XX Fluid Mechanics Conference KKMP2012,
Gliwice, 17-20 September 2012

EFFECT OF INFLOW VARIATION ON COMPRESSOR SECONDARY FLOW BEHAVIOR

Pingping Chen [1,2], Karsten Liesner[1], Robert Meyer[1], Weiyang Qiao[2]
1 DLR Institute of Propulsion Technology, 10623 Berlin, Germany;
2 Northwestern Polytechnical University, 710072 Xi'an, China
E-mail: pingping.chen@dlr.de

The application of lower aspect ratios and high loadings in the modern axial compressor leads to a much higher single-stage pressure ratio, whereas this also causes much more complex internal flow. Consequently, this leads to more complex secondary flow behavior, higher losses and a more complicated trigger mechanism for the flow instability. The study of secondary flow is usable to achieve a better compressor performance and avoid such instability problem. So the mechanism of secondary flow should be investigated firstly as a reference of the mechanism of flow instability and the further research of related flow control techniques.

Owing to the viscous flow and the non-uniform inflow, the secondary flow becomes substantial for the critical performance demands. To capture the internal flow mechanism well, numerical and experimental methods of a high-speed compressor cascade (consisting of NACA 65-K48 profiles, properties in Table 1) are carried out with inflow variations in the present investigation. In this paper, the influences of inlet boundary layer thickness and the inlet turbulence are investigated.

Table 1: Cascade properties at aerodynamic design point

Ma_1	Re	chord	span	pitch	β_1	β_s
0.67	560.000	40mm	40mm	22mm	132°	112.5°

In the experimental part of this investigation, the High Speed Wind Tunnel [1] of the German Aerospace Center (DLR) is used for validation of the calculations. Its robust measurement techniques allow for a qualified statement of flow quantities and topology. For the loss and flow turning investigation, a total pressure and angle probe rake is used in the measurement plane, 0.4 times the chord length downstream of the trailing edge. For the flow topology comparison, flow visualization techniques with oil-streak patterns are used on the stator vanes surfaces.

For the numerical simulations, the 3D RANS flow solver TRACE, developed by the German Aerospace Center (DLR), is used. With fully turbulent setting, a k-ω two-equation turbulence model is used in the steady simulations. According to the demands of flow solver and turbulence model, a multiple O-C-H type mesh is used. The block structured computational domain was generated for the half-span passage only, due to a symmetry boundary condition setting at mid span. The O-grid was utilized around the blade with 25 nodes along the wall-normal direction to resolve the flow in the vane's boundary layer, ensuring the dimensionless wall distance of $y^+ < 1$. The total number of grid nodes for the computational-domain is up to 1.4 million.

The variation of inlet boundary layer thickness was performed at the design point of the blade (T_u=2%), δ_{99} ranging from 1mm to 5mm. The fully turbulent steady simulations are mainly concentrated on the influence of the inlet boundary layer thickness on the secondary flow and flow loss. With the analysis, it can be concluded that, 1) The inlet boundary layer thickness has a pronounced effect on the secondary flow behaviors and loss; 2) The inlet boundary layer thickness influences the side wall flow loss a lot, while the profile loss remains nearly constant; 3) The total pressure loss is affected by the BL thickness because of the changes of the corner vortex and concentrated shed vortex on the suction surface; 4) The inlet boundary layer thickness has a small influence on the variation of outflow angle, which is below 1 degree at the most; 5) However, the inlet boundary layer thickness changes the three-dimensional distribution of the flow loss at the exit; 6) With the analysis of iso-surface of axial vorticity, the horseshoe vortex on the suction side is in the corner region of the profile and the sidewall, but the pressure-side branch induced by the pressure gradient between the neighboring two cascades tends to develop in the passage and then becomes part of the passage vortex; 7) Due to the development of horseshoe vortex, the wall streamlines on the profile pressure-side on the leading edge and trailing edge show a slight displacement, while the suction side differs on the intensity and size of the corner separation/vortex along span-wise and axial direction; 8) Regarding the 3D streamlines in the passage, when increasing the inlet boundary layer thickness, the passage vortex swirls much bigger, and the corner vortex is increased a lot to nearly half of the mid-span, while the wake vortex is similar to each case.

For the inflow turbulence variation, the simulations were based on the design inlet conditions with a boundary layer thickness of 4mm. With the results of simulations, it can be concluded that 1) The inlet turbulence has a noticeable influence on the profile loss due to increased surface friction; 2) Its effect on the secondary flow behavior can be observed in the changes of the secondary flow loss at the exit plane; 3) Compared to the experiment, the inlet turbulence affects the exit flow angle little, which is still of the similar trend with the experiment along span. The maximum value of the exit flow angle is varied a little with the inlet turbulence variation; 4) The inlet turbulence alters the development of the sidewall boundary layer thickness, which can be obtained from the passage turbulent kinetic energy contours; 5) The analysis of the blade suction-side wall shear stress contours and the surface streamlines shows that the inlet turbulence increases the wall shear stress on the suction side, but has little effect on the corner separation/vortex along the span-wise and axial direction.

In the full paper we will show mostly numerical simulations, some of which are highlighted with experimental results. The distribution of the total pressure loss is compared one to one for simulation and experiment while the flow visualization is used for a general description of the flow topology in comparison with the numerical results.

References

[1] Liesner, K. Meyer, R. (2008). "*Experimental setup for detailed secondary flow investigation by two-dimensional measurement of total pressure loss coefficients in compressor cascades*". VKI XIX Symposium on Measurement Techniques in Turbomachinery; Brussels, Belgium