Effect of Hot Spot Location on Flow Structure in Nozzle Guide Vane

Michal Piotrowicz, Pawel Flaszynski, Piotr Doerffer
Institute of Fluid-Flow Machinery, Polish Academy of Sciences,
Fiszera 14, 80-231 Gdansk, Poland
E-mail: michal.piotrowicz@imp.gda.pl

Abstract. Interaction of a combustor and turbine in aircraft engines generates still a lot of open questions. Investigation of combustor and turbine separately have been done for a years by research institutes and aircraft engine companies, but there is lack of knowledge about the interaction effect. In the paper, numerical simulations results for combustor simulator and nozzle guide vane (NGV) of the first turbine stage are presented. No combustion is modelled, but the hot flow at the swirler inlet is applied. Geometry and flow conditions are defined according to the turbine stage designed for experimental investigations in FACTOR (Full Aerothermal Combustor-Turbine Interactions Research) project within a framework of European Commission 7th Framework Programme. The main objective of the presented investigations is an analysis of the location of the high temperature zone upstream of the nozzle guide vane and its effect on the wall temperature. Numerical simulations are carried out by means of Fine/Turbo Numeca with two turbulence models: Spalart-Allmaras and Explicit Algebraic Reynolds Stress Model. As a model validation, a comparison of total pressure, total temperature and flow angles downstream of combustor simulator with experimental data from University of Florence is also presented.

1. Introduction
In order to reduce fuel consumption and improve efficiency, modern gas turbines operate at high speeds and high temperatures. The high uncertainty in predicting the interaction between the combustor and the turbine leads to the application of increased safety margins in the design process. Therefore, a better understanding of interaction of the combustor – turbine is required to maintain service life while optimizing high-pressure turbines (HPT) together with the lean burn combustion chamber.

Currently, quite often the design system is divided into two steps. As the first step, combustion chamber is designed [6,8], resulting in a temperature distribution at the outlet. Then, the turbine is designed according to boundary conditions defined at the combustor – turbine interface [7,10]. A disadvantage of such approach is very weak interaction between the combustor and the turbine, what can be highly important for a durability and performance of the first stage of gas turbine.

There is great experience developed over the years by the aircraft engines companies and different test facilities are available to investigate combustor and turbine separately, but there was no test rig allowing for the interaction analysis. Such gap in the knowledge drove the idea to build a test stand giving opportunity for experimental and numerical investigations of combustor-turbine interaction. The two test sections were designed and built within a framework of the FACTOR (Full Aerothermal
Combustor-Turbine Interactions Research) project: at the German Aerospace Center (Deutsches Zentrum für Luft- und Raumfahrt; DLR) in Goettingen and at the University of Florence. DLR test rig is large facility including combustor simulator and turbine stage, while at the University of Florence a tri sector rig (3 swirlers and 6 nozzle guide vanes downstream) is available.

Last years a few investigations focused on a similar problem are carried out [5,11]. Results presented by authors are related to the interaction between the combustor and nozzle guide vane (NGV), but no effect of hot spot circumferential location was investigated. Below, numerical simulations results for combustor simulator and NGV for the two circumferential locations and the effect of hot spot (high temperature zone) location on NGV wall temperature is presented. As the first step, a comparison of numerical results with experimental data from University of Florence for combustor without NGV is shown.

2. Geometry

Computational geometry (figure 1) is defined according to the design calculations [2] and final test section configuration.

Validation of the calculation model shown in figure 1 (left) is based on a comparison with measurements [3] carried out by University of Florence (UNIFI).

Further analysis of the influence of the nozzle guide vane (NGV) relative position to the swirler on the flow structure is investigated by means of model shown in figure 1 right.

![Figure 1. Computational domain left side only combustor, right combustor simulator + NGV](image)

Figure 2 shows two configurations that differ from each other in the position of the NGV relative to the swirler. In the LEC case, the leading edge of NGV 1 is fixed right downstream of the swirler. In the second configuration (PAC) the blade passage is located downstream of the swirler.

![Figure 2. Two relative positions NGV downstream of swirler: LEC (left) and PAC (right).](image)
3. Numerical model description
Numerical simulations were carried out for the flow domain shown in figure 1 by means of Fine™/Turbo NUMECA. The flow solver is a structured, three-dimensional density-based, multi-block Reynolds Averaged Navier Stokes code based on a finite volume method. Presented results are obtained for Spalart-Allmaras (SA) and Explicit Algebraic Reynolds Stress Model (EARSM) turbulence models [12, 13]). For spatial discretization (central, 2nd order) with Jameson type artificial dissipation is applied, while for the temporal discretization a four-stage explicit Runge-Kutta scheme is employed. In order to increase the convergence rate the local time-stepping, implicit residual averaging and full multi-grid techniques are used in the solver.

The mesh has been generated with IGG™ and AutoGrid5™. The mesh size varies from 7.75∙10^6 grid cells for the combustor chamber simulator only, up to 33.7∙10^6 grid cells for the model including cooled nozzle guide vanes. Full non matching boundary connection (FNMB) is applied at interfaces: inlet duct-swirler, swirler-combustor simulator, combustor simulator – NGV. The mesh is refined close to the solid walls in order to keep y⁺ below 1. An example of a mesh for a swirler, combustor and NGV is shown in figure 3.

The unsteady simulations were performed with a time step 5e-05s. The simulation time corresponding to 10 full flow-through the domain is used for the time statistics.

At the main inlet the, mass flow rate conditions is set to ~0.155 kg/s, total temperature is 512K, the turbulence intensity 1% with turbulent/laminar viscosity ratio 10. Cooling of the combustor simulator has been simplified from the flow through the perforated plate to the surface on which the parameters are distributed uniformly. Detailed information are shown in table 1.

For configurations with NGV, additional cooling needs to be considered where mass flow rate for each plenum channel is: cooling 1 = 0.009 kg / s and cooling 2 = 0.0045 kg / s. The temperature, turbulence intensity and viscosity ratio are the same as shown in table 1.

4. Model validation - comparison with measurements
The validation of the numerical model was carried out by means of comparison of experimental data in the plane P40. At this location, the interface between combustor simulator (CS) and nozzle guide vane (NGV) is defined.

Table 1. Operating conditions at combustion chamber

| mass flow (main inlet (%)) | static temperature [K] | angle/wall [deg] | turbulence intensity (%) | viscosity ratio(-) |
|----------------------------|------------------------|------------------|-------------------------|------------------|
| 15                         | 297.65                 | 60               | 1                       | 10               |
| 6                          |                         | 30               |                         |                  |
| 19                         |                         | 60               | 1                       | 10               |
| 12                         |                         | 30               |                         |                  |

For configurations with NGV, additional cooling needs to be considered where mass flow rate for each plenum channel is: cooling 1 = 0.009 kg / s and cooling 2 = 0.0045 kg / s. The temperature, turbulence intensity and viscosity ratio are the same as shown in table 1.
The normalized distribution of total pressure normalised by the mean total pressure is shown in plane 40 (figure 4). Low pressure area can be observed in the middle of the control plane what arises from the wake generated downstream of the swirler. Moving away from the core of the vortex generated by swirler, pressure is increasing reaching maximum closer to the endwalls and then it decreases. The distribution is highly non-uniform and it is driven by the interaction of the swirling flow in combustor middle zone and cooling flow ejected at the upper/lower endwalls.

The EARSM turbulence model better predicts the level of losses both in the vortex generated by swirler and outside. Additionally, it correctly predicts the minimum value as well as a circumferentially averaged results as presented in figure 4 (right). In the case of SA turbulence model, the distribution of these parameters is more uniform which may result from less accurate prediction of swirling flow.

![Figure 4. Normalized pressure distribution at P40 with circumferential averaging](image)

The temperature distribution shown in figure 5 has been defined according to the following equation:

$$T_{nd} = \frac{T - \bar{T}}{\bar{T} - T_{cool}}$$

where $T$ - total temperature, $\bar{T}$ - averaged total temperature and $T_{cool}$ – total temperature of coolant.

The EXP results indicate that this geometry allows to transport “hot-spot” directly to the analyzed plane. It is important effect which needs to be modelled as no combustion is considered and temperature is much lower than in the real combustor. There are two zones where the maximum can be seen (figure 5). The high temperature in the center is related to the hot flow downstream of the swirler, while the zones at the sides/outline result from the presence of neighbouring swirler. It should be remembered that the results for three sectors limited by side walls are presented for EXP, as mentioned in the article describing the experiment [3].

Likewise the EARSM model seems to be closer to measurements and again it is noticeable in the centre of the plane. Additionally this model correctly predicts the areas with local minimum. This may suggest that intensity of mixing inside the combustor simulator is closer to the existing in the test section. Much larger discrepancies are shown by the SA model. It can be observed that the hot air is transported further than in experimental data and its maximum is closer to the hub. It can also be clearly seen that the distribution of the temperature in close proximity to the hub and shroud is much lower than measured. This is confirmed by the circumferential average shown in plot of figure 5.

![Figure 5. Non dimensional temperature distribution at P40 with circular averaging](image)
Swirl angle distribution (figure 6) indicates that both turbulence models correctly predicts distribution in the middle of the channel, the differences starts appearing in the zones close hub and shroud. These differences may result from the simplifications of boundary conditions at perforated plates in CS, what is mentioned in section 3.

![Swirl angle distribution at P40 with circular averaging](image)

Figure 6. Swirl angle distribution at P40 with circular averaging

Finally, the pitch angle distribution confirms earlier observations that the EARSM model seems to give results closer to experiment. This is indicated by the deviation of the flow figure 7. The areas with the positive (red) and negative (blue) components are compared in size and value in relation to EXP. These differences are not so noticeable when they are averaged circumferentially (plot in figure 7).

![Pitch angle distribution at P40 with circumferential-averaging](image)

Figure 7. Pitch angle distribution at P40 with circumferential-averaging

The complex geometry of combustor simulator creates a strongly three-dimensional flow structure. Applied simplifications of the existing test section to only one sector (periodic conditions) and replacing perforated plates for CS cooling flow with uniform mass flow distribution does not cause significant discrepancies, as proven above. Satisfactory compliance of EARSM model with the experimental data allows the use this model for analysis of NGV position influence on flow structure in cascade.

5. **Hot spot location effect on Nozzle Guide Vane**

Results presented above are obtained for combustor simulator without NGV. In such situation a question arises if there is potential effect of NGV located downstream of plane 40 and if this affects flow conditions in plane 40. As shown in figure 8, there is insignificant effect. It should be emphasized that the control plane is very close to NGV, approximately 50% of NGV chord upstream.

![Normalized pressure distribution at P40](image)

Figure 8. Normalized pressure distribution at P40
Comparing the non-dimensional temperature distribution in the plane P40 (figure 9), more differences can be seen, nevertheless the main flow features are the same. The similar shape of the high temperature areas is the result of swirl generated by the swirler. The vortex direction is marked with a dashed line in figure 9.

The presence of the blades causes the vortex to be stretched and transported more intensively towards the hub.

![Figure 9. Non-dimensional temperature distribution at P40](image)

The evolution of hot air zone, visible in the plane at leading edge location (figure 10), indicates that the flow core in the case of LEC is separated by the leading edge of NGV 1 into two structures transported at both sides of one NGV. A significant part of the hot stream migrates towards the suction side of the NGV 1. In the PAC configuration the hot stream is in the middle of the blade passage and also affects both NGVs.

![Figure 10. Non dimensional temperature distribution at LE plane](image)

On the figure 11, it can be observed that despite the presence of NGV in LEC configuration, the major part of hot stream is transported within only one blade-passage. Additionally, in this case the hot spot affects the turbine rotor in the zone of gap and shroud location. This may cause faster wear of turbine blades as a consequence of thermal loading.

In the PAC configuration, the temperature at the NGV outlet is lower so the temperature distribution becomes more uniform. It can also be noted that the hot stream zone within the hub is smaller, which may be important for the turbine durability.

The iso-surface $T_{nd}=0.01$ is shown below (figure 12), where one can observed the hot zone penetration in the blade-passage.

![Figure 11. Non-dimensional temperature distribution at TE plane](image)

![Figure 12. Iso surface of non-dimensional temperature $T_{nd}=0.01$](image)
The change in the flow structure, forced by the NGV location, directly affects the wall temperature on NGV. In the LEC case, the temperature distribution on the pressure side of both NGVs is very similar (figure 13 and figure 14). Larger differences appear when the area near the NGV 2 hub is compared (figure 13). In case of PAC position, the wall temperature on NGV2 is much higher than on NGV 1. As in the previous configuration (LEC), it can also be noticed that higher temperature zone is located close to NGV 2 (PAC) hub region.

Comparison of the temperature distribution on the suction side (figure 15) indicates that on NGV 1 for the LEC configuration, located downstream of the swirler, temperature is higher than for the PAC configuration. On the suction side of NGV 2 differences are lower, but slightly higher temperature is for PAC case (figure 16).

6. Conclusions
Comparison of numerical simulations results against experimental data in plane upstream of nozzle guide vane allows to validate the model applied to investigation of flow structure for selected configurations. Both turbulence models (Spalart-Allmaras and Explicit Algebraic Reynolds Stress Model) show good prediction of the total pressure and flow angles. Some differences in the predicted and measured temperature are shown, but in both cases the hot spot existence is clearly visible. Nevertheless, better prediction of the total temperature upstream of the nozzle guide vane is obtained by EARSM model, so this one was selected for flow structure investigations and analysis of wall temperature in turbine cascade.

Circumferential position of guide vanes downstream of the swirlers strongly affects the wall temperature and flow conditions upstream of rotor cascade. Effect of the hot spot location upstream the leading edge or upstream the passage is presented on locally higher temperature on both sides of the neighbouring vanes. It influences on thermal loading of turbine stator, but has also strong impact on temperature level and flow non-uniformity at turbine rotor inlet. Future investigations will be focused on unsteady simulations for the investigated configurations including rotor cascade.
Acknowledgements.
The presented analysis are carried within a framework of EU FP7 FACTOR project and authors are particularly grateful to Tommaso Bacci from University of Florence for the sharing of experimental results. Numerical simulations were carried out in Computational Centre of TASK (Trojmiejska Akademicka Siec Komputerowa) and supported by PL-Grid Infrastructure.

References
[1] Barringer M, Thole K and Polanka, M 2004 Developing a Combustor Simulator for Investigating High Pressure Turbine Aerodynamics and Heat Transfer, Proceedings of ASME Turbo Expo 2004: Power for Land, Sea, and Air, June 14-17, 2004 Vienna, Austria, GT2004-53613
[2] Flaszynski P, Piotrowicz M 2017 An investigation of hot spot location at turbine inlet – numerical simulations for factor project test rig, Transactions of the Institute of Fluid-Flow Machinery, No. 135, 2017, 101–115
[3] Andreini A, Bacci T, Insinna M, Mazzei L and Salvadori S 2016 Hybrid RANS-LES Modeling of the Aero-Thermal Field in an Annular Hot Streak Generator for the Study of Combustor-Turbine Interaction, Proceedings of ASME Turbo Expo 2016: Turbine Technical Conference And Exposition, June 13-17, Seoul, South Korea, GT2016-56583
[4] Tsao J, and Lin C 1999 Reynolds stress modelling of jet and swirl interaction inside a gas turbine combustor, International Journal for Numerical Methods in Fluids, vol. 29, 1999, pp. 451–464.
[5] Klapdor E 2010 Simulation of Combustor - Turbine Interaction in a Jet Engine, PhD thesis, TU Darmstadt
[6] Qureshi I and Povey T 2011 A combustor-representative swirl simulator for a transonic turbine research facility, Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, vol. 225, 2011, pp. 737–748.
[7] Rahim A, Khanal B, Romero E and He L 2015 Effect of NGV Lean Under Influence of Inlet Temperature Traverse, Journal of Turbomachinery, vol. 136, 2013, pp. 1–12.
[8] Cresci I, Ireland P, Bacic M, Tibbott I and Rawlinson A 2015 Realistic Velocity and Turbulence Intensity Profiles At the Combustor-Turbine Interaction ( Cti ) Plane in a Nozzle Guide Vane Test Facility, European Turbomachinery Conference, 2015, pp. 1–11.
[9] Koupper C, Bonneau G, Gicquel L and Duchaine F 2016 Large eddy simulations of the combustor turbine interface: study of the potential and clocking effects, Proceedings of ASME Turbo Expo 2016: Turbine Technical Conference And Exposition, June 13-17, 2016, Seoul, South Korea, GT2016-56443
[10] Barigozzi G, Mosconi S, Perdichizzi A and Ravelli S 2017 The Effect of Hot Streaks on a High Pressure Turbine Vane Cascade with Showerhead Film Cooling, International Journal of Turbomachinery, Propulsion and Power, vol. 2, 2017, p. 15.
[11] Legrenzi P A 2017 Coupled CFD Approach for Combustor-Turbine Interaction, PhD thesis, Loughborough University
[12] Spalart P and Allmaras S 1992 A One-Equation Turbulence Model for Aerodynamic Flows, Proc. of 30th Aerospace Sciences Meeting and Exhibit
[13] Menter F, Garbaruk A and Egorov Y 2012 Explicit algebraic reynolds stress models for anisotropic wall bounded flows, Progress in Flight Physics 3 (2012) 89-104