TASK QUARTERLY vol. 19, No 2, 2015, pp. 141–152

INFLUENCE OF LIMITING WALLS ON SHOCK WAVE STRUCTURE IN SINGLE PASSAGE TEST SECTION WITH COMPRESSOR PROFILE

MICHAŁ PIOTROWICZ[1], PAWEŁ FLASZYŃSKI[1]
AND RYSZARD SZWABA[2]

[1] *Aerodynamics Department*
The Szewalski Institute of Fluid-Flow Machinery (IMP PAN)
Fiszera 12, 80-231 Gdansk, Poland

[2] *Experimental Aerodynamics Department*
The Szewalski Institute of Fluid-Flow Machinery (IMP PAN)
Fiszera 12, 80-231 Gdansk, Poland

(received: 21 January 2015; revised: 23 February 2015;
accepted: 25 February 2015; published online: 25 March 2015)

Abstract: The shock wave boundary layer interaction on the suction side of a transonic compressor blade is one of the main objectives of the TFAST project (Transition Location Effect on Shock Wave Boundary Layer Interaction). In order to look more closely into the flow structure on the suction side of the blade, a design of a turbine passage model in a rectilinear transonic wind tunnel was proposed. The model which could reproduce the flow structure, the shock wave location, the pressure distribution and the boundary layer development similar to the obtained in a reference cascade profile is the main objective of the design presented here. The design of the proposed test section is very challenging, because of the existence of a shock wave, its interaction with the boundary layer and its influence on the 3-D flow structure in the test section. The paper presents the influence of the test section geometry configuration on the flow structure as an effect of the shock wave boundary layer interaction.

Keywords: shock wave, laminar-turbulent transition, transonic compressor

1. Introduction

The shock wave boundary layer interaction on the suction side of a transonic compressor blade is one of the main objectives of the TFAST project (Transition Location Effect on Shock Wave Boundary Layer Interaction). The test section was designed and assembled in the IMP PAN laboratory in order to reconstruct the flow structure existing in the real transonic reference cascade.

The paper presents the influence of the test section geometry configuration on the flow structure as the effect of the shock wave boundary layer interaction. The shock wave location and secondary flows depend on the geometry of the side walls and the downstream boundary conditions. One of the test section design criteria is the inlet The Mach number and the inflow uniformity which should be adequate to the real conditions. In order to maintain the required inflow conditions the nozzle designed for the UFAST Project [1, 2] is used. The results of CFD calculations lead to identification and analysis of complex flow phenomena and show that the limiting side walls have great influence on the secondary flow behaviour, especially on the corner vortex and separation. The suction slots are applied to diminish the corner separation in the zone near the side wall and the profile.

2. Geometry

A sketch of the test section is shown in Figure 1. The upper and lower compressor profile is located in the test section downstream of the convergent-divergent nozzle. The nozzle is designed to maintain the uniform distribution of the Mach number upstream the profiles. It is a very important feature because it allows a translation of both profiles without changing the inlet conditions. The nozzle was designed for the UFAST project (Unsteady Effects of Shock Wave Induced Separation) [1, 2].

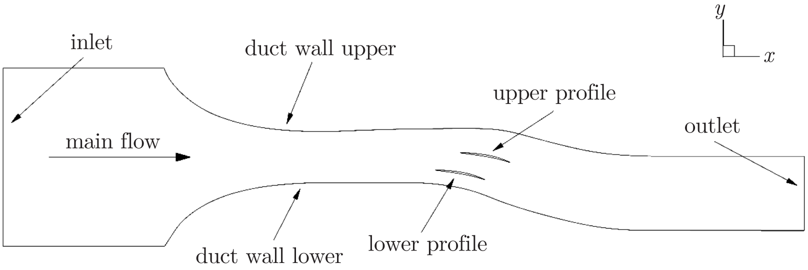

Figure 1. Geometry test section

The transonic reference compressor cascade was designed by Rolls-Royce Deutschland. The required parameters from the cascade profile are shown in Table 1.

Table 1. TFAST Compressor Cascade Profile

name	unit	value
Inlet Mach	—	1.22
Real Chord	mm	100
Pitch to Chord Ratio	—	0.6
Thickness to Chord Ratio	—	0.03
Blade Inlet Angle	deg	50.9
Blade Exit Angle	deg	33.2
Flow Inlet Angle	deg	55.5

The reference conditions [3] for the test section design are obtained from the numerical simulations for the cascade configuration. The numerical simulations were performed with a FINE/Turbo Numeca solver using the Explicit Algebraic Reynolds Stress Model (EASRM) of turbulence. The Mach number in the cascade and isentropic Mach number distribution on the suction side of a profile is shown in Figures 2–3.

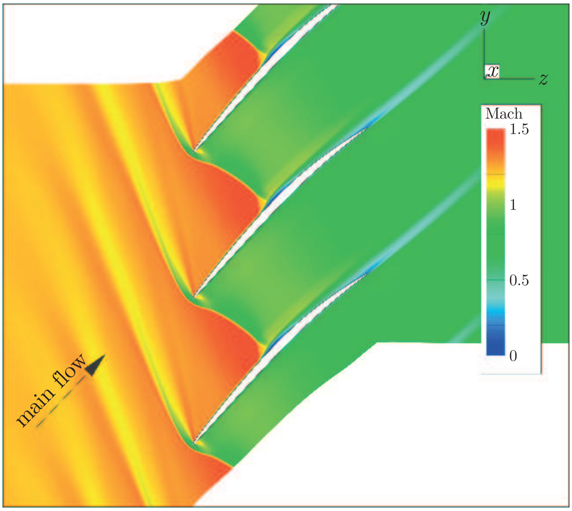

Figure 2. Mach number at mid span

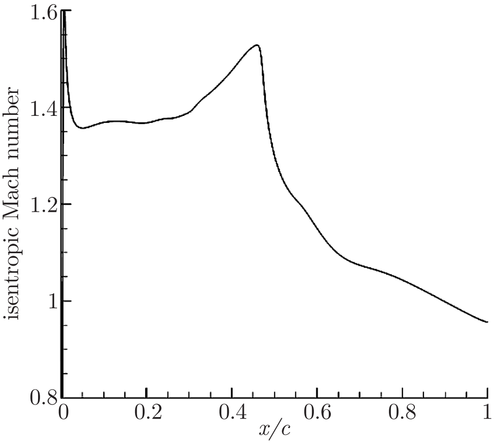

Figure 3. Isentropic Mach number

The numerical results presented below are obtained for the configurations shown in Figures 4–5. In the first case, the two profiles are located in the test section. The second configuration differs from the first one with the installed supports (red) and suction slots. The props allow mounting the profiles in the test section. The profiles cannot be fastened to the sidewalls because the window for optical measurements is mounted at the same height.

Additionally, the suction slots located at the upper and lower walls of the test section (slot 1 and slot 3) and suction slots (slots 2) at the upper side (suction) of the lower profile. The adjustment of the static pressure at the suction slots allows controlling the mass flow and influencing the flow structure in the test section.

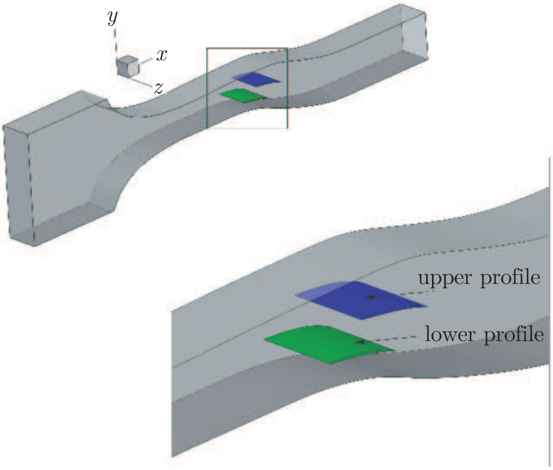

Figure 4. Basic geometry

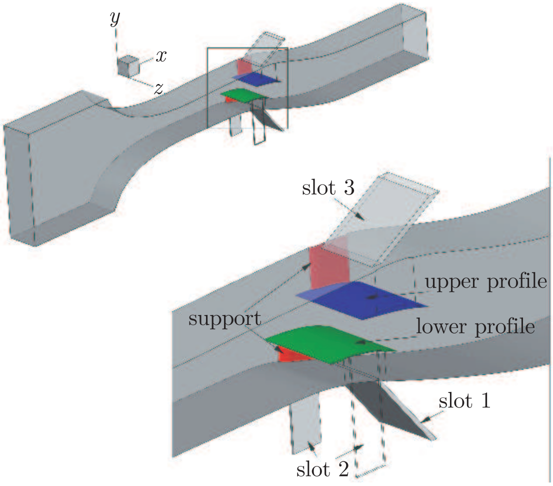

Figure 5. Geometry with additional supports and slots

3. Numerical model description

The numerical simulations were carried out with the FINE/Turbo Numeca code. The two-equation nonlinear eddy viscosity turbulence model of the Explicit Algebraic Reynolds Stress Model (EASRM) was applied [4]. The set of equations is closed by a perfect gas equation and Sutherland's law for viscosity. Spatial discretization using a 2$^{\text{nd}}$ order central difference scheme with scalar artificial dissipation formulated by Jameson, Schmidt and Turkel (1981) was applied. The structured mesh was generated by means of IGG Numeca (Interactive Geometry Modeler and Multi-Block Structured Grid Generator) [5]. Simulations were carried out for the three cases:

- Model 1: profiles mounted in the test section without supports – basic geometry (Figure 6).
- Model 2: profiles mounted in the test section with a support near sidewalls (Figure 7).
- Model 3: the model with suctions slots 1, 2 and 3 (Figure 5).

The multi-block topology for geometry (cases 1 and 2) consists of 31 blocks and the total number of cells is $13.5 \cdot 10^6$. The third case, where the slots (slots 1, 2

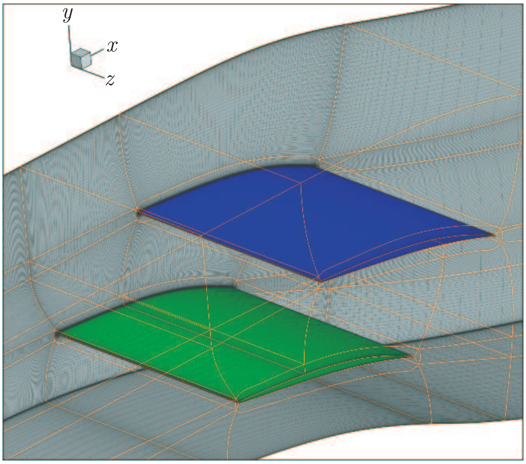

Figure 6. 3D mesh for the basic model

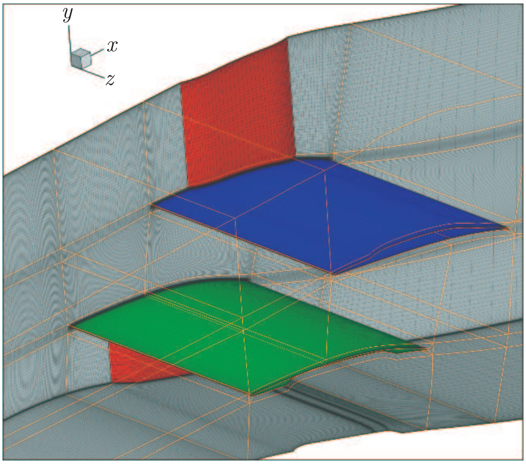

Figure 7. 3D mesh for geometry with

and 3) are taken into account, consists of 35 blocks and the mesh size is increased to $14\cdot10^6$ cells. The grid is refined close to the wall in order to obtain the first grid point near the wall related to $y^+=1$. An example of the mesh is shown in Figures 6–7.

Full non matching boundary (FNMB) [6] connections between blocks with different node distribution were applied. The full non matching connection allows keeping the mass flow conservation, the momentum and energy through the interface between two or more blocks. An example of a mesh connection is shown in Figure 8. The red mesh represents the bottom wall and the blue one shows the connection to the additional slot.

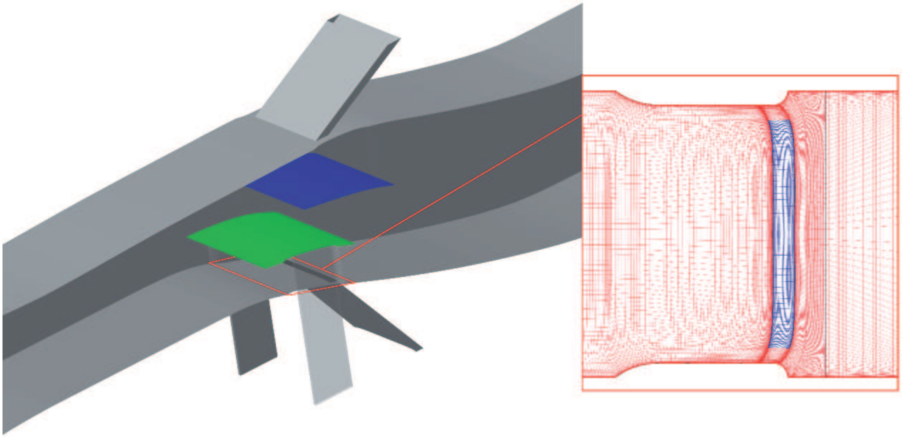

Figure 8. Full non matching boundary between bottom wall of test section and slot 1

The total pressure of 101 000 Pa and the temperature of 293 K are set at the inlet. The turbulence kinetic energy and dissipation rate were set assuming that the viscosity ratio was 10 and the turbulence intensity was equal to 0.4%. The static pressure at the outlet boundary condition was set as shown in Table 2.

Table 2. Outlet boundary conditions

case	outlet	slot 1	slots 2	slot 3
model 1	70 kPa	—	—	—
model 2	70 kPa	—	—	—
model 3	76 kPa	50 kPa	38 kPa	10 kPa

4. Numerical results

The shock wave generated at the leading edge of the lower profile is reflected from the upper wall of the test section and propagates downstream (Figure 9). The supports mounted above the upper profile influence the cross section reduction and a decreased Mach number. It has to be emphasized that the upstream effect of the existing supports is very strong. As shown in Figures 9–10, the shock wave above the upper profile has moved upstream. Downstream of the shock wave generated on the leading edge of the upper profile, the flow accelerates to supersonic conditions ($\mathrm{Ma} = 1.1$), but the velocity is much lower than in the case without supports.

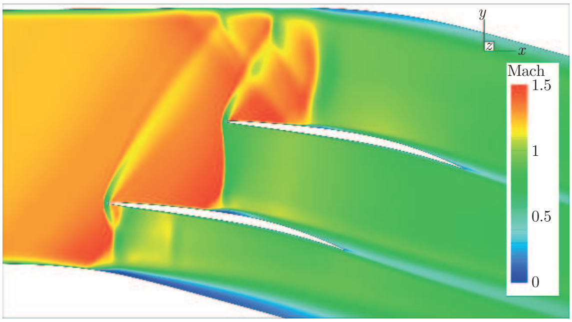

Figure 9. Mach number distribution in basic model

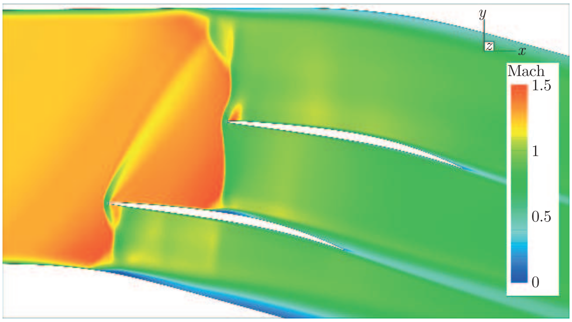

Figure 10. Mach number distribution in geometry with support

Additionally the decreasing velocity over the upper profile affects the subsequent formation of corner flows (blue zones represent isosurfaces of x component of velocity equal to $-0.01\,\mathrm{m/s}$). Due to a sudden change in the geometry behind the supports, the recirculation zone areas are formed (red circle in Figure 12). The blue zones differ between the cases (1 and 2, Figures 11–12), what can be noticed near the corner above the profile near the sidewall. If the sidewalls are flat without supports, the corner separation (blue zone) is created downstream of the leading edge shock wave as an effect of interaction with the boundary layer. The application of supports influences the translation separation zone more downstream.

The lower profile is also held by the supports. They influence the flow structure below the lower profile. The upstream effect is weaker than in the case of the upper supports. However, downstream of the supports near the bottom wall the separation zone differs in both cases. If the sidewalls are flat without supports, then the separation is created near the corners (green circle on Figure 11). In case

2 (with supports) the separation is created downstream of the supports along its height. However, a more pronounced difference exists near the bottom wall, where one can see separation zones created farther from the sidewalls, out of the corners. These zones are highlighted by yellow circles in Figure 12.

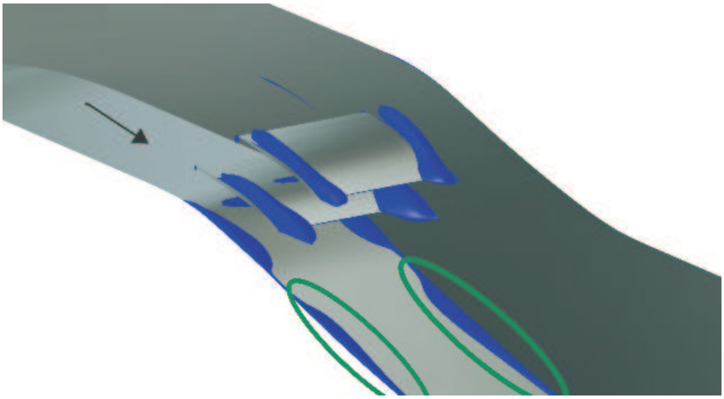

Figure 11. Reverse flows on basic model

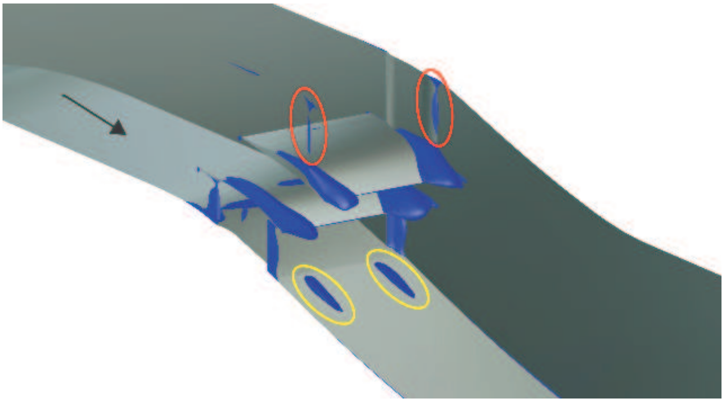

Figure 12. 3D Reverse flows on geometry with support

The supports mounted in the test section do not affect the flow structure between the upper and lower blades. The position of a normal shock wave generated on the leading edge and beneath the upper profile is at the same location (on the suction side of the lower profile). The mass flow in the blade passage is constant for both cases, and it is equal to 0.76 kg/s. Therefore, the corner flows near the suction side of the lower profile are the same in both cases.

One of the design criteria of the test section for the TFAST project is the similarity of the distribution of flow parameters on the lower profile. Such requirement can be fulfilled, if the upstream and downstream conditions are properly controlled. The upstream conditions, the Mach number and the shock wave structure are controlled by the above mentioned convergent-divergent nozzle. The downstream flow parameters are adjusted by the outlet pressure and the suction system realised by slots applied at the upper and lower test section walls. Suction at slots 1 and 3 (Figure 5) allows influencing the boundary layer and finally the shock wave location on these walls. On the bottom wall, suction is important for separation control and the amount of the sucked air affects the lower profile wake deflection. If the separation close to the bottom wall is larger, then the wake is more deflected upward, if it is reduced – the wake deflects downward. Such deflection is very important for the shock wave location on the lower profile, *i.e.* on the investigated suction side of the blade.

Another parameter strongly influencing the shock wave location on the profile is the outlet pressure. Its influence is a classical outlet pressure effect in the nozzle, the shock wave moves upstream or downstream, when pressure rises or decreases. The proper location of the shock wave generated on the leading edge of the lower profile depends on the interaction with the boundary layer at the bottom wall [7]. The required effect can be obtained, if the correct shape of the wall and the suction slot 1 is applied [8, 9].

Figures 13–14 show the streamlines on the lower profile and separation zones for two different flow cases. The streamlines show the contraction downstream of the shock wave. The blue iso-surface in the middle represents the separation zone on the profile. The positive pressure gradient has influence on the corner flow separation located near the sidewalls and a different upstream effect on the flow near the corners can be noticed. For the flow case 2 (Figure 13) the corner separation appears much upstream than the separation in the middle of the passage. Corner vortices and their influence on the overall structure can be controlled by suction slots 2. The effect of the side wall suction application is shown in Figure 14. The applied lower pressure at slots 2 reduces the corner separation while the separation in the middle of the span increases.

Suction reduces also the contraction downstream of the shock wave. The described flow control technique by means of suction is very important for flow

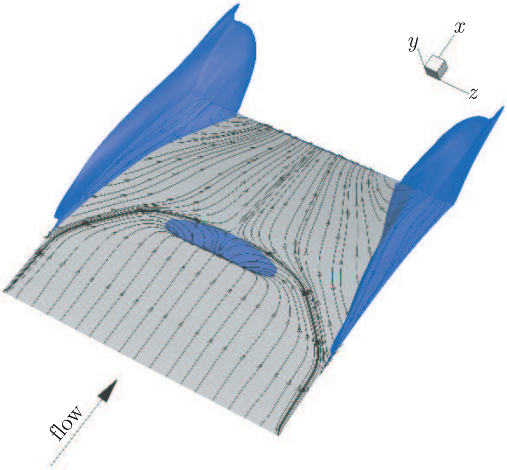

Figure 13. Secondary flows on model without suction (model 2)

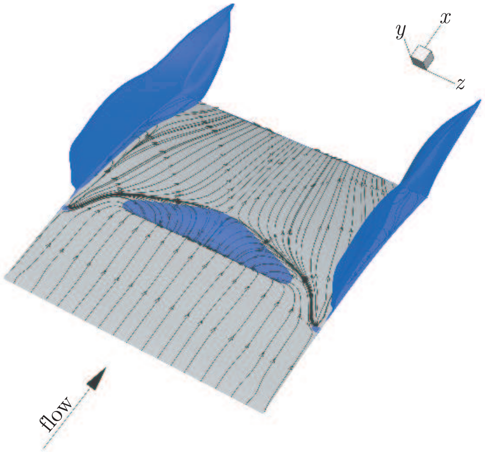

Figure 14. Secondary flows on model with suction (model 3)

parameters adjustment and final comparison with the required test section design criteria, *i.e.* similarity with the reference cascade flow structure. The influence of suction is also shown in Figure 15, where the isentropic Mach number at the middle span along lower profile is compared for the case with and without applied slots 2. The dashed line indicates the results for the model without active flow control while the solid line represents the case with active suction. As a result of the secondary flow reduction, the acceleration zone at 60% of the chord length is eliminated.

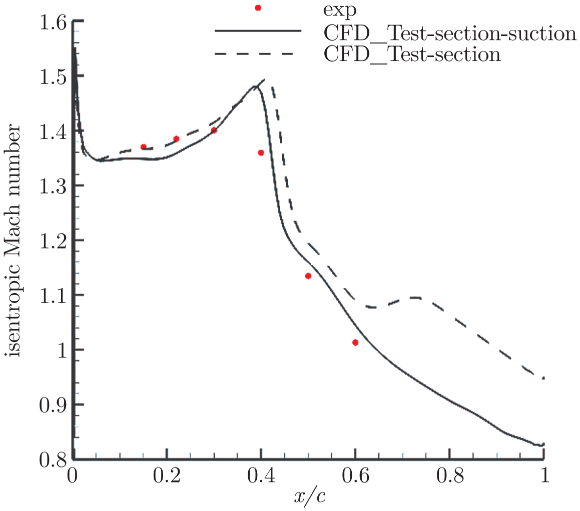

Figure 15. Isentropic Mach number distribution on suction side on lower profile

The solid line, representing the numerical simulation results for the case with suction slots fits to the experimental points very well. It confirms that the real flow at the test section can be well predicted by the numerical simulations.

The schlieren picture from experimental data (Figure 16) showing the flow pattern in the investigated passage can be compared with a similar picture based on the numerical results as the magnitude of density gradient (Figure 17). As shown in both Figures, the complex flow structure can be reproduced and the position of the shock waves predicted numerically is consistent with that obtained from the investigated test section.

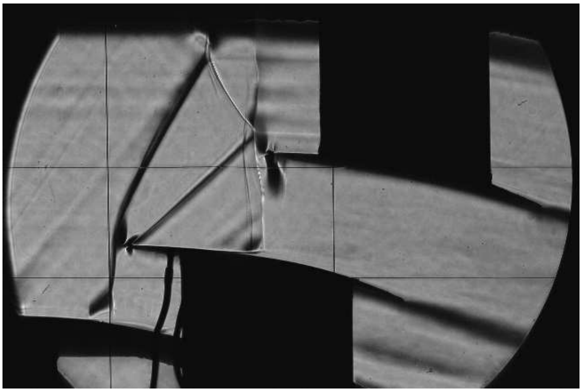

Figure 16. Schlieren picture

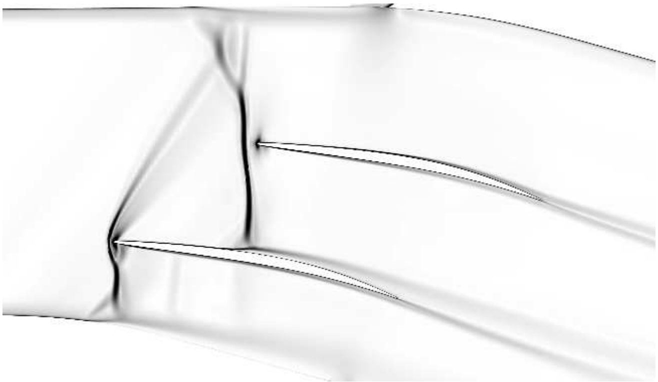

Figure 17. Numerical schlieren picture

5. Conclusions

Application of supports near sidewalls holding blades affects the flow structure. The flow structure can be controlled by properly adjusted suction at side and limiting channel walls. The suction influences the corner flows and it has strong impact on the entire flow field.

The designed test section allows investigation and analysis of a complex flow structure existing in a transonic compressor cascade in the transonic wind tunnel in IMP PAN. The similarity of the obtained flow structure is very important for further investigations of the shock wave boundary layer interaction on the transonic profile and investigations of the laminar-turbulent transition effect in such interaction. Finally the transition control methods can be examined in the designed test section (turbine compressor cascade model).

Acknowledgements

This is research was supported by the 7 EU framework project and was carried out within the research project with the acronym **TFAST** (Transition Location Effect on Shock Wave Boundary Layer Interaction).

This research was supported by CI TASK and in part by PL-Grid Infrastructure.

References

[1] Doerffer P 2009 *UFAST Experiments Data Bank: Unsteady Effects of Shock Wave Induced Separation*, IMP PAN Publishing

[2] Doerffer P, Hirsch M, Dussauge J, Babinsky H and Barakos G 2010 *Unsteady Effects of Shock Wave Induced Separation*, Springer

[3] Piotrowicz M, Flaszyński P and Doerffer P 2014 *Journal of Physics: Conference Series* **530**

[4] Numeca FINE^{TM}/Turbo v9.1 2014 Teoretical Manual *Documentation v9.1a*

[5] Numeca IGG^{TM}/Turbo v9 2014 User Manual *Documentation v9.1d*

[6] Numeca FINE^{TM}/Turbo v9.1 2014 User Manual *Documentation v9.1a*

[7] Babinsky H and Harvey J 2011 *Shock Wave-Boundary-Layer Interaction*, Cambridge University Press

[8] Lemke M, Gmelin C, Thiele F 2013 *Numer. & Exp. Fluid Mech NNFM*, Springer, **121**

[9] Liesner K, Meyer R, Chen Ping-Ping and Qiao Wei-Yang 2014 *ASME Turbo Expo 2014: Turbine Technical Conference and Exposition* 25043