Volume 16 Issue 26
Originally issued as A4

Global Journal of Researches in Engineering: a Mechanical and Mechanics Engineering
Volume 16 Issue 4 Version 1.0 Year 2016
Type: Double Blind Peer Reviewed International Research Journal
Publisher: Global Journals Inc. (USA)
Online ISSN: 2249-4596 Print ISSN: 0975-5861

Leakage and Cavity Shape Studies of Labyrinth Seals

By R Mohana Rao & Dr. Manzoor Husain
JNTU

Abstract- Labyrinth seals are a non-contacting sealing device consists of a series of cavities connected by small clearances. They are used in many places in gas turbine engine to optimize and improve their design using CFD, is of interest to this study with newer designs of labyrinth seal. Preliminary investigations were carried out to establish the baseline capability for CFD analysis of labyrinth seals using Fluent and also to finalize the turbulence model with mesh type, thereafter detailed 2-D axi-symmetric analyses with different geometries and configurations. The applications of the new labyrinth seal designs are important to meet future performance of gas turbine goals. This paper presents improved design of canted seal design using RANS equations, with Turbulence two equations k-w turbulence model using Computational Fluid Dynamics (CFD).

Keywords: *labyrinth seal, CFD, ansys workbench, canted shape*

GJRE-A Classification: *FOR Code: 290501*

Strictly as per the compliance and regulations of:

© 2016. R Mohana Rao & Dr. Manzoor Husain. This is a research/review paper, distributed under the terms of the Creative Commons Attribution-Noncommercial 3.0 Unported License http://creativecommons.org/licenses/by-nc/3.0/), permitting all non commercial use, distribution, and reproduction inany medium, provided the original work is properly cited.

Leakage and Cavity Shape Studies of Labyrinth Seals

R Mohana Rao[α] & Dr. Manzoor Husain[σ]

Abstract- Labyrinth seals are a non-contacting sealing device consists of a series of cavities connected by small clearances. They are used in many places in gas turbine engine to optimize and improve their design using CFD, is of interest to this study with newer designs of labyrinth seal. Preliminary investigations were carried out to establish the baseline capability for CFD analysis of labyrinth seals using Fluent and also to finalize the turbulence model with mesh type, thereafter detailed 2-D axi-symmetric analyses with different geometries and configurations. The applications of the new labyrinth seal designs are important to meet future performance of gas turbine goals. This paper presents improved design of canted seal design using RANS equations, with Turbulence two equations k-w turbulence model using Computational Fluid Dynamics (CFD).

Keywords: labyrinth seal, CFD, ansys workbench, canted shape

Nomenclature

ε - Dissipation of turbulent kinetic energy
κ - Turbulent Kinetic energy
s- Tooth pitch, mm
A: Area [m2]
H: Step height [mm]
h: Teeth height [mm]
k: Specific heat ratio
m: Mass flow rate [kg/s]
N: Number of teeth
Po: Total pressure [kPa]
PR: Pressure ratio, Poin/Pout
s: radial clearance [mm]
To: Inlet total temperature [K]

Abbreviations
RANS – Reynolds-Averaged Navier- Stokes

I. Introduction

Aviation industry face challenges when crude oil price increase that influence the economic conditions of world. This will directly impact on the engines running on crude oil products which convert chemical energy into mechanical energy for transportation, power generation etc., lead to optimization of turbo machinery future fuel conservation requirements. Studies show that reducing the high pressure turbine seal leakage of engine airflow would produce significant improvement of engine specific fuel consumption.

Gas path sealing is therefore a fundamental area of interest when seeking improvements in the efficiency and performance of aircraft engines. By reducing the level of leakage from the gas flow, efficient sealing helps retain the energy. As the performance improvement becomes marginal, reduction in leakage flows becomes more important. Therefore, labyrinth seals are used more intensively, their clearances are more tightly designed hence configurations are evolving continuously. Therefore, the requirement for an accurate leakage prediction is becoming more crucial. Labyrinth seals are the most common flow path seals applied to turbine engines. They consist of several knife edges in close clearance in a number of Configurations. Labyrinth seals rely on controlled leakage across the seal driven by the pressure difference between the seal ends. The design of the seal forces the flow to separate at the knife edge causing loss of kinetic energy and pressure from the gas flow.

II. Present Work

In order to finalize the combination of parameters to be changed to improve the seal design, the experimental case of advanced seal configuration, Design 5; reported by H. L. Stocker in [3] is considered as baseline case. In the present work, the authors have conducted a CFD investigation on different configurations of canted teeth. The design evolved while conducting a parametric study on teeth height, teeth tip thickness, stepped teeth, inclined teeth etc. The baseline configuration is a simple sharp teeth labyrinth seal. The results obtained for the baseline configurations using this methodology were validated by comparing against 2D and 3D experimental data on stationary labyrinth seals with smooth land.

a) Baseline geometry: Canted Sharp Teeth

Fig. 1 shows labyrinth seal geometry with dimensions. The labyrinth seal test section consists of an upper part, stepped, and a lower part with teeth. In a real engine, the upper and lower parts correspond to the stationary and rotating parts, respectively, have 4 teeth with inclined called Canted Seals. The two seals have almost similar teeth dimensions. Geometry definition:

Author α: Post graduate from BITS Pilani, India pursuing PhD from JNTU Hyderabad. e-mail: mohan.ramanadham@gmail .com
Author σ: PhD, Professor, JNTU Hyderabad, Telangana. e-mail: manzoorjntu@gmail.com

© 2016 Global Journals Inc. (US)

Pitch – 7.62 mm; step height – 3.05 mm; Knife height – 3.81 mm and seal canted angle 50 deg

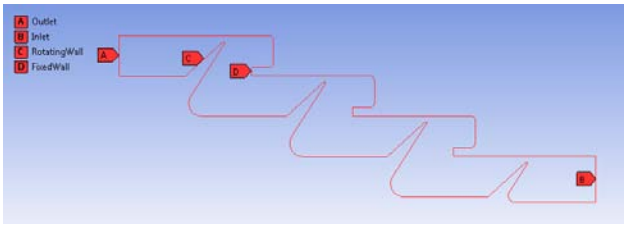

Fig.1: Base line teeth with 50 degree canted

b) New geometry: Canted Advanced Teeth

Fig.2 depicts the advanced sharp teeth are modified to increase the base of seal called Canted Advanced teeth. Except base width rest of parameters are kept same as original design.

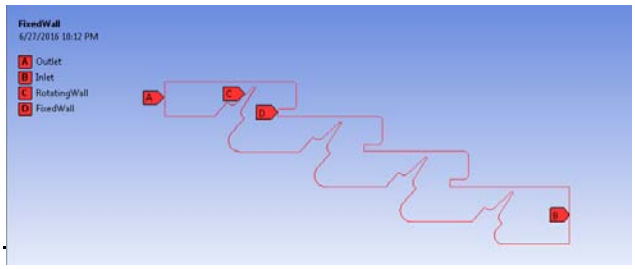

Fig.2: Modified advance seal teeth with 50 degree canted

c) *Performance Parameters*

The performance of a seal can be described by the relation between the pressure ratio and a flow parameter. The most common flow parameter is the following flow function:

$$\frac{m\sqrt{T_o, in}}{A_c P_O, in}$$

III. Analysis

a) Numerical analysis

Computational Fluid Dynamics (CFD) is extensively used because its capability to analyze a large number of design configurations and parameters in a relatively short period of time. Therefore, with the development of commercial codes, the use of CFD analysis has been increasing rapidly in recent years in 2 or 3 dimensional analysis. The test cases adopted in this work are two-dimensional with number of different operating conditions are analyzed.

b) Boundary Conditions

A commercial finite volume code, Ansys Workbench with Fluent 16.1v [10] is used. This commercial tool has wave linking geometry that eliminates loss of geometry while importing from CAD model to Fluent, manages entire problem in project charter. It was assumed that air was an incompressible ideal gas and the flow was steady and adiabatic. Various turbulence models available in Fluent [10] were considered for the current simulations.

A realizable two-layer k-ε turbulent model was used closest to Stocker [3], This model combines the realizable k-ε turbulent model with the two-layer approach. The realizable k-ε turbulent model uses equivalent kinetic energy and dissipation rate equations, but has additional flexibility of all y+ wall treatment that gives reasonable results for intermediate meshes where the cell falls in the buffer layer. Polyhedral mesh elements were used to create unstructured meshes in the entire domain. Fig.3 & 4 shows an example of generated computational grids.

The grid density in the clearance area was refined to locate sufficiently large number of meshes, this is done using inflation of smooth transition with 8 ~ 10 layers. For a given geometry, the inlet temperature is kept ambient and the exit pressure was essentially ambient (99.5kPa) and inlet pressure is varied to corresponding inlet total pressure was obtained thru calculation.

c) Mesh

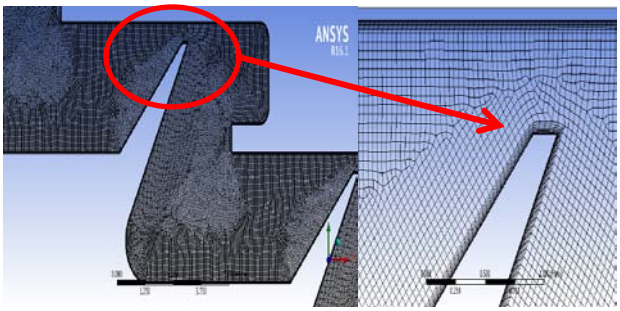

Fig. 3: Baseline geometry mesh (Model A)

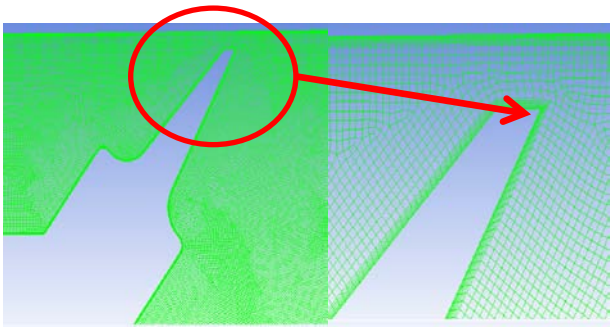

Fig. 4: Modified Advance seal mesh (Model B)

© 2016 Global Journals Inc. (US)

d) Mesh Sensitivity

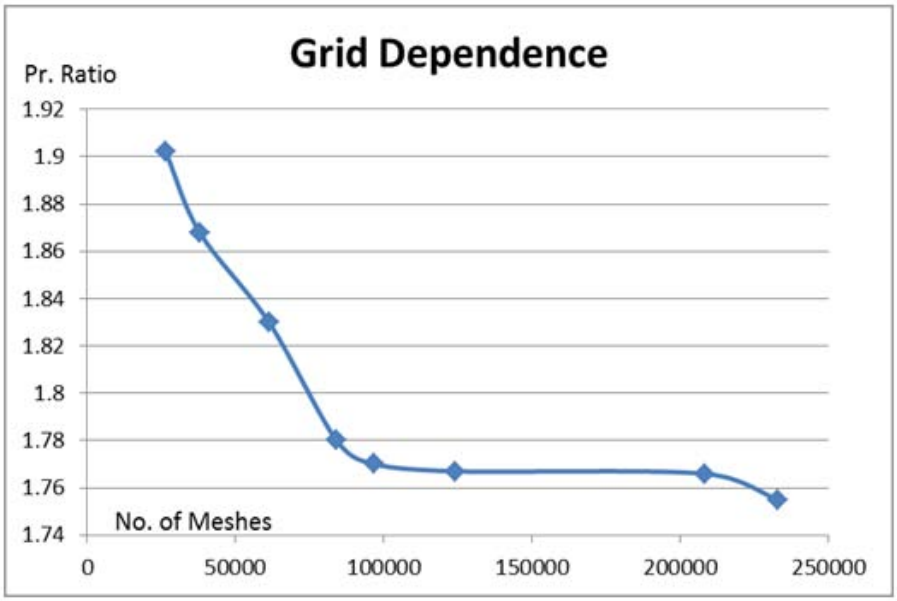

Fig. 5: Grid dependence of the CFD result

Grid dependence was checked to produce sufficiently converged solutions according to mesh size. Fig. 5 shows an example of grid dependence in the case of the Base geometry seal, presenting a variation of pressure ratio with the number of meshes for a given mass flow rate. The number of meshes ranges from 26,000 to 220,000 for the sharp teeth seal and from 43500 to 250,000 for the modified stepped seal, depending on the clearance size.

e) Single Vs Double Precision Solver Comparison Study

To study use of single and double precision solvers, baseline case was analyzed using single precision solver. The results from single precision solver were similar to those obtained except inlet velocity from the double precision solver. The CPU time for single precision solver was found more than double precision solver.

Table 1

			At Inlet			At Exit					
S.no	Analysis Description	No. of Cells	Mass flow (m/s)	V (m/s)	Temp (k)*	V (m/s)	Mass flow (m/s)	Pt (psa)*	Temp (k)	Del. T (k)	Pr = Pt(in)/ Ps(out)
1	Single Precision Solver	177644	0.0073	9.5577	295.14999	14.203	-0.007	99500	323.14679	27.997	2
2	Double Precision Solver	177644	0.0077	10.047	295.14999	12.977	-0.008	99500	323.14999	28	2

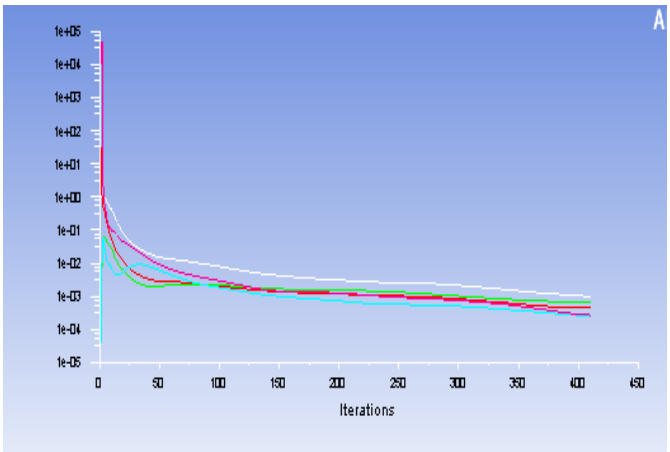

Fig. 6: Convergence plot for Double Precision

© 2016 Global Journals Inc. (US)

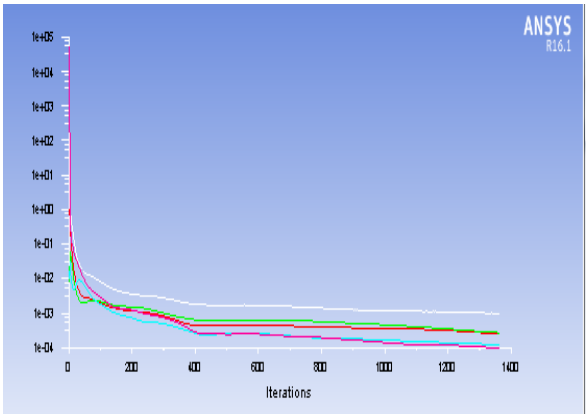

Fig. 7: Convergence plot for single Precision

f) Model A case with Experimental data

The Model A and Model B are run through by varying inlet pressure with ambient temprature and at exit ambient pressure. Models are run the pressure ratio reaching 3.0

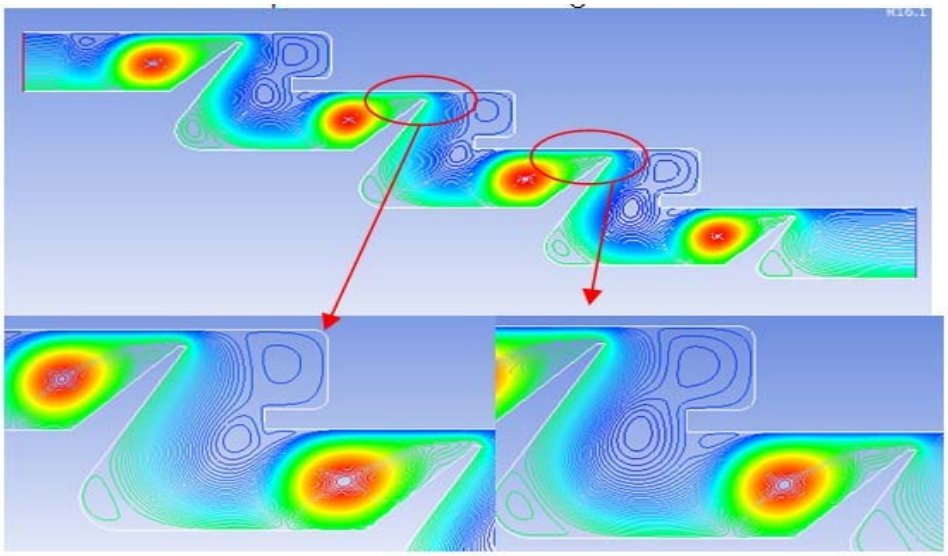

Fig. 7: Model A velocity contours

The model A results are plotted with experimental results. This is in line with the data. Refer the fig. for the graph showing experimental results CFD data which is ~ 5% variation.

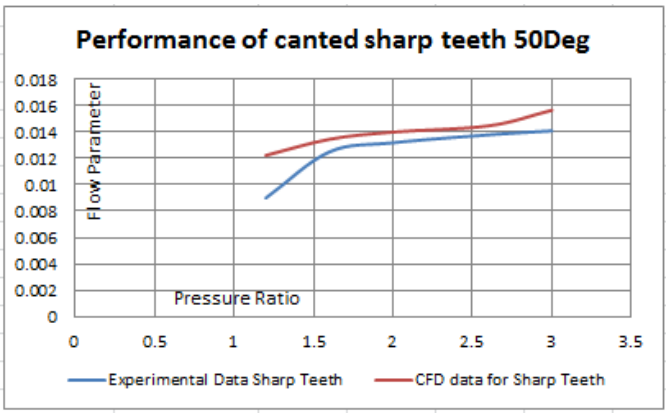

Fig. 8: Model a results with Experimental results

© 2016 Global Journals Inc. (US)

IV. Results and Discussion

Analysis of Results with different Seal Configurations

In this paper following studies are carried:

- Various Turbulence Models Studied
- Model A case with canted angle:70, 60, 40
- Step height: 2.0 mm and 3.05 mm
- Knife-edge clearance: (0.25 and 0.51 mm)
- Rotor speeds (rpm): 0,10000, 20000, 30000

a) Turbulence Model Study

2D axi-symmetric analysis of the modified Labyrinth Seal geometry (with rotation speed of 10,000 rpm) was first carried out. The objective was to establish baseline capability to run lab seal CFD analysis using Fluent and validate the experimental results.

Table 2: Turbulence Models

S.no	Analysis Description	No. of Cells	At Inlet				At Exit		Del. T (deg R)	Pr = Pt(in)/ Ps(out)
			Pin (Pas Cal)	temp	Mass flow	Vin (m/s)	Vout (m/s)	Tout (k)		
1	Std. k-e (T.I-10%,L.s-5%),2layer zornal model	197061	199200	295.14999	0.00754186	10.37968	15.023524	300	4.85001	2
2	Std. k-e (T.I-3%,L.s-3%),2layer zornal model	197061	199200	295.14999	0.00753624	10.37896	15.023502	300	4.85001	2
3	Std. k-e (T.I-1%,L.s-3%),2layer zornal model	197061	199200	295.14999	0.00754114	10.38819	15.04188	300	4.85001	2
4	Spalant - Allmaras (T.I-3%,L.s-3%),strain Vorticity based production	197061	199200	295.14999	0.00475953	6.729263	11.53694	300	4.85001	2
5	K-omega (T.I-3%,L.s-3%),Compressibility effects, shear flow correction	197061	199200	295.14999	0.00600749	8.296661	39.94806	300	4.85001	2
6	R.S.M (T.I-3%,L.s-3%),Wall B.C from K equation, Wall reflection effects	197061	199200	295.14999	0.00613915	9.127268	28.85895	300	4.85001	2

The Fluent results matched with the experimental results. This analysis was followed by turbulence models studies. The baseline model was analyzed first with Standard k-ε model (Turbulence Intensity = 10% and Length Scale = 5%). The same model was then analyzed with standard k-ε turbulence model but for different turbulence parameters, namely, Turbulence Intensity (T.I.) of 3% & 1%, and Length Scale (L.S.) 3% & 1% respectively. Standard k-ε, Spalart-Allmaras, k-omega, Reynolds's Stress Model were tried out with the baseline geometry. All the turbulence models showed consistent results except RSM & Spalart-Allmaras, with less mass flow and low velocities. It was decided to use Standard k- ε model for all future analyses.

b) Model A case with Canted angle

For Base line case (Model A) the canted angle is varied with 40, 60 and 70 degree and the analysis is carried with same boundary condition but the results show no improvement in flow parameter. Flow parameter values are much higher (15 % to 60%) compared with base line canted 50 degrees. Refer fig. 9 for velocity contours. The studies reveal that 50 degree is optimal canted angle for seal design for leakage flow.

© 2016 Global Journals Inc. (US)

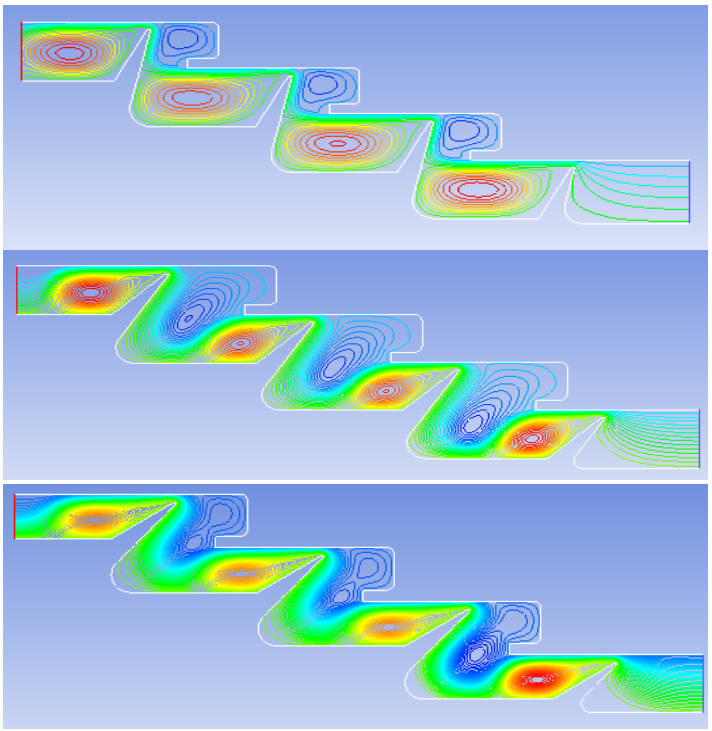

Fig. 9: Model A with various canted angles of velocity contours

c) Model B studies with step height and clearance

Analyses with step height and wall clearance between fixed wall and rotating wall are carried. Refer Fig. 10, 11,12, 13 with wall clearance (0.25, 0.51) and step heights (2.0 & 3.05) are analyzed based on review of minor differences in flow feature, the spacing of 0.25 was finalized for future axi-symmetric analyses. The models are analyzed for 10000 rpm. This analysis was considered as baseline analysis for all subsequent analyses. To improve the sealing efficiency, several labyrinth seal designs with varying geometries were analyzed

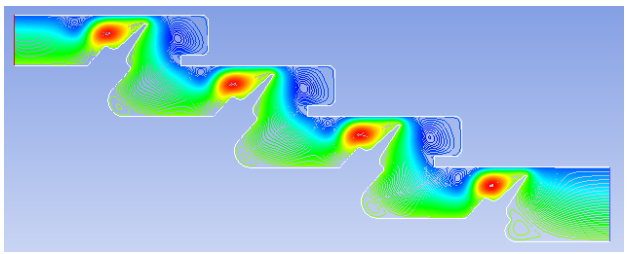

Fig. 10: Model B with .51 clearance and 3.05 step height- velocity contours

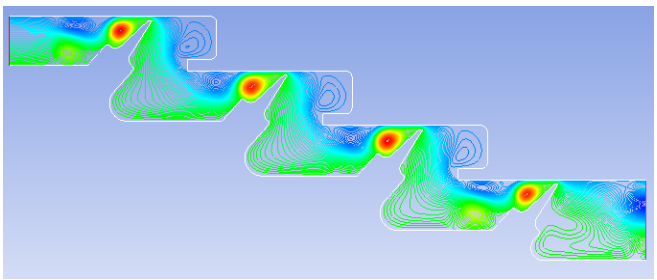

Fig. 11: Model B with .25 clearance and 3.05 step height- velocity contours

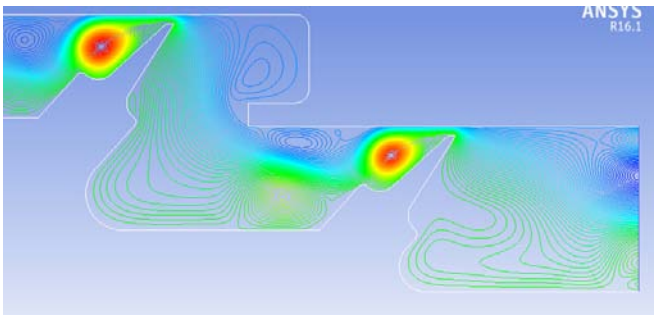

Fig.12 : Model B with .25 clearance and 3.05 step height- velocity contours of Tooth 1 and 2.

© 2016 Global Journals Inc. (US)

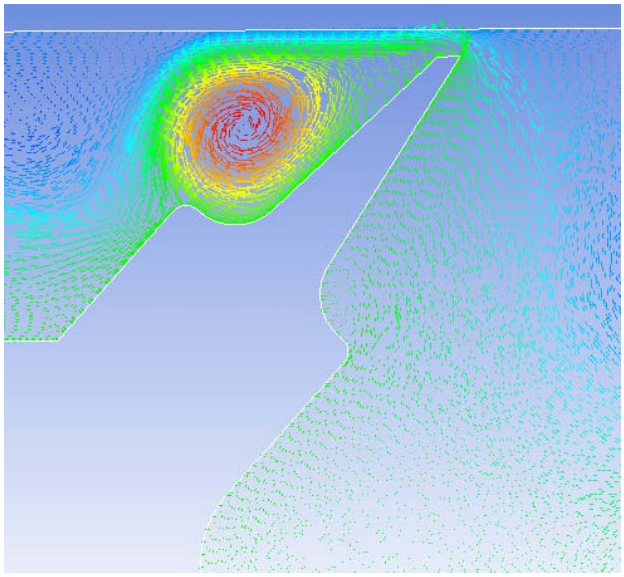

Fig. 13: Model B with .25 clearance and 3.05 step height- velocity vectors (stream function) of Tooth 2.

Comparing Figures 7, 10 and 11 the model B with .25 with step height 3.05 has created more turbulence in the flow thereby creating frictional flow resistance. Also since the stepped walls are inclined in the direction opposing the flow, there is direct resistance created for the flow to move downstream.

As observed in Figures 12 and 13 vortices are formed in the stepped wall zone at the outermost portion of the cells between sharp teeth also the vortices in the lower zone of the cells appear to be intensified. The step disrupts flow entering the clearance region and increases flow lockage upstream of and in the clearance area thereby reducing the exit pressure at each tooth (following the second tooth) overall leakage as compared to the baseline sharp teeth design.

Fig. 14 shows the seal performance of various models. Model B with .51 clearances has not shown much improvement. The flow parameter vs pressure ratio for this case is very high compared to base line sharp teeth model but Model B with .25 clearances is shown much higher improvement than any model.

Model B with .25 clearances shows 14.9% improvement when compared to Baseline Model A.

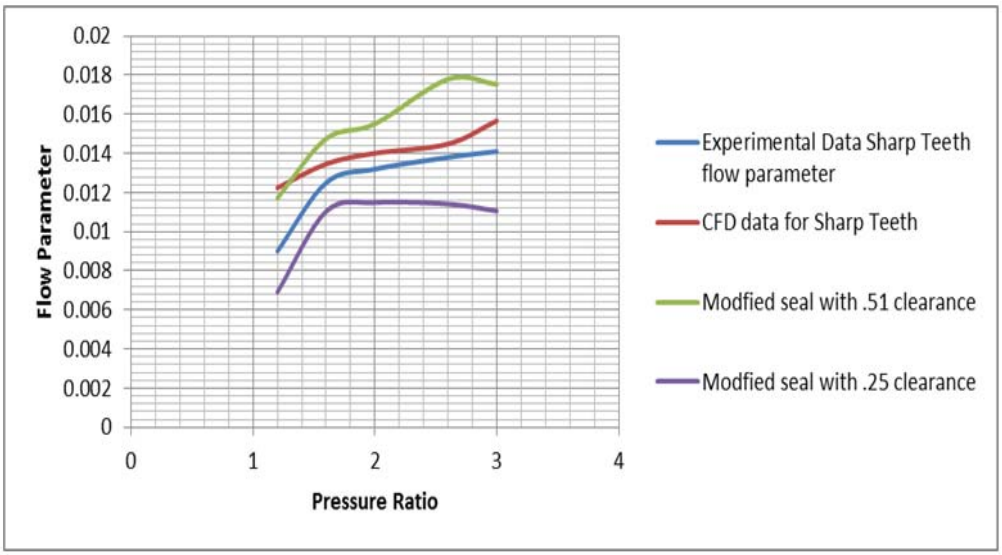

Fig.14: Seal performance of various models

The high performance seal Model B is further studied with various speed parameters. It shows the speed increase performance further improved this is because of radial velocity creates turbulence flow in the flow path that restricts movement. Refer Fig. 15 for seal performance at various speeds.

© 2016 Global Journals Inc. (US)

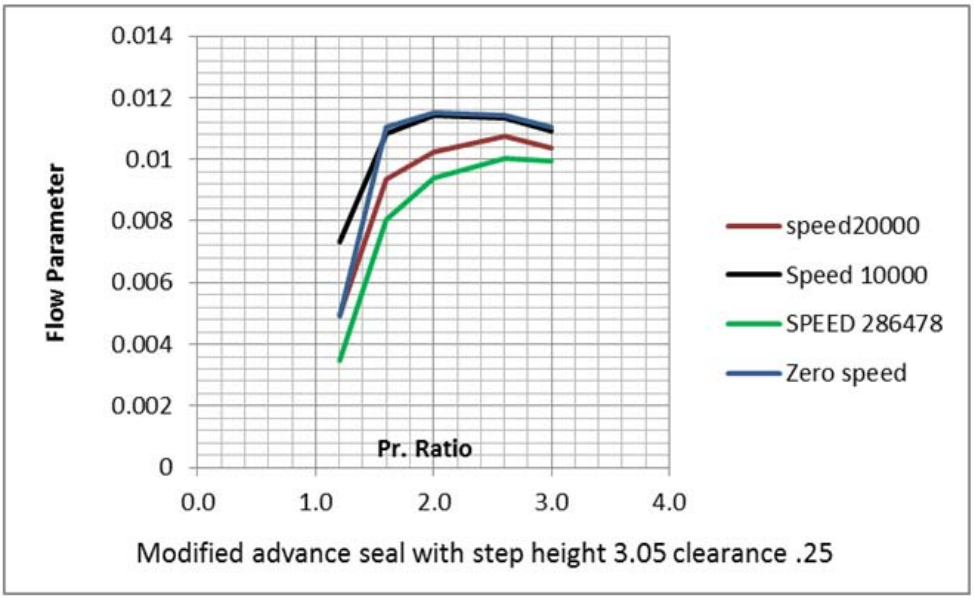

Fig.15: Seal performance of various models with different speeds

V. Conclusions

a) Standard k-ε, Spalart-Allmaras, k-omega, and RNG k-ε models give similar flow field for the analyzed lab seal configuration. Reynolds Stress Model gives elevated temperature field inside the domain. RNG k-ε turbulence model predicted less temperature rise and more pressure drop. The effect of throttling process in tooth clearance and the vortex flow are two main factors which influence flow resistance and leakage in seal.
b) The solution time is more with single precision solver than double precision solver
c) Model B with .25 clearance shows 14.9% improvement than baseline Model A.

Further work may be carried out for honeycomb wall with various diameters. Also air injection study with holes (modeled as slots in the 2d axi-symmetric analysis) to introduce in between the knife-edges in order to disturb the jet studies may be carried out.

VI. Acknowledgment

The authors wish to thank Kondalarao N and Suresh B for their contributions during case studies.

References Références Referencias

1. Suryanarayanan S., and Gerald M., 2009, "Analysis of Flow Parameters Influencing Carry-Over Coefficient," GT2009-59245, Proceedings of ASME Turbo Expo 2009, Orlando, Florida, U.S.A., June 8–12.
2. Bambang I. Soemarwoto, Johan C. Kok, Koen M.J. de Cock, Arjen B. Kloosterman and Gerrit A. Kool 2007, "Performance Evaluation of Gas Turbine Labyrinth Seals using computational Fluid Dynamics," GT2007-27905, Proceedings of GT2007 ASME Turbo Expo 2007: Power for Land, Sea and Air May 14-17, 2007, Montreal, Canada
3. H. L. Stoker, Determining and improving labyrinth seal performance in current and advanced high performance gas turbines, AGARD Conference Proceedings 237, Paper 13, (1978).
4. Jun Li, Shengru Kong, Shinnosuke Obi, Zhengping Feng, 2010, "Numerical Investigations on Leakage Performance of the Rotating Labyrinth Honeycomb Seal" journal of engineering for Gas Turbines and Power.
5. Sir David King, Dr Oliver Inderwildi, Dr Chris Carey, " Sealing technologies –signed, sealed and delivering emissions savings," Aviation and the Environment, 04/09
6. Hasham H Chougule., Alexander Mirzamoghadam, Douglas Ramerth, 2014, "CFD simulation of Single step and Double Notched tooth for labyrinth seals," Proceedings of ASME Turbo Expo 2014: Turbine Technical Conference and Exposition, GT2014-26286.
7. J. T. Han, A Fluid mechanics model to estimate the leakage of incompressible fluids through labyrinth seals, Date Published - August 1978
8. Pawel Kaszowski, Marek Dzida, Piotr Krzyslak, 2013, "Calculations of labyrinth seals with and without diagnostic extraction in fluid-flow machines," POLISH MARITIME RESEARCH 4(80) 2013 Vol 20; pp. 34-38 10.2478/pomr-2013-0038
9. N. Herrmann, K. Dullenkopf, H.-J. Bauer, 2013, "Flexible seal strip design for advanced labyrinth seals in turbines" Proceedings of ASME Turbo Expo 2013: Turbine Technical Conference and Exposition GT2013 June 3-7, 2013, San Antonio, Texas, USA Ansys workbench V 16.0

© 2016 Global Journals Inc. (US)