

Numerical investigations of flow structure in gas turbine shroud gap

F Wasilczuk^{1,2}, P Flaszyński¹ and P Doerffer¹

¹Institute of Fluid-Flow Machinery, Polish Academy of Sciences, Fiszerza 14, 80-952 Gdańsk, Poland

²Interdepartmental PhD Studies at the Gdansk University of Technology Narutowicza 11/12, 80-233 Gdańsk, Poland

Abstract. The structure of the flow in the labyrinth sealing of an axial gas turbine was investigated by means of numerical simulations. Additionally, the flow structure for two- and three-dimensional axisymmetric models was compared. The porous disc as a model for the pressure drop relevant to the obtained in the cascade was proposed and tested. Several flow structure features existing in the sealing cavities are investigated: vortical structure and separation bubble on the rib and the correlation between the pressure drop and the clearance size. The carried out investigations indicate that the innovation aimed at decreasing the leakage flow through implementation of the flow control devices is possible. Furthermore the comparison between 2D and 3D models shows good agreement, thus application of less demanding 2D model introduces negligible differences. It is shown that the proposed porous disc model applied to mimic pressure drop in cascade can be effectively used for rotor blade sealing simulations.

1. Introduction

The leakage between the shroud and the casing is one of the most important sources of losses in shrouded cascades of gas turbines. It influences on the loss coefficient increase. Firstly it reduces the working potential, by decreasing the mass flow rate in blade passages [1]. Secondly, there is a difference of velocity distribution downstream of the rotor blade between the fluid passing through the gap and the blade passage. This results in mixing in highly non-uniform flow zone and entropy production [1,2,3]. Moreover, the vortices created in shear layer can affect the efficiency of the subsequent stator [2,3] and they can have negative influence on thermal conditions on following nozzle guide vane. The leakage losses can be limited by reduction of the amount of the mass flow in the gap between the shroud and the casing.

One of the most popular methods to decrease the gap flow between the shroud and the casing is a labyrinth seal. It is made of ribs mounted on the shroud, creating cavities along the gap. The kinetic energy fluid is dissipated in those cavities influencing on mass flow reduction in the gap.

The ultimate aim of the research, INNOLOT-COOPERNIK is the investigation of the labyrinth seal modifications that will limit the mass flow through the seal, decreasing losses caused by the leakage. There is a lot of studies aimed at leakage loss reduction by analysing of the 3-dimensional flow structure and application flow control devices to match the fluid velocity at the gap and in the mainstream, to limit the mixing [4,5,6,7]. This paper presents results of flow structure investigations in the labyrinth seal as the first step of the further modifications and flow control methods application.



2. Numerical model description

2.1. Geometry

The results presented in the paper are obtained for the geometry created as the simplification of the configuration for a large-scale, low-speed gas turbine rotor stage, investigated in European project AITEB-2 [8]. The main objective of the numerical investigations is to study the effect of applied model simplifications on the flow structure. An important result of the leakage arises from the passage flow interaction with leakage flow downstream of the sealing. Simulations for full cascade configuration should include passage flow and sealing. However, if the investigations are focused on the sealing configuration only, the cascade is simplified or modelled. The results in the paper are carried out for the model where blade row is substituted with porous disc which enables to model the effect of pressure drop in cascade. The porous disc model is described in the next chapter. Some changes were also applied to the geometry of the labyrinth seal, which is located on the disc. Firstly, the last of the three seal ribs was neglected in the model. Secondly, the roundings were simplified by corners, which significantly reduce the complexity of the mesh, but not influence on these basic investigations. The sketch of the cross section model with the most important dimensions is shown in Figure 1.

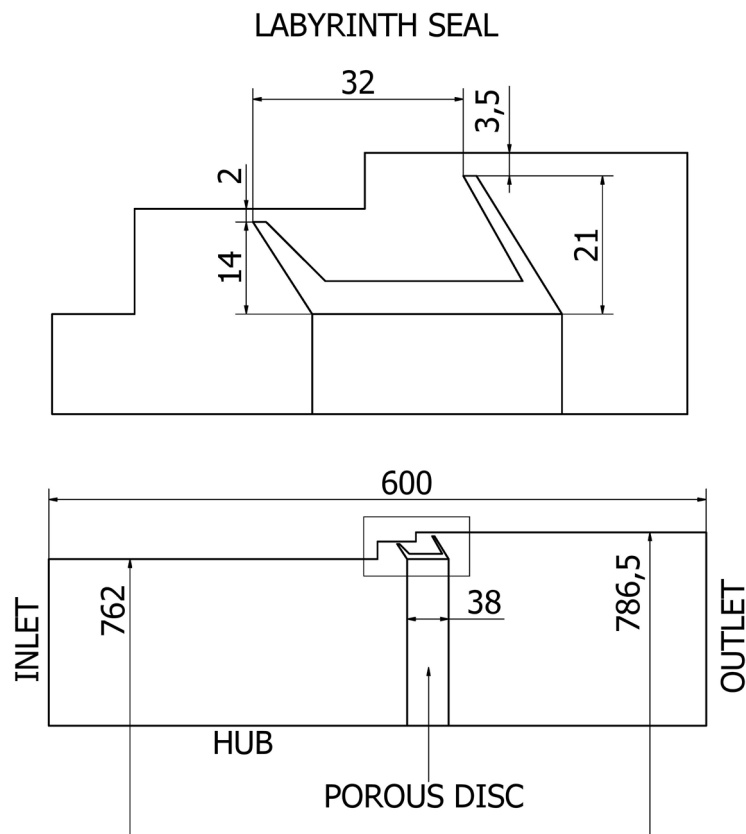


Figure 1. The geometry of the model.

2.2. Boundary Conditions

The boundary conditions were set according to the operating conditions of test rig [8]. –At the inlet, total temperature is 293K, the turbulence viscosity is 0.0001. Five cases for different inlet total pressures were analysed (Table 1). The outlet static pressure was set for all cases to 98325 Pa. The speed of rotation at labyrinth walls was assigned 359 RPM, corresponding to the rotational velocity at the test section.

Table 1. Boundary conditions.

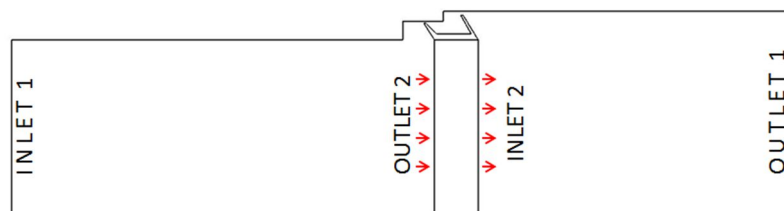
Case No.	Inlet [Pa]	Outlet [Pa]	Pressure ratio (π)
1.	99419	98325	1.01
2.	101419	98325	1.03
3.	103419	98325	1.05
4.	105419	98325	1.07
5.	107419	98325	1.09

2.3. Numerical parameters

Two structured meshes were used for the numerical simulation: 2D and 3D. 2-dimensional, axisymmetric mesh contains ~70000 elements. 3-dimensional consists of 1.3 million cells with the 2° angle of extrusion. The calculations were carried out in Fine/Turbo Numeca as steady ones by means of Spalart-Allmaras turbulence model [9]. The solver uses multigrid technique with coarse grid initialization. In order to improve convergence Merkle low-speed preconditioning and CFL number of 1 was used. The perfect gas model is used.

3. Porous disc

Simulating the whole blade row including all 3 dimensional features is time and resource consuming. Thus, due to the investigations effectiveness of wide range of various seal configurations, the simplified model was applied. The analysis of the flow over rotating, impermeable disc [10,11,12] showed formation of the vortex upstream of the disc that changed the flow conditions in a gap. Thus in the numerical investigation the labyrinth seal cannot be just attached to the impermeable disc. There is a need to simulate the main features of the blade with a less complex model. There are two methods that can be utilized. First is to take data of flow features upstream and downstream of a blade row from the numerical simulation or experimental measurements, and apply them as additional outlet and inlet in the area, where blades are supposed to be (Figure. 2) [13]. However, this approach requires accurate data to set as boundary conditions. Moreover, with the flow becomes constrained with extra boundary conditions, and the impact of the seal flow on the main flow is difficult to observe.

**Figure 2.** Sketch of the model with additional inlet and outlet instead of the blade row.

Another considered approach is to model the pressure drop in the blade row by means of the porous disc. Based on appropriately chosen porosity parameters, one can achieve a pressure drop similar to the one in a blade row, keeping the same the mean axial velocity downstream of cascade. The main drawback of this solution is that it is only two-dimensional, all of the complex 3-D flow structures are not modelled, and one cannot study the interaction of the leakage flow with the main flow. However, since the ultimate goal of the investigation is to study the features of the flow within the seal, this drawback is not much affecting the main objective. Thus, the porous disc approach was chosen for this study.

Fine/Turbo allows various porous media formulation, but for this case the Darcy law was used to determine the pressure drop on the disc (1).

$$\vec{\nabla} p = -\frac{\mu}{\kappa} \vec{v} \quad (1)$$

where:

$\bar{\nabla}p$ – pressure drop

μ – viscosity

\vec{v} – velocity

κ – intrinsic permeability given by (2)

$$\kappa = \frac{\varepsilon^3 \left(\frac{6}{A}\right)^2}{180(1 - \varepsilon)^2} \quad (2)$$

where A is the specific surface area and ε is the porosity. [14]

In the specific case of blades row simulation, the entire pressure drop will occur on the porous disc, independently of its parameters and its value will depend on the boundary conditions. However, the porosity parameters will significantly affect velocity of the flow. Using the relation between pressure drop and velocity it is impossible to estimate both porosity and specific surface area. For that reason, porosity was constrained to 0.9, and from that the estimate specific surface area was calculated. Then it was tuned using numerical calculations to mimic the velocity from the experiment [8]. Figure 3 shows the pressure distribution in the model, with visible pressure drop in the porous zone, proving that the model is working as intended.

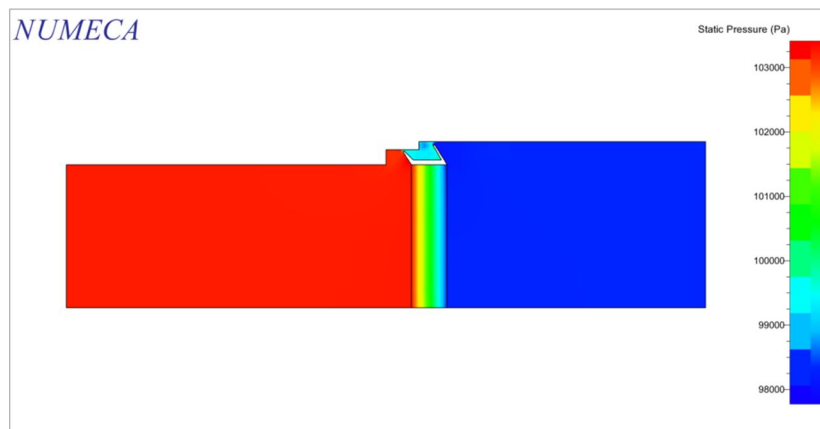


Figure 3. Pressure contour in the meridional view.

4. Results

The static pressure distribution in the whole computational domain is shown in the figure 3. Application of the porous disc model leads to the uniform pressure upstream and downstream of the disc as assumed at inlet and outlet plane. The same pressure decrease occurs inside the modelled cascade (disc) domain and in the labyrinth seal.

As mentioned above, the presented results are considered as the reference flow structure investigations, which is planned to be modified and controlled in the next step by means of flow control methods. Due to that, selected flow features are analysed below.

4.1. Pressure drop in the gaps between the ribs and the casing

In the figure 4, the pressure drop in each gap between the rib and the casing is shown. The pressure drop above each rib is proportional to the pressure ratio in the cascade (figure 5) and the gap height (distance between rib and shroud). In the investigated case, the clearance is different above both ribs (2 mm for the first gap and 3.5 mm for the second), what influences on the pressure drop differences. It is several times bigger in the first gap than in the second one. The first rib is responsible for approximately 80% of the pressure drop, while the second one for the remaining 20%.

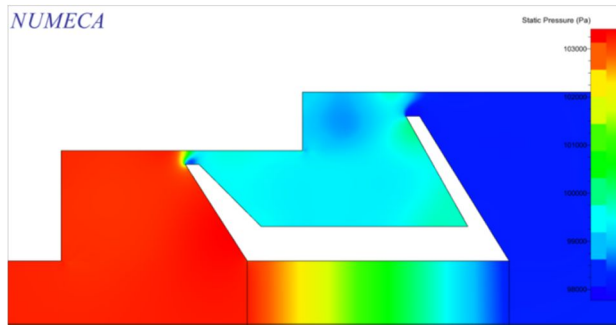


Figure 4. The pressure distribution in the labyrinth seal

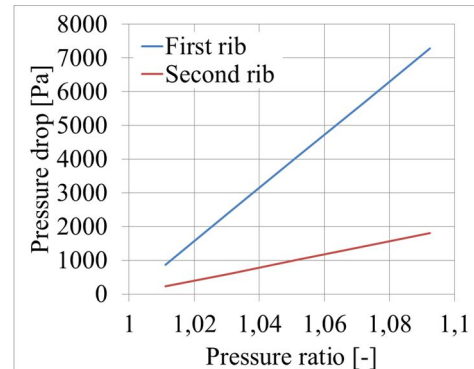


Figure 5. Pressure drop in the gap between the rib and the casing as a function of pressure ratio.

4.2. Acceleration and separation in the contractions

The clearance between the first rib and the casing influences on the flow acceleration. As the effect of the local acceleration, the jet is generated which interacts with the second rib. Such interaction is dependent on the second rib location, its shape and sealing geometry (figure 6). The maximum velocity of the jet as the function of pressure ratio is shown in figure 7. Moreover, the separation is induced at the leading edge of the rib, which reduces the effective area of the gap. This phenomenon can be also noticed in the flow over rotating disc with sharp edges [11, 12]. Similar situation occurs on the second rib, although the gap is higher, so the jet velocity is lower. Anyway, one has to emphasize that this flow feature is strongly dependant on the inflow velocity, flow direction and the geometry details of the rib (inclination, sharp or rounded edge). [12, 15]

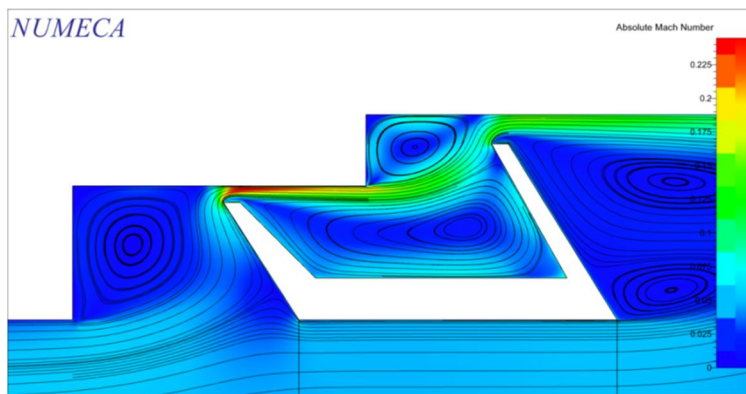


Figure 6. Streamlines and absolute Mach number contour.

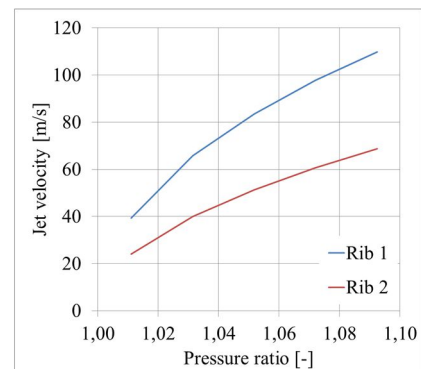


Figure 7. Maximum velocity of the jet.

4.3. Pressure rise on the surface of the second rib.

The pressure rise can be observed on the surface of the second rib at the upstream side, next to the tip of the rib (figure 8). It is caused by the jet generated by the contraction above the first rib influencing on the created stagnation zone on the second rib (figure 8). The pressure distribution on the second rib and the location of the maximum value (stagnation point) is dependent on the pressure ratio in cascade (figure 9). This feature is very important if flow control method based on the local pressure difference is considered to apply.

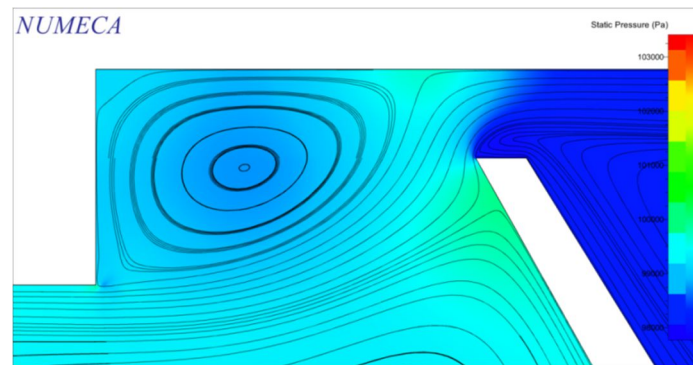


Figure 8. Streamlines and static pressure contour.

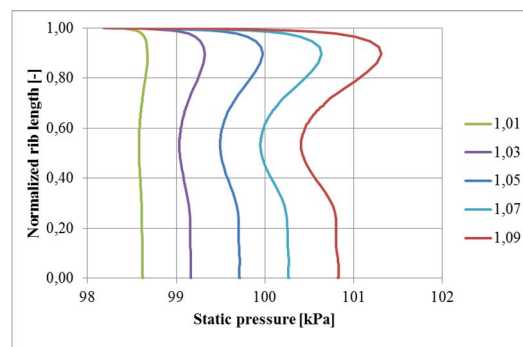


Figure 9. Static pressure on the wall of the second rib.

4.4. Vortex structure

Vortex structure of the flow upstream and inside the labyrinth seal was analyzed. The flow downstream of the sealing in real cascade is strongly affected by interaction with the passage flow, so its analysis downstream of the blade row on the simplified model could not lead to any meaningful conclusions. However, the flow structure inside the cavity is weakly affected by the outer (passage) flow and it is similar for different pressure ratio in all analyzed cases.

Upstream of the seal, there is one large vortex created by low velocity, which located in the corner of a casing (figure 6). The size and shape of this vortex is highly dependent on the casing geometry. The large vortex that is powered by the jet generated above the first rib is created in the cavity between the two ribs. Due to the viscosity, the circumferential velocity of the shroud drives the fluid in this direction. For that reason the vortex has significant tangential velocity, unlike the two others, which is shown in figure 10.

The last major vortex is located in the corner of the casing, upstream of the second rib. Similarly to the main vortex in the cavity, it is driven by the jet and its intensity (strength) is dependent on kinetic energy transfer between the jet and the corner vortex. However, because it is separated from the rotating seal by the high velocity jet, its tangential velocity component is not significant. In the center of this vortex, there is a region of low pressure (figure 6).

It is worthwhile to note, that various labyrinth seals differ from each other significantly. The configuration depends on the stage geometry: rotor, upstream and downstream nozzle guide vanes and casing. Finally, the vortex structure, its intensity and location in the sealing zone is reliant on the investigated geometry.

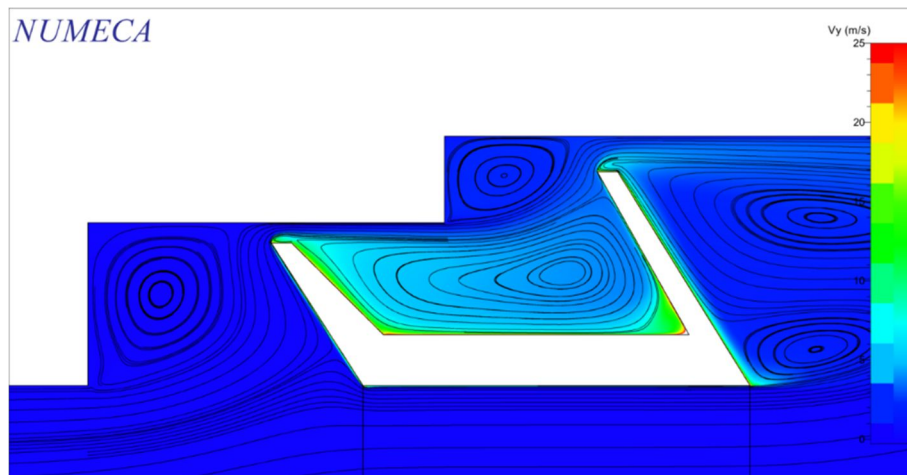


Figure 10. Streamlines and circumferential velocity contour

4.5. 2D and 3D model comparison

The velocity components distribution at four traverses (figure 11) for 2D and 3D models were compared, to compare the effect of the model simplification. The example of the velocity components comparison at traverse 1, which is located in the gap above the first rib, is shown on figures 12, 13 and 14. The curves are marked by different colors, the 3D model with brighter line and dots while 2D model with darker solid line. In case of all traverses, for all velocity components, results obtained by means of two-dimensional and three-dimensional model are very similar, showing always the same trend in spite of little local differences. Thus, using substantially less computationally demanding 2D axisymmetrical model for further studies is justified and does not lead to significant differences in comparison with 3D model.

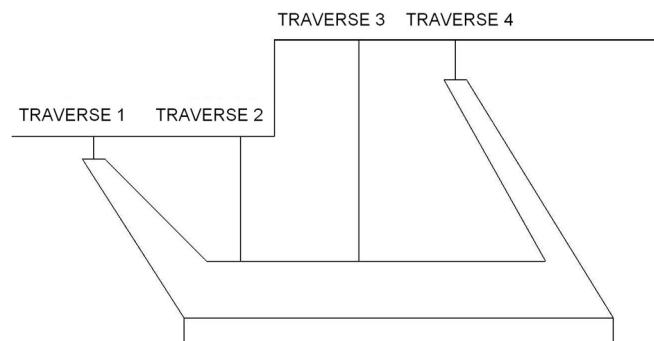


Figure 11. Traverses used for velocity comparison

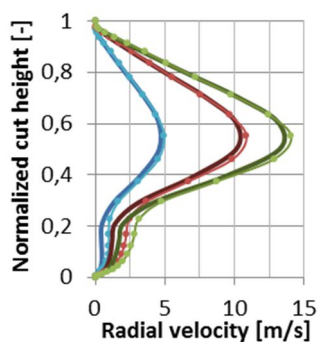


Figure 12. Radial velocity component on traverse 1.

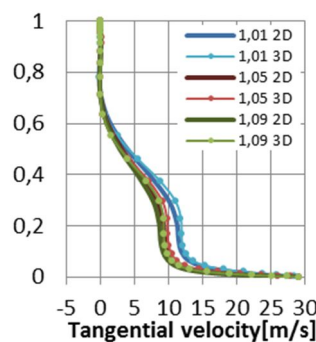


Figure 13. Tangential velocity component on traverse 1.

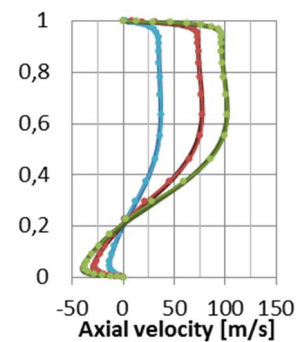


Figure 14. Axial velocity component on traverse 1.

5. Summary, conclusions and future work

The flow features of the rotor blade labyrinth sealing were investigated. It was discovered that the vortical structure relies heavily on the geometry and its location and intensity is dependent on the pressure ratio and rotational speed of cavity walls. Moreover the jet created downstream of the contraction above the first rib is a source of the pressure rise at the upstream side of the second rib. This feature is specific for the investigated labyrinth sealing, but it can be harnessed for flow control application. Finally, the comparison of 2D and 3D models shows the same trends with little local differences.

Several important conclusions, valid for the future implementation of flow control devices aimed at decreasing the gap flow, can be drawn from this study. Firstly, for the purpose of preliminary investigations of flow control devices much less computationally demanding 2D axisymmetrical model can be used. Secondly, most of the flow features are highly dependent on the sealing cavities geometry. Thus the geometry modification or introducing a flow control device is likely to change the flow structure significantly and possibly reduce the leakage flow. For this particular case, the pressure rise on the upstream side of the second rib surface is promising, as the potential location the flow control device based on local pressure difference.

Future studies will be focused on various air jet flow control devices, which will be optimized and implemented on ribbed configuration.

Acknowledgments

This work was supported by the Coopernik project “Cooperative Research for Next Generation High Efficiency LP Turbine” INNOLOT/I/11/NCBR/2014. This research was supported by CI TASK and by PL-Grid Infrastructure in part.

References

- [1] Denton J D 1993 *J. Turbomach* **115**(4), 621–656
- [2] Rosic B, Denton J D and Pullan G 2005 *J. Turbomach* **128**(4), 699–707
- [3] Lampart P, 2006 *Task Quarterly* **10**(2), 139-175
- [4] Wallis A M, Denton J D and Demargne A A J 2000 *J. Turbomach* **123**(2), 334-341
- [5] Porreca L, Kalfas A I and Abhari R S 2008 *J. Turbomach* **130**(3)
- [6] Rosic B, Denton J D, Curtis E M and Peterson A T 2008 *J. Turbomach* **130**(4)
- [7] Rosic B, Denton J D 2008 *J. Turbomach* **130**(2)
- [8] Lehmann K, Thomas R, Hodson H and Stefanis V 2009 *ASME Paper No. GT2009-59531*, 515-525
- [9] Spalart, P R and Allmaras S R 1992 *AIAA Paper* **92**
- [10] Wasilczuk F, Flaszyński P and Doerffer P 2015 *Task Quarterly* **19**(2), 89-100
- [11] Puzyrewski R and Jakubek A 1989 *Prace Instytutu Maszyn Przepływowych*, **89**
- [12] Puzyrewski R and Jakubek A. 1991 *Prace Instytutu Maszyn Przepływowych*, **93**
- [13] Frączek D, Wróblewski W and Chmielniak T 2015 *Proc. of 11th European Conf. On Turbomach. Fluid Dyn. & Thermodynamics ETC11*
- [14] Numeca International 2014 *Theoretical Manual* Brussels
- [15] Zimmerman H, Kammerer A and Wolff K H 1994 *ASME Paper No. 94-GT-131*,