LABYRINTH SEALS FLOW FIELD EVALUATION WITH OPTICAL METHODS

Szymański A.*, Wróblewski W., Dykas S., Majkut M., Strozik M. and Marugi K.[1]
*Author for correspondence
Institute of Power Engineering and Turbomachinery,
Silesian University of Technology,
Gliwice, 41-250,
Poland,
E-mail: artur.szymanski@polsl.pl

[1]Avio Polska Sp. z o.o.
Michała Grażyńskiego 141, Bielsko-Biała, 43-300
Poland,
E-mail: krzysztof.marugi@avioaero.it

ABSTRACT

This work aims to perform the detailed experimental investigation of the flow field in labyrinth seal specimen using optical methods: LDA (Laser Doppler Anemometry) and schlieren visualization. Preliminary tests were performed on a stationary (rotor model with labyrinth does not move), linear – where the curvature of the specimen is omitted – measuring stand supplied by a vacuum pump. The installation makes it possible to achieve critical pressure ratios, up to two. This investigation was also supported by CFD (Computational Fluid Dynamics) calculations performed using the Ansys CFX v.17 commercial code with a flow model based on the RANS equations. Prediction scheme simulated the experimental campaign parameters. In CFD study, different types of mesh resolution were tested, with variable volume discretization in the area of labyrinth fin tip. Presented study shows challenges as well as the possibilities of flow field visualization including three-dimensional vortexes and strong jets occurring downstream the fin tips. Some limitations of LDA method application were pointed out, especially in areas of rapid fluid expansion. Moreover paper presented that schlieren method is a very efficient way of giving the turbulence structures in linear labyrinth seal fins. In the end, experimental results were compared with CFD study, which reviled the best method for labyrinth seal structures flow field simulation. Comparison of experimental and computed results showed some agreement between those two approaches. Flow visualization also allowed to understand better the flow behavior in cavities, which is crucial for design tools development.

INTRODUCTION

Thermodynamic engines, such as steam and gas turbines are the most popular sources for the electricity production in the world. It is estimated that more than 70% of electricity generated in 2014 [1] comes from fossil fuels combustion and nuclear power, which is directly associated with their application. Isentropic efficiency of state-of-the-art high power gas and steam turbines is close to 90%. The rest of the energy is dissipated as heat. Ever-increasing requirements imposed on the manufacturers assumes reduced emission of pollutants, noise and improved safety comfort of the operation, and reliability. The main cause of loss generation are secondary flows. They are associated with all flows and flow phenomenon which are not taking a part in energy conversion process. Two types of secondary flows can be pointed out: a turbulent phenomenon that appears in the main flow channel, like separation in boundary layer or horseshoe vortexes generation; as well as internal and external leakages. They are present due to a number of gaps and clearances, needed from a constructional point of view for proper rotating machinery work. It is estimated that mentioned gaps and cavities are responsible for about half of the losses present in turbomachinery.

To prevent from leakages to occur, many types of seal can be applied. Besides state-of-the-art techniques of reducing leakages, such as film riding face seals, finger seals or brush seals, labyrinth seals remain the most common solution used in turbines. It is a prime turbine sealing solution because of low price, low maintenance, minimal rub particulate contamination and high-temperature capability. [2] One of the methods of improving machine efficiency is to optimize gas turbine sealing operating conditions. Type of applied sealing has an impact not only on the size of leakage but also on the dynamic behaviour of rotor and entire engine fuel consumption, the temperature distribution downstream the stage, power decrease of an engine, as well as the risk of creep phenomenon. The results of investigation presented in [3] may be an example here. They indicate that a rise in the leakage through the gas turbine stage sealing from 3% to 4.5% of main mass flow (a relative 50% rise), involves an increase in the turbine outlet temperature by 15 ◦C. This, in turn, decreases durability by approximately 30% due to the risk of creep occurrence [3]. Therefore, the knowledge of the discharge characteristics and flow phenomenon in labyrinth seals becomes a key issue.

Optical measurements in fluid dynamics provide very useful information about many properties of the flow field. One the most relevant methods that are used in turbomachinery related experiments are Particle Image Velocimetry (PIV), Laser Doppler Anemometry (LDA) or schlieren method. In research associated with labyrinth seals and turbomachinery cavities testing, literature provides results for flow field visualisation with PIV method [4] [5] where complex labyrinth seal are investigated. Also the usage the LDA method is presented [6] [7] in labyrinth seals and simple cavities .

Institute of Power Engineering and Turbomachinery of the Silesian University of Technology has a long experience in optical measurement in rotating machinery. It specialises in LDA unsteady flow field evaluation in axial compressor stages [8][9] and schlieren pictures of condensing steam flow in blade cascades and nozzles [10]. Therefore, basing on this experiences, experimental campaign aiming the flow field evaluation by means of LDA and schlieren method was taken up.

To support the experiment, the 2D CFD (Computational Fluid Dynamics) calculation has been taken up, exploiting steady state Reynolds Averaged Naiver-Stokes scheme in Ansys CFX v17. Based on the owned experience in simple labyrinth seal testing [11] as well as complex sealing geometries [12], the k-omega SST turbulence model was selected. Proposed Calculation scheme showed good agreement with the experiment, in discharge parameters prediction, as well as in flow field visualisation.

The research procedure described herein is the stage of research comprising a detailed study of the labyrinth seal structures – in the reference configuration – and their optimization. Therefore, the development of measurement and calculation procedures becomes a key issue.

NOMENCLATURE

A	[m^2]	Cross section area
b	[m]	Labyrinth seal fin tip width
c	[m s^{-1}]	Velocity
C_D	[-]	Discharge coefficient
H	[m]	Labyrinth seal fin height
L	[m]	Labyrinth specimen length
m	[kg s^{-1}]	Mass flow rate
p	[Pa]	Pressure
R	[J kg^{-1} K^{-1}]	Gas constant
s	[m]	Labyrinth seal clearance size
$schn$	[kg m^{-4}]	Numerical schlieren
t	[m]	Labyrinth seal fins pitch
T	[K]	Temperature
u	[m s^{-1}]	Circumferential velocity
Special characters		
ρ	[kg m^{-3}]	Density
κ	[-]	Specific heat ratio
π	[-]	Pressure ratio
∇	[-]	Gradient
Subscripts		
ax		Axial component
id		Ideal (in terms of isentropic flows without loss)
in		Inlet parameter
out		Outlet parameter
s		Static parameter (pressure or temperature)
x		Component along x axis
y		Component along y axis
0		Total parameter (pressure or temperature)

EXPERIMENTAL SETUP

The experimental part of the testing was performed using the air installation in the Turbomachinery Laboratory of the IPET of the SUT. The installation includes a Roots air blower with the output of 600 Nm^3 min^{-1} (0.2 kg s^{-1}), a 3 m^3 pressure vessel and a pipeline system connecting the air installation to the research stand equipped with a measuring system. The minimum achievable pressure at the air blower inlet is 50 kPa(a). The total capacity of the installation (vessel and pipelines) exceeds 3.5m^3, which ensures maintaining stable and repeatable pressure distributions at the measuring stand inlet. Fig. 1 presents a simplified diagram of the installation. The diameter of the pipelines is DN100. Secondary air is sucked in from the surroundings through DN100 and DN 50 inlets with a throttling valve (5 and 6 in Fig. 1, respectively). The valve makes it possible to additionally regulate the pressure value in the vessel. On the stand side, the air is sucked in through an inlet preceded by a 4 m long pipe, which makes it possible to create appropriate conditions for the mass flow measurement upstream the test rig. After the air passes through the stand, it gets into the vessel and then – through the blower – into the environment. The mass flow is evaluated on the inlet pipe, 3 m downstream the pipe inlet.

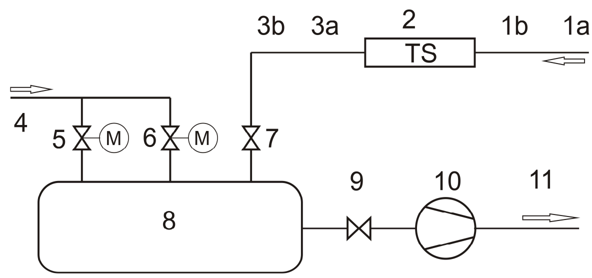

Figure 1 Experimental vacuum installation intended for the labyrinth seal testing in the IPET of the SUT.

1a – test section inlet, T_0, p_0 evaluation, 1b – HWA probe no. 1 (3.5m downstream the pipe inlet, ambient conditions), 2 – test section with the measurement system, 3a – HWA probe no. 2 (7m downstream the test rig, low pressure conditions), 3b – ISA orifice plate (2m downstream the HWA probe no. 2), 4 – secondary air inlet, 5 – DN 100 valve, 6 – DN 50 valve, 7 – cut-off valve, 8 –3m^3 pressure vessel, 9 – cut-off valve, 10 – Roots air blower, 11 – exhaust to the environment

TEST SECTION

A linear specimen approach test section is used to perform the experimental study. In this methodology, the circumferential shape of the labyrinth and its movement is omitted. This simplification is justified basing by Waschka research [13], which compared the experimental study results between stationary linear labyrinth with the circular rotating specimen. Cited research indicate that when axial velocity component of flow above the fin tip – c_{ax} is higher than the circumferential velocity of rotor tip – u, and the condition $u/c_{ax} < 1$ is met, this simplification does not impact the results. Moreover, the planar design keeps the design reasonable in terms of financial outlays and manufacturing, it also enables very accurate control of the clearance.

The cross section of this is presented in Figure 2. The clearance is regulated by moving the labyrinth specimen and controlled with blade feelers (with accuracy ±0.005 mm). Replaceable labyrinth and the labyrinths facing allow performing a wide range of research in presented solution.

The test section width is more than 200 times bigger than the examined clearances, which minimizes the effect of the side walls. Based on CFD studies, the boundary layer thickness at the side walls, near the fins, is negligible compared to the specimen width and the fluid velocity.

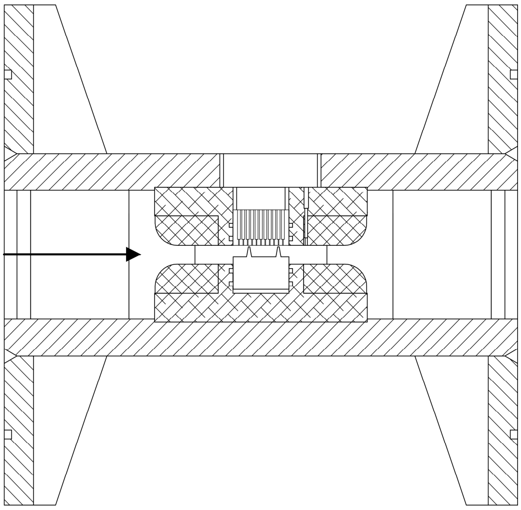

Figure 2 Test section cross-section

Side walls of the test rig are equipped with optical glass panes enabling visual access to the test section interior. **Figure 3** presents the photograph of the test rig with described peephole and investigated geometry.

Figure 3 View of the test rig with the peephole intended for optical measurements

Investigated labyrinth seal geometry is shown in the Figure 4. It consists of two straight fins, against the smooth land. This structure has been previously the subject of various studies, including Karlsruher Institut für Technologie (KIT) research of the honeycomb application impact [14], IPET of the SUT validation study of presented herein experimental methodology [11] against literature survey results, as well as CFD study details [15].

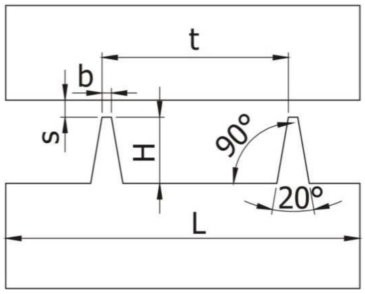

Figure 4 Examined specimen dimensions

Mentioned dimensions are: H = 7 mm, b = 1 mm, t = 20 mm, L = 38 mm and clearance s = 0.4, 0.9, 1.5 mm

OPTICAL MEASUREMENT SETUP

Schlieren (from German, singular "Schliere", meaning "streak") photography flow visualization method is one of the oldest, and the simplest optical method, which bases on the variation of the light refractive index in function of fluid density. The classical implementation of an optical schlieren system uses light from a single collimated source shining on, or from behind, a target object. Variations in refractive index caused by density gradients in the flowing fluid distort the collimated light beam. This distortion creates a spatial variation in the intensity of the light, which can be visualized directly with a shadowgraph system. The main advantage of this method is its non-invasive, short time of measurement and simplicity. This method characterises also with significant susceptibility to very weak flow disruption. The main idea of application schlieren photography was to observe the vortex and separation structures occurring during the flow through the labyrinth seal, which is the major source of pressure losses.

The second method used for flow field visualisation was Laser Doppler anemometry. This technique uses the Doppler shift in a laser beam to measure the velocity in the fluid with seeding, in one specified point. In contrast to schlieren photography, this measurement method provides a very accurate value of velocity (simplest and most precise method, requiring no pre-calibration). Thus, it requires more time to evaluate the larger flow area, also in gas application seeding is required (0.4 - 4μm oil droplets or powder particles).

2D NUMERICAL METHOD

Currently, ongoing development in the field of computer technology allows us to perform complex calculations once impossible. CFD methods can be applied as a fairly quick way to evaluate the flow field in the analysed area, accelerating the design or optimization of different structures in terms of flow [16]. The flow in the labyrinth seal channel is very complex. It involves strong jets, flow separation and appearance of vortex structures. The modelling of all these phenomena is very important for the evaluation of the sealing because the labyrinth seal effectiveness is a direct result of the possibility of kinetic energy dissipation in the seal cavities. In order to support the experiment, a two-dimensional CFD study is made based on the RANS scheme with the SST turbulence models and using the Ansys CFX v.17 commercial code. The turbulence model selection and mesh study for this research approach have been previously presented in [17].

The computational domain is two-dimensional, as presented in Figure 5. The numerical model also includes an inlet and an outlet channel. The modelled inlet channel (30 mm long) is to align the velocity profile. The outlet channel (120 mm long) is to reduce the influence of the flow recirculation above the fins on the outlet boundary condition and eliminate the impact of boundary conditions on the flow structure.

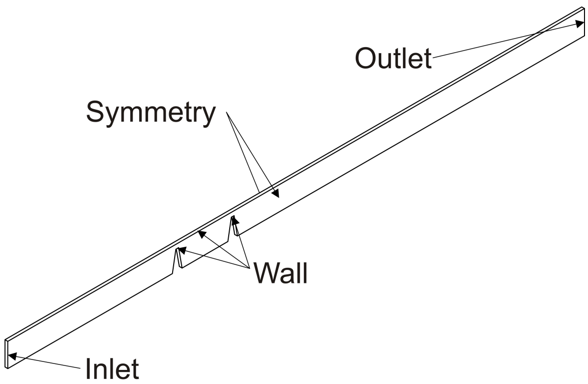

Figure 5 CFD fluid domain with boundary conditions explained

Table 1 CFD study boundary conditions

Inlet	Total temperature	293 K
	Total pressure	100 kPa(a)
	Turbulence intensity	5%
Outlet	Static pressure	60 - 95 kPa(a)
	Pressure profile blend	0.05
Global	Turbulence model	SST (Shear Stress Transport)
	Timescale	Automatic, local
	Heat transfer	Total energy including Viscous Work.
	Material	Air as ideal gas. Dynamic viscosity with Sutherland correlation [7].
	Wall model	Adiabatic, smooth
	Mass imbalance residual criteria	< 0.01 %

The aim of the calculations, as well as the measurement, was to visualise the flow field and determine the dimensionless flow rate, C_D, which is a measure of the sealing efficiency. This parameter depends on a number of parameters, and often due to unsteady nature of the flow phenomenon, fluctuates during the measurement or calculation. Therefore, the discharge coefficient was determined for the parameters measured in about 90 s. When calculations are considered, there is also required a considerable accuracy, providing a constant value of flow field. For this purpose, it was established that the parameter terminating calculation is the difference between the mass flow rate evaluated on the boundary conditions – inlet and outlet. Maintaining the mass imbalance parameter for following 100 iterations below 0.01% resulted in fluctuations of the C_D parameter lower than 0.001. This approach guaranteed the numerically independent result, uninterrupted by mathematically generated unsteady flow phenomena. The place of measurement of the static pressure for the purpose of determining pressure ratio is located 50 mm downstream the labyrinth specimen. It corresponds to the place where the static pressure was measured at the test rig. The boundary conditions of presented calculations are presented in Table 1.

CFD AND EXPERIMENT DISCHARGE RESULTS

To evaluate results, the algorithm presented below was applied. In this study dimensionless flow parameter – discharge coefficient versus operational pressure ratio was used. This approach was applied both for experimental and computational studies. Static pressure measurement, which is used for the evaluation of the operating pressure ratio, was set behind the fin, 50 mm downstream the labyrinth specimen. The pressure ratio is defined as a ratio of the inlet total pressure (in this case ambient pressure) to the static pressure measured in the presented point. It is described by the following equation:

$$\pi = \frac{p_{0,in}}{p_{s,out}} \tag{1}$$

Analysing the possibility of limiting the leakage through the tested structure of the seal, it turns out that the value of the mass flow rate as an indicator for the evaluation and comparison is insufficient. It depends not only on the architecture of the seal itself, but also on the total inlet parameters, which - due to the characteristic of the installation - are variable in time. Therefore frequently cited in the literature dimensionless flow coefficients are used [18]. These allow for comparison of the leakage through the different structures of different shapes and sizes by varying values of fluid parameters. The first is discharge coefficient C_D defined as a ratio of the mass flow rate through the model of the seal to that of an isentropic nozzle with the same surface area.

$$C_D = \frac{\dot{m}}{\dot{m}_{id}} \tag{2}$$

The mass flow rate under the perfect fluid condition is defined as:

$$\dot{m}_{id} = A\frac{p_{0,in}}{\sqrt{T_{0,in}}}\sqrt{\frac{2\cdot\kappa}{R(\kappa-1)}\left[\left(\frac{1}{\pi}\right)^{\frac{2}{\kappa}}-\left(\frac{1}{\pi}\right)^{\frac{\kappa+1}{\kappa}}\right]} \tag{3}$$

Results show two-finned labyrinth seal behaviour (Figure 6), indicate similar flow trends obtained both by means of measurement and CFD calculations. At a high value of clearance (s = 1.5 mm), some difference between the C_D obtained by means of experiment and calculations is visible. The CFD results are overestimated up to 10% of leakage value comparing to the experiment results. However, for lower values of clearance (0.9 and 0.4 mm), the convergence of results is higher. The best results were obtained for the clearance s = 0.4 mm, where the discrepancies were lower than 1%. For this reason, geometry with the smallest clearance was selected for further research, aiming schlieren photography.

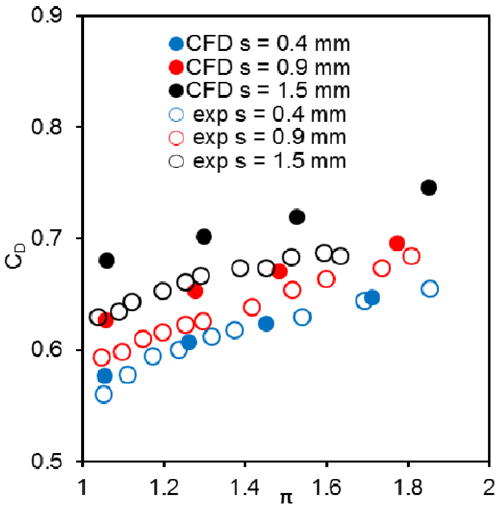

Figure 6 Discharge coefficient in function of pressure ratio. Experiment vs 2D CFD.

OPTICAL MEASUREMENTS RESULTS

In this subsection, the results of experimental (Figure 7) and numerical schlieren (Figure 8) are presented. The numerical schlieren is defined as the root mean square value of density gradient, as follows:

$$schn = \sqrt{\nabla\rho_x^2 + \nabla\rho_y^2} \qquad (4)$$

This equation was applied in CFX and the results are plotted in Figure 8. In the case of three-dimensional flow (this CFD study was assumed as two-dimensional), the third component in z-direction should be applied. Both methods predicted strong turbulent structures in front of the second labyrinth seal fin, rising in a function of pressure ratio. However, the shape of the vortex is different – the CFD calculations predict the circular one, while the experiment shows significantly stochastic behaviour. However, the general flow phenomenon is similar. Both method also predicted expanding jet angle downstream the labyrinth fin tip – which shows the place of rapid acceleration and density drop. The effect is stronger for high pressure ratios, and the mentioned angle is wider. Schlieren photographs show flow field averaged in some period, associated with the equipment parameters. CFD calculation use automatic timescale, which can be the source of potential differences seen in figures.

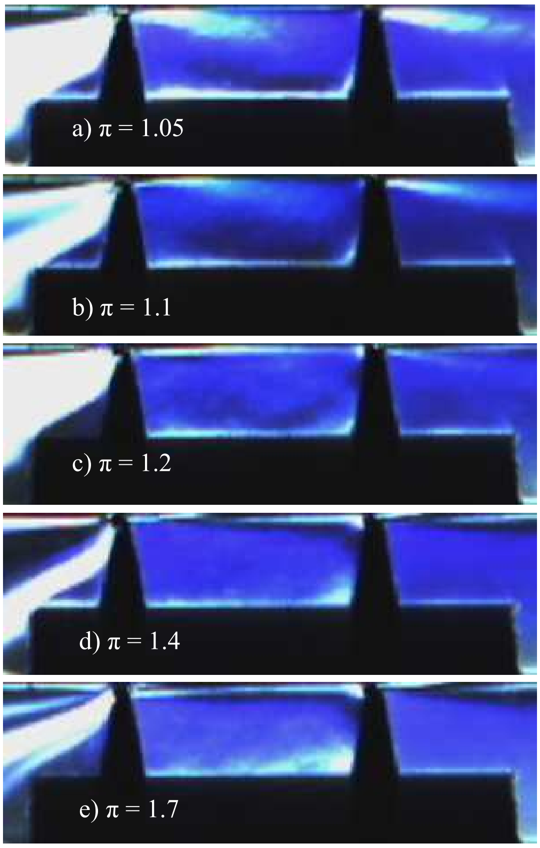

Figure 7 Schlieren pictures of two finned labyrinth seal against a smooth land, with clearance s = 0.4 mm, at various pressure ratios (1.05 – 1.7)

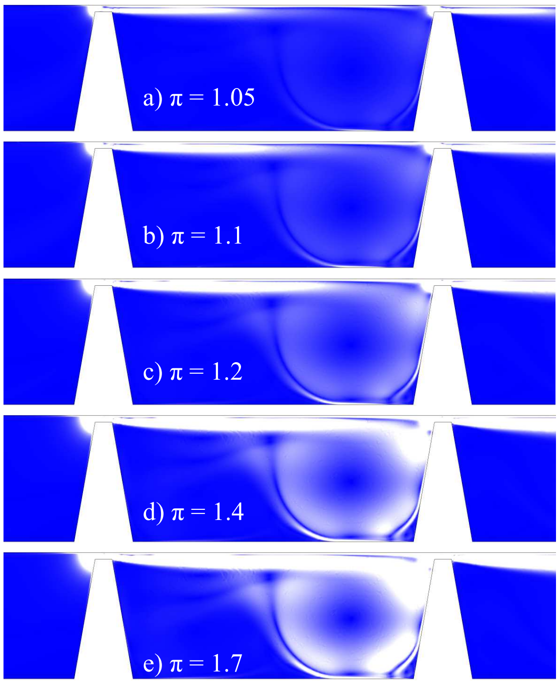

Figure 8 Numerical schlieren pictures of two finned labyrinth seal against a smooth land, with clearance s = 0.4 mm, at various pressure ratios (1.05 – 1.7)

The results obtained by the LDA method show very turbulent flow behaviour in the cavity between fins. The preliminary test showed that turbulence intensity in this volume exceeds 30%, what makes the LDA measurement impossible to perform. Also, due to a large stream acceleration behind fins, and the seeding conglomeration in larger droplets, the velocity measurement in this area became difficult. LDA measurement results showed significant statistic error and will be furtherly analysed in future.

CONCLUSION

This paper presents the approach in flow field of two-finned labyrinth seal against a smooth land visualization using schlieren photography and 2D RANS calculation scheme. The study covers a wide range of clearances and pressure ratios, simulating the real engine work. Presented simplified calculation scheme is sufficient for the mass flow evaluation, however, some discrepancies are visible, especially in the case of large clearance. The maximum error reaches 9% of C_D – dimensionless discharge coefficient applied for to evaluate the sealing structure characteristics. It is worth mentioning, that for smaller sizes of the gap between fin tip and the facing, the discrepancies minimize.

Optical measurement, basing on the Laser Doppler anemometry provided some information about the flow field, but the exact values were burdened with errors, mostly because of very turbulent phenomenon occurring in the flow field, as well as relatively small size of the probe. The turbulence intensity in area between the fins exceeded 40%, which caused large measurement discrepancies.

Schlieren pictures showing the phenomenon determining the pressure loss mechanism in labyrinth seals explains well flow behavior in cavities. Also, some agreement of numerical schlieren pictures with experiment is present, however the steady-state limitations are pointed out.

Future work aims performing more detailed CFD studies, with unsteady, three-dimensional flows modelling. This procedure may upgrade the flow field comparing to the empirical ones. From experimental point of view, the schlieren technique upgrade is planned, to provide better quality images, taken at higher frequency. Also, more sophisticated geometries will be tested, including labyrinth fi higher number of fins, against the honeycomb land, commonly applied in turbomachinery.

The present study provides important data for the design

of future turbo machines, because the exact knowledge of the labyrinth phenomenon, improving the design of sealing components and CFD tools.

ACKNOWLEDGEMENTS

This work was supported by the National Centre for Research and Development and Avio Polska within the Innolot Programme, Coopernik Project

REFERENCES

[1] International Energy Agency, 2014 Key World Energy Statistics, 2014

[2] Chupp R., Hendricks R., Lattime S. and Steinz B., Sealing in Turbomachinery, NASA/TM—2006-214341, 2006

[3] Zimmermann H., Some Aerodynamic Aspects of Engine Secondary Air Systems, *The American Society of Mechanical Engineers*, 1989

[4] Vakili A. D., Meganathan A. J., Michaud M. and Radhakrishnan S., An Experimental and Numerical Study of Labyrinth Seal Flow, *Proceedings of ASME Turbo Expo 2005*, 2005.

[5] Micio M., Facchini B., Innocenti L. and Simonetti F., Experimental Investigation on Leakage Loss and Heat Transfer in a Straight Through Labyrinth Seal, *Proceedings of ASME Turbo Expo 2011*, 2011

[6] Denecke J., Dullenkopf K., Wittig S. and Bauer H., Experimental Investigation of the Total Temperature Increase and Swirl Development in Rotating Labyrinth Seals, *Proceedings of the ASME Turbo Expo 2005*, 2005

[7] Moore J. and Palazzolo A., CFD Comparison to 3D Laser Anemometer and Rotordynamic Force Measurements for Grooved Liquid Annular Seals, *Transactions of the ASME*, 1999, pp.306-314

[8] Witkowski A., Ziach M., Majkut M. and Strozik M., Experimental Investigations of the Flow Phenomena in the Rotor Blades Of The Axial Flow Low Speed Compressor Stage at the Unstable Part of the Overall Performance Characteristic, *The Archive of Mechanical Engineering*, Vol. 55, No. 4, 2008

[9] Witkowski A. and Majkut M., Experimental investigation if inlet guide vane-rotor interaction in a low speed axial flow compressor stage. *The Archive of Mechanical Engineering*, Vol. 53, No. 1, 2006

[10] Dykas S., Majkut M., Strozik M. and Smołka K., Experimental study of condensing steam flow in nozzles and linear blade cascade, *International Journal of Heat and Mass Transfer*, Vol. 80, 2014, pp. 50-57

[11] Szymański A., Dykas S., Wróblewski W., Frączek D. and Marugi K., Experimental and numerical validation study of the labyrinth seal configurations, *Proceedings of the 12th European Turbomachinery Conference, Stockholm, Sweden*, 2017

[12] Wróblewski W. and Bochon K., Conjugate heat transfer analysis of the tip seal in the counter rotating low pressure turbine, Archives of Mechanics, Vol. 67, No. 3, 2015, pp. 253-270

[13] Waschka W., Zum Einfluss der Rotation auf das Durchflussverhalten und Wärmeübertragungsverhalten in Labyrinthdichtungen und Wellendurchführungen, *Dissertation, Institut für Thermische Strömungsmaschinen der Universität Karlsruhe*, 1991.

[14] Weinberger T., Einfluss geometrischer Labyrinth- und Honigwabenparameter auf das Durchfluss- und Wärmeübergangsverhalten von Labyrinthdichtungen, *Dissertation*, *Institut für Thermische Strömungsmaschinen der Universität Karlsruhe*, 2014.

[15] Frączek D. and Wróblewski W., Validation of numerical models for flow simulation in labyrinth seals, *Journal of Physics: Conference Series*, Vol. 760, 2016

[16] Wróblewski W., Dykas S., Bochon K. and Rulik S., Optimization of tip seal with honeycomb land in LP counter rotating gas turbine engine, TASK Quarterly, Vol. 14, No. 3, 2010, pp. 189-207

[17] Szymanski A., Dykas S., Majkut M. and Strozik M., The assessment of the calculation method for determining the flow characteristics of one straight fin labyrinth seal, *Transactions of the Institute of Fluid-Flow Machinery*, Vol. 134, 2016, pp. 89-107

[18] Braun E., Dullenkopf K. and Bauer H. J., Optimization of Labyrinth Seal Performance Combining Experimental, Numerical and Data Mining Methods, *Proceedings of the ASME Turbo Expo 2012*, 2012.