along the cavity, resulting in pressure drop inside the disc cavity below the main annulus pressure. This allows for ingress or ingestion of the external fluid into the cavity through the rim seal [3], [8], [9].

![Figure 3. The schematic of the flow structure in a shrouded disc cavity with a supply of a superimposed flow](image)

### 4.1.1 The influence of the gap ratio

Four flow regimes have been identified by experiments done by Daily and Nece [10] for the flow between an enclosed rotor and stator disc cavity with no superimposed flow, depending on the gap ratio, G and the nature of flow (laminar or turbulent). These regimes which are shown in Figure 4, as a function of G and rotational Reynolds number ($Re_\theta$), are:

- **Regime 1:** laminar flow, small clearance, merged boundary layers
- **Regime 2:** laminar flow, large clearance, separate boundary layers
- **Regime 3:** turbulent flow, small clearance, merged boundary layers
- **Regime 4:** turbulent flow, large clearance, separate boundary layers
As can be seen, for large distance between the discs (Regimes 2 and 5), the flow structure resembles more the Batchelor-type flow, consisting of an inviscid core in the wheel-space and the radial flow occurring only in the boundary layers of the discs [3].

A series of tests by Bayley and Owen [11] on an unshrouded system and gap ratios between 0.008 and 0.03, showed a decrease in the angular velocity of the rotating core as the gap ratio increased. At sufficiently high gap ratios, the core region totally disappeared [2]. When the gap between the discs increases above a certain limit, the effect of the stator decreases and the flow behaves similar to the flow close to a free disc [2].

4.1.2 The influence of the superimposed flow

In absence of any superimposed flow, the flow inside an enclosed or shrouded disc cavity, i.e., with no or restricted radial outflow, follows the Batchelor model, with a rotating core in between the rotor and stator boundary layers [2]. Supply of a superimposed air which moves through the cavity and radially outward, disturbs the core rotation inside the cavity and at high gap ratios can completely suppress it. In that case, the flow structure changes more toward the Stewartson flow model [2].

4.1.3 The influence of the shroud

Measurements have shown that the core rotation is negligible when the confined space between the discs is open to the atmosphere, while if the space is enclosed, e.g., with a peripheral shroud, there would be a rotating core [2], [3]. Hence the open system can be better characterized by Stewartson model while Batchelor model is more valid for the enclosed system.
4.2 Axial turbine cavity flow

The turbine system is a practical example of the rotor-stator system described in section 4.1, sharing the same flow characteristics.

The volume confined between the rotating and the stationary disc in a turbomachine is called wheel-space or rotor-stator cavity. Figure 5 is an illustration of a typical axial gas turbine stage, showing the rim seal and the wheel-space.

![Figure 5. An overview of one stage in an axial turbine](image)

The presence of this clearance is necessary in order to make relative motion of the rotating and stationary parts feasible, and prevent wear of components [2].

Since the cavity is open to the external environment, i.e., the main annulus, flow exchange, although restricted by the rim seal which partially shrouds the cavity at its outer radius, can occur between the mainstream and the wheel-space [2]. Studies have shown that ingress is usually induced mainly due to the main annulus flow, rather than by the disc rotation. As the flow passes the vanes and the blades in the annulus, a pressure gradient is formed radially outward through the rim seal at different circumferential locations. These pressure asymmetries in the main flow result in inflow and outflow through the rim seal. Where the pressure in the cavity is locally lower than the main annulus pressure, ingress occurs [8], [9], [12]. The dominance of the effect of the external flow on the ingress was first suggested by Abe et al. [13]. A series of experiments conducted by Phadke and Owen [14] also showed that above a certain values of external flow Reynolds number to rotational Reynolds number ratio \((Re_e/Re_\phi)\), the effect of the external flow on the ingress becomes dominant.

Ingestion of hot gas from the main annulus into the wheel-space is a big concern related to the safety of gas turbines and is shown previously to be influenced by the ratio of the sealing air and the main flow velocities, and the clearance and the shape of the rim seal [8]. The effect of the ingress on the flow structure is to reduce the core rotation and to increase the moment on the rotor [3].

As mentioned earlier, the characteristics of cavity flow is influenced by the distance between the discs, the presence of a stationary or rotating shroud, and supply of any superimposed flow [2]. The flow in a gas turbine is highly turbulent, with rotational Reynolds numbers being usually in the range of \(2\times10^7\) to \(3\times10^7\) for a high-pressure stage. Besides, the disc cavity spacing is usually sufficiently large to allow two separate...
boundary layers to be formed. Hence, among the four regimes mentioned above, regime 4 is thought to be more relevant to this case [2].

4.2.1 Pressure fluctuation inside the disc cavity

Unsteady pressure measurements in earlier studies have detected low frequency pressure fluctuations. Several theoretical, experimental and computational studies have been conducted during the past ten years in order to better understand these pressure structures. These studies included different geometries of the cavity and the rim seal, full geometry including the hot gas path with the vanes and blades, or considering the cavity alone. CFD calculations showed a rotating flow structure inside the cavity, which has been detected by measurements.

A study by Jakobi et al. [15] detected rotating low pressure regions consisting of three large scale low pressure regions. These low-frequency fluctuations were not induced by vanes and blades interaction since their frequency was an order of magnitude less than the blade-passing frequency. The hot gas was mostly ingested into these rotating low pressure areas. The low pressure structures which rotated at about 80% of the rotor speed were completely suppressed above a certain cooling air flow rate. This upper limit was believed to depend on the seal configuration and turbine’s operating conditions. Similar time-dependent unstable flow structures have been observed previously, both numerically and experimentally, for the case of the flow inside the cavity between two co-rotating discs with throughflow. ([2] has given a review over these studies.)

In a similar study, Cao et al. [4] modeled the flow inside an unshrouded disc cavity together with a section of the annulus, without considering the vanes and blades. Their study also showed unsteady rotating structures with an angular velocity of 90-97% \( \Omega \). The large-scale structures were believed by the authors to be associated with the main annulus and cavity flow interaction. The results showed a reduction in the scale and strength of these structures, and as expected, the flow ingestion from the main annulus to the cavity for a smaller gap ratio.

In another study by Julien et al. [16] a 74 degree sector of the turbine was modeled including 9 vanes and 12 blades and the rim cavity. The study investigated the flow structure for three different purge flows. Six fluctuating regions were detected inside the rim cavity, and around 30 structures were suggested by the authors to be detectable for a full 360 degree model.

A very recent numerical study by Wang et al. [17] included the entire annulus with all vanes and blades together with a double seal forming an outer rim and an inner disc cavity. Between the rotor and stator boundary layers, the results showed 12 cells, rotating at the same direction and at about 86% of the rotor speed. These structures were observed in the outer rim cavity at low and medium purge flow rate. For low purge flow rate however, the cells were found to be slightly disturbed by the ingress and egress through the rim. At high purge flow, these structures were suppressed close to the inner radius of the rim cavity, while still distinguishable close to the rim seal. Farther from the rim seal, inside the inner cavity, these structures were less prominent.
4.3 Test turbine

The study is based on an experimental rig consisting of a newly designed blisk for the turbomachinery lab, Energy Department, KTH. Presently, the turbine is in its final production stage, and is to be installed in a few months.

The turbine is a one-stage subsonic steam turbine model, containing 42 stator vanes and 60 rotor blades, and is operated with air. A schematic of the test turbine is shown in Figure 6, where the front cavity in the new design which is modeled in this study is marked. The casing design includes a recess 5mm downstream of the vane, as can be seen in the magnified picture of the casing.

![Figure 6. Schematic of the one-stage turbine rig](image)

The rim seal design is of overlapping type with radial clearance.

The radius of the hub is 177.5 mm. The purge air enters the cavity through a labyrinth seal at the radius of about 94 mm. The purge Air is supplied from the same source to the main channel and the disc cavity. The air is fed to the cavity through a labyrinth seal. More information on the geometry and operating conditions of the turbine is given in Table 1 and Table 2.
Table 1. Geometrical characteristics of the turbine

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tip-hub ratio</td>
<td>1.148</td>
</tr>
<tr>
<td>Blade pitch-chord ratio (mid)</td>
<td>0.85</td>
</tr>
<tr>
<td>Blade aspect ratio (mid)</td>
<td>1.47</td>
</tr>
<tr>
<td>Vane axial chord [mm]</td>
<td>24.1</td>
</tr>
<tr>
<td>Blade axial chord (mid) [mm]</td>
<td>17.81</td>
</tr>
<tr>
<td>Distance between vane trailing edge and blade leading edge (mid) [mm]</td>
<td>10</td>
</tr>
<tr>
<td>Gap ratio (G) [-]</td>
<td>0.05-0.078</td>
</tr>
<tr>
<td>Seal clearance (Gc) [-]</td>
<td>0.027</td>
</tr>
</tbody>
</table>

Table 2. Operating condition of the turbine

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total to total isentropic efficiency (ηis) (%)</td>
<td>91.1</td>
</tr>
<tr>
<td>Total to static pressure ratio (Π) [-]</td>
<td>2.139</td>
</tr>
<tr>
<td>Degree of reaction (Λp,mid) [-]</td>
<td>0.389</td>
</tr>
<tr>
<td>n [rpm]</td>
<td>10270</td>
</tr>
<tr>
<td>Re [ -]</td>
<td>6.9×10^4</td>
</tr>
</tbody>
</table>
4.4 Governing equations

CFD computations are based on solving the main fluid flow equations, i.e., the mass conservation (continuity) and conservation of momentum (Navier-Stokes) equations which describe the flow motion. An overview of the governing equations is given in this chapter. The content of this chapter is mainly based on the introductory book to CFD by [18].

Navier-Stokes equation is obtained by applying the Newton’s second law of motion to the flow, assuming a Newtonian fluid, i.e., viscous stresses are proportional to the rates of deformation. These equations are given below in a Cartesian coordinate system:

**Continuity equation:**

\[ \frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{u}) = 0 \]  \hspace{1cm} (4.1)

**Momentum equations:**

\[ \frac{\partial (\rho u)}{\partial t} + \text{div}(\rho u \mathbf{u}) = -\frac{\partial p}{\partial x} + \text{div}(\mu \text{grad } \mathbf{u}) + S_{Mx} \]  \hspace{1cm} (4.2a)
\[ \frac{\partial (\rho v)}{\partial t} + \text{div}(\rho v \mathbf{u}) = -\frac{\partial p}{\partial y} + \text{div}(\mu \text{grad } v) + S_{My} \]  \hspace{1cm} (4.2b)
\[ \frac{\partial (\rho w)}{\partial t} + \text{div}(\rho w \mathbf{u}) = -\frac{\partial p}{\partial z} + \text{div}(\mu \text{grad } w) + S_{Mz} \]  \hspace{1cm} (4.2c)

where \( S_M \) is the momentum source term, including mainly the contribution due to the body forces such as gravity, but also a small contribution due to viscous stress terms. \( p \) refers to pressure which is a normal stress. The divergence term on the left hand side is referred to as convective or advection term, and the divergence term on the right hand side is called diffusive term.

**Energy equation:**

The energy equation is derived from the first law of thermodynamics:

\[ \Delta E = \Delta Q + \Delta W \]  \hspace{1cm} (4.3)

where \( Q \) is the heat transfer to the fluid particle due to conduction, and \( W \) is the work done on the fluid particle by surface stresses. The fluid energy, \( E \), is furthermore defined as the sum of internal, kinetic and potential energy.

The energy equation can be derived as:

\[ \frac{\partial (\rho i)}{\partial t} + \text{div}(\rho i \mathbf{u}) = -p \text{div } \mathbf{u} + \text{div } (k \text{ grad } T) + \Phi + S_i \]  \hspace{1cm} (4.4)

where \( i \) refers to internal energy. Potential energy which is related to gravitational forces is considered as a body force, and hence included in the source term. Fourier’s law of heat conduction is used to relate the heat flux to temperatures. The first term on the right hand side is due to the work done by pressure. The
terms related to the viscous stresses included in the kinetic energy are deleted with the equal terms in the
work after being subtracted from the right hand side of the equation. The effects due to the viscous
stresses, remaining in the work term, are described here by the dissipation function, $\Phi$. For more details
regarding the derivation of the equations, the reader is referred to [18].

In addition to the main flow equations mentioned above, transport equation for scalar quantities such as
temperature or species concentration might be required to be solved depending on the nature of the flow
problem.

Based on the commonalities between the equations above, transport equation for a general property $\phi$
which could be a vector or a scalar quantity can be written as below:

$$\frac{\partial (\rho \phi)}{\partial t} + \text{div}(\rho \phi \mathbf{u}) = \text{div}(\mu \text{grad } \phi) + S_\phi$$  \hspace{1cm} (4.5)

The terms in this equation can be expressed from left to right as: rate of increase of $\phi$ of fluid element, net
rate of flow of $\phi$ out of fluid element, rate of increase of $\phi$ due to diffusion and rate of increase of $\phi$ due
to sources.

4.4.1 Turbulent flow

Above a certain Reynolds number known as critical Reynolds number, there will be a transition from
laminar to turbulent flow which is characterized by a random state of motion in which the velocity and
pressure change continuously with time. Many engineering problems have a turbulent nature.

Various properties of turbulent flow can be expressed as a sum of a steady mean value and a fluctuating
component, using the so called Reynolds decomposition. For velocity and pressure we can write:

$$\mathbf{u} = \mathbf{U} + \mathbf{u}'$$ \hspace{1cm} (4.6)

$$p = P + p'$$ \hspace{1cm} (4.7)

Substituting these terms in the continuity and Navier-Stokes equations (5.2a) to (5.2c), and time averaging
the equations using the rules for time-averaging the fluctuating properties, the so called Reynolds-averaged
Navier-Stokes (RANS) equations can be obtained for a flow with constant viscosity as below:

**Continuity:**

$$\text{div } \mathbf{u} = 0$$ \hspace{1cm} (4.8)

**Momentum:**

The density-weighted averaged form of the momentum equations for compressible turbulent flows are
given below for the case where mean density changes but the effect of density fluctuations are negligible:

$$\frac{\partial (\rho \mathbf{U})}{\partial t} + \text{div}(\rho \mathbf{U} \mathbf{U}) = -\frac{\partial P}{\partial x} + \text{div}(\mu \text{grad } \mathbf{U}) - \text{div}(\rho \mathbf{u}' \mathbf{u}') + S_{Mx}$$ \hspace{1cm} (4.9a)

$$\frac{\partial (\rho \mathbf{V})}{\partial t} + \text{div}(\rho \mathbf{V} \mathbf{U}) = -\frac{\partial P}{\partial y} + \text{div}(\mu \text{grad } \mathbf{V}) - \text{div}(\rho \mathbf{v}' \mathbf{u}') + S_{My}$$ \hspace{1cm} (4.9b)
\[
\frac{\partial (\bar{\rho} W)}{\partial t} + \text{div}(\bar{\rho} U) = -\frac{\partial \bar{P}}{\partial z} + \text{div}(\mu \text{ grad } \bar{W}) - \text{div}(\bar{\rho} w' u') + S_Mz \quad (4.9c)
\]

Compared with the Navier-Stokes equations, one extra term (third term on the right hand side) appears in the RANS. The new term which includes the fluctuating velocity components is associated with convective momentum transfer due to turbulent eddies. The extra turbulent stresses including three normal and three shear stresses are called Reynolds stresses.

RANS equations are very useful for solving engineering problems with turbulent nature, where the details of the turbulent fluctuations can be neglected. It should be mentioned that these forms of equations are widely used in commercial CFD software for compressible turbulent flow.

### 4.4.2 Turbulence models

Among different numerical methods which has been developed for solving turbulent flow problems, the most popular and cost effective approach is based on RANS. Turbulence models need to be developed to compute turbulent flows with the RANS equations. The extra terms which appear in RANS due to turbulent fluctuations interactions are modelled with the turbulence models. This approach is what is used today in commercially available CFD codes. Two most widely used RANS turbulence models are k-\(\varepsilon\) and k-\(\omega\) models, which are briefly explained in this section.

#### 4.4.2.1 Standard k-\(\varepsilon\) model

The k-\(\varepsilon\) model is a two- equation model which solves the transport equations for turbulent kinetic energy and dissipation of turbulent kinetic energy.

The instantaneous kinetic energy, \(k(t)\) of a turbulent flow is the sum of the mean kinetic energy and the turbulent kinetic energy.

\[
k(t) = K + k \quad (4.10)
\]

where,

\[
k = \frac{1}{2} (u'^2 + v'^2 + w'^2) \quad (4.11)
\]

\[
K = \frac{1}{2} (U^2 + V^2 + W^2) \quad (4.12)
\]

The deformation rate of a fluid element can also be written as a sum of a mean and a fluctuating component:

\[
s_{ij}(t) = S_{ij} + s_{ij}' \quad (4.13)
\]

The work done by the smallest eddies against viscous stresses results in dissipation of turbulent kinetic energy. The rate of dissipation is written as:

\[
\varepsilon = 2\nu s_{ij}' s_{ij}' \quad (4.14)
\]

The turbulent or eddy viscosity can be written as:
\[ \mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \]  

(4.15)

The standard \( k-\varepsilon \) model uses two transport equations, one for the turbulent kinetic energy \( (k) \), and one for the rate of dissipation \( (\varepsilon) \):

\[
\frac{\partial (\rho k)}{\partial t} + \text{div}(\rho k \mathbf{U}) = \text{div} \left[ \frac{\mu_t}{\sigma_k} \text{grad} k \right] + 2 \mu_t S_{ij} \cdot S_{ij} - \rho \varepsilon \tag{4.16}
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \text{div}(\rho \varepsilon \mathbf{U}) = \text{div} \left[ \frac{\mu_t}{\sigma_\varepsilon} \text{grad} \varepsilon \right] + C_{1\varepsilon} \frac{\varepsilon}{k} 2 \mu_t S_{ij} \cdot S_{ij} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \tag{4.17}
\]

The terms in these equations can be described from left to right as rate of change of \( k \) or \( \varepsilon \), transport of \( k \) or \( \varepsilon \) by convection, transport of \( k \) or \( \varepsilon \) by diffusion, rate of production of \( k \) or \( \varepsilon \), and rate of destruction of \( k \) or \( \varepsilon \).

The constants in the equation above are obtained by data fitting for a wide range of turbulent flow and are assigned the following values:

\[
C_\mu = 0.09 \quad \sigma_k = 1.00 \quad \sigma_\varepsilon = 1.30 \quad C_{1\varepsilon} = 1.44 \quad C_{2\varepsilon} = 1.92 \tag{4.18}
\]

More details regarding these constants and derivation of the equations above can be found in [18].

The standard \( k-\varepsilon \) model is the simplest turbulence model. In general it has poor performance for cases such as some unconfined flows, swirling and rotating flows. It gives inaccurate predictions at the layer between the fully turbulent region and the viscous sub-layer in low Reynolds number flows. The model over predicts the shear stress and suppresses separation. It is therefore not appropriate for aerospace applications where flows over airfoils are involved. However due to its simplicity and robustness the \( k-\varepsilon \) model is still widely used in many industrial flow computations.

Several advanced variation of the standard \( k-\varepsilon \) model has been proposed in order to resolve some of the weaknesses of this model. These model will however not be discussed here.

### 4.4.2.2 \( k - \omega \) model

The standard \( k - \omega \) model is one of the more recent turbulence models compared to standard \( k-\varepsilon \) model. \( k - \omega \) model uses turbulence frequency \( \omega = \frac{\varepsilon}{k} \) as the second variable instead of \( \varepsilon \). The eddy viscosity is then given by:

\[ \mu_t = \frac{\rho k}{\omega} \]  

(4.19)

The transport equation for \( k \) and \( \omega \) for turbulent flows at high Reynolds number can be written as:

\[
\frac{\partial (\rho k)}{\partial t} + \text{div}(\rho k \mathbf{U}) = \text{div} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \text{grad}(k) \right) \right] + P_k - \beta^* \rho k \omega \tag{4.20}
\]
where $P_k$ is the rate of production of turbulent energy, and

$$
\frac{\partial (\rho \omega)}{\partial t} + \text{div}(\rho \omega \mathbf{U}) = \text{div} \left[ \left( \mu + \frac{\mu_t}{\sigma_\omega} \text{grad}(\omega) \right) \right] + \gamma_1 \left( 2\rho S_{ij} \cdot S_{ij} - \frac{2}{3} \rho \omega \frac{\partial U_i}{\partial x_j} \delta_{ij} \right) - \beta_1 \rho \omega^2 \quad (4.21)
$$

where the constants of $k$- and $\omega$-equation are:

$$
\sigma_k = 2.0 \quad \sigma_\omega = 2.0 \quad \gamma_1 = 0.553 \quad \beta_1 = 0.075 \quad \beta^* = 0.09 \quad (4.22)
$$

The $k-\omega$ model has much better near-wall performance compared to the standard $k-\varepsilon$ model, but it has the disadvantage of producing results which are dependent on the assumed free stream value of $\omega$.

Shear stress transport (SST) $k-\omega$ is a more recent model, and is a hybrid model between the $k-\varepsilon$ model and the $k-\omega$ model. This model uses a transformation of the $k-\varepsilon$ model into $k-\omega$ model in the near-wall region, and the $k-\varepsilon$ model in the fully turbulent region far from the wall. The $k$-equation in SST model is the same as in the $k-\omega$ model mentioned above, but the $\varepsilon$-equation is transformed into an $\omega$-equation using the substitution of $\varepsilon = k\omega$. This results in appearance of one extra source term in the equation (5.21).

Both the $k-\omega$ model and the SST $k-\omega$ model have good performance for aerospace applications [18].

### 4.4.2.3 Near-wall modeling

Flow close to a wall boundary can be divided into the inner region and outer region. The inner region can itself be sub-divided into a viscous sub-layer which is closest to the wall, where the viscous effects are dominant, and the so called log-law region, where the turbulent stresses are larger than the viscous stresses. Between these two layers a buffer layer is identified, where effect of the turbulent and viscous stresses are of the same magnitude. Far from the wall, in the outer layer, the flow is fully turbulent and free from the viscous effects. The inner region constitutes about 10-20% of the total thickness of the wall layer [18].

In order to capture the near wall behavior of the flow, finer grids close to the wall are required. The wall function method, which is used in CFD codes together with the turbulence models, uses empirical formulas to model the high gradient shear close to the wall, when the mesh is relatively coarse. This can save significant computational time and resources [19].

### 4.4.3 Finite volume method

The finite volume method which is the most widely used CFD technique for solving engineering problems, consists of three main steps. First, the domain is divided into discrete control volumes via the process of grid generation. In the second step, the governing equations are integrated over the finite control volumes, and also over time in case of unsteady problems, and the integral equations are discretized, i.e., converted to a system of algebraic equations, using an appropriate discretization method. Finally, the algebraic equations are solved using an iterative method [18].

The discretization schemes in CFD approximate the gradients based on series expansion such as Taylor series approximation. The error of the approximation which is due to the neglected truncated terms of the
expansion can be reduced by reducing the grid size. The accuracy of the results is also dependent on the order of the approximation used by the discretization scheme. Higher-order accuracy schemes result in faster reduction of error with grid refinement, and hence increasing the accuracy of the solutions. However, high-order accuracy schemes have the disadvantage of being less robust besides increasing the computational time. They can produce numerical stability problems, and result in oscillatory behavior in the region with high gradients. This can be due to the fact that the boundedness criterion of the scheme is not satisfied.

Transient discretization schemes can be either explicit or implicit. Explicit method solves the equation for the value of the variable at the new time using its value at the old time, whereas the implicit method solves the equation by involving the states at both the old and new time.

5 CFD modeling

5.1 Main annulus model

5.1.1 Computational domain

The main annulus of the turbine is modeled in order to obtain the boundary conditions to the cavity model. The geometry of the blade and vane passages was created in TurboGrid 14 (ANSYS), using the hub, shroud and blade profile curves imported from the CAD files. A periodic sector of the annulus, including one stator blade passage (8.57°) and one rotor blade passage (6°) is modeled instead of the full (360 degree) annulus. The computational domain is shown in Figure 7. The domain consists of a stationary subdomain including the stator vane and a rotating subdomain which includes the rotor blades. The outlet of the rotating subdomain is considered stationary. The stationary and rotating subdomains are connected with an interface which is located 4.6 mm (19% of the vane axial chord) downstream of the vane trailing edge.

There is clearance of 0.3 mm between the blade tip and the shroud. It should be mentioned that the original design of the blade included fillets at the junction of blade and hub, which were, for sake of simplicity, neglected in this model. The channel casing design included a recess 5 mm downstream of the stator (see Figure 6), which was replaced with a ramp, as can be seen in Figure 7, in order to prevent problems with convergence. Moreover, as can be seen in Figure 5, the rotor hub at leading edge is curved, but is considered cylindrical in this model in order to avoid problems with the placement of the interface later on.

![Figure 7. The 3D view of the main annulus computational domain](image)
5.1.2 Grid structure

The hexahedral mesh was created in TurboGrid 14.0 (ANSYS) using the Automatic Topology and Meshing (ATM) feature. This method provides high quality structured meshes with automatic refinement in areas with large variations in the geometry. The grid structure is shown in Figure 8. The stationary and rotating mesh were created separately and then merged together in the solver using a proper interface.

The initial mesh was refined in two steps in order to perform grid sensitivity analysis. The results of the analysis are presented in section 5.1.5. The mesh statistics for the coarse, medium and fine meshes are given in Table 3.

Table 3. The mesh statistics for three meshes

<table>
<thead>
<tr>
<th>Mesh type</th>
<th>No. of nodes</th>
<th>No. of cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>250848</td>
<td>230504</td>
</tr>
<tr>
<td>Medium</td>
<td>701039</td>
<td>660588</td>
</tr>
<tr>
<td>Fine</td>
<td>1128983</td>
<td>1072100</td>
</tr>
</tbody>
</table>

5.1.3 Operating and boundary conditions

The rotational speed of the rotor is 10270 rpm. The boundary conditions to the main annulus model were taken from a previous design study [20]. Total pressure boundary condition was applied at the inlet to the main annulus flow domain, and static pressure boundary condition was set to the domain outlet (see Table 4). All other boundaries were set to no-slip walls.
5.1.4 Interfaces

Since only one passage of the turbine is modeled, periodic boundaries are used for the vane and the blade passages on circumferential direction, using the rotational periodicity interface model. There is also a general connection interface at the blade shroud tip. Moreover, there is a frame and pitch change between the stator and the rotor. A mixing plane interface is therefore required to connect the two domains. The location of the interfaces is shown in Figure 9.

The interface connects the mesh at two sides. Proper mesh connection needs to be chosen for each interface. If the nodes at one side of the interface are in correspondence with the other side, and there is no domain change across the interface, direct (one to one) mesh connection is used. If the nodes location on either side of the interface does not match, General Grid Interface (GGI) method is used for grid connection at the interface. Setting the mesh connection option to automatic allows the solver to choose the most appropriate type of connection.

5.1.5 CFD calculations

ANSYS-CFX (version 15.0), which is a finite volume commercial CFD code was used as solver. The (SST) k-ω was selected as turbulence model, since it has been a common choice for rotor-stator flow modeling.

A summary of the model setup is given in Table 4.

In order to enable transition between the stationary and rotating domain, the so called stage interface model is used for steady-state modeling. Using the stage model, the flow properties between the rotating and stationary frame are circumferentially averaged in a mixing plane interface. This model requires only a single periodic passage to be modeled. While often giving reasonably good results compared to transient simulations, the model does not account for circumferential distortions at the inlet to a blade row or the interaction between the rotor and stator [21].

Figure 9. The boundary conditions to the computational domain
5.1.5.1 Transient blade row model

Unsteady calculations were performed using the new methods for transient blade row model, which is a new development available since ANSYS CFX 14.0.

In the conventional methods of modeling transient rotor-stator with unequal pitch increasing the number of vanes or blades modeled was necessary in order to approach a pitch ratio close to one. The disadvantage of these methods is that they are very costly since they sometimes require addition of several vanes and blades. The transient blade row model uses instead the principle of so called “phase shifted periodic boundary conditions” to overcome the non-unity pitch problem, and hence allowing less number of vanes and blades being modeled. Using this principle, pitch-wise boundaries are periodic to each other at different time instances. The new methods provide accurate results while at the same time reduce the simulation time considerably [22].

The transient blade row model in CFX offers two new methods: Fourier transformation and Time transformation methods. Both of these methods use the common principle.

Time transformation method, which is only valid for compressible flow, is used in this study. In this method the problem of unequal pitch is handled by applying a time transformation to the flow equations so that simple periodic boundary conditions on the pitch-wise boundaries can be used. There is however some limitations in using the time transformation method. The pitch ratio between the blade and vane passages should not deviate much from unity and should fall within a certain range, as described below:

\[
1 - \frac{M_\omega}{1 - M_\theta} < \frac{P_S}{P_R} < 1 - \frac{M_\omega}{1 + M_\theta},
\]

where \( M_\omega \) is the Mach number related to the rotor rotational speed, \( M_\theta \) is the tangential Mach number and \( \frac{P_S}{P_R} \) is the pitch ratio between the stator and rotor.

This criterion can be achieved by choosing appropriate number of passages for each component.

Instabilities in the solution can occur if the pitch ratio is too close to the limits. With the conditions included in this study, the lower and upper limits for the pitch ratio were 0.65 and 1.54 respectively. When only a single passage for both blade and vane was chosen, the pitch ratio was equal to 1.43, which is considered by the solver to be reasonably good.

5.1.5.2 Spatial discretization

The high-resolution scheme is chosen for discretization of advection terms. The scheme is second-order accurate, but changes toward the first-order scheme when necessary, in order to maintain the boundedness [22]. The scheme is therefore more accurate than the first-order upwind scheme.

For approximations of spatial derivatives of the diffusion and pressure terms, ANSYS CFX uses finite-element shape functions [22].

5.1.5.3 Temporal discretization

Second-order backward Euler scheme is used for discretization of the transient terms in the governing equations. The scheme is implicit and second-order accurate.
Time steps are chosen based on the stator blade passing period since it has the longest passing period. Forty time steps per passing period is chosen, which gives the time step of $6.9551 \times 10^{-5}$ s. The simulation is further performed for the total time of 0.00278 s equivalent to 20 passing period. This duration corresponds to about half of one revolution of the rotor.

Table 4. The main annulus model setting

<table>
<thead>
<tr>
<th>Fluid model</th>
<th>Fluid</th>
<th>Heat transfer model</th>
<th>Turbulence model</th>
<th>SST k-ω</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interface model</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mixing plane interface</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interface model</td>
<td>General connection</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mixing model (steady state case)</td>
<td>Stage</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mixing model (transient case)</td>
<td>Transient rotor stator</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mesh connection</td>
<td>GGI</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interface model</td>
<td>Rotational periodicity</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mesh connection</td>
<td>Automatic</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shroud tip interface</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Interface model</td>
<td>General connection</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mesh connection</td>
<td>GGI</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Inlet boundary condition</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Total Pressure</td>
<td>2.167 bar</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Total temperature</td>
<td>345 K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Outlet boundary condition</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Static pressure</td>
<td>1.013 bar</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Transient setting</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Analysis type</td>
<td>Transient blade row model</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Transient method</td>
<td>Time transformation</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>No. of time step per passing period</td>
<td>40</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>No. of passing period per run</td>
<td>20</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Solver control</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Advection scheme</td>
<td>High resolution</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Temporal scheme</td>
<td>2nd order backward Euler</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>RMS Residual target</td>
<td>0.00001</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Convergence was controlled by monitoring various flow properties such as pressure, mass flow rate and velocity at different radial and circumferential positions throughout the domain: the inlet and outlet, and downstream and upstream of the vane and blade leading and trailing edges. For the steady-state simulation, convergence was confirmed to be achieved when all the residuals fell below $10^{-5}$, and the monitored properties reached steady values.

The results from the steady-state calculations were used as initial condition for the unsteady calculations.

For the transient simulation, it was ensured that as the simulation was in progress, the value of monitored properties fluctuated in a repeating pattern, corresponding to the blade passing frequency.

### 5.1.6 Grid sensitivity study

Grid sensitivity study was performed on the main annulus model for three meshes using the steady model, in order to ensure that the solutions are independent of the mesh size.

As an example, the total to total efficiency and mass flow rate are compared for the three meshes in Figures 10 and 11 respectively. As can be seen, when the mesh is refined from 660588 cells to 1070100 cells, the changes in the efficiency and mass flow rate are less than 0.1%. The mesh with medium number of cells was hence used for the transient simulations.

![Figure 10. Total to total efficiency compared for three meshes](image-url)
The pressure profiles over the span at the vane trailing edge are also compared for the three different meshes (Figure 12). As can be seen the values for the medium and fine meshes are very close. However, a slight oscillatory convergence can be seen between the two meshes at certain span positions.

![Static pressure profile 1 mm downstream of the vane trailing edge for three meshes](image-url)

**Figure 12.** Static pressure profile 1 mm downstream of the vane trailing edge for three meshes
5.1.7 Main annulus solution

Convergence was achieved for the main annulus simulation. Figure 13 shows the pressure distribution over the passage at the mid-span. The velocity profile downstream (1.4 mm) of the vane trailing edge and the static pressure profile upstream (~1 mm) of the blade leading edge are shown in Figure 14. These profiles are visualized at the boundary planes to the cavity domain and are used as boundary conditions for modelling the cavity in the next step.

Figure 13. Instantaneous static pressure distribution along the passage at mid-span

Figure 14. Instantaneous velocity and pressure contour downstream of the vane trailing edge and upstream of the blade leading edge, respectively.
5.2 Disc cavity model

5.2.1 Geometry creation

The geometry of the disc cavity (shown in Figure 6) is created in ANSYS Design Modeler. The labyrinth seal is not included in this model. The purge air enters at the disc inner radius. The axial part is taken slightly away from the vane trailing edge and the blade leading edge. The distance of the axial part which is modeled is 7.7mm.

The geometry of the cavity is shown in Figure 15. Figure 15(a) shows the geometry of the cavity with the original rim seal design. A modified rim seal geometry, with larger clearance is shown in Figure 15(b). The idea for modeling the cavity with wider gap was to increase the probability of capturing the unsteady phenomenon inside the cavity.

![Figure 15. The lateral cross section view of the computational domain with (a) the original design of the rim seal and (b) the larger rim seal gap](image)

5.2.2 Grid structure

ANSYS ICEM (v14.0) was used to produce the grids. This was done using block-structured grids, producing quadrilateral cells in 2D mesh and hexahedral cells in 3D mesh. In this method the flow domain is divided into several blocks and each block is meshed separately. Each block has a structured mesh, and is joined with the adjacent blocks [18].

The 2D mesh is shown in Figure 16. It is consists of 1993 nodes and 1845 quadrilateral cells. The 3D mesh was obtained by rotating the 2D mesh about the axis of rotation (Figure 17). The 3D mesh is composed of 717480 nodes and 664200 hexahedral cells.
Figure 16. The 2D mesh of the disc cavity with larger rim seal gap

Figure 17. The 3D mesh of the disc cavity, obtained by rotation of the 2D mesh
Mesh quality

The near-wall region was treated by refining the mesh close to the walls, using the bunching function in ICEM for structured meshing. Further refinement adversely affected the mesh quality, by increasing the skewness at regions with curvatures.

The quality of the mesh is important in order to ensure numerical stability and convergence. The mesh quality was checked in ICEM. The CFX solver also does an automatic evaluation of the mesh quality. The Mesh orthogonality, expansion ratio and aspect ratio which are among important mesh quality indicators, are evaluated by the solver.

Orthogonality was poor for only two cells (with minimum angle of 13.5 degree), in acceptable range for 2% of the cells, and good for rest of the cells. The aspect ratio and expansion ratio were in good range for all cells.

5.2.3 Boundary conditions

The boundaries of the fluid domain are shown in Figure 18 in cross section view.

Figure 18. The disc cavity domain boundaries

The main flow path inlet and outlet boundary conditions were imported from solution of the main annulus modeling. Time-averaged velocity and total temperature profiles were used at the main inlet boundary, and time-averaged static pressure profile was used at the outlet. The profiles obtained from the main annulus were expanded to the entire periphery using the right number of vanes and blades. The imported boundary contours are shown in Figure 19.

For the purge air at the inner inlet, mass flow rate and total temperature were used.

The non-dimensional cooling air mass flow rate is defined as:

\[
C_w = \frac{\dot{m}_{purge}}{\mu R_{hub}}
\]  (5.1)
where $R_{hub}$ is the hub radius.

Two different purge flows were tested:

Low purge: $C_w=1000$, equivalent to 0.105% of the main flow,
High purge: $C_w=10000$, equivalent to 1.05% of the main flow

All other boundaries were set to no-slip wall.

5.2.4 CFD calculation

ANSYS CFX was used for simulation of the flow inside the rotor-stator disc cavity. $k-\varepsilon$ model with wall functions was used as turbulence model. High resolution scheme and second-order backward Euler schemes were used for discretization of the advection and transient terms. The chosen time step was 0.00005 s, which corresponds to 116 time steps per disc revolution. The total simulation time was set to 0.1 s, corresponding to about 17 disc revolutions.

A summary of the model setting parameters is given in Table 5.

A steady-state simulation was first performed. Flow properties such as pressure and mass flow rate were monitored during the simulation at points above the rim seal (in the main annulus) and below the rim seal (inside the cavity). Convergence was confirmed when the flow properties reached a steady-state value.

The converged steady-state results were used as initial values for the unsteady simulation.
A scalar variable was added to the model, for which the transport equation was solved. The value of this variable was set to 1 at the main annulus inlet and 0 at the purge flow inlet, in order to track the ingested flow through the rim seal down into the disc cavity.
6 Results and Discussion

The flow inside the disc cavity of the test turbine was simulated, using the boundary conditions from the solution of main annulus flow simulation. To validate the cavity model solution, the pressure structure in the cavity mid-plane is compared with the main annulus model solution (see Figure 20). The slight difference seen should be related to influence from the wheel-space and the difference in the extension of the two models at the hub and shroud. As shown previously by the geometry of the two models, the hub curvature on rotor side is excluded in the main annulus model, while in the cavity model, a cylindrical shroud is considered.

![Figure 20. The instantaneous pressure structure at cavity mid-plane for the main annulus and cavity models](image)

The rotational Reynolds number ($Re_p = \rho \Omega R_{hub}^2 / \mu$) for both low and high purge flow cases is found to be $2.5 \times 10^6$. The overall pressure distribution for the main annulus and the disc cavity, at the mid-plane is shown for the two purge flow cases in Figure 21 and Figure 22. As expected, the pressure inside the cavity has increased at higher purge. No unsteady low pressure regions, as shown by some previous studies were detected inside the cavity. While this could partly be due to different cavity geometry and seal configuration, the model setting and accuracy of the simulation can also be a reason for not observing such structures in the results.

![Figure 21. Instantaneous pressure distribution at mid-plane for low purge flow case (left) in the main annulus and the disc cavity, (right) inside the disc cavity only](image)
Figure 23(a) and 23(b) show the streamlines inside the main annulus and the disc cavity in cross sectional view, for cases with low and high purge flow respectively. As can be seen flow recirculates inside the disc cavity, moving upward along the rotor disc and downward along the stator disc (this is more clearly seen by the velocity vectors in Figure 26 and Figure 28), meaning that the cavity’s gap ratio allows for two separate boundary layers to form on the stationary and rotating discs. As can be seen for both low and high purge cases, the purge flow, after entering axially, is drawn to the rotor boundary and moves upward along the rotor disc. A recirculation of a separated flow can also be seen below the purge air inlet for both cases. In case of low purge, the ingested main flow is drawn to the stator wall, but then after crossing the core region moves towards the rotor wall. Similar flow pattern has been found by earlier studies [3], [23], [24]. This is while no ingestion can be seen in high purge flow case, at the location shown in Figure 22b, as it is inhibited by the high superimposed flow. A so called source region is formed where the purge flow is supplied, from which the flow is fed to the boundary layers on the discs. As expected a much larger source region, filling above half of the cavity, due to the superimposed (purge) flow, can be seen in case of high purge flow compared to low purge case, which has disturbed the core region. As it can be seen by the streamlines, the outward flow along the rotor wall has become stronger while the inflow along the stator disc has weakened. The general structure of the flow is comparable with the structures found in previous studies [23], [24].
Figure 23. Streamlines inside the disc cavity for the case of (a) low purge flow and (b) high purge flow.
The time-averaged pressure variation in circumferential direction, 0.5 mm above the hub is shown in Figure 24 at several axial locations downstream of the vane trailing edge. The pressure coefficient \( C_p \) is calculated as below:

\[
C_p = \frac{(p - p_{avg})}{q}
\]  

(6.1)

where \( q \) is the dynamic pressure of fluid at rotor hub speed:

\[
q = \frac{\rho U^2}{2}
\]  

(6.2)

and \( U = \Omega \times R_{hub} \)  

(6.3)

As expected, the peak-to-peak variation has decayed as moving farther away from the vane. Figure 25 and Figure 27 show the pressure distribution in the main annulus, downstream of vanes, close to the rim seal, together with the streamlines inside the disc cavity at two different locations around the annulus, for low and high purge cases respectively. Figure 26 and Figure 28 show the velocity vectors on the same planes around the seal. For the case of low purge, ingress and egress of flow through the rim seal can be seen depending on the pressure distribution in the main annulus, near the seal region. This alternating inflow and outflow around the annulus through the seal can be more clearly seen by the velocity vectors on the same planes, as shown in Figure 26. As can be seen for case (a), where the pressure above the seal is high, flow is forced into the disc cavity. For case (b), a simultaneous ingress and egress can be seen, resulting in a recirculation at the seal.

At high purge flow, as can be seen in the Figures 27 and 28, egress occurs at both locations, due to higher cavity pressure. However as can also be seen by the velocity vectors, the egress is somewhat suppressed for case (a), where the hub pressure is higher.
Figure 24. Circumferential variation in time-averaged pressure coefficient, 0.5 mm above the hub at mid-plane and (a) $x=2.7$ mm (b) $x=3.1$ mm (c) $x=4.7$ mm downstream of the vane trailing edge.
Figure 25. The pressure contours in the main annulus at mid-plan, and the streamlines on two planes at two different pitch locations, for low purge case.

Figure 26. Velocity vectors near the rim seal on the same planes shown in Figure 25.
Figure 27. The pressure contours in the main annulus at mid-plan, and the streamlines on two planes at two different pitch locations, for high purge case.

Figure 28. Velocity vectors near the rim seal on the same planes shown in Figure 27.
A passive variable was added to track the flow ingested through the rim seal. Figures 29a and 29b show the value of this variable changing from 0 to 1 for the case of low and high purge respectively. As can be seen for low purge case, the ingested flow is driven to the stator disc. The concentration of the ingested flow is high at the seal, but decreases as moving downward inside the cavity. The ingested flow then moves toward the rotor disc, as could also be seen by the streamlines shown in Figure 23a. For the case of high purge, no ingestion occurred below the rotor hub, meaning high sealing effectiveness up to the rim seal at this flow rate.

The sealing effectiveness was calculated using the scalar variable value at two different radii as shown by locations 1 and 2 in Figure 29. The definition used here for the effectiveness is simply:

$$\eta = 1 - \frac{C}{C_{\text{inlet}}}$$

where \(C\) is the local concentration of the variable and \(C_{\text{inlet}}\) is the concentration of the variable at the main inlet, which is equal to 1. This is calculated using the average values for \(C\) around the periphery at each radius and over one revolution.

The sealing effectiveness is calculated according to the definition above to be 89% and 100% for high purge, and 31.2% and 58.4% for low purge at \(R=176\) mm and \(R=173\) mm respectively.

Figure 29. The contours for concentration of the scalar variable for (a) low purge flow and (b) high purge flow
7 Conclusion and future work

A preliminary study was conducted in order to study the flow structure inside the disc cavity of a test turbine, consisting of a newly designed blisk. A method was developed in order to reduce the computational resources necessary. The main annulus of the stage was first modeled separately. The boundary conditions to the cavity model were then exported from this model. According to the previous studies, using a simplified 360 degree model can still capture the transient phenomenon of rotating low pressure regions in the cavity, which was not possible to detect using a more detailed but sector model. This study was therefore limited to a 360 degree model including only a section of the main flow. A model which includes the hot gas path (including the blades) and the cavity could possibly improve the estimations by obtaining more accurate results.

The effect of the purge flow on the flow structure and ingestion of flow from the main annulus was studied using low and high purge flows. The high purge flow used in this study seemed to completely prevent ingestion. However, the purge flow which enters the main annulus should be minimized in order to reduce the losses and the disturbance of the main flow by this secondary flow. It is therefore important to find the optimum purge flow required to prevent ingestion, by testing lower purge flows than the high purge flow used in this study. This can be also good to be compared with the empirical correlations from predictive models reported in the literature [25].

The $k-\varepsilon$ model which was used in this study for modeling the cavity flow, is not a very good choice for flows with high swirl, but has been used due to its simplicity and moreover, since it has been successfully used in previous studies of rotor-stator cavities, where no blade was included. Other turbulence models are recommended to be tried and compared with the results of this study.

The time step used for the cavity model was not small enough to properly capture the changes in flow for each blade passing. A smaller time step could be used in order to get a better resolution in changes in the flow.

No unsteady rotating pressure structures, as was observed in several previous studies were detected by the results of this study. The mesh used for the cavity model in this study was coarse in some areas close to the walls, due to negative effect of finer mesh in areas with curvatures on the mesh quality, leading to instabilities in the model. Finer mesh close to the hub also resulted in problems with backflow. The first problem might be resolved using a better blocking method, or switching to unstructured mesh with automatic prism layers close to the walls. The problem with the backflow however seems to be related to the boundary condition at the outlet. Due to the coarse mesh, especially close to the rotor wall, the boundary layers on the discs and the associated shear forces might have not been captured accurately. This is thought to have influenced the pressure structures inside the cavity. The mesh, especially close to the walls, need definitely to be refined. The total time of simulation was also limited in this study, due to limitation with resources. Longer time simulation could be tried to see if these instabilities can occur after a larger number of revolutions.

Using the results of this study, it is difficult to conclude if the unsteady phenomenon observed by previous studies occurs for the specific configuration and operating conditions studied here. It should be noted that the flow inside the rotor-stator disc cavity is quite complicated, and a proper modeling of this flow requires more insight. Further and deeper studies are required in order to more accurately demonstrate the flow patterns.
References


[22] "ANSYS CFX Theory Guide".

