Influence of the Radial Gap Under the Stator Blade on Flow Around the Hub-End of the Rotor

Petr Straka[a*], Martin Němec[b]

Výzkumný a zkušební letecký ústav, a.s., Beranových 130, 199 05 Praha-Letňany, Czech Republic

[a]straka@vzlu.cz, [b]nemec@vzlu.cz

Keywords: turbomachinery, axial turbine stage, drum-type rotor, secondary flow

Abstract: The contribution deals with experimental and numerical investigation of flow-field in an axial turbine stage with prismatic blades which are not equipped with a shroud. Main attention is paid to the influence of the radial clearance under the stator blade on generating of the secondary flow structure which dramatically affect flow-field in the hub part of the rotor blade. Results of the numerical simulation are compared with the experimental data. Both numerical and experimental data point to a significant drop in efficiency of the stage due to interaction of the rotor blades with the secondary vortices generated behind the radial clearance under the stator blade.

Introduction

The axial steam turbines of low power (around hundreds kilowatts to units megawatts) are typically designed with the drum-type rotor as shown in Fig. 1 (left). For reduction of a production cost the shroud-less stator and rotor wheels are still in use. In this configuration it is necessary to maintain the radial clearance under the hub-end of the stator blades and above the tip-end of the rotor blades. However, flow through the radial clearance generates large secondary flow structures which has negative influence on the efficiency.

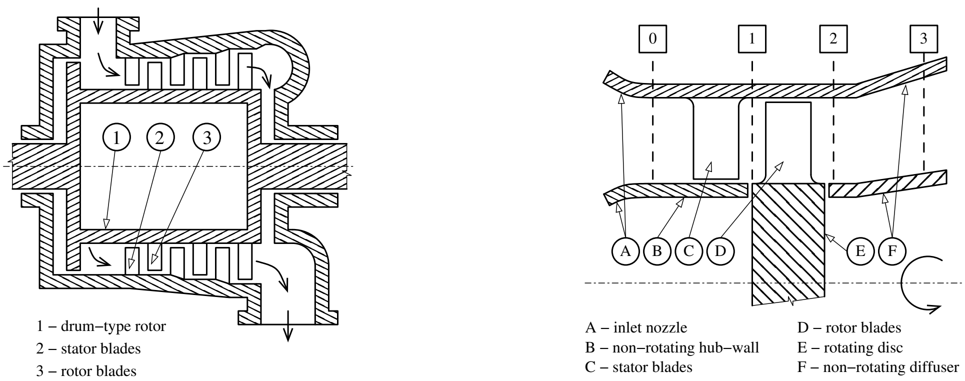

Fig. 1: Simplified scheme of axial turbine with drum-type rotor (left). Scheme of experimental stage (right).

The effect of the radial clearances is investigated experimentally and numerically on the testing stage which is schematically shown in Fig. 1 (right). Tested regimes are defined by total/static pressure ratio in range $1.4 \leq p_{T0}/p_{S2} \leq 1.9$ and velocity ratio in range $0.35 \leq u/c_{2is} \leq 0.65$, where p_{T0} is the total pressure in plane 0, p_{S2} is the static pressure in plane 2, u is the circumferential velocity at the middle diameter and c_{2is} is the isentropic velocity in plane 2. Planes 0 and 2 are defined in Fig. 1.

The test rig (see. [1, 2]) is a part of the high-speed closed-loop wind tunnel allowing independent setting of the Mach and Reynolds numbers. Testing stage is loaded by hydraulic dynamometer. A concept of the rotatable stator was used in consequence of linear probe manipulators. Fast response pressure probes were also used in order to study the unsteady flow field at the outlet of the rotor. The mass flow rate is measured using the orifice plate.

The in-house CFD code (see. [3, 4]) is used for simulation of the unsteady, compressible, viscous, turbulent flow of the perfect gas in testing stage. The CFD code is based on the finite volume method for discretization of the RANS equations closed by $k - \omega$ turbulence model. Multi-block structured mesh of hexahedral elements is used for discretization of the computational domain which is bounded by planes 0 and 3 (as shown in Fig. 1) and contains periodical repeated section containing five stator and six rotor blades. Interface between stator part and rotating rotor part of the computational domain is implemented via the sliding mesh technique.

Results and discussions

Left part of Fig. 2 shows secondary vortices generated behind the radial clearance under the hub-end of the stator blades. These vortices are drifted on the higher diameter along the suction side of the rotor blades, which leads to decreasing of the efficiency at lower forty percent of the span as shown in Fig 2 right. The efficiency drop at lower forty percent of the span is of about ten percent. The efficiency at upper twenty percent of the span is negatively influenced by secondary vortices generated behind the radial clearance above the tip-end of the rotor blades.

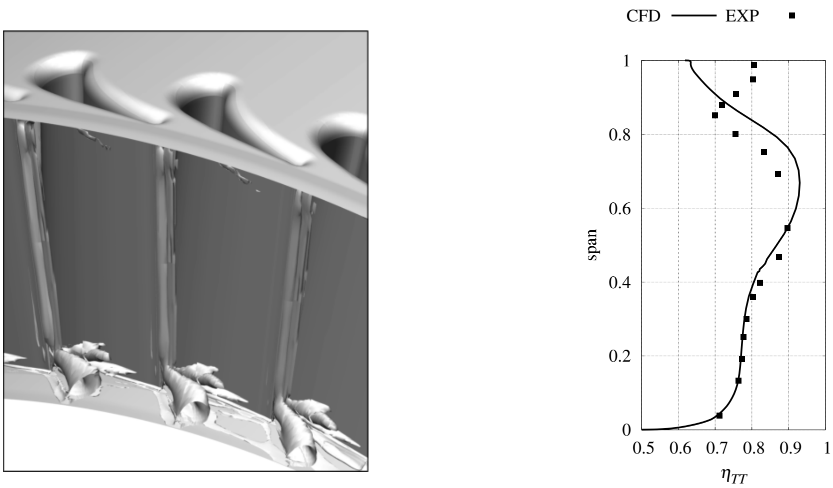

Fig. 2: Vortices generated behind the hub-end of the stator blades (left), span-wise distribution of the total-total efficiency (right).

Acknowledgments: This work was supported by the grant project TA02021336 of the Technology Agency of the Czech Republic.

References

[1] T. Jelíınek, M. Němec, The axial turbine stage test rig, Turbomachinery 2010, Plzeň, (2010) (in Czech).

[2] M. Němec, T. Jelínek, J. Benetka, Test turbine VZLÚ integral characteristics measurement, VZLU report R-4939, Prague, (2010) (in Czech).

[3] P. Straka, Simulation of a 3D unsteady flow in an axial turbine stage, in: Proc. Conf. Exp. Fluid Mech. 979-987, (2011).

[4] P. Straka, Calculation of 3D unsteady inviscid flow in turbine stage ST6, VZLÚ Report R-4910, Prague, (2010) (in Czech).