Effect of Load Distribution at the Roots among Stages on Performance of a High Bypass-ratio Fan/Booster

Xiaoxu Ding¹, Donghai Jin¹,² * and Xingmin Gui¹,²

¹ School of Energy and Power Engineering, Beihang University, Beijing, China.
² Collaborative Innovation Center for Advanced Aero-Engine, Beijing, China.

*Corresponding author e-mail: 13146014223@163.com

Abstract. The modern high-performance aero-engines have raised higher requirements on the fan/booster, such as high throughput, high pressure ratio, high efficiency and a wide range of stable operation. In this paper, a method called arbitrary camber lines blading design method for both axial and centrifugal compressors and three dimensional CFD (computational fluid dynamics) calculation are used to design and analyse the results of the aerodynamic design of a high bypass-ratio fan/booster. By changing load distribution on the roots among the rotors, the flow of the fan/booster can be affected. The results of the study show that increasing the load on upstream stages roots can help relieve the separation at the roots of the downstream stages, which leads to the increase of the total pressure, adiabatic efficiency and stall margin of the core duct of the fan/booster.

1. Introduction
Turbofan engines are widely used in military and civil aviation fields with advantages, such as low fuel consumption and high cycle thermal efficiency. As one of the components of a turbofan compressor, fan/booster’s characteristics at different operating points have a direct effect on the engine performance. The modern high-performance aero-engines have raised higher requirements on the fan/booster, such as high throughput, high pressure ratio, high efficiency and a wide range of stable operation [1].

But there are many difficulties to meet the requirements. At first, the rotating speed of a fan/booster is low, so the roots of the fan and the booster have a weak ability to increase the total pressure of the fluid. What’s more, because the flow path of the booster is complex and the blades are short, the ratio of the boundary layer to the blade height is high, which increases the difficulty to improve efficiency and stall margin [2].

Smith diagrams show the relationship between flow coefficient and load factor on different degrees of reaction published in Turbomachinery Performance Analysis [3], which reveal that the matching of flow coefficient, load factor and degree of reaction has a crucial influence on the efficiency and the stall margin of the compressor stage. Fang Z [4] investigated the effect of end-bending and bow on the flow separation in corners of blades in a high throughput-flow fan/booster, which shows that the end-bent fan could improve the flow field and increase the adiabatic efficiency and that the end-bending and bow for stators could reduce choking and losses of the corner flow.
This paper uses the arbitrary camber lines blading design method [5] to optimize the original fan/booster. By changing the load distribution among the roots of the rotors to alter the matching of the load factor and the flow coefficient, the performance of the fan/booster has been improved.

2. Research object and tools

2.1. Research object

In this paper, the fan/booster is optimized based on the 11km, 0.75Ma flight condition, whose flow path is shown in Fig. 1. And the table 1 shows the nomenclature.

![Flow path of the fan/booster](image)

**Figure 1.** Flow path of the fan/booster

| Nomenclature | Meaning                  | Nomenclature | Meaning                  |
|--------------|--------------------------|--------------|--------------------------|
| RF           | Fan                      | S2           | Second stage stator      |
| SF           | Outlet guide vane        | R3           | Third stage rotor        |
| SB           | Vane assisted to calculate| S3           | Third stage stator       |
| S0           | Inlet vane of the booster| R4           | Fourth stage rotor       |
| R1           | First stage rotor        | S4           | Fourth stage stator      |
| S1           | First stage stator       | SC           | Support board            |
| R2           | Second stage rotor       |              |                          |

**Table 1. Nomenclature**

2.2. Design Tool and Three-Dimensional Navier-Stokes Computations

The fan/booster is designed by the streamline curvature through-flow method and arbitrary camber line blading method[5,6]. Three-dimensional numerical simulation has been widely used in turbomachinery and proved to be of acceptable accuracy [7~9]. In this paper, commercial software Numeca is used to conduct the three-dimensional numerical simulation analysis. The computational mesh is shown in Fig. 2. The fan/booster is meshed with 3557311 cells. The turbulence model is the Spalart-Allmaras model. Total pressure, total temperature and the direction of the flow are imposed at the inlet of the
computational domain. At the outlet, static pressure ruled by the simple radial equilibrium is set. Adiabatic non-slipping condition is applied to the solid boundaries.

**Figure 2.** Computational grid for the fan/booster

3. **The analysis of the characteristics and flow field of the original fan/booster**

Fig.3 and Fig. 4 show the performance characteristics of total pressure and efficiency, which are normalized by the design requirements. As can be seen from the performance characteristics, the total pressure ratio and the efficiency of the bypass duct met the requirement, while the core duct did not.

![Performance curve of core duct](image1)
![Performance curve of bypass duct](image2)

**Figure 3.** Performance curve of core duct

**Figure 4.** Performance curve of bypass duct

Fig.5~Fig.7 show the flow fields at the near design point, which are respectively the circumferentially averaged relative Mach number distribution of the fan/booster, the relative Mach number distribution of
the fan in the 5% percent span and the limiting streamlines on the suction surface of fan, the relative Mach number distribution of the booster. From Fig.5 and Fig.7, it can be seen that there was a large low-Mach-number area at the roots of the R3, S3, R4, S4. In addition, it can be seen from Fig.5 and Fig.6 that the flow at the root of the fan was in good condition.

Figure 5. Relative Mach number distribution on meridional plane at near design point

Figure 6. Relative Mach number distribution of the root of RF and the limiting streamlines on the suction surface of RF

Figure 7. Relative Mach number distribution of the booster

4. The results of the optimization

4.1. The redistribution of the load on the roots in the core duct
From the analysis above, we can know that the reasons for the lack of the pressure ratio and efficiency is that the separation from the R3 root. From the paper of Fang Z [4], we can know the separation at the root of R3 could not be controlled by end-bending. So it is reasonable to consider reducing the loading at the root of R3. But there is another matter which should be taken into account that the stagnation
temperature and the stagnation pressure should be as equal as possible from hub to casing of the booster’s outlet in order to reduce the distortion of the inlet of the high compressor from hub to casing. Therefore, when reducing the load on the downstream stages roots, the load on the upstream stages roots should be increased. As can be seen from Fig.5 and Fig.6, the Mach number at the inlet of the booster is 0.62, which is not too high and the flow condition of the fan in the core duct is good. So it can be assumed that the load on the downstream stages can be reduced and the load on RF in the core duct can be increased. Fig.8 and Fig.9 show the changing of the load on RF and R3.

4.2. Comparison of the characteristics
In the optimization process, the same grid structure and control parameters as the original fan/booster were used to make the comparison of calculation results more convincing. The comparison of the characteristics is shown in Fig.10. As can be seen from the figures, compared with the original, the total pressure and adiabatic efficiency have been improved, and so has the stall margin. Table 2 shows the comparing between the original and the modified.

|                      | Original | Modified | Difference |
|----------------------|----------|----------|------------|
| Pressure ratio       | 0.9657   | 1.0075   | +4.77%     |
| Efficiency           | 0.9824   | 1.0153   | +3.31%     |
4.3. Comparison of the flow field
This section compares the flow fields between the original and the modified at near design point. Fig.11 shows the relative Mach number distribution of the modified on meridional plane. Compared with the original, the Mach number at the entrance of the booster has been increased, but it is still within the acceptable range of the selected profile (controlled diffusion profile). The low-Mach-number area at the roots of the third and fourth stages has disappeared. Fig.12 and Fig.13 show the spanwise distribution of adiabatic efficiency of RF (within 50%span) and the relative Mach number distribution at the roots of the booster, which reveal that the efficiency of RF has increased and that the flow field of the booster has been improved.

Figure 11. Relative Mach number distribution on meridional plane at near design point

Figure 12. Spanwise distribution of efficiency of RF (within 60%span)

Figure 13. Relative Mach number distribution of the booster roots of the original and the modified

5. Conclusion
In this paper, optimization has been conducted of a fan/booster by redistributing the load on the roots of the rotors. Main results are summarised as follows:
1. The distribution of the load on the roots of the rotors in a fan/booster is important to achieve high pressure ratio and high efficiency.
2. The results of 3D CFD show that the total pressure and adiabatic efficiency of the core duct of the modified fan/booster has increased by 4.77% and 3.31%, meeting the design requirement, and that the stall margin has also been improved.
3. It is helpful to improve the efficiency of the fan to increase the load on the fan root within a certain range, when taking Mach number at the entrance of a booster into consideration.

4. Raising the loading of the fan root can reduce the loading of the downstream stage roots, which helps relieve the separation at the roots of the downstream stages.

References

[1] Xingmin Gui, Fang Zhu, Donghai Jin, Report of Aerodynamic Performance of Scaled Fan. Beijing: Beijing University of Aeronautics and Astronautics, 2012.

[2] Baoshi Chen, Analysis of Difficulties in Designing A High Bypass-ratio Fan/booster. High-level Forum on Key Technologies of Large Aircraft and Academic Essays of the 2007 Annual Conference of Chinese Society of Aeronautics and Astronautics 2007.

[3] Lewis R I. Turbomachinery Performance Analysis. 1996.

[4] Fang Z, Gui X. Corner Flow Control in High Through-Flow Axial Commercial Fan/Booster Using Blade 3-D Optimization. Journal of Thermal Science, 2012, 21(1):32-41.

[5] Guofeng Ji, Xingmin Gui, A Blading Design Method for Axial/Centrifugal Compressor Airfoils Using Arbitrary Camber Lines, Journal of Aerospace Power, 2009, 24(1):150-156.

[6] Frost G R, Wennerstrom A J. The Design of Axial Compressor Airfoils Using Arbitrary Camber Lines. ARL-73-0107, AD-756165, 1973.

[7] Xiaojuan Li, Xingmin Gui, Numerical Simulation of Fan/compressor Design and Off Design Performance, Journal of Propulsion Technology, 2005, 26(6):522-525.

[8] Xingmin Gui, Xiaojuan Li, Aerodynamic Design of Fan/Compressor Operated at Low Reynold Number, Gas Turbine Experiment and Research, 2007, 20(3):18-22.

[9] Donghai Jin, Xingmin Gui, Numerical Simulation of Highly Loaded Multi-stage Fan/Compressor of A Turbofan Engine, Journal of Aerospace Power, 2011, 26(5): 4059-1065.