Investigations of shock wave boundary layer interaction on suction side of compressor profile

M Piotrowicz, P Flaszyński and P Doerffer

Transonic Flows and Numerical Methods Department, The Szewalski Institute of Fluid-Flow Machinery (IMP PAN), 14 Fiszera Street, Gdańsk 80-231, Poland

E-mail: michal.piotrowicz@imp.gda.pl

Abstract. The shock wave boundary layer interaction on the 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 the suction side of a blade, a design of a generic test section in linear transonic wind tunnel was proposed. The test section which could reproduce flow structure, shock wave location, pressure distribution and boundary layer development similar to the obtained on a cascade profile is the main objective of the presented here design. The design of the proposed test section is very challenging, because of shock wave existence, its interaction with boundary layer and its influence on the 3-D flow structure in the test section.

1. Introduction

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

In the paper, the geometry configuration influence on the flow structure is presented. Shock wave location and separation bubble depends on the profile location in respect to the upper and lower tunnel walls. One of the design criteria is the inlet Mach number and inflow uniformity which should be adequate to the real conditions. In order to maintain the required inflow conditions the nozzle designed during UFAST Project is used.

2. Geometry

A sketch of the test section is shown in figure 1 the upper and lower profiles are the main part of the test section. A reference conditions for the test section design are obtained from the numerical simulations for cascade configuration (figure 2). Mach number contours and isentropic Mach number distribution on the profile should be reproduced in the designed test section. Transonic compressor profile is delivered by Rolls-Royce Deutschland. The main parameters of cascade are shown in table 1.

Relative location of the profiles in the test section is the same as in the cascade. Upstream of the profiles, a convergent-divergent nozzle is located. Upper wall shape of was designed (UFAST Project) in order to obtain a uniform Mach number at the inlet stream. It is useful test section feature, because it allows a translation of both profiles without changing inlet conditions.
3. **Numerical model description**

Numerical simulations have been performed by means of Fine/Open Numeca solver. At the first step, the geometry parameters influence on the shock wave location are investigated. Unstructured 2D mesh (Hexpress/Numeca) consists of $\sim 2.8 \times 10^5$ cells. The grid is refined ($y^+ \sim 1$) close to the profiles, mainly close to the suction side of lower profile and pressure side of the upper one. An example of the mesh is shown in figure 3.

---

**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  |

---

**Figure 1.** Geometry test section.

**Figure 2.** Mach number in cascade and isentropic Mach number on profile.
One-equation turbulence model Spalart-Allmaras is applied. In all test cases used, the total pressure and temperature with turbulent viscosity at inlet boundary conditions was assumed. At outlet a constant static pressure was applied. Spatial discretization using 2nd order central difference scheme with scalar artificial dissipation formulated by Jameson, Schmidt and Turkel (1981) is applied

4. Numerical results
As it was mentioned, the main objective of the design is to obtain the same flow conditions as in the cascade. The first results of numerical analysis allow to draw important conclusions. The position of the shock wave is dependent on the relative profile position to the fixed duct walls. The shock wave generated upstream of the leading edge of the lower profile is reflected from the upper wall. The design should cause that further this shock reflects from the suction side of the upper profile, avoiding its penetration into the blade passage. Vertical translation of the profiles influences the position of the reflected shock wave. The upper location of the lower profile (maximum distance from the lower wall) is limited by this reflection. The profile cannot be translated too much, otherwise the reflected shock wave appears between profiles. Such shock wave configuration is highly undesirable because such interaction does not take place in blade passages (figure 4).
Horizontal translation affects Mach number distribution close to the lower wall (figure 5). The lower wall shape is defined according to the streamline curvature obtained for cascade configuration. However, the final shape of the lower wall has to be modified due to the boundary layer effect and its development downstream of the shock wave. Modification of vertical/horizontal location influences the static pressure distribution on lower profile, what is shown on isentropic Mach number plot (figure 6).

**Figure 5.** Influence of horizontal location.

**Figure 6.** Isentropic Mach number (lower profile).
vertical shift – left, horizontal shift – right (“+5” – 5mm translation).
Shock wave boundary layer interaction on the lower wall is very important for the overall flow structure in the test section. Depending on the lower wall shape and upstream shock wave location, the separation can appear. Similar effect can be noticed at the upper wall. Modification of the upper and lower walls (figure 7) allows to minimize or totally remove the separation bubble (blue area marked with red dots). Size of the separation has a strong impact on the flow structure downstream of profile, because it deflects the lower profile wake. It negatively influences on the separation downstream of the shock wave on the lower profile suction side.

![Mach number](image)

**Figure 7.** Influence of lower wall.

Effect of pressure at the outlet is shown in figure 8. Increased outlet pressure moves the shock wave further upstream, it is a classical outlet pressure effect in the nozzle. Finally, increased or decreased outlet pressure influences shock wave reflection from the upper wall, so it is an additional important parameter which should be properly adjusted if one has to find the required shock wave.
In the compressor cascade, intensity of the shock wave generated at the leading edge, then pressure distribution on the profile, transition location and separation size (if exists) is dependent on the inflow angle. The test section is designed for the certain, operating condition, but the influence of the inflow angle should be taken into account. Such degree of freedom is also envisaged in the test section and one degree rotation at the leading edge is allowed. Mach number for two different inflow angles is shown in figure 9.

**Figure 8.** Influence of pressure outlet.

**Figure 9.** Influence of profile rotation.
One has to emphasize, that the whole flow structure is highly sensitive and modification of any mentioned geometrical parameters influences on the whole flow field.

![Figure 10. Isentropic Mach number on lower profile.](image)

The next step of the test section design is an analysis of the 3D effects. The structural mesh for the test section is generated in IGG/Numeca and it consists of $9.6 \times 10^6$ cells (figure 11). Simulations were carried out by means of Fine/Turbo Numeca with Spalart-Allmaras turbulence models. An example symmetric solution of flow structure is shown in figure 12 the main difference between 2D and 3D approach is the effect of sidewalls. In figure 12, isosurfaces of velocity component in X direction equal to zero are shown. They represent corner separation downstream of the shock wave boundary layer interaction zones.
5. Conclusions
The numerical simulations results for designed test section are presented. It was shown, that the shock wave configuration is highly dependent on the investigated profiles location. The main objective of the design, similarity of the flow parameters distribution on the suction side of the profile in the test section and in cascade configuration, can be obtained by appropriate geometry definition. Discussed results are obtained by means of two-dimensional model, so it can be considered as the first step of design. Next one, mentioned at the last part of the paper, it is three-dimensional approach. Investigations within a framework of two-dimensional model are useful and very helpful if one needs to limit the number of cases for three-dimensional simulations. However, two-dimensional model does not include sidewall effect, very important for shock wave boundary layer interaction and flow structure development.

6. References
[1] Doerffer P, Hirsch M, Dussauge J, Babinsky H and Barakos G 2010 Unsteady Effects of Shock Wave Induced Separation Springer
[2] Doerffer P 2009 UFAST Experiments Data Bank: Unsteady Effects of Shock Wave Induced Separation IMP PAN Publishing
[3] Prosnak W 1971 Mechanika Płynów tom II Dynamika Gazów PWN
[4] Babinsky H, Harvey J 2011 Shock Wave-Boundary-Layer Interaction Cambridge University Press

Acknowledgments
This is research was supported by 7 EU framework project and was carried out within research project of acronym TFAST (Transition Location Effect on Shock Wave Boundary Layer Interaction). This research was supported in part by PL-Grid Infrastructure.