TRANSACTIONS OF THE INSTITUTE OF FLUID-FLOW MACHINERY

No. 135, 2017, 101–115

Paweł Flaszyński* and Michał Piotrowicz

An investigation of hot spot location at turbine inlet – numerical simulations for factor project test rig

Institute of Fluid-Flow Machinery, Polish Academy of Sciences, Fiszera 14, 80-231 Gdańsk, Poland

Abstract

The main objective of the presented paper is the investigation of a flow structure and parameters distribution downstream of combustor simulator and its effect on turbine inlet. The investigations are carried out by means of numerical simulations for combustor and turbine nozzle guide vane configuration existing at test rig in DLR Goettingen. As the effect, the hot spot location for different relative positions of the swirler and nozzle guide vanes is shown. The presented results are obtained within a pre-test simulations supporting the final design of the test section and they allow to draw conclusions for the required limiting cases investigated experimentally. The location of the hot spot is highly important for thermal loading of the first stage rotor.

Keywords: Turbomachinery; Gas turbine; Hot spot; Combustor-turbine interaction

1 Introduction

In order to reduce fuel consumption, modern turbomachinery operates at high velocities and high temperature conditions. The lack of confidence in the prediction of combustor-turbine interaction leads to apply extra safety margins on components design. Consequently, the understanding of combustor-turbine flow field interactions is mandatory to preserve high-pressure turbine (HPT) life and performance when optimising the design of new high-pressure turbine and combustors (e.g., lean burn combustors). A lot of projects have investigated combustor

*Corresponding author. E-mail address: pflaszyn@imp.gda.pl

technologies and others addressed the challenge of understanding the behaviour of hot flow structures in the HPT. All those projects gave a better understanding of the physical behaviour of the combustor and the turbine and brought improvements on the designs of both modules. However industrial experience shows that the separate optimisation of the two modules – combustor and turbine – does not ensure that the system in which they are embedded will also be optimum. The link between the combustor and the turbine in an engine is very tight and all engine manufacturers are putting a strong effort to master this interface.

Historically, one of the first investigations on combustor turbine interaction were carried out by Stabe *et al.* in 1984 [8]. The influence of non-uniform radial temperature profiles on the performance of a turbine compared to the assumed uniform inlet temperature was experimentally studied. The overall performance of the turbine was found not to be affected. In 1995 Shang *et al.* [9] introduced a test rig enabling investigations of hot streaks and radial temperature distortions in a transonic turbine stage. In the early years of combustor hot streaks effect on the turbine was considered in terms of its influence on the rotor, the review presented by Dorney *et al.* in 1999 [3]. In 2004 Jenkins *et al.* [4] experimentally investigated a hot-streak impinging on the nozle guide vane (NGV) leading edge, which was film-cooled by a showerhead configuration. One of the more complex rigs is operating at the Wrigth-Patterson Air Force Base in Dayton, Ohio. Results of investigations carried out in a combustor simulator [1] (it is operated nonreacting) have been published focusing on the flow field within the NGV in [2,6,7].

The main objective of FACTOR project (*Full Aerothermal Combustor-Turbine Interactions Research*), funded by European Commission, is to optimise the combustor-HPT interaction design. In order to get a detailed understanding of the combustor-HPT interactions, a very challenging test rig is designed and manufactured, which will be hosted by DLR Goettingen (Deutsches Zentrum für Luft- und Raumfahrt). The designed test stand is very challenging task because it has to meet technological requirements and it should also enable possibly high flexibility for measurement techniques access. Measurement techniques require limited temperatures in the investigated zones. Due to such limitations it was decided to design and manufacture combustor simulator instead of real combustor. It means that the combustor simulator should be enable to generate velocity and temperature distribution upstream of nozzle guide vane similar to the real case. Combustor simulator requires very careful design in order to obtain the 'realistic' hot spot. In the paper, results for the two relative locations of combustor simulator and NGV are presented. Numerical simulations have been carried out in order to find the two limiting cases. The first one, the hot spot is located in front of leading edge and the second one, the hot spot is located in the middle of the blade passage.

2 Geometry and numerical model description

Numerical simulations were carried out for the flow domain shown in Fig. 1 by means of two commercial solvers. In both cases Reynolds averaged Navier Stokes method is applied, which is based on the mass, momentum and energy conservation equations:

- mass conservation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \ , \quad i,j = 1,2,3 \ , \tag{1}$$

- momentum conservation

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) + \frac{\partial p}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} = 0 \ , \tag{2}$$

- energy conservation

$$\frac{\partial}{\partial t}\left[\rho(e + \frac{1}{2}u_i u_i)\right] + \frac{\partial}{\partial x_j}\left[\rho u_j\left(h + \frac{1}{2}u_i u_i\right)\right] + \frac{\partial q_j}{\partial x_j} - \frac{\partial}{\partial x_j}(u_i \tau_{ij}) = 0 \ , \tag{3}$$

where ρ — density, t – time, u – velocity component, p – pressure, τ_{ij} – viscous stress tensor, e – internal energy per unit mass, h – enthalpy per unit mass, q – heat flux, subscripts x, y, z denotes Cartesian coordinates, and closing equations according to the selected turbulence models, state equation for perfect gas and Sutherland formula for viscosity. Results obtained for k-w SST [5] (Ansys/Fluent) [13] and Spalart-Allmaras (Fine/Turbo Numeca) [12] are presented.

As shown in Fig. 1, computational domain includes one swirler and two nozzle guide vanes. Inlet conditions are applied upstream of the swirler at the inlet duct and on the combustor limiting surfaces described as mp-*. The additional inlet condition is located inside the NGV, where the coolant is supplied (LE, TE plenum). The outlet conditions are applied at the plane downstream of NGV. The structured mesh was generated with IGG/Numeca (Interactive Geometry Modeler and Multi-Block Structured Grid Generator) and AutoGrid5 [11], automatic full hex multiblock mesh generator for multi-stage turbomachinery.

The multiblock topology consisted of 3016 blocks and 33.6×10^6 mesh cells. An example of a mesh for a swirler, combustor and NGV is shown in Fig. 2.

Full not matching boundary connection (FNMB) was applied at interfaces: inlet duct-swirler, swirler-combustor simulator, combustor simulator – NGV.

In case of Fluent 2nd order upwind and central difference scheme with scalar artificial dissipation for spatial discretization [10] in case of Fine/Turbo were employed.

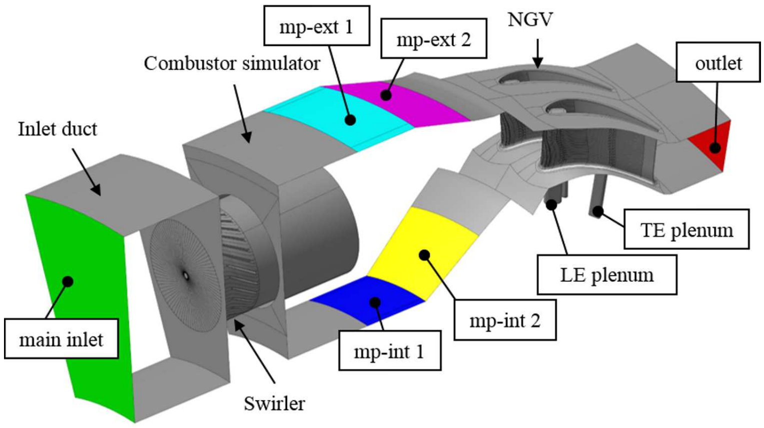

Figure 1: Computational domain.

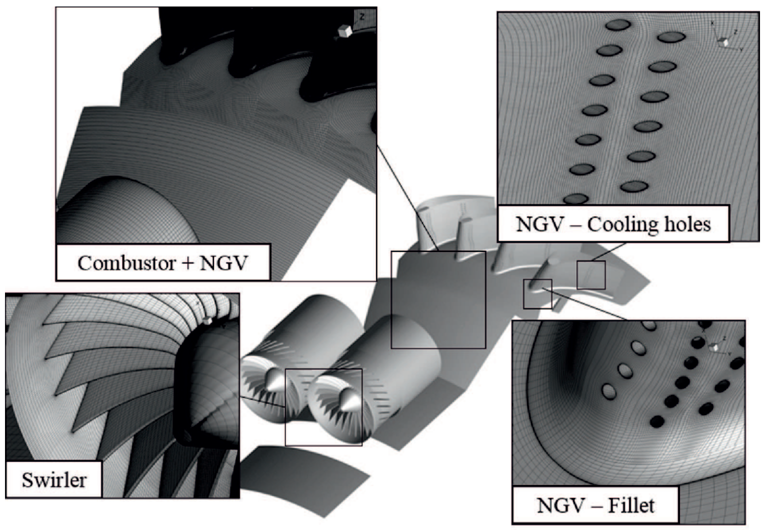

Figure 2: View of mesh details.

Boundary conditions were set as follows: total mass flow rate of 4.8 kg/s in the test section with air ratio: 65% hot flow + 35% cold flow. At the inlet, upstream of the swirler mass flow rate 0.156 kg/s (per one pitch), total temperature 530 K, turbulence intensity 1%, and viscosity ratio 10 are set. Inlet conditions were applied at liner (perforated plates): mass flow rate 0.084 kg/s (per one pitch), total temperature 300 K, turbulence intensity 1%, and viscosity ratio 10. Massflow rate distribution at liner surfaces is shown in Tab. 1. There are two patches at the inner and outer surfaces with uniform massflow rate distribution with different inflow angles.

Table 1: Cold flow distribution.

Combuster liner	Mass flow (%)	Velocity angle at combustor liner [deg]
mp-ext 1	13	60
mp-ext 2	8	30
mp-int 1	10	60
mp-int 2	4	30

At the outlet plane static pressure 79.4 kPa is set. The outlet plane was located downstream of NGV where the swirl highly influences on the pressure distribution, so radial equilibrium condition also was applied. At all walls adiabatic conditions are applied.

An important feature of the geometry are cooling holes in NGV. Each NGV contains eight rows of cooling holes, in total there were more than 200. Rows from 1 to 6 were supplied by leading edge (LE) plenum channel and other rows were connected to trailing edge (TE) plenum (Fig. 3). In case of Fine/Turbo cooling holes were calculated together with plenum and the boundary conditions for the individual plenum channel were defined as following: total temperature 300 K, turbulence intensity 2%, turbulent viscosity ratio 10, mass flow 0.006 kg/s (LE plenum) and 0.012 kg/s (TE plenum). In case of Fluent, boundary conditions were applied at the cooling holes inlet: total pressure 155 000 Pa, total temperature 300 K, turbulence intensity 2%, turbulent viscosity ratio 10.

As mentioned above, the test section was designed for the combustor turbine interaction, but actually no combustion is present. Similar distribution of temperature and velocity has to be provided by correct design of combustor simulator. In order to control the temperature distribution upstream of NGV cascade, swirler

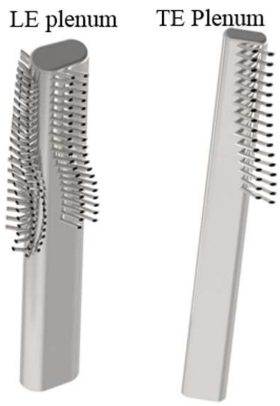

Figure 3: NGV cooling holes.

duct (Fig. 1) was designed. Modification of the swirler duct length influences on the flow conditions at combustor simulator outlet.

The two relative swirler and NGV locations were investigated. The two configurations are shown in Fig. 4. In the first case, swirler is located upstream of NGV leading edge. The location is marked by swirling line downstream of swirler hub, in figure on left side, NGV1. The second case concerns of swirler located upstream blade passage (NGV2). It means that in both cases every second nozzle guide vane or blade passage is located downstream of swirler.

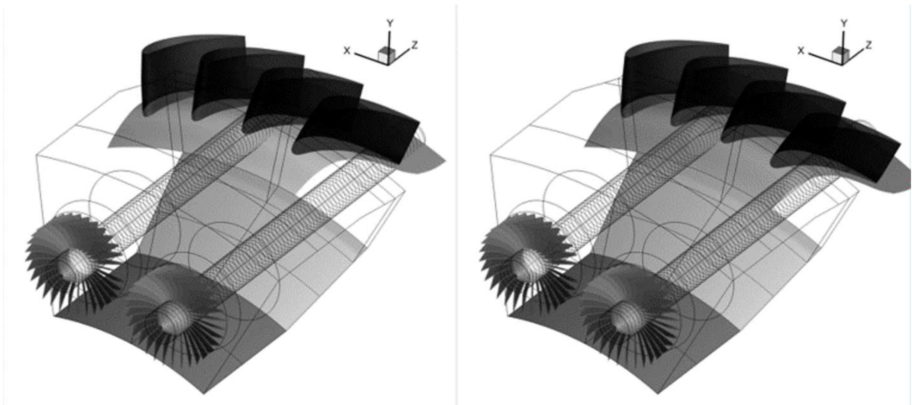

Figure 4: Two relative positions NGV downstream of swirler: NGV1 (left) and NGV2 (right).

3 Velocity distribution upstream of NGV

General view of the velocity distribution in meridional plane in combustor simulator is shown in Fig. 5. Shown velocity distribution is normalized by the area averaged velocity in plane upstream of the NGV. One can notice high velocity downstream of swirler, in swirler duct. In the centre, low velocity zone indicates existence of the swirler hub wake. Downstream of the swirler duct, due the shear layer and the influence of the inflow through inner and outer liner surfaces, velocity distribution is highly nonuniform. In the figure on the right, velocity magnitude is also presented at the plane upstream of the NVG (shown in the plot). Velocity distribution arises from the swirling flow and indicates existence of the vortex core with lower velocity. Such effect is important for temperature distribution and its further interaction with NGV and film cooling. Circumferentially areas of averaged velocity magnitude and total temperature are shown in Figs. 6 and 7. In both plots, results for both solvers are presented. Circumferentially averaged velocity is more uniform in case of Fine/Turbo with Spalart-Allmaras model due to difference close to inner conical casing. It arises from differences in prediction of mixing of main flow and coolant in combustor liner zone. On the other side, total temperature distribution indicates existence of the hot spot upstream of NGV, which is crucial for thermal loading of guide vane and rotor blade of gas turbine.

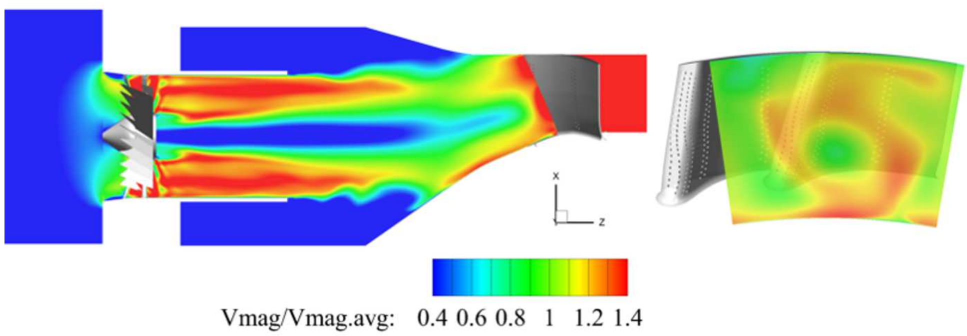

Figure 5: General view of velocity distribution in meridional section and plane upstream of nozle guide vane.

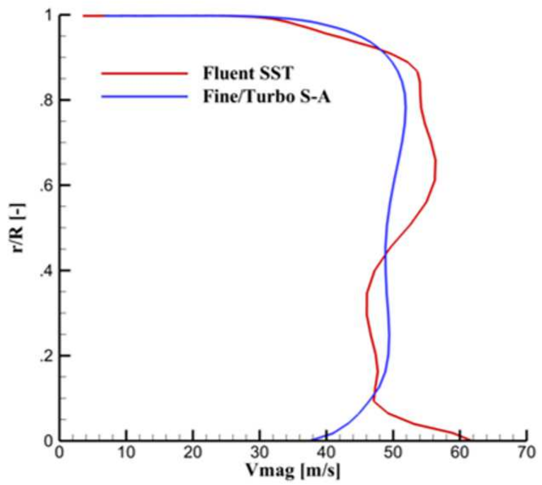

Figure 6: Circumferentially averaged velocity magnitude upstream of NGV.

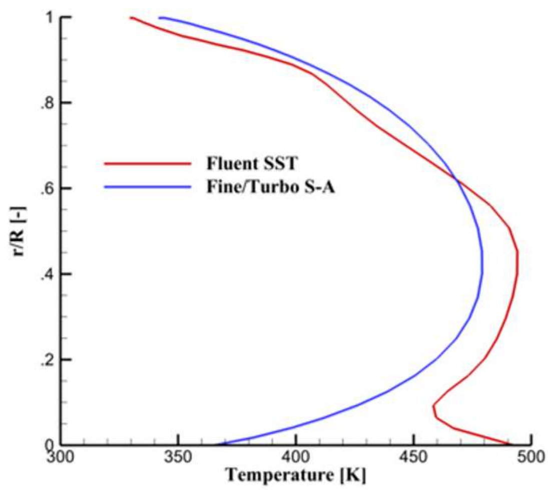

Figure 7: Circumferentially averaged total temperature upstream of NGV.

4 Hot spot location

The two objectives of the presented simulations can be distinquished. The first one, the prediction of the hot spot location upstream of NGV and the second one, investigation if the NGV leading edge has any potential effect on such location and if the hot spot migrates in pitchwise direction. Below, the two final solutions are shown which are taken into account for the test section definition (Fig. 8). There are total temperature isosurfaces for both cases. Hot flow comes from swirler duct domain, then it interacts with low velocity zone in combustor simulator. The first case (NGV1) is shown in left figure, the centre of hot flow is located upstream of leading edge. The location of the hot flow centre and zone with maximum total temperature normalized by total temperature at the swirler inlet is presented in Fig. 9. The arrow indicate of the swirl centre and hot spot location. On the right side of Figs. 8-9, hot spot is shown upstream of blade passage in the middle. The plane is located just upstream of the NGV, one can notice some differences in total temperature distribution. On the right side for NGV2 case, lower temperature zone at outer radius zone is lightly wider and lower values are obtained. Close to the lower radius, higher values of total temperature are spread wider in circumferential direction. Such temperature distribution is governed by vorticity generated by the swirler and then the interaction with secondary air flow delivered by inner and outer liner surfaces.

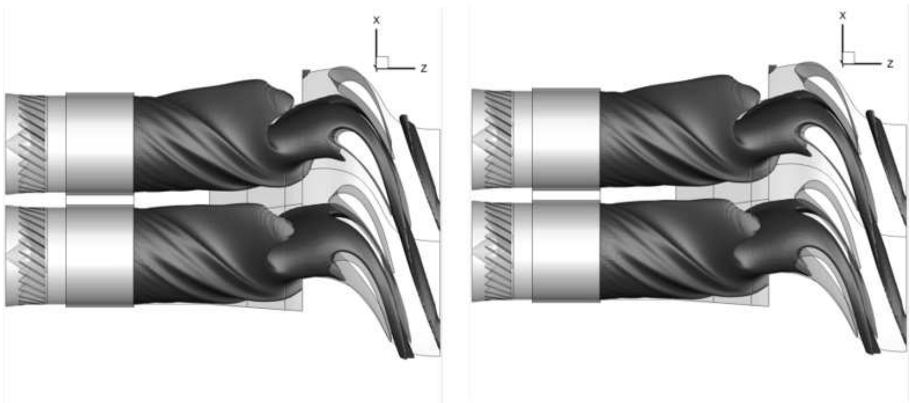

Figure 8: Isosurface of total temperature downstream of swirler duct: NGV1 (left) and NGV2 (right).

Swirl generated by the swirler can be visualised by an axial component of vorticity vector. In Fig. 10, vorticity at plane upstream of NGV is shown. Axial vorticity generated by swirler is influenced by the NGV location. For NGV2 case, the vortex is located in the middle of blade passage and it is weakly influenced by NGV leading edge. Contrary to the NGV2 case is NGV1, where vorticity is highly affected by leading edge and secondary vortices are created in this zone.

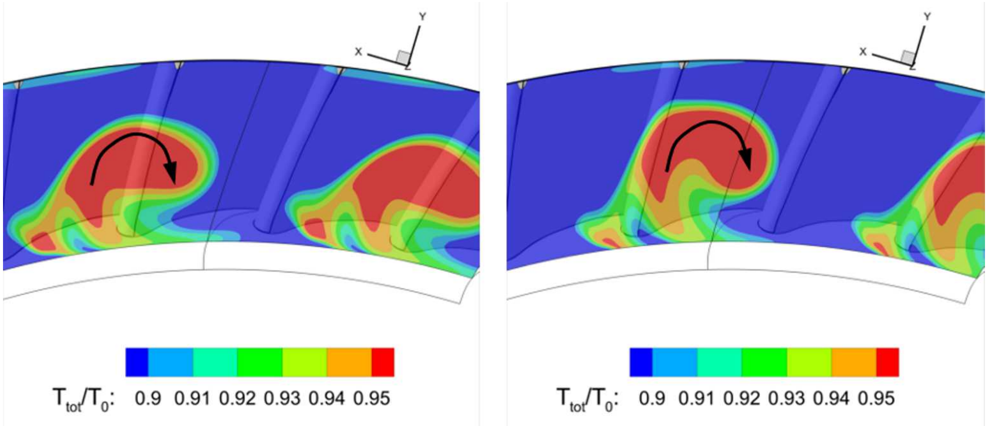

Figure 9: Total temperature upstream of NGV – NGV1 (left) and NGV2 (right).

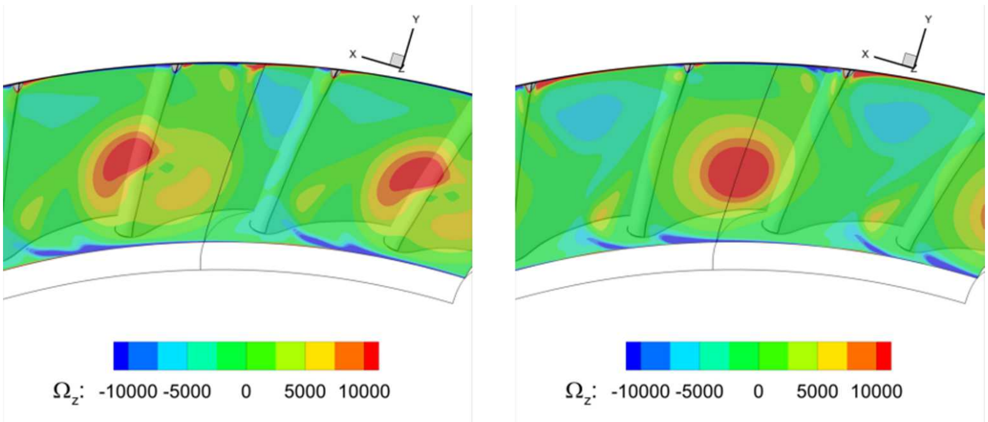

Figure 10: Axial vorticity component, Ω_z, close to NGV leading edge: NGV1 (left) and NGV2 (right).

Further downstream of leading edge, total temperature distribution highly differs to each other. Total temperature distribution in neighbouring blade passages are shown in Fig. 11. In case of NGV1 (figure on the left), hot spot migrates toward suction side and hot zone is seen in the lower part of the passage, mainly. The second case (NGV2) indicates evolution of the hot spot in middle of blade passage. It is slightly moved toward suction side, but the maximum total temperature is located further from the blade suction side.

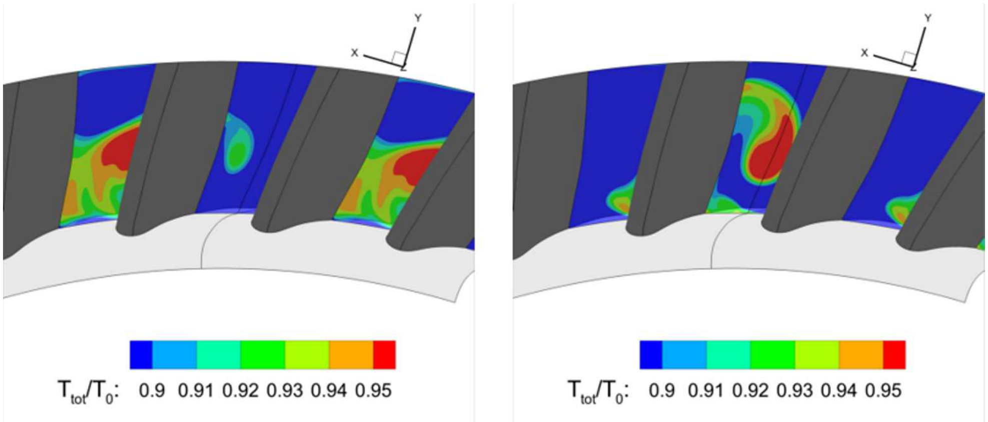

Figure 11: Total temperature downstream of NGV leading edge: NGV1 (left) and NGV2 (right).

Evolution of the hot spot and its location is very important for the rotor blades. Presented results do not include upstream effect of the rotor. Anyway, the obtained results indicate of total temperature distribution difference downstream of NGV depending on the inlet conditions. In case of NGV1, hot zone is located near the hub and the second zone with increased temperature is located at middle height of the passage. In case of NGV2, there are two hot zones. The first one is near the hub, but further than it is observed in case of NGV1. The second one is moved to the blade tip zone.

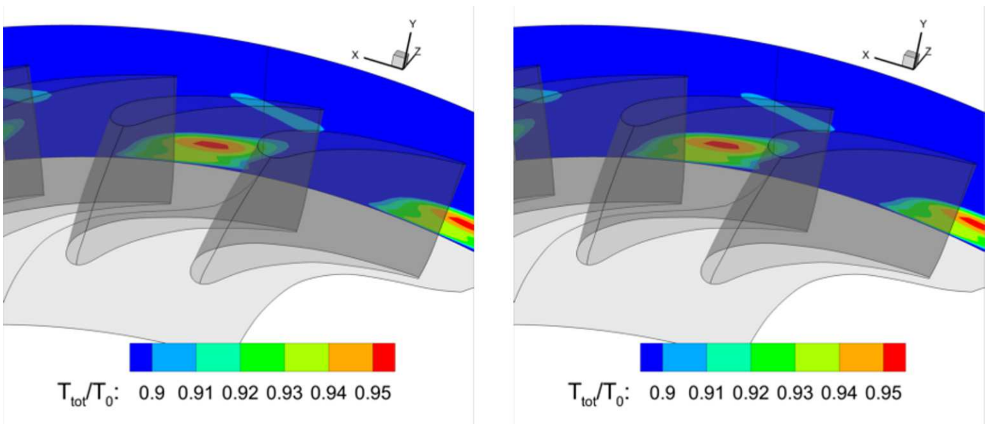

Figure 12: Total temperature downstream of NGV: NGV1 (left) and NGV2 (right).

The simulations have been carried out for two configurations: with or without film cooling on NGV. It was combined with swirler/NGV relative location and finally, the effect on the neighbouring nozzle guide vanes are presented. In Figs. 13–16 the wall temperature in NGV domain is presented. One has to emphasize that presented results are obtained as a pretest simulation and no comparison with experimental is available at this stage of the project. Anyway, the qualitative effect of hot spot location can be presented.

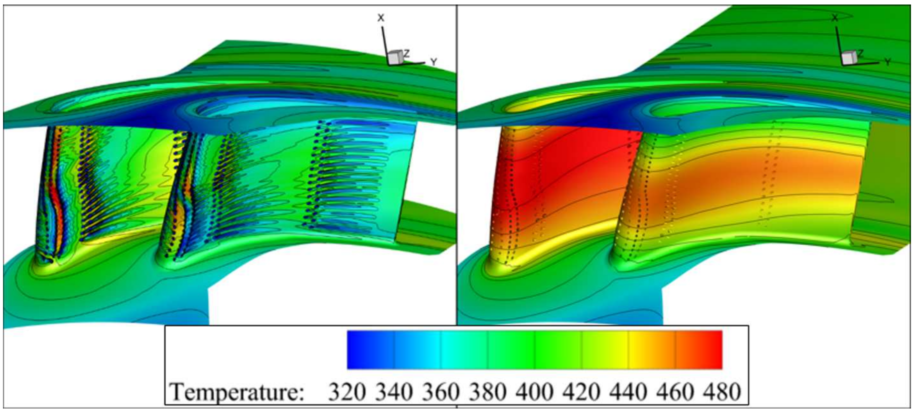

Figure 13: Wall temperature for NGV1 and Fine/Turbo Numeca (Spalart-Allmaras): cooling on (left) and cooling off (right).

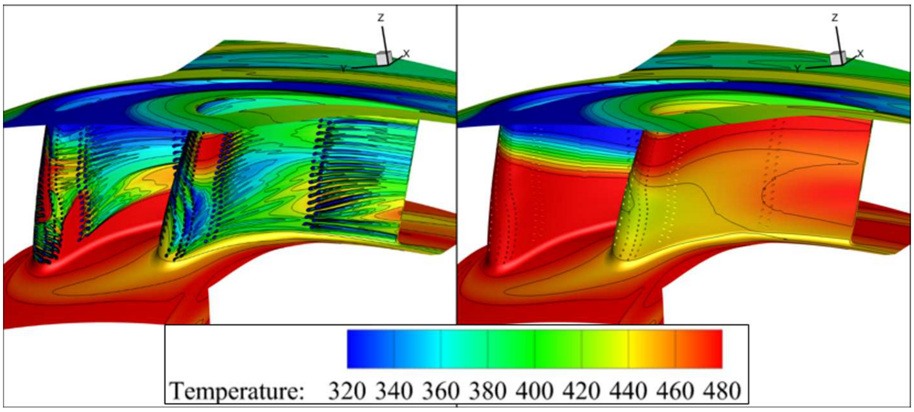

Figure 14: Wall temperature for NGV1 and Fluent (k-w SST): cooling on (left) and cooling off (right).

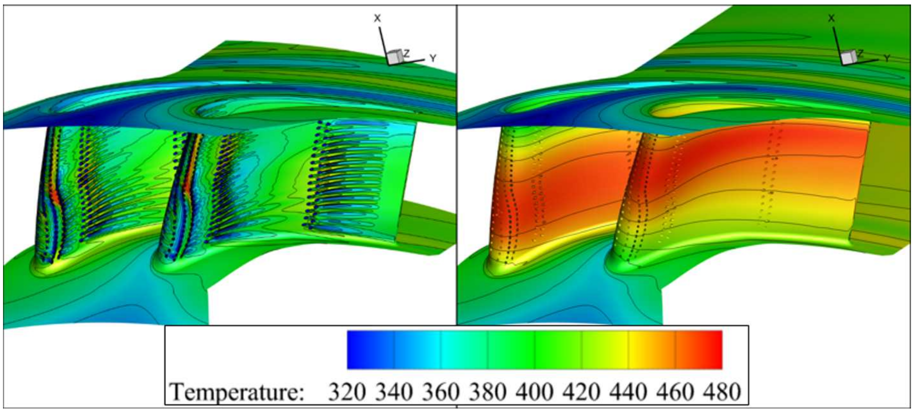

Figure 15: Wall temperature for NGV2 and Fine/Turbo Numeca (Spalart-Allmaras): cooling on (left) and cooling off (right).

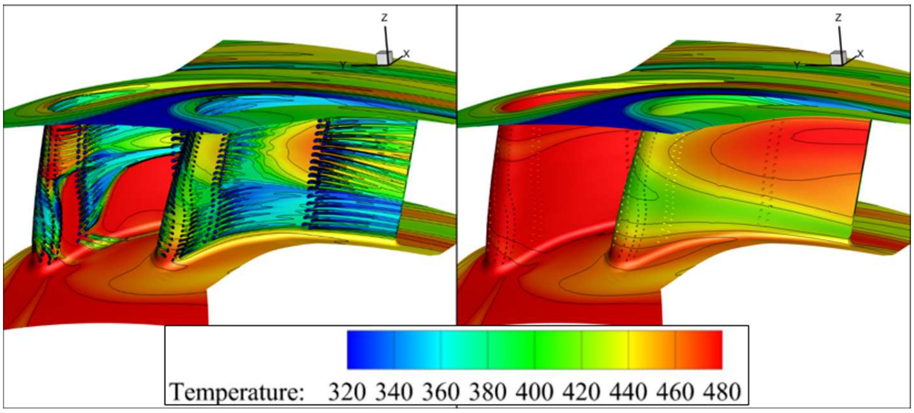

Figure 16: Wall temperature for NGV2 and Fluent (k-w SST): cooling on (left) and cooling off (right).

Each set of plots represents results for NGV with film cooling and without. One can notice significant difference in temperature distribution and effect of film cooling. Downstream of cooling holes, the traces of coolant, as low temperature zones, are visible. Another important effect, directly arising from the hot spot location, is the high difference of temperature distribution on the neighbouring NGVs. Larger zone of higher temperature is observed on the vane located downstream of the hot spot. In both cases, independently on the hot spot location wall temperature distribution is highly nonuniform. Although, obtained results are dependent on the turbulence model and no conjugate heat transfer approach is applied, the locally increased temperature exists which impact on the thermal stresses and durability of the gas turbine can be significant.

5 Summary

Numerical simulations results obtained within a framework of FACTOR project (Full Aerothermal Combustor-Turbine Interactions Research) are presented. Shown results are considered as the pre-test, because no experimental data are available, so no comparisons with the measurements can be done. Anyway, the investigations of a flow structure and temperature or velocity distribution downstream of combustor simulator and its effect on turbine inlet are very important for the final test section configuration. Presented results for the two limiting cases of hot spot location indicate high differences in the hot spot evolution in the blade passage and its effect on the wall temperature distribution. Finally, the hot spot effect on the rotor blades has to be considered if swirler location leads to hot spot location in the middle of the NGV passage.

Acknowledgments Pre-test numerical simulations results have been obtained by IMP PAN within a framework of FACTOR project (*Full Aerothermal Combustor-Turbine Interactions Research*) are presented in the paper. The research leading to these results has received funding from the European Union 7th Framework Programme (FP7/2007-2013) under grant agreement No. 265985. Numerical simulations were carried out in Computational Centre of TASK (Trójmiejska Akademicka Sieć Komputerowa) and supported by PL-Grid Infrastructure.

Received in 15 August 2017

References

[1] Barringer M.D., Thole K.A., Polanka M.D.: *Developing a combustor simulator for investigating high pressure turbine aerodynamics and heat transfer.* Proc. ASME TurboExpo 2004: Power for Land, Sea and Air, June 14-17, 2004, Vienna, GT2004-53613.

[2] Barringer M.D., Thole K.A., Polanka M.D.: *Experimental evaluation of an inlet profile generator for high pressure turbine tests.* Proc. ASME TurboExpo 2006: Power for Land, Sea and Air, Barcelona, May 8–11, 2006, GT2006-90401.

[3] Durney D.J., Gundy-Burlet K.L., Sondak D.L.: *Survey of hot streak experiments and simulations.* Int. J. Turbo Jet Engines **16**(1999),1, 1–15.

[4] Jenkins S., Varadarajan K., Bogard D.G.: *The effects of high mainstream turbulence and turbine vane film cooling on the dispersion of a simulated hot streak.* ASME J. Turbomachinery, **126**(2004),1, 203–211.

[5] Menter F.R.: *Two-equation eddy-viscosity turbulence models for engineering applications.* AIAA J. **32**(1994),8, 1598–1605.

[6] Polanka M.D., Anthony R.J., Bogard D.G., Reeder M.: *Determination of cooling parameters for a high speed, true scale, metallic turbine vane ring.* In: Proc. ASME TurboExpo 2008: Power for Land, Sea and Air, Berlin, June 9–13, 2008, GT2008-50281.

[7] Shyy W., Braaten M.E., Burrus D.L.: *Study of three-dimensional gas-turbine combustor flows.* Int. J. Heat Mass Trans. **32**(1989), 6, 1155–1164.

[8] Stabe R.G., Whitney W.J., and Moffitt T.P.: *Performance of a high-work low aspect ratio turbine tested with a realistic inlet radial temperature profile.* Techn. Rep., NASA TM 83655, 1984. AIAA Paper 84-1161.

[9] Shang T., Guenette G.R., Epstein A.H., Saxer P.: *The influence of inlet tmperature distortions on rotor heat transfer in a transonic turbine.* In: Proc. 31st Joint Propulsion Conf. and Exhibit, San Diego, 1995, AIAA 95-3042.

[10] Jameson A., Schmidt W., Turkel: *Numerical solutions of the Euler equations by finite volume methods using Runge-Kutta time-stepping schemes.* Palo Alto 1981, AIAA Paper 81-1259.

[11] Numeca IGG/Turbo v10 2016 User Manual Documentation.

[12] Numeca FINE/Turbo v10 2016 Theoretical Manual Documentation.

[13] Ansys FLUENT v16 2015 Theoretical Manual Documentation.