Leakage and Cavity Shape Studies of Labyrinth Seals

R Mohana Rao[1] and Dr. Manzoor Hussian[2]

[1] JNTU

Received: 6 December 2015 Accepted: 31 December 2015 Published: 15 January 2016

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).

Index terms— labyrinth seal, CFD, ansys workbench, canted shape

1 Introduction

viation 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.

2 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.

3 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: Pitch -7.62 mm; step height -3.05 mm; Knife height -3.81 mm and seal canted angle 50 deg 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:

4 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.

5 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-? turbulent model was used closest to Stocker [3], This model combines the realizable k-? turbulent model with the two-layer approach. The realizable k-? 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. **??** & 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. Grid dependence was checked to produce sufficiently converged solutions according to mesh size. Fig. **??** 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.

6 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. 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. 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-? model (Turbulence Intensity = 10% and Length Scale = 5%). The same model was then analyzed with standard k-? 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-?, 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? model for all future analyses.

7 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. 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. 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. [1]

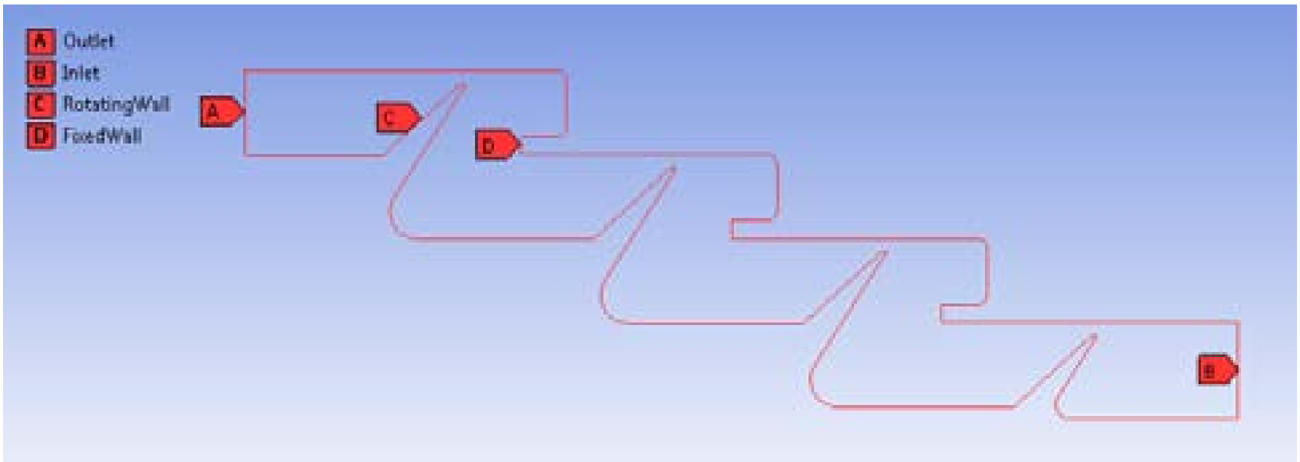
1 

Figure 1: Fig. 1 :

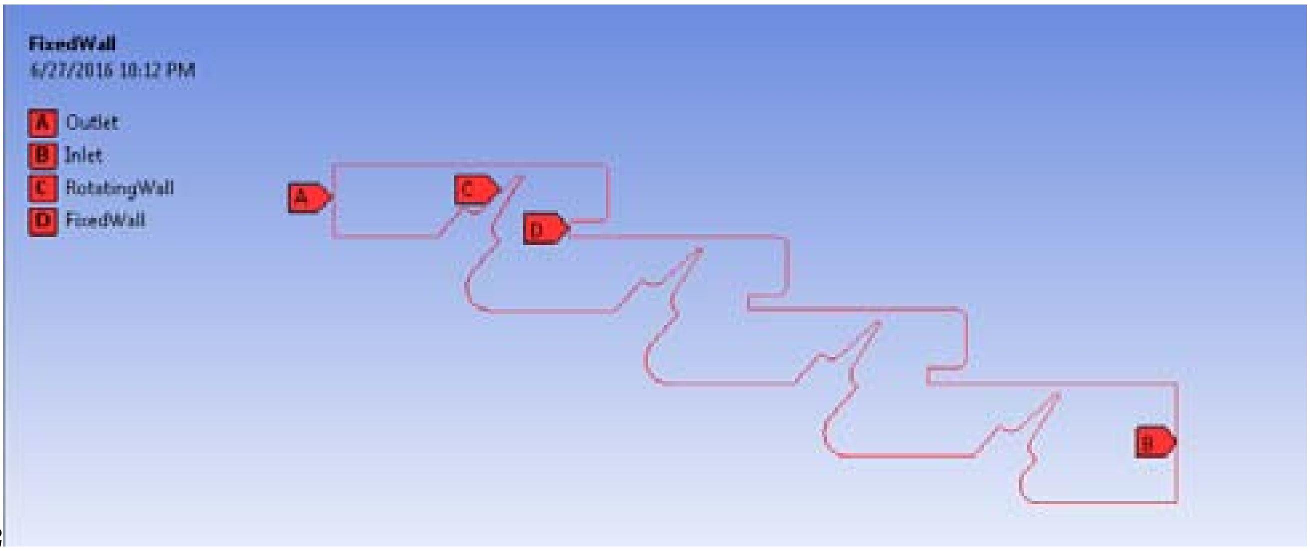
2 

Figure 2: Fig. 2 :

34

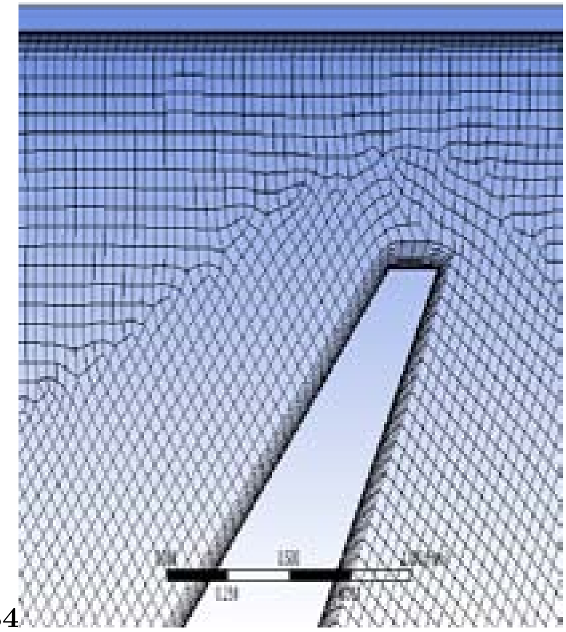


Figure 3: Fig. 3 :Fig. 4 :

7

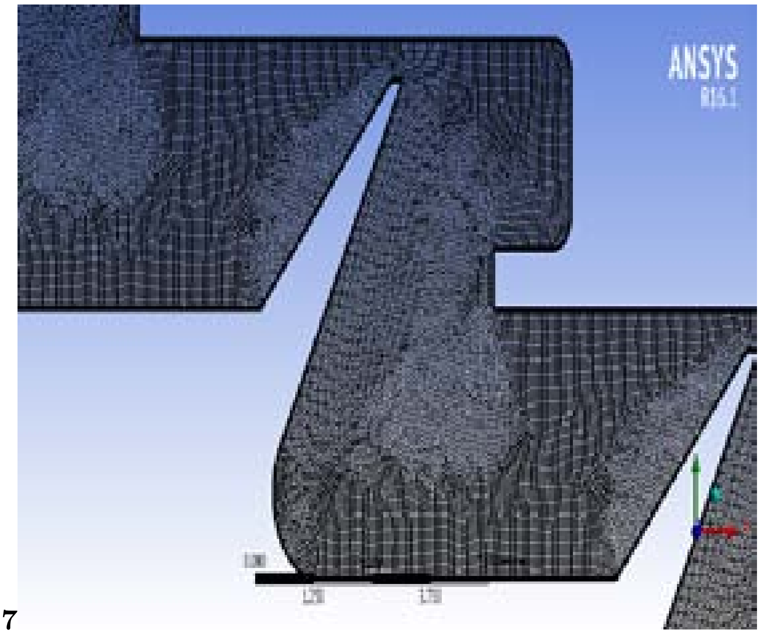


Figure 4: Fig. 7 :

820167

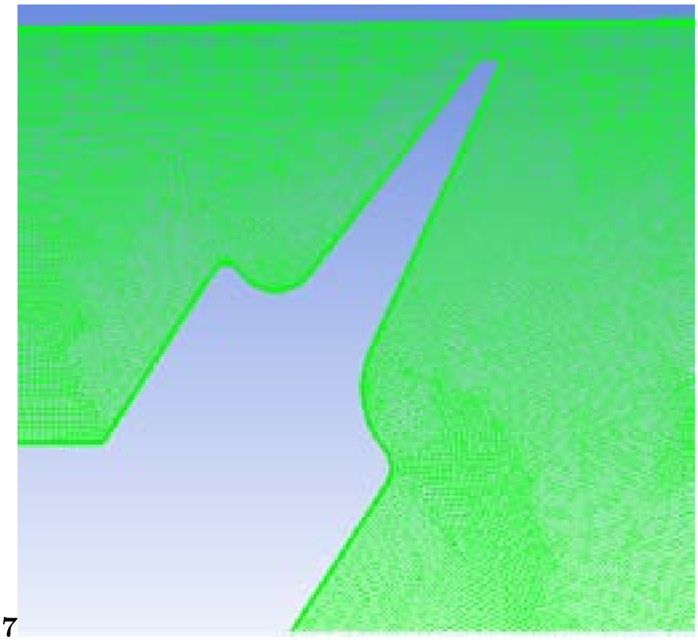


Figure 5: Fig. 8 :Year 2016 AFig. 7 :

9

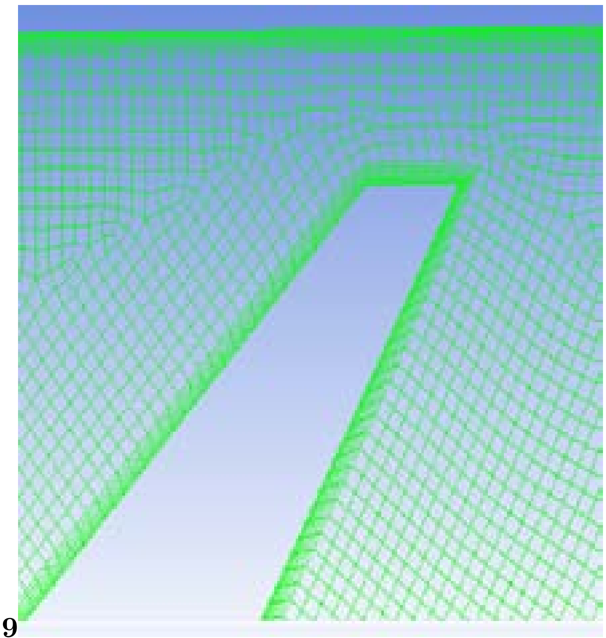


Figure 6: Fig. 9 :

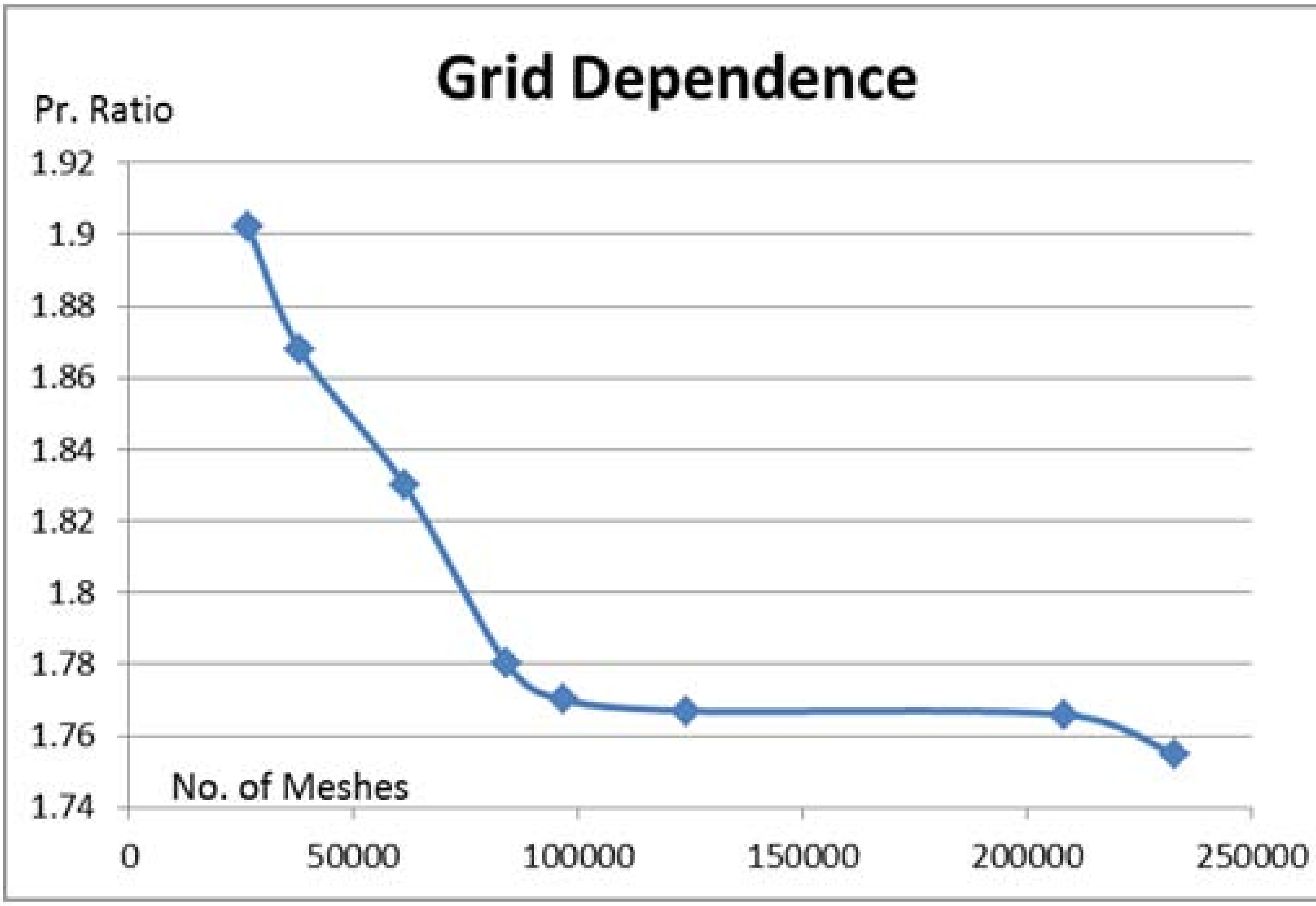
10


Figure 7: Fig. 10 :

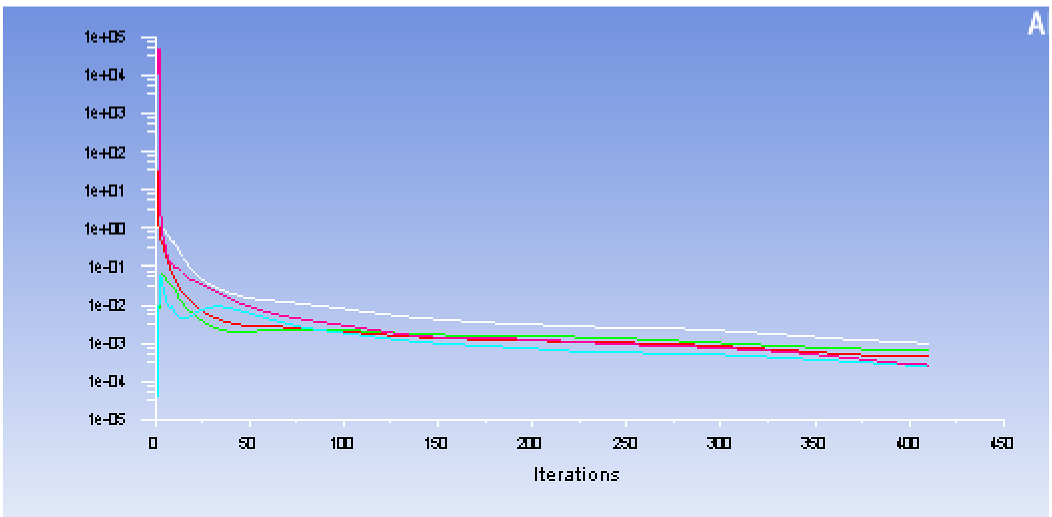
14


Figure 8: Fig. 14 :

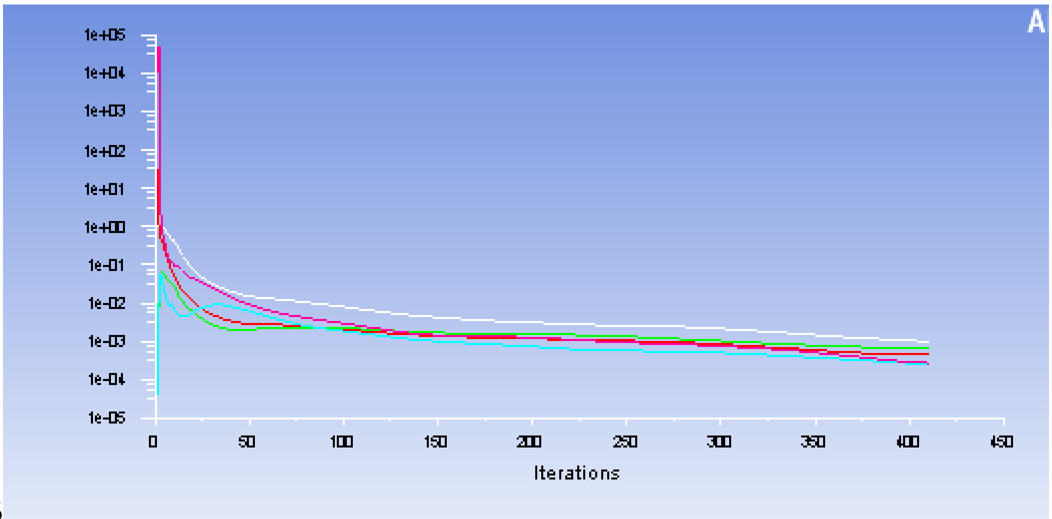
15 

Figure 9: Fig. 15 :

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(ou
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

[Note: Fig. 6: Convergence plot for Double Precision]

Figure 10: Table 1

2

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

Figure 11: Table 2 :

.1 Acknowledgment

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

[Han (1978)] *A Fluid mechanics model to estimate the leakage of incompressible fluids through labyrinth seals*, J T Han . August 1978. Date Published.

[Suryanarayanan (2009)] ‘Analysis of Flow Parameters Influencing Carry-Over Coefficient’. S Suryanarayanan , GeraldM . GT2009-59245. *Proceedings of ASME Turbo Expo*, (ASME Turbo ExpoOrlando, Florida, U.S.A.) 2009. 2009. June 8-12.

[Kaszowski et al. ()] *Calculations of labyrinth seals with and without diagnostic extraction in fluid-flow machines*, Pawel Kaszowski , Marek Dzida , Piotr Krzyslak . 2013.

[Hasham H Chougule et al. ()] ‘CFD simulation of Single step and Double Notched tooth for labyrinth seals’. Alexander Hasham H Chougule , Douglas Mirzamoghadam , Ramerth . *Proceedings of ASME Turbo Expo 2014: Turbine Technical Conference and Exposition*, (ASME Turbo Expo 2014: Turbine Technical Conference and Exposition) 2014. p. .

[Stoker ()] ‘Determining and improving labyrinth seal performance in current and advanced high performance gas turbines’. H L Stoker . *AGARD Conference Proceedings*, 1978. 237.

[Li et al. ()] *Numerical Investigations on Leakage Performance of the Rotating Labyrinth Honeycomb Seal” journal of engineering for Gas Turbines and Power*, Jun Li , Shengru Kong , Shinnosuke Obi , Zhengping Feng . 2010.

[Bambang et al. (2007)] ‘Performance Evaluation of Gas Turbine Labyrinth Seals using computational Fluid Dynamics’. I Bambang , Johan C Soemarwoto , Kok , M J Koen , De Cock , B Arjen , Gerrit A Kloosterman , Kool . *GT2007-27905, Proceedings of GT2007 ASME Turbo Expo*, (Montreal, Canada) 2007. 2007. May 14-17, 2007. Power for Land.

[Sir et al.] *Sealing technologies -signed, sealed and delivering emissions savings*, David Sir , Dr King , Oliver Inderwildi , Chris Dr , Carey . 04/09.