ISSN (Online) 2394-2320

International Journal of Engineering Research in Computer Science and Engineering (IJERCSE)
Vol 5, Issue 5, May 2018

Study on Mass Flow Rate in Labyrinth Seal using CFD Analysis

[1] Aprameya C R, [2] J Sharana Basavaraja, [3] Rajesh P
[1] P. G. Scholar, M. Tech. in Machine Design, [2] Associate Professor, [3] Assistant Professor
[1][2][3] BMS College of Engineering (BMSCE), Bengaluru

Abstract: **- The present study aims to understand the leakage flow rate parameter of labyrinth seals with respect to the different profiles and validation of leakage flow rate through experimental setup known as seal test rig for labyrinth seals. The different labyrinth seal profiles are designed using CATIA software and CFD analysis is performed to study the mass flow rate at inlet and outlet of the labyrinth seal, results in terms of pressure, velocity distribution profiles, pressure, velocity contours, turbulence dissipation rate, turbulence kinetic energy, velocity field are tabulated, studied to understand the mass flow rate through a labyrinth seal and study aims to conduct static analysis, modal analysis using ANSYS software & CFD analysis using COMSOL Multiphysics software. The results and observations are tabulated and compared to analyze the leakage flow rate of labyrinth seal for different configurations and design changes.**

Keywords— Computational Fluid Dynamics (CFD), Labyrinth seal, static analysis, CATIA software, ANSYS software, Mass flow rate, leakage, COMSOL Multiphysics, etc.

I. INTRODUCTION

Seals are integral parts of modern gas turbine engines; they are used in many locations between the stationary and rotating parts of the shaft, compressor, and turbine; over the tip of compressor and turbine as shown in the Sealing in such a location becomes significant on engine performance and efficiency, wear and leakage effect on engine thrust to weight ratio, specific fuel consumption [SFC]. Thus the seal deterioration in large turbofan engine can generate annually as 1% increase in SFC. According to Ludwig (1980), greater deterioration in smaller engines can account up to 17% power loss and over 7% increase in SFC. Sealing between the stationary and rotating parts in gas turbine engines was accomplished by utilizing rigid and non-complaint seal such as cylindrical and labyrinth seal. A labyrinth seal is widely used and is also a standard to verify the performance of the new design. The conventional labyrinth seal requires radial operating clearance to prevent the rubbing action between the stationary and rotating parts, if the surface speed exceeds 457 m/s, there exists a continuous contact between rotating and stationary parts that will produce wear and sometimes even cause catastrophic failure of components due to thermal growth and dynamic loading. The rigidity of seal would further experience wear and erosion over the continuous use of the engine and generates a large percentage of leakage flow which causes an adverse change in SFC, engine effectiveness and overhaul times.

A. Introduction to seals

Seals are used in gas turbines to restrict gas leakage, provide thrust balancing, maintain thermal gradients, meter cooling gas flow, and protect bearing sumps. The clearance between two components will allow the passage of fluid molecules in particular direction; this direction depends on the pressures and momentum associated with the fluid flow as shown in below Figure 1.

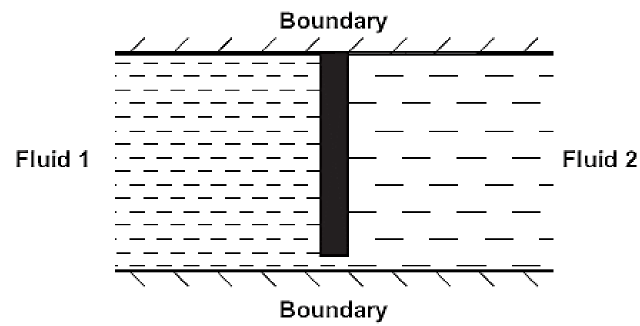


Figure: 1. Basic sealing geometry

B. Introduction to labyrinth Seals

Labyrinth seals are non-contacting seals that provide an expensive and simple method of restricting the secondary leakage flow from high-pressure region to low pressure region in gas turbine and are used both static as well as dynamic applications.

The leakage flow is controlled by dissipation of flow energy through series of knives (or sharp edges or tooth's) and corresponding chambers, by inducing turbulence in each chamber and static pressure difference decelerating the flow. The three knife labyrinth structure is as shown in the

ISSN (Online) 2394-2320

International Journal of Engineering Research in Computer Science and Engineering (IJERCSE)
Vol 5, Issue 5, May 2018

Figure: 2

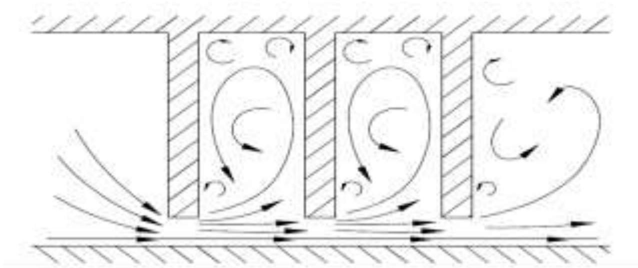


Figure 2. Flow restriction in three Knives of the labyrinth seal

The flow restriction depends primarily on the radial clearance between rotating and stationary parts and number stages associated with seal. The labyrinth sealing technique consists of a number of labyrinths or knives in a close clearance with a number of configuration [straight, stepped, staggered] on stationary and rotating parts, which create an air flow restriction between high pressure to low pressure region. Figure: 3 show the three knives labyrinth seal.

Figure 3. Three knives labyrinth seal

The configurations of labyrinth seals are of three types they are straight through, stepped and staggered configuration. The performance of any labyrinth seal is largely determined by the actual clearance between the tips of the teeth and the opposing surface. The seal clearance maintained between the rotating and the stationary surface plays a vital role during operation.

II. LITERATURE REVIEW

Labyrinth seals, innovated by C.A.Parsons, can mainly be divided into 3 categories; stepped, straight and staggered seals. Moreover, the tooth has two common profiles; rectangular and triangular teeth. Among them, straight-through triangular profiles are one of the most common ones in steam turbines due to their high sealing capacity and ease of manufacture.[2] These seals have triangular teeth over the rotor or stator part with cavities in between them. The first technical paper attempting to describe the fluid flow through the labyrinth seals was introduced by Becker. He modeled the flow through the labyrinth seal as Poiseuille flow and tried to find a coefficient of friction

Tong Seop Kim and Kyu Sang Cha,[1] has analyzed the influence of configuration and clearance on the leakage behavior of labyrinth seals. Both computational fluid dynamics (CFD) and an analytical tool were used to predict the leakage flow of two different (straight and stepped) seal configurations with various clearances. The predicted results were compared with experimental data. The CFD gives a better agreement with the experimental result than the analytical model on average. In the straight seal teeth, the dependence of the discharge coefficient on the clearance is considerable, while it is much smaller in the stepped seal.

Pawel Kaszowski and Marek Dzida [3], in their study, has done the CFD (computational fluid dynamics) analysis using ANSYS Fluent software to analyze the fluid flow through the labyrinth seal and paper describes the results using CFD simulation software with the help of contained a computational k – ε model. Where – 'k' is the energy of kinetic turbulence & 'ε' is energy dissipation.

Jan SIMAK, Petr STRAKA, Jaroslav PLANT [4] in the paper titled as "Numerical Solution of a Flow inside a labyrinth Seal" studies behavior of a flow inside a labyrinth seal on a rotating shaft. As per the authors, the labyrinth seal is a type of a non-contact seal where a leakage of a fluid is prevented by a rather complicated path, which the fluid has to overcome. In the presented case the sealed medium is the air and the seal is made by a system of 20 teeth on a rotating shaft situated against a smooth static surface.

III. METHODOLOGY

After going through the detailed literature survey, learning's from previous experiences and discussions we framed our methodology for this study as follows. First In order to understand the behavior and flow profiles of fluid inside the different labyrinth seal, we started with designing the different possible profiles of labyrinth seal using CATIA software, then we wanted to conduct the static structural analysis in order to endure that the seal teeth are capable of withstanding the 10 bar pressure as we limited our study for maximum 8 bar inlet pressure boundary condition.

Followed by the static structural analysis using ANSYS software, we wanted to conduct the MODAL analysis of seal using ANSYS software to ensure that the operating frequency of shaft is well less than the natural frequency of the seal we designed for study purpose, later we planned to conduct the CFD analysis.

We have designed 3 different seal designs as shown in the following table 1.below

Table 1. Design of labyrinth seal profile

Design	Description of design
01	Seal profile with 0.5mm width teeth tip
02	Seal profile with 1mm width teeth tip
03	Seal profile with a 13-degree angle between teeth tip

The thickness of seal in 15mm, 12mm height, and all the three profiles are designed for 5mm length. Stainless steel is the material considered for the seal. For CFD modeling we have considered the region of fluid flow between the rotating shaft and seal interference.

A. Designing the labrynith seal using CATIA software

By taking the learnings from the literature review first we started to design geometric model of the labyrinth seal for the required dimension based on experimental setup available. We have considered basic configurations for the design of labyrinth seals as follows. Model 1. OD 110mm, ID 80 mm & 0.5mm equal steps teeth as shown in below figure 4.

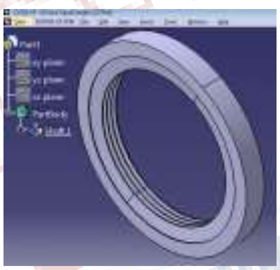


Figure 4. Model 1 sketch and 3D model with 0.5mm equal steps

B. Static structural analysis using ANSYS software

ANSYS static structural analysis calculates the effects of steady loading conditions on a structure while ignoring inertia and damping effects, such as those caused by time-varying loads. ANSYS static structural analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed. Stainless steel is applied as material and physical properties are volume 40680 mm³ and mass 0.31934 kg, density 7.85e-006 kg mm^-3, Poisson's ratio 0.3 etc.,

C. Meshing and Boundary condition

Model is meshed used software controlled smooth and fine mesh with minimum element size as 1mm and applied with the boundary condition as 10 bar pressure on seal teeth as shown in Figure 5 and 6 below,

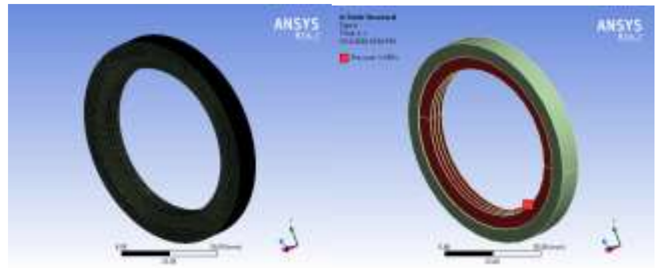


Figure 5.Meshed model & Figure 6. Applied boundary condition 10 bar

Below Table 2. Show the mesh details.

Table 2. Showing Mesh (Quadrilateral elements) details:

Sl. no	Parameters	Values
1.	Nodes	606978
2.	Elements	138645
3.	Element Size	1mm
4.	Analysis Type	Static Structural

IV. RESULTS AND DISCUSSIONS

ANSYS static structural analysis results for total deformation, equivalent von-Mises stress, equivalent von-Mises Strain are tabulated and results show good strength of seal teeth to withstand the applied pressure of 10 bar, the results are very important to know the initial behaviour of the seal teeth once applied with inlet pressure, assuming 10 bar as maximum pressure that can be applied analysis is carried out and results are tabulated in table 3 below.

Table 3. ANSYS results

Sl.No	Parameters	Results
1.	Total Deformation	5.0553e-005 [mm]
2.	Equivalent Stress	1.5064 [M pa]
3.	Equivalent Elastic Strain	7.5318e-006

Below figure 7. and figure 8. shows the deformation and stress developed for the initial and applied boundary condition on labyrinth seal.

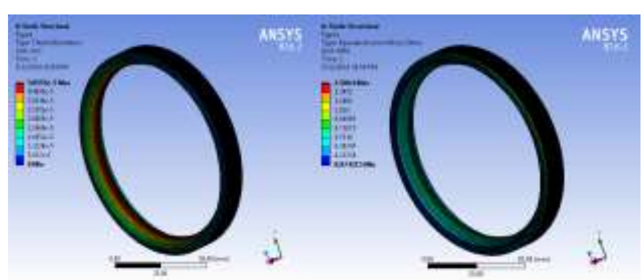


Figure. 7. Total deformation & Figure 8. Eq. Stress in

IFERP
ISSN (Online) 2394-2320
International Journal of Engineering Research in Computer Science and Engineering (IJERCSE)
Vol 5, Issue 5, May 2018

the model after analysis

A. Modal analysis of labyrinth seal

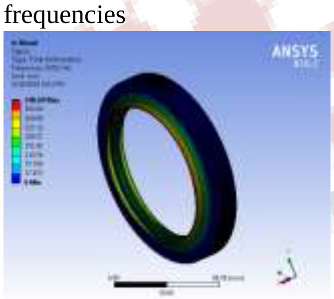
Modal analysis is to define the vibration characteristics (natural frequencies and mode shapes) of an assembly or a machine element, while it is being designed. It also can be a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a spectrum analysis. Here we have analyzed the modal behavior of the labyrinth seal in order to ensure the natural frequency of the seal are very less than the operating frequencies of the shaft with which seal may interact.

Modal analysis is carried out using the ANSYS software and following are the natural frequencies of the seal which are tabulated in Table 4. below.

Table 4. Modal frequencies of seal designed

Mode	Frequency (Hz)	Mode	Frequency (Hz)
1.	36522	5.	36937
2.	36618	6.	37453
3.	36630	7.	37465
4.	36928	8.	38212

Following Figures from 9. to Figure 16. shows the different modes of vibration of a seal designed with respect to frequencies



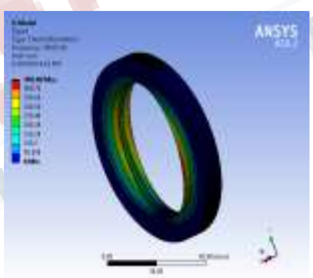
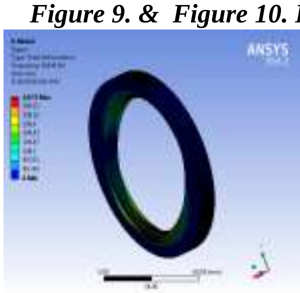


Figure 9. & Figure 10. First and second mode shapes



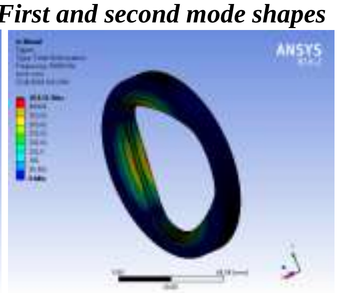
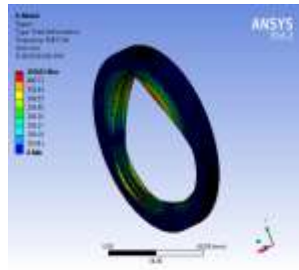


Figure 11. & Figure 12. Third and fourth mode shapes



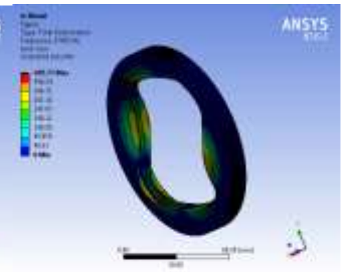
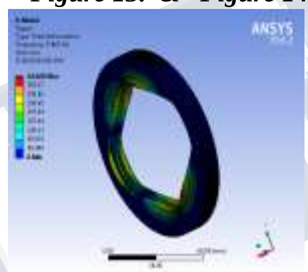


Figure 13. & Figure 14. Fifth and sixth mode shapes



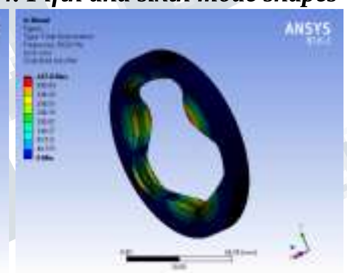


Figure 15. & Figure 16. Seventh and eight mode shapes

B. Introduction to Computational fluid dynamics (CFD)

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations like conservation of mass, momentum, energy, species mass, etc. The results of CFD analyses are relevant in Conceptual studies of new designs, detailed product development, Troubleshooting, Redesign. Etc.,

CFD analysis complements testing and experimentation by:

1. Reducing total effort
2. Reducing cost required for experimentation, etc.,

C. Introduction to COMSOL Multiphysics (CFD) Analysis

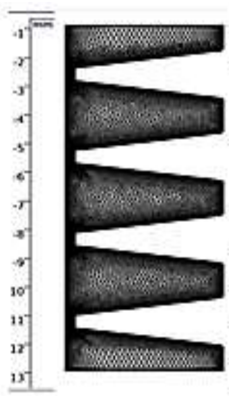
COMSOL Multiphysics® is general-purpose platform software for modeling engineering applications. We can use the core package on its own or expand its functionality with any combination of add-on modules for simulating designs and processes based on electromagnetics, structural mechanics, acoustics, fluid flow, heat transfer, and chemical engineering behavior.

D. CFD analysis of Labrynith seal

Here we have considered the 2D profile of labyrinth seal to perform CFD analysis, as CFD analysis demands more time and computer with a higher configuration to simulate the 3D model. Using COMSOL software we analyzed the 2D profiles of the seal by applying necessary initial and boundary conditions for inlet pressure compressed atm. air as a fluid. Mesh details are given below Table 5.

International Journal of Engineering Research in Computer Science and Engineering (IJERCSE)
Vol 5, Issue 5, May 2018

Table 5. Mesh statistics for seal geometry in COMSOL software



Mesh statistics:	
Description	Value
Minimum element quality	0.0954
Average element quality	0.7533
Triangular elements	16707
Quadrilateral elements	5121
Edge elements	1153
Vertex elements	20
Maximum element size	0.175mm

Figure 17. Meshed seal 2D profile with details of meshing

For all the simulations results are tabulated and different parameters like pressure distribution, velocity distribution, pressure contours, velocity contours, turbulent energy dissipation rate, turbulent kinetic energy dissipation rate, velocity field are tabulated and comparison plots are drawn to understand the mass flow rate of the compressed air inside the profiles of the labyrinth seal.

E. CFD analysis and plotting of seal with 0.5mm width teeth tip using COMSOL software

Following Figures 18 to Figure 22. shows the Pressure and velocity profiles and contours for 8 bar inlet pressure.

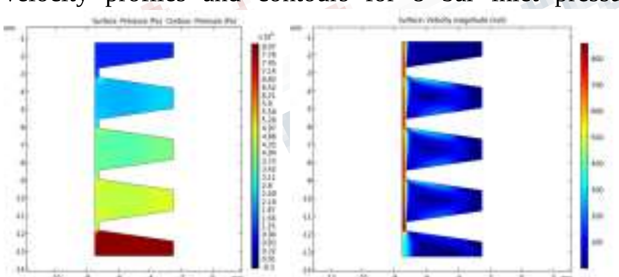


Figure 18. & Figure 19. Pressure & Velocity profiles for 8 bar inlet pressure

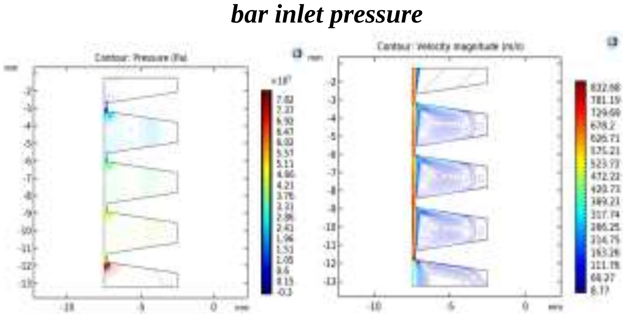


Figure 20. & Figure 21. Pressure & Velocity contours for 8 bar inlet pressure

Here in CFD analysis of labyrinth seal, we assume that the flow is compressible turbulent flow, a turbulent flow exhibits small-scale fluctuations in time. It is usually not possible to resolve these fluctuations in a CFD calculation. So the flow variables such as velocity, pressure, etc. are time-averaged.

We have observed the pressure, velocity profile and pressure, velocity contours for 2 bar, 4 bar, 6 bar of same nature for inlet boundary conditions, here we displayed the results for 8 bar inlet condition for all three seal profiles.

F. CFD analysis and plotting of seal with 0.5mm width teeth tip using COMSOL software

Once we run the CFD simulation analysis the COMSOL software will provide the different plots to plot turbulence dissipation, turbulence Kinetic energy, velocity field, pressure variation along the arc length. As per requirement of observation, we can select the different plots to understand the behavior of the fluid flow inside the seal.

Below Figure 22 to Figure 25 shows the comparison of parameters for 0.5mm thickness teeth tip with different inlet pressure. i.e. 2 bar,4 bar,6 bar,8 bar.

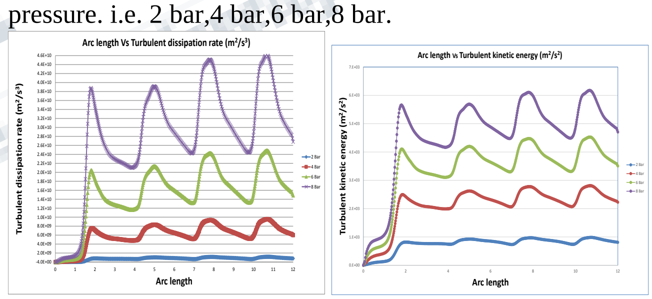


Figure 22. & Figure 23. Comparison plots for turbulence dissipation and turbulence kinetic energy

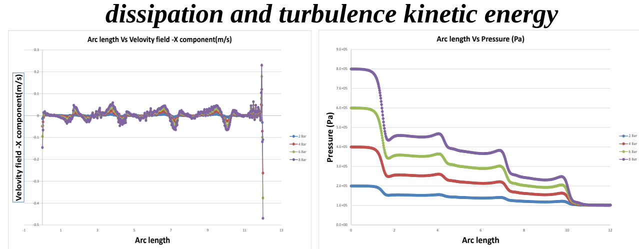


Figure 24. & Figure 25. Comparison plots for velocity field along axis and pressure variation

G. CFD analysis and plotting of seal with 1mm width teeth tip using COMSOL software

Following Figures 26 to Figure 29. Shows the Pressure and velocity profiles and contours for 8 bar inlet pressure.

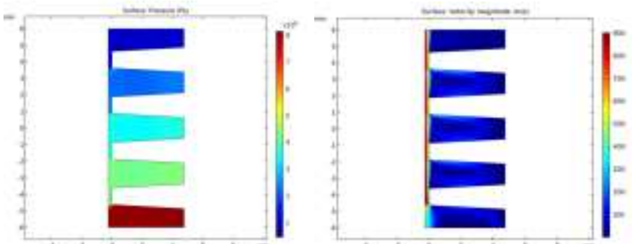


Figure 26. & Figure 27. Pressure & Velocity profiles for 8 bar inlet pressure

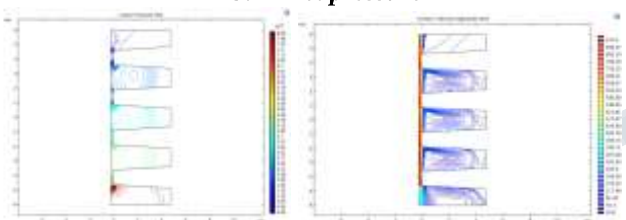


Figure 28. & Figure 29. Pressure & Velocity contours for 8 bar inlet pressure

H. Comparision plots for different input conditions for 1mm width teeth tip

Below Figure 30 to Figure 33 shows the comparison of parameters for 1 mm thickness teeth tip with different inlet pressure. i.e. 2 bar, 4 bar,6 bar,8 bar.

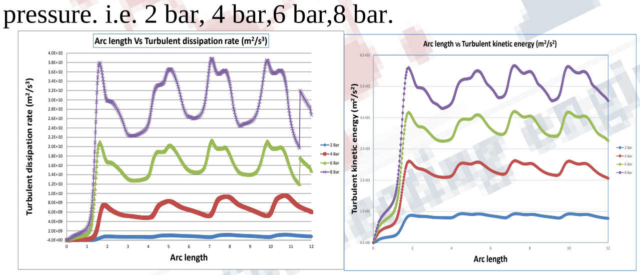


Figure 30. & Figure 31. Comparison plots for turbulence dissipation and turbulence kinetic energy

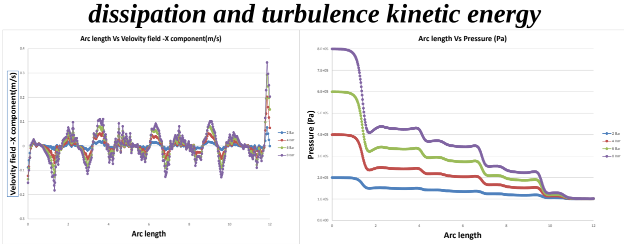


Figure 32. & Figure 33. Comparison plots for velocity field along axis and pressure variation

I. CFD analysis and plotting of seal with a 13-degree angle between teeth tip using COMSOL software

Following Figures 34 to Figure 37. Shows the Pressure and velocity profiles and contours for 8 bar inlet pressure.

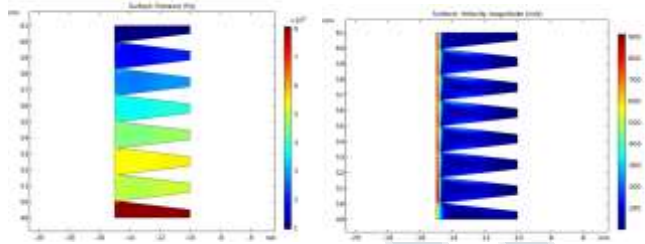


Figure 34. & Figure 35. Pressure & Velocity profiles for 8 bar inlet pressure

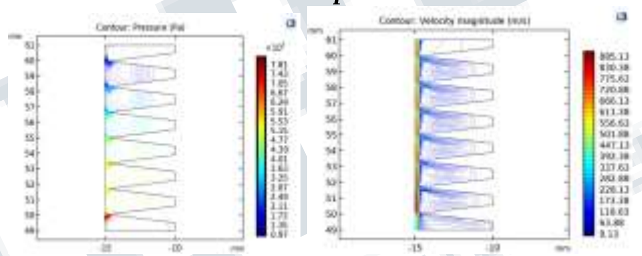


Figure 36. & Figure 37. Pressure & Velocity profiles for 8 bar inlet pressure

J. Comparision plots for different input conditions of the seal with a 13-degree angle between teeth tip

Below Figure 38. to Figure 39. shows the comparison of parameters for the 13-degree angle between teeth tip with different inlet pressure. i.e. 2 bar, 4 bar,6 bar,8 bar.

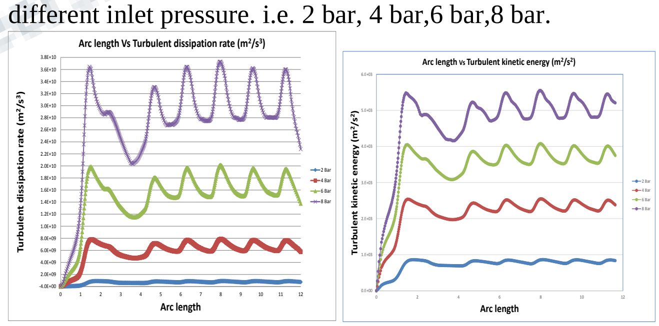


Figure 38. & Figure 39. Comparison plots for turbulence dissipation and turbulence kinetic energy

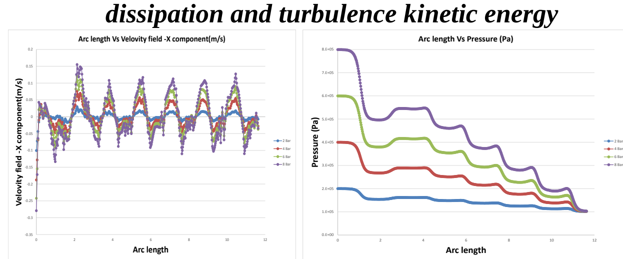


Figure 40. & Figure 41. Comparison plots for velocity field along axis and pressure variation

K. Comparision plots for mass flow rate

Below plot Figure 42, Figure 43, Figure 44 below. shows the comparative values for leakage from seal measured as mass flow rate at the outlet of the seal, for 0.5mm and 1mm width teeth tip seal profiles. We can observe that there is a increase in the mass flow rate as the inlet pressure is increased. And the plots clearly reflects that in case of 0.5mm and 1mm width labyrinth seal teeth tip profile the mass flow rate is more at inlet and correspondingly the mass flow at the out let is also more when compared to the plots of 13-degree teeth labyrinth seal profile.

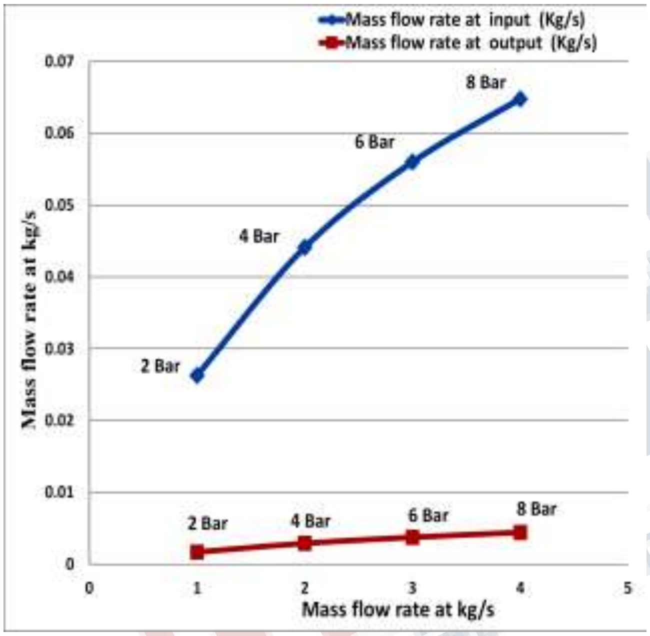


Figure 46. Comparison plots for mass flow rate inside 0.5mm width of teeth seal profile.

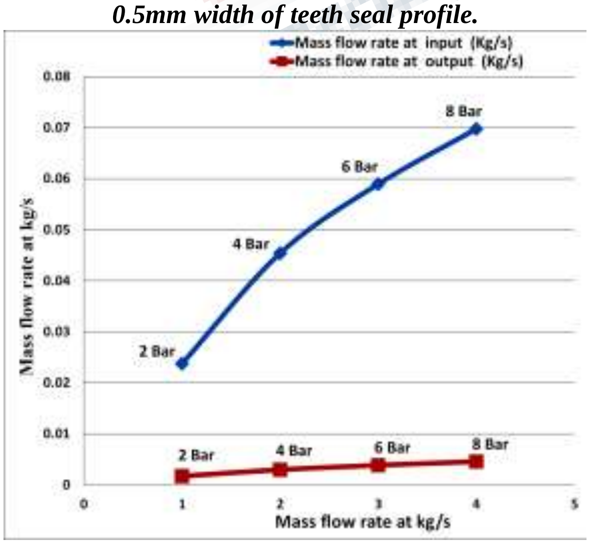


Figure 45. & Comparison plots for mass flow rate inside 1mm width of teeth seal profile.

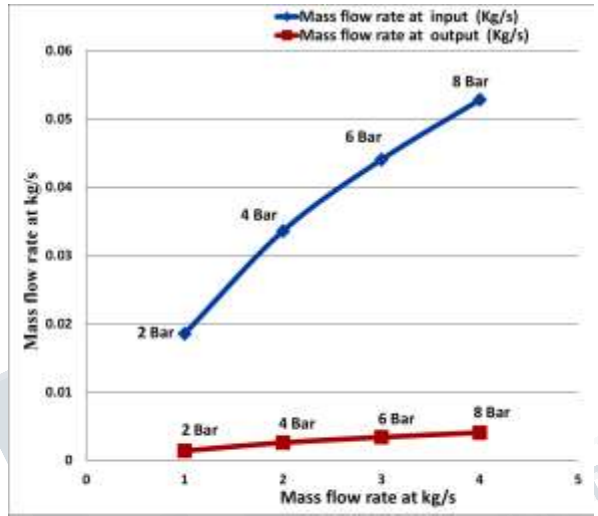


Figure 47. Comparison plots for mass flow rate inside seal for the 13-degree angle between teeth seal profile

V. CONCLUSIONS

The presented work consists static, modal analysis and CFD simulation to understanding the fluid flow behavior in a labyrinth seal, different profiles of labyrinth seal were analyzed and results were tabulated, compared As Labrynith seals are used in turbomachinery to reduce the leakage, mas flow rate at the outlet is considered. From this study, the following conclusions are drawn, Pressure and velocity profiles and contours clearly reflected that as the inlet pressure increases mass flow into the seal profile also increases.

The turbulence dissipation curve clearly reflects that in every teeth step there is a development of turbulence which also affects the mass flow rate inside the seal. All the plots and graphs indicate that in case of the labrynith seals, the clearance between the shaft and seal is the major contributor to mass flow rate and hence the leakage from the seal. The study can be extended by varrying the profile design of seal and for different boundary conditions like pressure variation, and for different materials for the seal. From this study it can be concluded that by varying the seal teeth profile design, depth of cavity or the length of the seal teeth, there is a possibility of changes in the turbulence dissipation rate and turbulent kinetic energy rate of fluid flow, and it is important to select the profile of seal such that, it should minimise the clearance between shaft and seal for effective sealing [5].

VI. ACKNOWLEDGMENT

The authors like to thank BMS college of Engineering, Bengaluru, for giving the opportunity to work on the above project and providing the lab facilities and technical guidance. Authors acknowledge the aid of TEQIP III for funding and supporting this work.

references

[1] T. S. Kim and K. S. Cha, “Comparative analysis of the influence of labyrinth seal configuration on leakage behavior,” J. Mech. Sci. Technol., vol. 23, no. 10, pp. 2830–2838, 2009.

[2] H. J. Sneck, “Labyrinth Seal Literature Survey,” J. Lubr. Technol., vol. 96, no. 4, pp. 579–581, 1974.

[3] P. Kaszowski and D. Ph, “Calculations of labyrinth seals with and without diagnostic extraction in fluid-flow machines,” vol. 20, no. 4, pp. 34–38, 2013.

[4] Jan SIMAK, Petr STRAKA, Jaroslav PELANT "Numerical Solution Of A Flow Inside A Labyrinth Seal", EPJ Web of Conferences 25, 01087 (2012)

[5] M Neeharika, & Prabhat Kumar Hensh, “Estimation of Leakage Mass Flow Rate through labyrinth seal using CFD techniques” Corporate Research & Development Division, Bharat Heavy Electricals Limited, Hyderabad. International Journal of Science and Engineering Invention (IJSEI) Volume 03 Issue 02 March 2017.