# Investigation of Flow through a Labyrinth Seal

Marius ENACHE[1], Razvan CARLANESCU[1], Andreea MANGRA*,[1],
Florin FLOREAN[1], Radu KUNCSER[1]

*Corresponding author
[1]National Research and Development Institute for Gas Turbine COMOTI,
220D Iuliu Maniu Bd., Bucharest 061126, Romania,
andreea.petcu@comoti.ro*

DOI: 10.13111/2066-8201.2021.13.2.6

*Received: 17 February 2021/ Accepted: 08 April 2021/ Published: June 2021*

***Abstract:*** *Growing performance requirements for gas turbines have led to a continuous increase in gas temperature and pressure ratios. Together with the resulting increase in cooling flows, this requires more and more minimization and control of internal gas leaks. To meet future performance goals, the application of a new seal design and an improved understanding of leakage flow characteristics are of particular importance. The air mass flow through a labyrinth seal designed for a low-pressure turbine has been determined both through analytical calculus and CFD modeling. Different radial clearances and different air temperatures have been considered. In the next stage, the results will be validated through experiments.*

***Key Words:*** *labyrinth seal, leakage flow, turbo-machinery, Computational Fluid Dynamics (CFD)*

## 1. INTRODUCTION

The current gas turbine engine utilizes a multi-shaft approach with a separate high-pressure turbine (HPT) and a low-pressure turbine (LPT). Therefore, the LPT is an important component in determining the overall engine performance and efficiency. The optimization of the lower pressure turbine by minimizing the leakage flow of burning gases between the casing and rotor has become very important. While the total elimination of these leakages is not possible, it has become a necessity to reduce the clearances between the tip of the rotor blade and the casing by finding new labyrinth geometries that lead to a minimum leakage flow. Sealing technology is a key technology for developing high-performance gas turbine engines.

The leakage flow behavior of labyrinth seals has often been the subject of investigations in the past through numerical and experimental methods. An experimental study has been presented in [1], where the leakage rate has been measured on a high-speed straight-through labyrinth seal. Paolillo et al. [2] have conducted experiments for various stepped labyrinth seal designs and focused in particular on the effect of rotational speeds on the discharge characteristic. McGreehan et al. [3] have published the experimental results of windage heating for different labyrinth seal geometries and a tooth-to-tooth calculation algorithm has been derived from shrouded-disk correlations. Kong et al. [4] have investigated the effect of tip clearance by testing labyrinth rings with different rotor tip radii. Leakage flow rate and swirl ratios in the outlet cavity were measured at different rotating speeds. Childs et al. [5] have studied the dynamic characteristics of the labyrinth seal under high pressure and speed conditions. The CFD analyses were used to predict rotor-dynamic coefficients of the labyrinth seal with a negative-swirl brake. The results showed that the negative-swirl brake improved the seal stability. Untaroiu et al. [6] have studied the geometry of a labyrinth with a negative-swirl brake. The objective has been to suppress the circumferential flow and improve the seal stability. Cížek et al. [7] have presented the results on the influence of the labyrinth seal radial clearance on the air mass flow.

A generic labyrinth seal of a turbine aircraft engine was modeled. The target was to compare and analyze the influence of the radial clearance and teeth location. Bochon et al. [8] have conducted experiments to identify the basic flow parameters and flow structures in the seal and compare them with CFD results. A straight-through seal with two leaning fins and smooth or honeycomb land was analyzed.

Also, in the last decades, the usage of CFD flow simulations in the design of turbomachinery has grown very fast, as a consequence of the available computer power increase. Experimental methods of predicting the performance of a turbine are costly and time-consuming compared to the Computational Fluid Dynamics (CFD) approach. Monteiro et al. [9] have analyzed the first stage performance of an axial flow turbine, using a computational tool for simulating the steady state 2D/ 3D viscous flow.

The results were compared with those obtained from the mean line loss model code. The internal flow in turbo machines is extremely complex due to the level of details of the blades, with different angles and different levels of curvature and thickness, as well as the characteristics of the channel between blades, added to the phenomena caused by high speed rotation. Amaral et al.[10] seeked to optimize an axial turbine from a small gas turbine engine using ANSYS and Multi-Objective Optimization techniques focused on geometry changes to maximize the turbine performance.

In [11] Adnan and Hartono explain the taken steps for designing a single stage gas generator axial turbine for a small turbojet engine using CFD. A 3D simulation of the turbine full configuration is conducted to evaluate its overall performance.

The observed parameters including axial gap, stagger angle and tip clearance affect the output power of the turbine.

Gherman et al. [12] have described the improved test bench for rotating labyrinth seals developed at COMOTI. On the existing testing rig, only a seal configuration in a simplified environment (air temperature max.150°C, labyrinth diameter max. 300 mm, airflow rate max. 0.35 kg/s, pressure max. 9 bars) can be tested.

The newly improved research infrastructure will be able to increase these limits up to 800°C, 600 mm labyrinth diameter, 10 kg/s airflow rate and, 50 bars.

The goal of the investigations presented in this article is to determine the air mass flow through the labyrinth seal designed by the project topic manager, using analytical calculus and CFD modeling. Over the years, the CFD techniques have proven useful in designing different parts of a gas turbine, including the labyrinth seals [13, 14, 15, 16, 17].

At this stage, the proposed labyrinth seal teeth height, pitch, inclination, and number have been maintained constant, only the radial clearance has been varied, four values being taken into consideration.

These results will be validated later through experiments, using the test bench (Fig. 1) manufactured during the AIRSEAL project from the CLEAN SKY program. The test bench will be used to perform experiments regarding leakage in labyrinth seals for a low-pressure turbine. The test bench has been designed according to the requirements received from the project topic manager, namely: air pressure 4 bars, air temperature 150°C, labyrinth seal module rotation speed 15000 rev/min.

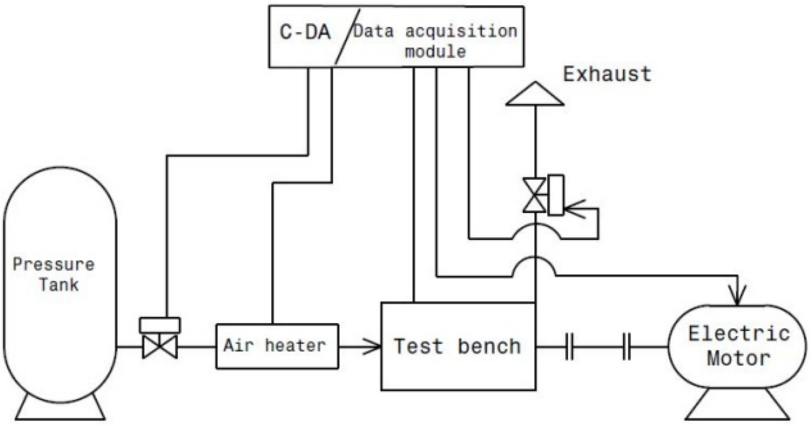

Fig. 1 AIRSEAL test bench

## 2. ANALYTICAL CALCULUS

To calculate the mass flow through the labyrinth seal, the following relation (eq. 1) has been used [18]:

$$\dot{m} = 0.685 \cdot \mu \cdot A \cdot \sqrt{\frac{P_0}{v_0}} \cdot \sqrt{\frac{1-\varepsilon_z^2}{z \cdot (1-\varepsilon_*)} - \frac{\varepsilon_* \cdot (1-\varepsilon_z)^2}{z^2 \cdot (1-\varepsilon_*)^2}} \tag{1}$$

where: $\mu$ - flow coefficient, $A$ – minimum section area, $P_0$ – pressure before the minimum section, $v_0$ – specific volume before the minimum section, $\varepsilon_z$ - pressure ratio, $\varepsilon_*= 0.13$ [18], $z$ – number of teeth.

If the edges of the labyrinth seal teeth are well rounded then $\varepsilon_*= 0.546$ [18]. For the given labyrinth seal, the values presented in Table 1 have been used for determining the air mass flow.

Table 1. Labyrinth seal input data

| Parameter | Value | Unit |
|---|---|---|
| $P_0$ (pressure before the labyrinth) | 400000 | Pa |
| $P_1$ (pressure after the labyrinth) | 100000 | Pa |
| $T_0$ (air temperature) | 293 / 373 / 423 | K |
| $v_0$ (specific volume) | 0.21 / 0.267 / 0.303 | $m^3/kg$ |
| $\varepsilon_z$ (pressure ratio) | 0.25 | - |
| $\varepsilon_*$ | 0.546 | - |
| $z$ (number of teeth) | 2 | - |
| $RC$ (radial clearance) | 0.3 / 0.5 / 0.8 / 1.5 | mm |
| $\mu$ (flow coefficient) | 1 | - |

The geometry of the labyrinth, the air pressure before the labyrinth and the air temperature of 423 K have been imposed by the project topic manager. In addition two air temperature values have been considered in this article, in order to observe this parameter influence on the air mass flow through the labyrinth.

The critical mass flow through the labyrinth seal is calculated using eq. 1 in which $\varepsilon_z$ is substituted with $(\varepsilon_*)_z$. $(\varepsilon_*)_z$ is calculated using eq. 2 [18]:

$$(\varepsilon_*)_z = \frac{\varepsilon_*}{z \cdot (1-\varepsilon_*) + \varepsilon_*} \tag{2}$$

Thus, the critical mass flow is calculated using eq. 3 [18]:

$$\dot{m}_c = 0.685 \cdot \mu \cdot A \cdot \sqrt{\frac{P_0}{v_0}} \cdot \sqrt{\frac{1-\left(\frac{\varepsilon_*}{z \cdot (1-\varepsilon_*) + \varepsilon_*}\right)^2}{z \cdot (1-\varepsilon_*)} - \frac{\varepsilon_* \cdot \left(1-\frac{\varepsilon_*}{z \cdot (1-\varepsilon_*) + \varepsilon_*}\right)^2}{z^2 \cdot (1-\varepsilon_*)^2}} \tag{3}$$

According to the algorithm presented in [18], if $\varepsilon_z \geq (\varepsilon_*)_z$, the air mass flow through the labyrinth seal is calculated using eq. 1.

Otherwise, the mass flow through the labyrinth seal stays constant and is equal to the critical mass flow calculated with eq. 3.

For the proposed labyrinth seal it was obtained $(\varepsilon_*)_z = 0.069519$. Thus $\varepsilon_z > (\varepsilon_*)_z$ and consequently, the air mass flow is calculate using eq. 1.

Fig. 2 presents the analytical calculated results regarding the air mass flow through the proposed labyrinth seal, for different air temperature values.

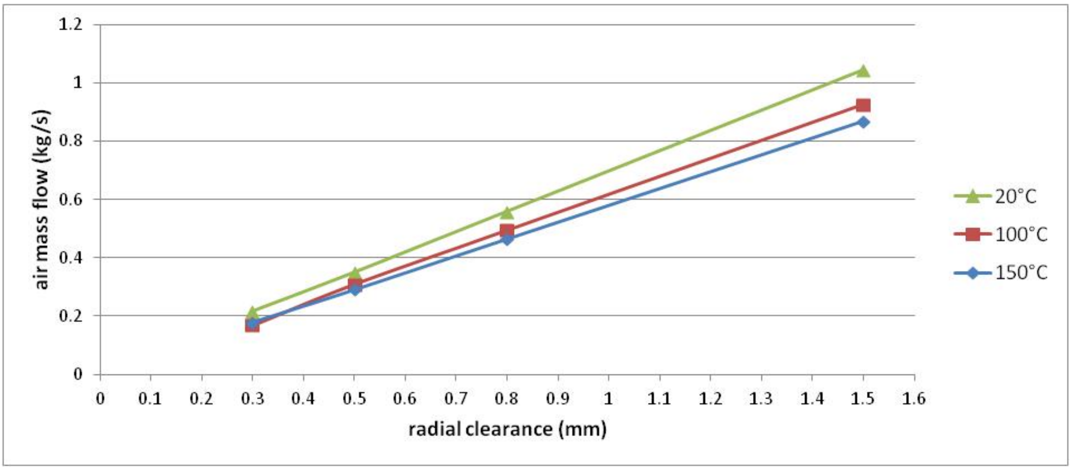

Fig. 2 Air mass flow through the labyrinth (analytical method)

As expected, the mass flow increases with the increase of the radial clearance. Also, the mass flow increases with the decrease of the air temperature.

This is explained by the fact that the air temperature decrease leads to an increase in air density, resulting a decrease in the air volume. Thus a greater air mass flow passes through the labyrinth.

## 3. CFD MODELING

Fig. 3 shows the geometry of the test bench on which the labyrinth module will be tested. The geometry of the labyrinth module is protected by copyright, thus more details cannot be provided/ therefore no further details can be provided. The test bench has been designed according to the requirements received from the project topic manager. The pressurized air enters axially the test bench. Then it goes through an inlet guiding vanes model, designed by the project topic manager, ensuring a desired air flow angle at the labyrinth seal entrance. The test bench continues with the rotating labyrinth seal module, a 90 degree vertical turn and the atmospheric outlet.

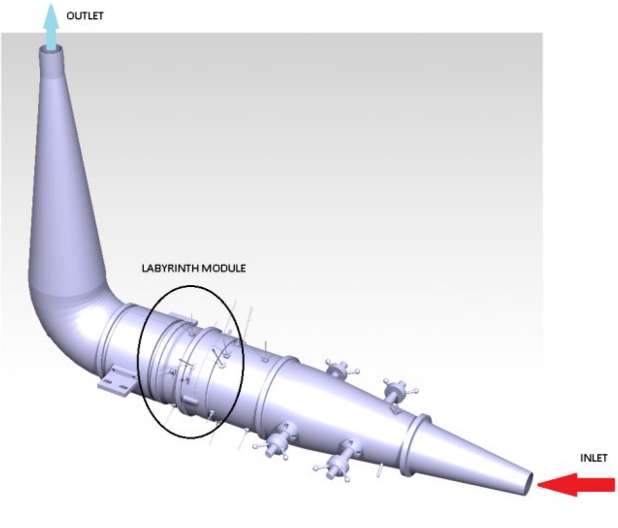

Fig. 3 Labyrinth seal test bench geometry

Based on the geometry presented in Fig. 3, the computational mesh has been generated using ICEM CFD. The geometry being symmetric, only half of it has been considered in the numerical simulations, thus shortening the computational time.

The mesh is of unstructured type. The domain is divided into 4 sub-domain. To better capture the flow through the labyrinth module, a boundary layer has been generated. The grid area-averaged y+ value for wall regions of importance, the labyrinth module walls, is 18 and for less important walls is 52.

A RANS type turbulence model has been chosen, namely, the k-ε model, which is a numerically stable and robust model and very popular in the realization of technical applications numerical simulations. Together with this turbulence model, the scalable wall function formulation developed by ANSYS CFX has been used.

Fig. 4 presents the computational domain together with the boundary conditions.

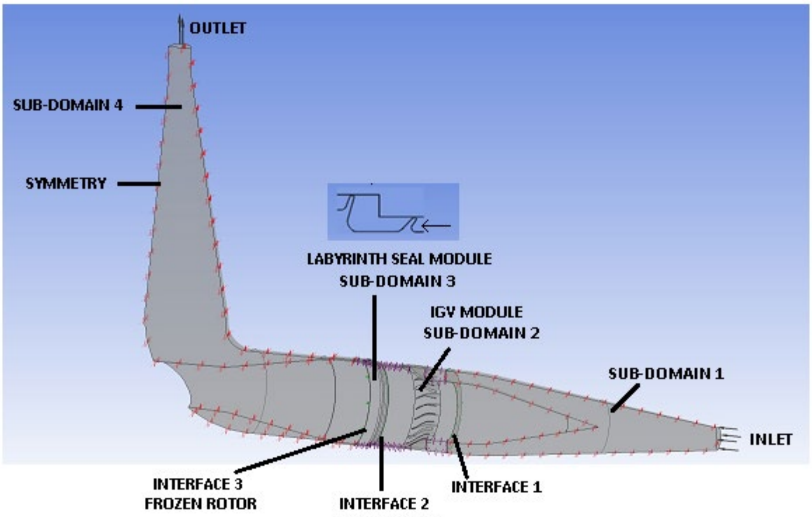

Fig. 4 The computational domain and boundary conditions

The reference pressure has been set to 1 bar. Sub-domains 1, 2, and 4 are stationary. Sub-domain 2 contains the inlet guiding vanes (IGV) module. This was included in the computational domain to ensure the air flow angle at the labyrinth seal entrance. Sub-domain 3, containing the labyrinth seal, is rotational.

The rotational speed has been set at 15000 rev/min. The walls are considered adiabatic and smooth. The interfaces between the stationary domains and the rotating domain are of frozen rotor type.

For each of the radial clearance considered values, the following boundary conditions have been set: at inlet the total relative pressure of 3 bars and the total temperature of 20°C / 100°C / 150 °C and at the outlet the static relative pressure of 0 bar.

## 4. RESULTS

The air mass flow through the labyrinth, obtained through numerical simulation, for each of the considered radial clearance values and air temperatures, is presented in Fig. 5.

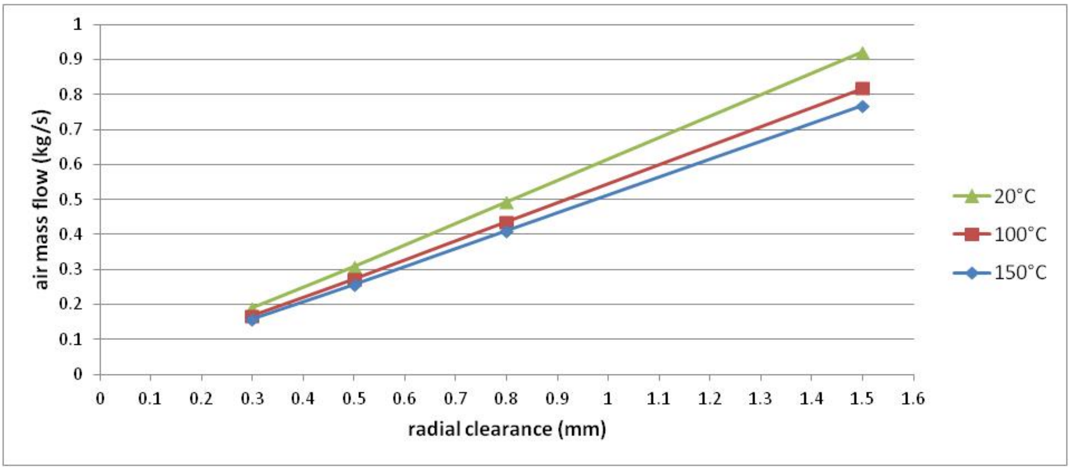

Fig. 5 Air mass flow through the labyrinth (CFD method)

Figures 6-8 compare the results obtained with the 2 methods.

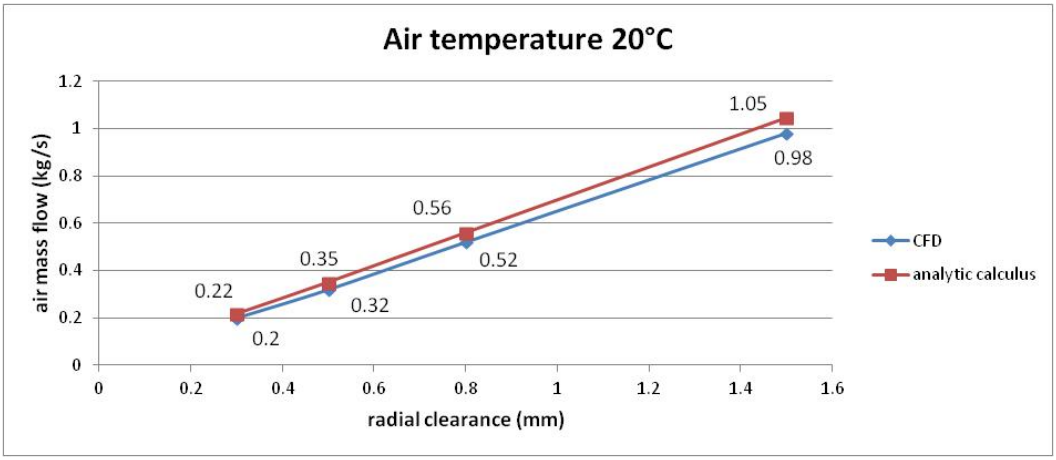

Fig. 6 Air mass flow - air temperature 20°C

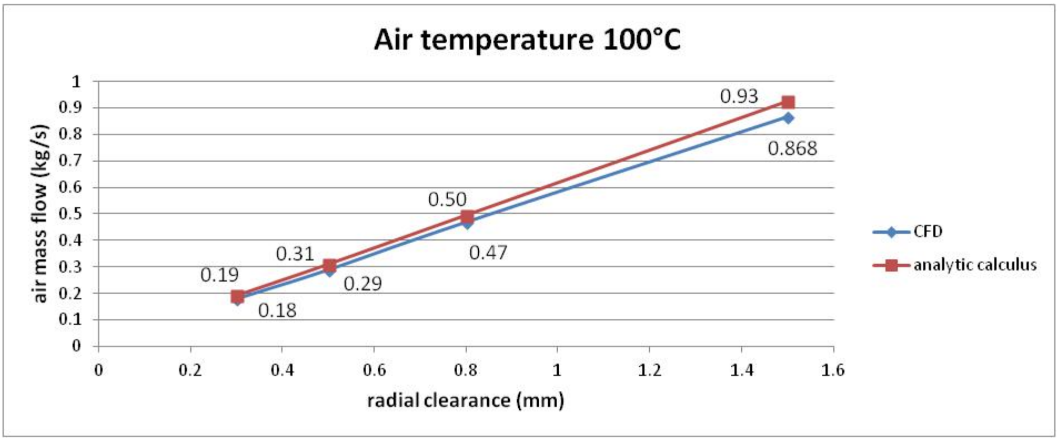

Fig. 7 Air mass flow - air temperature 100°C

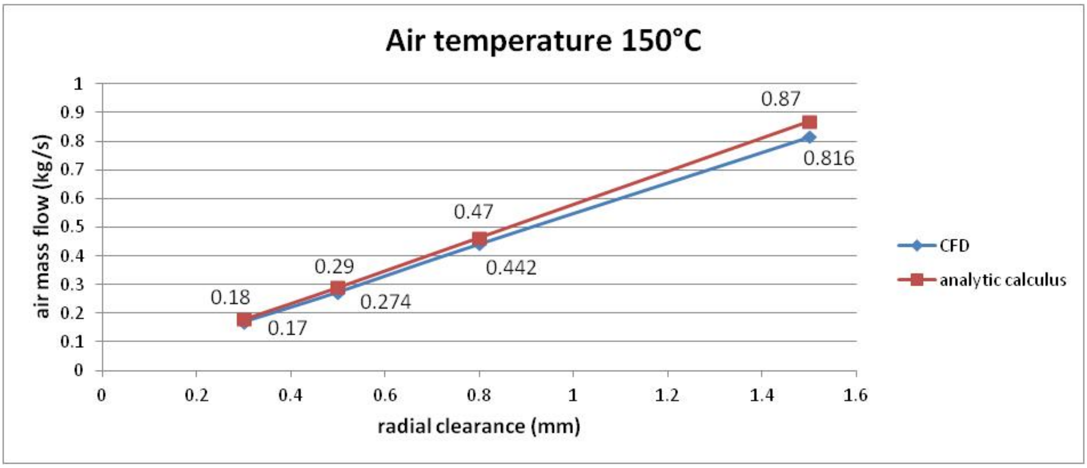

Fig. 8 Air mass flow - air temperature 150°C

As can be seen from figures 6-8, there is a difference of 5-7% between the mass flow values obtained through the 2 methods. This difference can be explained by the fact that in the numerical simulations the rotational speed has been taken into consideration. The turbulence induced by the high speed tangential component led to the increase of the labyrinth seal efficiency, thus decreasing the air mass flow rate.

## 5. CONCLUSIONS

The scope of the article was to determine, both through analytical calculus and numerical modelling, the air mass flow through a labyrinth seal designed for a low-pressure turbine. The labyrinth seal teeth height, pitch, inclination, and number have been kept constant while varying the radial clearance and the air temperature. Comparing the results obtained through the 2 methods, a difference of 5-7% has been obtained. This may be due to the fact that in the numerical simulations the rotational speed has been taken into consideration. Thus, because of the induced turbulence, leading to an increase of the labyrinth seal efficiency, the numerical simulation obtained mass flow is lower than the analytical calculus one. For future work, the rotational speed, and the air pressure ratio will also be varied to observe their influence on the air mass flow. The results will be validated through experiments which will be conducted on the test bench designed and manufacture within the AIRSEAL project. If the analytical and numerical results will be in good correlation with the experimental ones, than the above presented methods can be used in other labyrinth seal calculus, as well.

## ACKNOWLEDGMENTS

This work was carried out within the Clean Sky2 Program and European Structural and Investment Funds, Horizon 2020, supported by the European Commission and Romanian Minister of Research and Innovation through the implementation of the AIRSEAL project and the INFRASEAL project.

## REFERENCES

[1] W. Waschka, S. Wittig, S. Kim, Influence of High Rotational speeds on the Heat Transfer and Discharge Coefficients in Labyrinth Seals, *ASME Paper GT1990-330*, 1990.

[2] R. Paolillo, S. Moore, D. Cloud, et al., Impact of Rotational Speed on the Discharge Characteristic of Stepped Labyrinth Seals, *ASME Paper GT2007-28248*, 2007.

[3] W. McGreehan, S. Ko, Power Dissipation in Smooth and Honeycomb Labyrinth Seals, *ASME Paper 89-GT-220*, 1989.

[4] X. Kong, G. Liu, Y. Liu, L. Zheng, Experimental testing for the influences of rotation and tip clearance on the labyrinth seal in a compressor stator well, *Aerospace Science and Technology*, vol. **71**, pp. 556-567, 2017.

[5] D. W. Childs, E. James, Z. Ming, et al., Rotordynamic performance of a negative-swirl brake for a tooth-on-stator labyrinth seal, *J Eng Gas Turbines Power*, vol. **138**, no. 6, 2015.

[6] A. Untaroiu, H. Jin, G. Fu et al., The effects of fluid pre-swirl and swirl brakes design on the performance of labyrinth seals, *J Eng Gas Turbines Power*, vol. **140**, no. 8, 2018.

[7] M. Cížek, Z. Pátek, T. Vampola, Aircraft Turbine Engine Labyrinth Seal CFD Sensitivity Analysis, *Appl. Sci.*, vol. **10**, no. 19, 2020.

[8] K. Bochon, W. Wróblewski, A. Szymański, M. Majkut, M. Strozik, K. Marugi, Experimental and numerical tip leakage flow visualization in the LP turbine labyrinth seal, *E3S Web Conf*, vol. **137**, 2019.

[9] V. G. Monteiro, E. L. Zaparoli, C. R. de Andrade, R. C. de Lima, Numerical Simulation of Performance of an Axial Turbine First Stage, *J. Aerosp. Technol. Manag.*, vol.**4**, no. 2, pp. 175-184, 2012.

[10] L. G. F. Amaral, R. B. Diaz, J. T. Tomita, C. Bringhenti, I. B. Henriques, Optimization based on 3D CFD simulations of an axial turbine from a small gas turbine engine, *Aeronautics and Aerospace Open Access Journal*, vol. **3**, no. 4, 2019.

[11] P. F. Adnan, F. Hartono, Design of single stage axial turbine with constant nozzle angle blading for small turbojet, *J. of Physics: Conference Series*, vol. **1005**, 2018.

[12] B. Gherman, L. Flore, R. Carlanescu, M. S. Enache, O. Anghel, A. G. Radu, An overview of the new research infrastructure for rotating labyrinth seals at COMOTI, *IOP Conf. Series: Materials Science and Engineering*, 2021.

[13] W. Wang and Y. Liu, Numerical analysis of leakage flow through two labyrinth seals, *Journal of Hydrodynamics*, vol. **19**, no. 1, pp. 107-112, 2007.

[14] T. S. Kim and S. Y. Kang, Investigation of leakage characteristics of straight and stepped labyrinth seals, *International Journal of Fluid Machinery and Systems*, vol. **3**, no. 3, pp. 253-259, 2010.

[15] F. Cangioli, P. Pennacchi, L. Nettis and L. Ciuchicchi, Design and Analysis of CFD Experiments for the Development of Bulk-Flow Model for Staggered Labyrinth Seal, *International Journal of Rotating Machinery*, 2018.

[16] P. Kaszowski and M. Dzida, CFD analysis of fluid flow through the labyrinth seal, *Transactions of the Institute of fluid-flow machinery*, vol. **130**, pp. 71-82, 2015.

[17] M. Kamouni, Tridimensional analysis of a turbulent flow through an eccentric short labyrinth seal, *International Journal of innovation and Applied Studies*, vol. **19**, no. 1, pp. 37-45, 2017.

[18] G. Creta, *Turbine cu abur si cu gaze* (*Steam and gas turbines*), Editura Didactica si Pedagogica, Bucuresti, 1981.