Numerical investigations of shock wave interaction with laminar boundary layer on compressor profile

M Piotrowicz\(^1\) and P Flaszyński\(^1\)
\(^1\)Institute of Fluid-Flow Machinery, Polish Academy of Sciences,
Fiszera 14, 80-231 Gdańsk, Poland
E-mail: Michal.Piotrowicz@imp.gda.pl

Abstract. The investigation of shockwave boundary layer interaction on suction side of transonic compressor blade is one of main objectives of TFAST project (Transition Location Effect on Shock Wave Boundary Layer Interaction). In order to look more closely into the flow structure on suction side of a profile, a design of generic test section in linear transonic wind tunnel was proposed. The experimental and numerical results of flow structure on a suction side of the compressor profile investigations are presented. The numerical simulations are carried out for EARSM (Explicit Algebraic Reynolds Stress Model) turbulence model with transition model. The result are compared with oil flow visualisation, schlieren pictures, Pressure Sensitive Paint (PSP) and static pressure.

1. Introduction
Shock wave boundary layer interaction is a phenomena existing in all transonic flows. Such interaction can affect the boundary layer separation downstream of the interaction. In the case of a civil turbofan engine operating at particularly high altitudes the Reynolds number can drop by a factor of 4, when compared to the sea level values. The laminar boundary layer on the transonic compressor rotor blades interact with shock waves and as a result a strong boundary layer separation is formed. This can seriously affect the aero-engine performance and operation. One way to avoid strong separation is to ensure that the boundary layer upstream of the shock wave is turbulent. Shock wave boundary layer interaction was widely investigated and lot of results are published [1-2, 11].

Forcing transition within the boundary layer can be achieved by the application of a surface roughness or a turbulator patch. Although such passive control methods are already in use, the mechanism of the shock wave-laminar boundary layer interaction, and in particular the source of the strong shock unsteadiness are still not understood in detail. Furthermore, the benefits of boundary layer control obtained for low Reynolds numbers can turn into loss increase at the higher levels of Reynolds numbers. Another possibility of transition control is to use Vortex Generators driven by Air Jets (AJVGs). In the compressor application the jets may be driven by the pressure difference between the suction and pressure sides of the blade.

In order to analyze the flow structure as the effect of shock wave boundary layer interaction on transonic compressor profile, test section was designed in IMP PAN within a framework of EU FP7 TFAST project [3-4].

The paper presents the comparison of numerical results and experimental data for laminar boundary layer interaction with the shock wave. One has to emphasize that modelling of such phenomenon on transonic compressor profile is very challenging, due to dependence of the shock wave and flow structure on the secondary flows [4]. As the first step, the flow structure details are compared and validation of numerical model is presented.
2. Geometry

Computational geometry (figure 1) consists of several elements, such as convergent-divergent nozzle, transonic profiles, support for holding profiles and slots. The nozzle was designed during EU FP6 UFAST project (Unsteady Effects of Shock Wave Induced Separation) [1, 2]. The main feature of the nozzle is the uniform distribution of the Mach number (M=1.22) at the location of installed compressor profiles. The nozzle was selected for the investigations in order to fulfil design requirements defined by Rolls-Royce Deutschland, one TFAST project partners. Mach number 1.22 is representative for relative Mach number at aircraft engine inlet, upstream of fan blades. Cascade configuration was considered as the reference for test section design, so the relative location of the profiles in the wind tunnel is the same as in the cascade. It is essential condition if one needs to reproduce flow structure as the one obtained in cascade configuration (figure 2). Reference conditions for the single passage are shown in table 1. Additionally in test section, the slots are implemented (marked 1, 2 and 3) and used to control the secondary flows, especially corner flows which have strong impact on the axial velocity density ratio [10].

![Figure 1. Test section – general view](image)

---

[1] EU FP6 UFAST project
[2] Unsteady Effects of Shock Wave Induced Separation
3. Numerical model description

The numerical simulations were carried out by means of FINE/Turbo Numeca code. The two-equation nonlinear eddy viscosity turbulence model of the EARSM (Explicit Algebraic Reynolds Stress Model) [6] with transition model was applied. The transition model (Menter and Langtry [15]) is based on two transport equations for intermittency and transition momentum thickness Reynolds number. The set of equations is closed by a perfect gas equation and Sutherland’s law for viscosity. It is applied 2nd order central scheme with scalar artificial dissipation for spatial discretization formulated by Jameson, Schmidt and Turkel [7].
The structured mesh was generated by means of IGG Numeca (Interactive Geometry Modeler and Multi-Block Structured Grid Generator). The multi-block topology shown in figure 3 consists of 35 blocks and the total number of hexahedral cells is $14 \cdot 10^6$. The mesh resolution near the solid wall boundaries was assumed in such a way that $y^+$ was below 1.

Full non matching boundary connection (FNMB) between blocks with different node distribution where applied. The full non matching connection allows keeping the mass flow conservation, the momentum and energy through the interfaces between the blocks. An example of a mesh connection is shown in figure 4. The red mesh represents the bottom wall and the blue one shows the connection to the additional slot.
Total pressure 101 000 Pa and total temperature 293 K are set at the inlet. Turbulence kinetic energy and dissipation rate were set assuming that the viscosity ratio was 10 and the turbulence intensity was equal to 0.5% according to the measurements. The static pressure at the outlet boundary condition was set as shown in Table 2. It is adjusted properly in order to get the required shock wave location.

### Table 2. Outlet boundary conditions.

| Name    | Value   |
|---------|---------|
| Outlet  | 76 [kPa]|
| Slot 1  | 50 [kPa]|
| Slots 2 | 38 [kPa]|
| Slot 3  | 10 [kPa]|

4. Numerical results

The boundary layer upstream of the shock wave at four location is shown in figure 5. Traverses 1, 2, 3 and 4 are located at relative position x/C=0.089, 0.163, 0.242, 0.355. The integral parameters of the boundary layer profiles are shown in table 3 where $\delta^*$ is displacement thickness, $\delta^{**}$ momentum thickness and $H$ means shape factor.

![Boundary layer thickness graph](image-url)

**Figure 5. Boundary layer thickness**
Table 3. Integral parameters for velocity traverse.

| Traverse | δ*    | δ**   | H   |
|----------|-------|-------|-----|
| 1        | 0.063 | 0.017 | 3.717 |
| 2        | 0.085 | 0.022 | 3.772 |
| 3        | 0.101 | 0.027 | 3.726 |
| 4        | 0.112 | 0.03  | 3.691 |

Velocity profiles upstream of the shock wave are compared with laminar boundary layer profile (Blasius) and turbulent boundary layer Prandtl profile [9] on a flat plate (figure 6). In the equation for turbulent profile used coefficient n=7 which is valid for Re numbers in the range of $5 \cdot 10^5 - 10^7$ (in the investigated case Re=$1,27 \cdot 10^6$). Boundary layer profile at traverses 1-4 are similar to the laminar one therefore, one can conclude that the boundary layer downstream of the leading edge is laminar and shock wave interacts with laminar boundary layer. In the case of traverse 4, the profile with higher shear stress close to the wall can be noticed, what arises from local expansion zone. Nevertheless, the boundary layer starting at the leading edge up to the shock wave is laminar which is also confirmed by the intermittency distribution shown in figure 7 (0-laminar flow, 1-turbulent flow).

Figure 6. Normalized Velocity distribution

Figure 7. Shear stress distribution and intermittency at mid span on suction side on lower profile
The schlieren picture from experimental data (Figure 8) showing the flow pattern in the investigated passage can be compared with similar picture predicted by numerical simulations (Figure 9). As shown in both figures, the complex flow structure above suction side of lower profile can be reproduced and the position of shock waves predicted numerically is consistent with that shown in measurements. In both cases the shock wave generated at the leading edge of the lower profile is reflected from upper wall of test section and propagates downstream in the passage between the upper profile and the upper wall. The reflected shock wave penetration into the blade passage would be highly undesirable because such interaction does not exist in in cascade passages. In spite of differences of the shock wave interaction with the upper wall, different λ foot height and reflection angle, the shock wave structure between profiles is very similar to the measured one. In Figure 9, thicker boundary layer downstream of the shock than its influence on the wake is shown. The λ foot on the lower profile is not well visible in schlieren picture as in case of numerical results, but its height is similar.

Figure 8. Schlieren picture

Figure 9. Density gradient magnitude

Isentropic Mach number on suction side of lower profile at middle span is shown in Figure 10. Numerical simulations results (black line) are compared with isentropic Mach number obtained from the static pressure measurements (red dots) and data (blue line) delivered by Rolls-Royce as the reference for the test section design. Downstream of the shock wave, the flow accelerates slightly and Mach number predicted numerically is very close to the measured one. Similar agreement is obtained downstream of the shock wave. In the location shortly upstream of the shock wave, it is difficult to assess the numerical prediction due to the lack of measurement point in range x/C 0.3 – 0.4. However, general picture of the pressure distribution can be compared by means of Pressure Sensitive Paint measurements.

Figure 10. Isentropic Mach number distribution at mid span on suction side on lower profile
Static pressure distribution on the whole suction side of the investigated profile is shown in Figure 11, Pressure Sensitive Paint measurement on the left and numerical results on the right. As already shown in Figure 10, the pressure distribution in the middle section is predicted well. Discrepancy increases closer to the sidewalls, where the effect of suction slots can be noticed. These disparities arise from different size of corner vortices and mass flow at suction slots, which is adjusted in order to get the correct shock wave location and pressure distribution at middle section.

![Figure 11. Static pressure on suction side](image)

Different pressure distribution close to the sidewalls influences on the separation downstream of the shock wave, as shown in oil flow visualization (Figure 12 and 13). The separation length (streamwise direction) is the same in both cases, experimental and numerical. In the measurements, one can see a lower spanwise size of the separation bubble which is the effect of more intensive corner flows. At the leading edge, close to the sidewalls, the corner flows are generated and they develop further downstream. As shown by oil flow visualisation, their intensity is higher than predicted numerically what is visualised by streamlines near the wall. Finally, they influence on the separation size in spanwise direction, but its effect on the flow downstream of the shock wave is much weaker.

![Figure 12. Oil flow visualisation](image)

![Figure 13. Streamlines near the suction side](image)

5. Conclusions
In the paper, there are presented numerical simulation results carried out for the transonic compressor profile investigated experimentally in IMP PAN wind tunnel. Investigations of flow structure on suction side of compressor profile can be successfully done in single passage test section. The main objective of the investigations was the shock wave interaction with laminar boundary layer. Numerical prediction of such interaction enforces application of turbulence model including transition modelling effect. As indicated by velocity profiles in the boundary layer upstream of the shock wave,
shear stress distribution on the investigated profile and intermittency, the boundary layer is laminar from the leading edge up to the shock wave. Separation bubble length downstream of the shock wave is predicted properly. The difference arises from the corner flow existence and the application of suction at sidewall slots what affects less agreement than obtained in the middle section of the investigated profile. Anyway, the very good agreement in the middle of the profile allows for further investigations of flow control devices application to transition location control and its influence on shock wave boundary layer interaction.

Acknowledgements

This 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, Doerffer P 2014 Investigation of Shock Wave Boundary Layer Interaction on Suction Side of Compressor Profile Journal of Physics: Conference Series Vol. 530
[4] Piotrowicz M, Flaszyński P, Szwaba 2015 Influence of Limiting Walls on Shock Wave Structure in Single Passage Test Section with Compressor Profile Task Quarterly: Scientific Bulletin of Academic Computer Centre in Gdańsk Vol. 19(2) pp.141-152
[5] Flaszyński P, Doerffer P, Szwaba R, Kaczyński P, Piotrowicz M 2015 Shock Wave Boundary Layer Interaction on Suction Side of Compressor Profile in Single Passage Test Section Journal of Thermal Science Vol. 24(6) pp. 510-515
[6] Numeca FINE™/Turbo v9.1 2014 Theoretical Manual Documentation v9.1a
[7] Jameson A, Schmidt W, Turkel 1981 Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes AIAA Paper 81-1259, Palo Alto, CA
[8] Becker B, Rayer M, Swoboda M, 2007 Steady and unsteady numerical investigation of transitional shock-boundary-layer-interaction on fan blade Aerospace Science and Technology, Vol. 11, No 7-8, pp. 507-517
[9] Puzyrewski R, Sawicki J, 1987 Podstawy Mechaniki Płynów i Hydrauliki Państwowe Wydawnictwo Naukowe, Warszawa
[10] Bo Song, Wing F. Ng Influence of Axial Velocity Density Ratio in Cascade Testing of Supercritical Compressor Blades 40th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit 11-14 July 2004, Fort Lauderdale, Florida
[11] Babinsky H, Harvey J 2011 Shock Wave-Boundary-Layer Interaction, Cambridge University Press
[12] Langtry R, Menter F, 2005 Transition Modelling for General CFD Applications in Aeronautics AIAA 2005-522