24th International Conference
ENGINEERING MECHANICS 2018
Svratka, Czech Republic, May 14 –17, 2018
Paper **#109**, pp. 821–824, doi: 10.21495/91-8-821

NUMERICAL SIMULATION OF FLOW THROUGH A SIMPLIFIED MODEL OF THE SHAFT-SEAL

P. Straka*, J. Pelant**

Abstract: *The contribution deals with numerical modelling of flow through a simplified shaft labyrinth seal for various operating regimes. The objective is to obtain dependency of the mass flow rate through the seal and the outlet flow angle from the seal on the pressure ratio and the rotation. The results will be used as a background for modification of the test-rig for axial turbine stage.*

Keywords: CFD, internal fluid dynamics, turbulent flow, shaft-seal

1. Introduction

Experimental and numerical investigation of flow and energy properties of the stator-rotor stages plays a major role for increasing of rotary blade-machines efficiency. The experimental test rig was built (Jelínek and Němec, 2011 and Němec et al., 2012, Fig. 1a) for measurement of the aerodynamic and energy properties of the isolated axial turbine stages (Straka et al., 2015 and Straka and Němec, 2015).

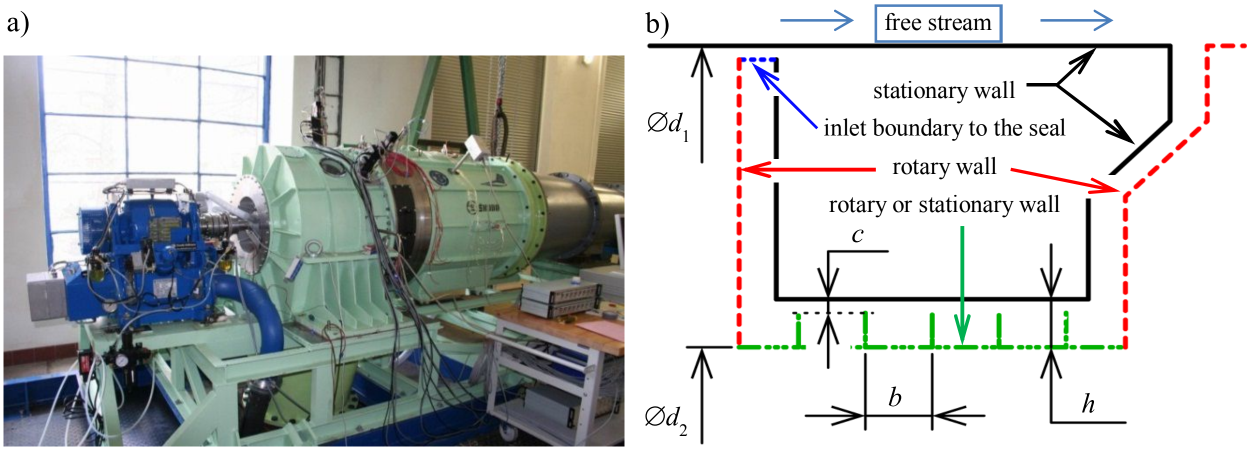

Fig. 1: a) turbine test rig, b) scheme of simplified model of the shaft-seal.

The axial turbine stages work mostly in multistage configuration. In this case the inlet flow-field is affected by flow through foregoing stages. The effect of the foregoing stages was partially simulated via manipulation with the inlet span-wise profiles of the velocity and turbulent intensity (Jelínek et al., 2016). Subsequently the ability of the test rig has been extended for measuring of so called *one and half stage* configuration which consists of one axial turbine stage (stator and rotor wheel) followed with next stator wheel. That configuration allows catching the effect of the foregoing turbine stage on the flow field in subsequent stator blade wheel (Němec et al., 2017/a and Němec et al., 2017/b).

One of the significant effects that influence the flow in peripheral parts of the blade-span, is the outlet stream from the labyrinth-seal (shaft seal at the root part or shroud seal at the tip part). For this reason, the effort is to extend the test rig capability to include the effect of the shaft- and shroud-labyrinth-seal.

* Ing. Petr Straka, Ph.D.: Czech Aerospace Research Centre, Beranových 130; 199 05, Prague – Letňany; CZ, straka@vzlu.cz

** RNDr. Jaroslav Pelant, CSc.: Czech Aerospace Research Centre, Beranových 130; 199 05, Prague – Letňany; CZ, pelant@vzlu.cz

Paper (Straka, 2017) deals with numerical study of flow through a simplified model of the shaft-labyrinth-seal for various geometry configurations and operating states, all for pure axial flow direction in the free stream region above the shaft-seal area (see fig. 1b or fig. 3a).

The interaction of the stream from the shaft-seal with the free stream has an important impact on the efficiency of the stage. Therefore, it should be studied an effect of the flow direction in the free stream area on mentioned interaction.

Present work deals with numerical simulation of flow through a simplified model of the shaft-seal (fig. 1b) under condition of non-axial flow direction in the free stream region above the shaft-seal. Presented results together with that from paper (Straka, 2017) are used as the basis for selection of the shaft-seal design variant for the turbine test rig. Figure 2 shows two possible variants of the shaft-seal design in *one and half stage* configuration: a) the shaft-seal rotating together with the rotor disc, b) stationary shaft-seal. The first variant allows to include the effect of the seal-fins rotation on flow through the shaft-seal, the second variant is structurally much simpler and more precise during assembly. The second variant also does not affect the dynamics properties of the rotor disc.

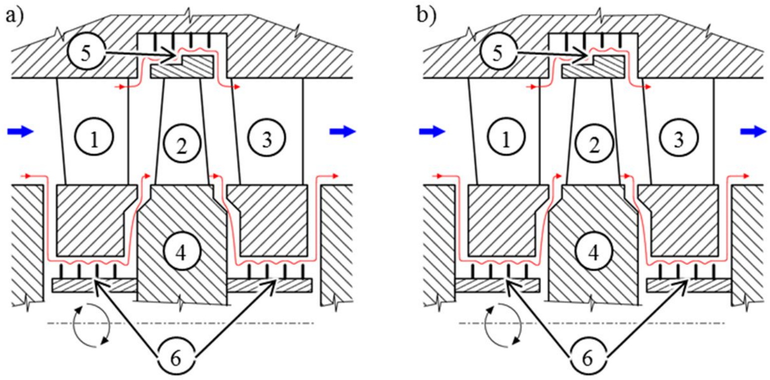

Fig. 2: Scheme of the one and half stage *with a) rotary shaft-seal, b) stationary shaft-seal; 1 – first stator blade, 2 – rotor blade, 3 – second stator blade, 4 – rotor disc, 5 – shroud-seal, 6 – shaft-seal.*

Figure 1b shows scheme of the simplified shaft-seal model with description of the boundaries. The outer shaft diameter is for all configurations $\varnothing d_1$ = 220 mm, diameter of the bottom wall is $\varnothing d_2$ = 177.5 mm, the seal-clearance is c = 1.2 mm, pitch of the seal-fins is b = 6 mm, the height of the seal area is h = 4.5 mm number of seal-fins is n = 4. The thickness of the seal-fins is 0.3 mm.

2. Numerical results

Flow through the shaft-seal is modelled as steady, 3D, compressible, turbulent flow of the ideal gas. Figure 3a shows a 3D computational domain which takes angular section related to one pitch of the first stator blades. At the inlet boundary to the shaft-seal there are prescribed the total pressure p_T = 102 kPa, the total temperature T_T = 300 K, turbulence intensity Tu = 2.2%, ratio of the turbulent to the molecular viscosity μ_t / μ = 100 and radial flow direction. The total pressure and temperature of the free stream are the same as at the inlet boundary to the shaft-seal. Flow direction of the free stream is α_1 = 72° from axial direction. The computations were carried out for the static to total pressure ratio in range $0.778 \le p_S / p_T \le 0.975$, rotation speed of the rotating walls in range $900 \le \mathrm{RPM} \le 5100$.

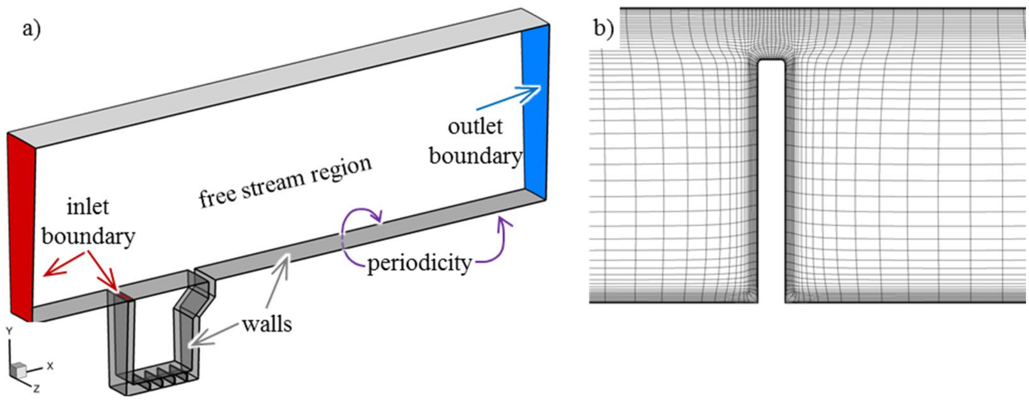

Fig. 3: a) computational domain; b) detail of the computational mesh.

Calculations of flow were performed using the in-house numerical software (Straka, 2016/a and Straka, 2016/b), which is based on solution of RANS equations closed with two-equations turbulence. The nonlinear explicit algebraic model of the Reynolds stress (Rumsey and Gatski, 2003) is used in this work.

The computational domain is discretized by multi-block structured computational mesh of hexahedral cells (detail in fig. 3b). The computational mesh is refined close to the walls to ensure that the viscous sub-layer is covered at least by five cells.

Figure 4a demonstrates different character of the stream from the shaft-seal interaction with the free stream for pure axial free stream direction and for free stream flow direction $\alpha_1 = 72°$ (which corresponds with the flow direction behind first stator wheel in *one and half stage* configuration mentioned above). Main difference is in size of the separation bubble behind the outlet slit of the shaft-seal. Bottom row of fig. 4 shows the mass flux density defined as: $\dot{m} = [(\rho u_{ax})^2 + (\rho u_{rad})^2]^{0.5}$, where ρ is the density and u_{ax} and u_{rad} are axial and radial components of the velocity vector.

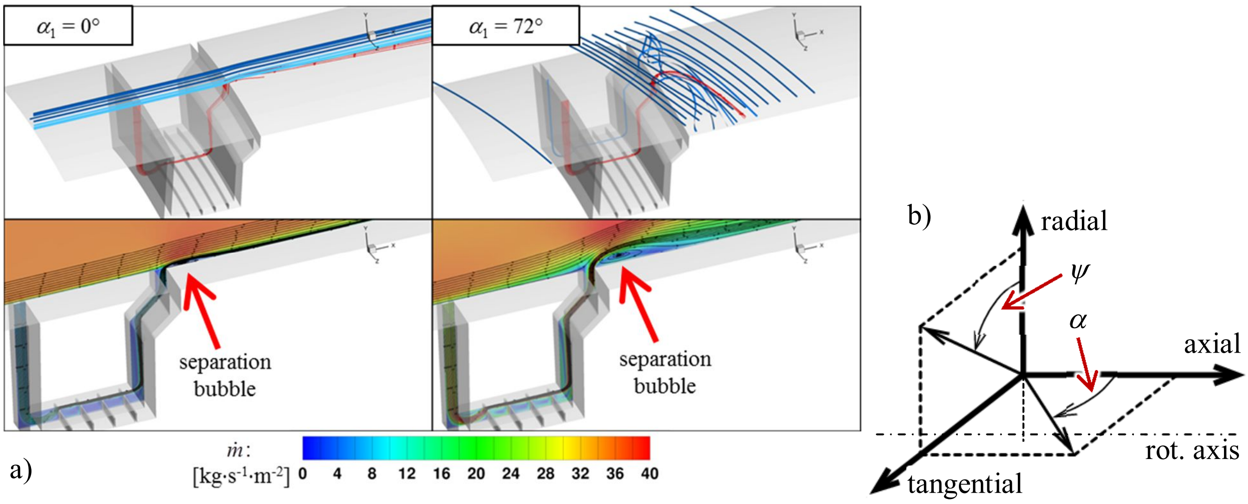

Fig. 4: a) interaction of the stream from the shaft-seal with the free stream for $\alpha_1 = 0°$ (left column) and $\alpha_1 = 72°$ (right column). Top row: stream lines, bottom row: contours of the mass flux density and the stream lines projection to the meridian plane. Regime: stationary shaft seal, $p_S / p_T = 0.925$, 900 RPM; b) the deflection angles definition.

Figure 5 shows the dependency of the mass flux Q through the shaft-seal on the rotation speed and on the static to total pressure ratio. It is evident that the rotation speed has only minor impact on the mass flux value. On the other hand, the difference between the stationary and rotary shaft-seal increase with increasing of the rotation speed. By other words, the effect of the seal-fins rotation is more significant at higher RPM. Rotation of the seal-fins leads to little decreasing of the mass flux. This is an effect of the centrifugal force which leads to narrowing of the effective radial clearance above the seal-fins.

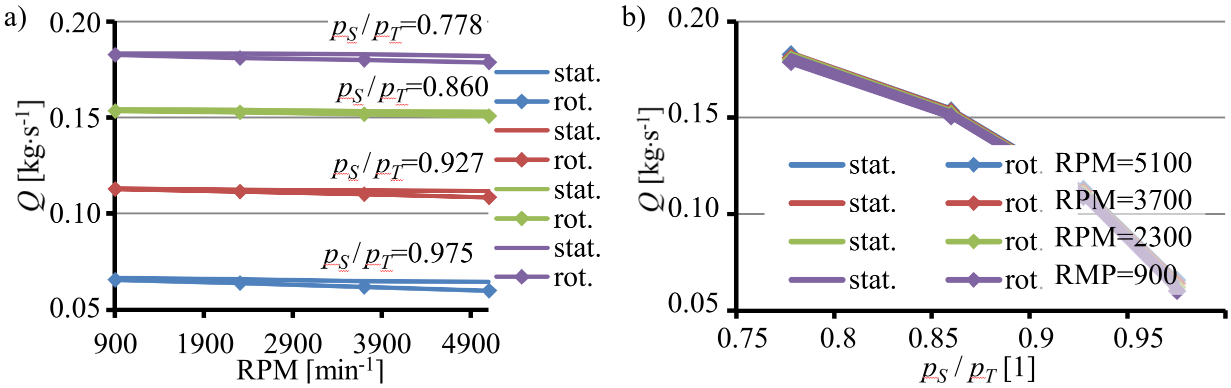

Fig. 5: Dependency of the mass flux on: a) the rotation speed, b) static to total pressure ratio.

Figure 6 shows dependency of the deflection angles at the outlet from the shaft-seal in the radial-tangential plane – angle ψ and in the axial-tangential plane – angle α (see fig. 4b). One can see that the higher rotation speed, the larger deflection angle. Also it can be stated that the deflection angles increase with increasing of the static to total pressure ratio. It is given by the fact that for higher values of the static to total pressure ratio the fluid spends longer period of time in the shaft-seal domain. The effects of the rotor disc and of the seal-fins rotation are therefore higher. Absence of the seal-fins rotation effect in case of the stationary shaft-seal leads to decreasing of the deflection angles in both radial-tangential and axial-tangential planes.

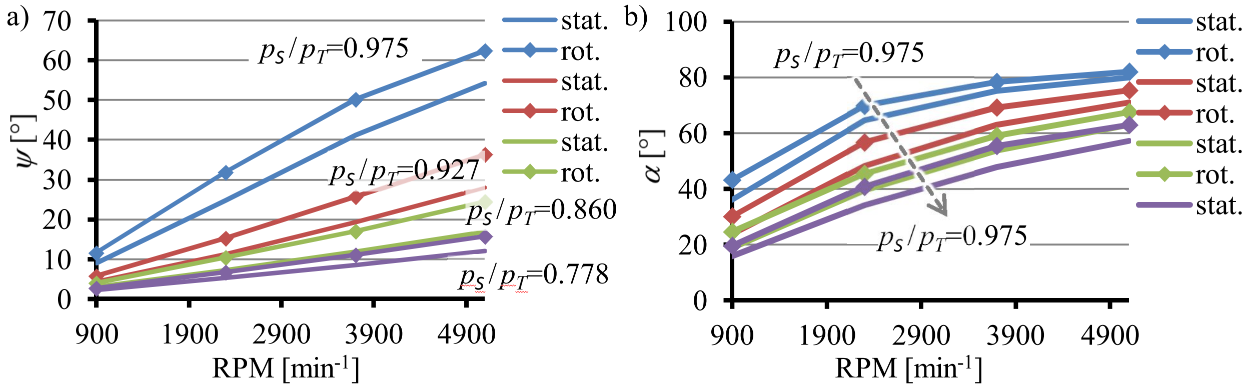

Fig. 6: Dependency of the deflection angle on the rotation speed in: a) the radial-tangential plane, b) the axial-tangential plane.

3. Conclusions

Based on performed simulations of flow through the simplified model of the shaft seal under the condition of the free stream angle α_1 = 72° it can be stated that:

- the mass flux is almost independent on the rotation speed in case of the stationary shaft-seal,
- the seal-fins rotation in case of the rotary shaft-seal leads to little decreasing (up to two percent) of the mass flux against the stationary shaft-seal,
- the deflection angles increase with increasing of the rotation speed and with increasing of the static to total pressure ratio,
- absence of the seal-fins rotation effect in case of the stationary shaft-seal leads to decreasing of the deflection angle against the rotary shaft-seal (up to ten degrees in the radial-tangential plane – angle ψ and up to eight degrees in the axial-tangential plane – angle α).

Acknowledgement

This work was supported by the project TH02020086 of the Technology Agency of the Czech Republic.

References

Jelínek, T., Ně mec, M. (2011) Investigation of Unsteady Flow in Axial Turbine Stage. In: *Proc. Conf. Experimental Fluid Mechanics 2011*, 2, Jičín, pp. 706-716.

Jelínek, T., Němec, M., Uruba, V. (2016) Flow Parameters Simulation Technique in Annular Turbine Cascade Inlet. In: *Engineering Mechanics 2016*, IT AS CR, Prague, pp. 254-257.

Němec, M., Jelínek, T., Babák, M. (2012) In-Scale and Up-Scale Full Turbine Stage Meassurements as a Support for Small Turbine Units Developement, In: *Proc. Conf. ASME Turbo Expo 2012*, Kobenhavn, GT2012-69693.

Němec, M., Jelínek, T., Milčák, P. (2017/a) An Investigation of a Flow Field in One and Half Axial Turbine Stage, *AIP Conference Proceedings* 1889, 020025.

Němec, M., Jelínek, T., Milčák, P. (2017/b) Clocking of Stators in one and Half Stage of Axial Steam Turbine, In: *Proc. Conf. Experimental Fluid Mechanics 2017*, Mikulov, pp. 465-469.

Rumsey, C.L., Gatski, T.B. (2003) *Summary of EASM Turbulence Models in CFL3D with Validation Test Cases*, NASA Report/TM-2003-212431.

Straka, P. (2016/a) Numerical Simulation of High-Swirl Flow in Axial Turbine Stage, *EPJ Web of Conferences* 114, 02115.

Straka, P. (2016/b) *Software for Calculation of Turbulent Unsteady Flow Using the Nonlinear Turbulence Model*, Report VZLÚ R-6381, Prague (in Czech).

Straka, P. (2017) Numerical Study of Shaft-Seal Parameters for Various Geometry Configurations and Operation Regimes, In: *Proc. Conf. Experimental Fluid Mechanics*, Mikulov, pp. 613-618.

Straka, P., Němec, M. (2015) Influence of the Radial Gap under the Stator Blade on Flow Around the Hub-end of the Rotor blade. In: *Engineering Mechanics 2015*, ITAM AS CR, Prague, Svratka, pp. 294-295.

Straka, P., Němec, M., Jelínek, T. (2015) Investigation of Flow in Axial Turbine Stage Without Shroud-Seal, *EPJ Web of Conferences* 92, 02088.