30th Congress of the International Council
of the Aeronautical Sciences
DCC, Daejeon, Korea ; September 25-30, 2016

MULTI-POINT DESIGN OPTIMIZATION OF A HIGH BYPASS RATIO FAN BLADE

Sheng Qiu *, Xiangyang Deng *
***AVIC Commercial Aircraft Engine CO.LTD**
Keywords: *high bypass ratio fan blade, aerodynamic, structure Mechanics, rain ingestion, multi-point design*

Abstract

The overall aerodynamic performance of the fan blade of a high bypass ratio aero engine is not only determined by its conventional aerodynamic parameters, such as pressure ratio and efficiency, but also the mass flow ratio of ingested water in the bypass duct during rain ingestion airworthiness test. To ensure the engine reliablibity, the larger percentage of ingested water passing through the bypass duct is always desirable. In conventional fan blade airfoil design practice, the pressure ratio and efficiency are first preliminarily determined for the design point, usually the cruise condition, and the mass flow ratio of ingested water in bypass duct is then calulated for the approach condition for the accessment of its rain ingestion performance. With this approach, many iterations are usually needed before the ideal design results can be obtained, and such interations can be a considerable part of the design cost. To accelerate the design procedure, a multi-point optimization method is proposed in this paper to consider the multipal factors, i.e. the pressure ratio, efficiency and the mass flow ratio of ingested water, all together. In this method, the Bezier curve is used to parameterize the fan blade. The geometrical parameters are used as variables while the pressure ratio, efficiency, and the mass flow ratio of ingested water in bypass duct of the fan blade are used as the objective functions. The flow-field of fan rotor under water ingestion conditions is evaluated by ANSYS CFX. An Eulerian-Lagrangian approach is used in formulating the flow and droplet governing equations in the rotating reference frame. In order to resolve the difficulty of this high-dimensional multi-objective optimization problem, the NSGA-II algorithm is used to obtain a more complete picture regarding the trade-off among pressure ratio, efficiency and water mass ratio in bypass duct under three typical conditions. A multi-point design optimization of a typical high bypass ratio fan blade is performed in this paper. The mass flow ratio of ingested water in bypass duct of the optimized fan blade is much larger than that of the baseline, while the pressure ratio and efficiency of the blade remain unchanged. The feasibility and efficiency of this optimization method are shown by this work.

1 Introduction

The geometry of a fan rotor is of fundamental importance in defining the internal and external flow characteristics. But the design of the higher bypass ratio fan rotor is not based solely on conventional aerodynamic issues. The design process for the transonic fan involves multidisciplines such as aerodynamics, rain ingestion and structure mechanics. The overall aerodynamic performance of the fan blade of a high bypass ratio aero engine is not only determined by its conventional aerodynamic parameters, such as pressure ratio and efficiency, but also the mass flow ratio of ingested water in the bypass duct during rain ingestion airworthiness test. The fan in the turbofan engine suffers the effects first, since it is directly exposed to water ingestion. In the case of the fan rotor, water drop is expected to impact on the blades and casing, followed by rebound, droplet break up and film formation. And high-speed droplets break-up into smaller secondary droplets. Due to the centrifugal action, there is radial displacement of this water film, which has a detrimental effect on the compressor's performance. Water drop may enter the by-pass duct or the core engine. The problem of water ingestion in the engine becomes a key technology of engine development. To ensure the engine reliablibity, the larger percentage of ingested water passing through the bypass duct is always desirable.

The ingestion of large quantities of water into the gas turbine engine has a damaging effect and can lead to performance deterioration. This performance deterioration may result in flameout or shutdown of the engine. There have been several instances where the engine lost its power during water ingestion [1, 2, 3, 4]. Water ingestion is drawing increasing attention to recent research. Extensive research efforts have been conducted to investigate the effect of water ingestion on the gas turbine compressor. Williams et al. [6] and Das et al. [7] have investigated the trajectories of water droplets on a rotor blade. Williams has used CFD and particle tracking calculations to show the computed droplet trajectories for various droplet sizes entering the first rotor row of the test compressor [8]. Quantifying the effects of water ingestion on engine performance has been addressed by Day et al. [9]. Mund and Pildis showed an application of CFD to predict the air flow in a gas turbine air intake and simulate the water injection for compressor washing [10]. Roumeliotis et al. performed stage tests on wet gas compression in an axial compressor [11]. White at al. demonstrates a marked reduction in aerodynamic efficiency as water injected into a gas turbine increases [12].

These disciplines are highly coupled, and the optimal design in one discipline may lead to the failure in other disciplines. And the design of an aircraft engine fan is a complex process and may be pertinently called a study into compromise. Consequently, a detailed analysis of the various components constituting the fan rotor is required to achieve the optimal design specification. In order to improve the overall fan performance, a lot of work about multi-objective and multidisciplinary optimization design of fan blade have been performed[13-21]. Siller and Aulich [15] performed multidisciplinary aero optimization on a high pressure ratio fan stage with the consideration of static and dynamic mechanical constraints. Later, Aulich and Siller [16] discussed the features and usage of Kriging surrogate model and Bayesian neural network model at different optimization phases, and applied Pareto optimization strategy to the same fan stage as in [15]. Astrua and Piola [17] did multi-point aero-mechanical optimization on the transonic compressor blade, in which the ANN model and random walking algorithm were used, and their final optimization design fully fits the aero-mechanical requirements. Kang [18] built the compressor MDO process in which the genetic algorithm and ANN were implemented. Both the 3D aero-only and 3D aero-mechanical optimizations for a transonic fan were executed by Joly [19] by utilizing the differential evolution method. Li and He [20] developed a blading design optimization system using aero-mechanical approach and harmonic perturbation method. Later, Wang and He [21] developed and applied the adjoint method to concurrent blading aerodynamic and aeromechanic design optimization.

In conventional fan blade airfoil design practice, the pressure ratio and efficiency are first preliminarily determined for the design point, usually the cruise condition, and the mass flow ratio of ingested water in bypass duct is then calculated for the approach condition for the accessment of its rain ingestion performance. With this conventional approach, many iterations are usually needed before the ideal design results can be obtained, and such interations can be a considerable part of the design cost. From the above studies, it can be seen that most of the optimum design methods in all published papers for fan blades were focused on the fan blade's pressure ratio,efficiency and machanical performance, but not so much on the rain ingestion performance or the rain problem isn't even considered in those multi-objective and multidisciplinary optimization. Since the aerodynamic optimization of the engine fan rotor against water ingestion is a challenge to developments of aero-engine and a refined design of the fan rotor can lead to lower water concentration in the core engine, it is necessary to explore a design optimization method which can take into account the aerodynamic, machanical performance and the rain ingestion performance simultaneously to improve the performance of high bypass turbofan fan blades more efficiently. With consideration of that, a multi-point design optimization method is proposed in this paper. This paper represents a natural extension of the work by Qiu et al. [22] and explores the use of the multi-point optimization method to maximize the aerodynamic,mechanical and rain ingestion performace of fan blade at the same time. In this method, the Bezier curve is used to parameterize the fan blade, and the geometry parameters are used as the variables for the optimum design. The four different objectives are introduced to achieve best performance under different conditions. The two-phase-flow simulations of the fan rotor are computed using the ANSYS CFX. Finally, a nondominated sorting genetic algorithm II is put forward to solve this multi-objective optimization problem. With the optimization, the fan blade with a better performance can be obtained.

The objective of the current paper is to describe an fast optimization strategy to design fan rotor to reduce the level of water ingestion in the core engine system of a turbofan engine in the approach condition while maintaining the fan rotor aerodynamic performance and mechanical performance at cruise point. The first contribution of this work is that the aerodynamic, structure mechanic and rain ingestion are considered together for the first time in order to improve the comprehensive performance of the fan blade.The second contribution lies in the use of multi-point design optimization method to tackle the water ingestion problem.

This paper is organized as follows. In Section II the numerical simulation methods are presented. Section III gives an overview of multi-point optimization design procedure based on Kriging model and Section IV presents the computational results using the multi-point optimization algorithm. Finally, Section V concludes this paper.

2 Numerical Simulation Methods

The model simulated is a single passage flow of the transonic fan with core engine and bypass duct, which is shown in Fig.1. Due to the confidential issue, the detailed geometric and aerodynamic design parameters are not given in this paper, but the parameters are typical for the state-of-the-art civil fan design. Three different resolution grids with the identical topology are generated using AutoGrid for the entire computational domain. The mesh topology and resolution of these three grids are shown in Fig. 2. The details of fan model and grid independence of the computational grid for CFD simulation can refer to published work of Qiu et al. [22]

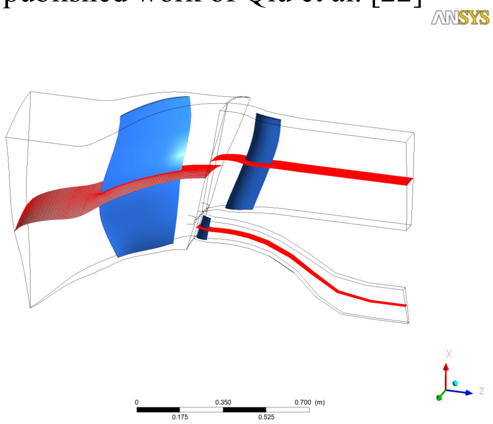
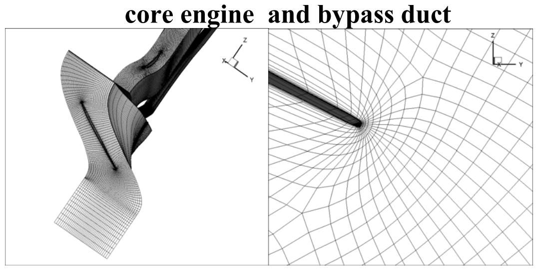


Fig. 1 a single passage flow of the transonic fan with core engine and bypass duct



(a) Medium grid (b) Tip clearance grid

Fig. 2 Computational grids for rain ingestion computation

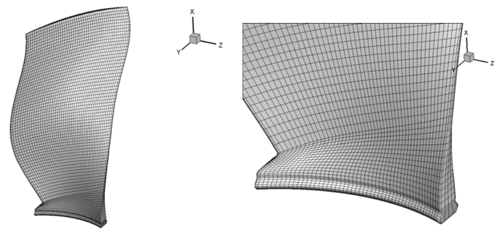
2.1 CFD approach for simulation of water drop in air flow

Computational Fluid Dynamics (CFD) is the basic methodology to examine the details of the flow in fan and how it is affected by the presence of water. In this study, a CFD investigation of water injection in a gas turbine inlet duct has been carried out. The numerical analysis has been performed by means of the commercial software ANSYS CFX 14.5. The standard k-ε turbulence model [23] has been activated to resolve the turbulent air-flow. The effect of the impaction of water drop on the blades and casing[24], and the process of rebound[25] and break up[26] of droplet are considered in this paper. Turbulence

interaction with particle trajectory calculation has also been included in the CFD simulation.
To model the particle trajectory, the Eulerian-Lagrangian approach is adopted for the continuous and discrete phases and the governing equations are solved in the rotating reference frame. The solver tracks the particle trajectories giving details about their behaviour in the flow field. What is more, the mass of water droplets and their position as they impact on the rotor blade are included in the flow field solution. This code makes it possible to solve the detailed turbulent thermal-flow-field in a given geometrical 3D domain. Furthermore, the presence of two-phase phenomena, as water injected in air, can be numerically reproduced. For this kind of problem, the equations of motion for the continuum phase (air) are solved based on a finite-volume approach, while the discrete phase (water) particles trajectories are calculated based on a Lagrangian approach. Mutual interactions between phases like momentum transfer have also been activated. The details of aerodynamic computation at cruise point, mechanical computation and rain ingestion computation is illustrated in detail in our paper[22][27].

2.2 Structure Simulations

Both the static stress analysis and the modal analysis are carried out using the commercial FEA software, ANSYS Mechanical. The mesh for the FEA is generated by an in-house tool FANE3D which is able to generate the hexa mesh for the fan blade with or without the arc dovetail root and shank. The mesh details of the fan blade with the arc dovetail root and shank are shown in Fig. 3. The total element number is about 17000. The details of structure simulation can refer to the work of Deng et al. [27].The static stress is evaluated at the red line rotation speed which corresponds to about 105% of the top of climb rotation speed. The eigenfrequencies of the first ten modes are calculated at 95%, 100% and 105% of the top of climb rotation speed, respectively. And the hot geometry is used for the structure simulation for avoiding a time-consuming hot-to-cold conversion. In each optimization cycle, the static pressure distribution on the blade surface from the CFD computation at the cruise operation point is transferred to FEA.



(a)The whole picture (b) the local region
Fig. 3 FEA mesh of the fan blade with the arc dovetail root and shank

2.3 The Optimization Design Based on Kriging model

2.3.1 Kriging Model

In this study, a multi-point optimization design process is carried out by using the commercial optimization software, ISIGHT 5.0. In the optimization process, the method of DOE used is the optimal Latin hypercube method, through which the sample points can be spread as evenly as possible within the design space. Kriging model[28] is selected as the surrogate model.

2.3.2 The Optimization formulation

2.3.2.1 The Multi-point optimization problem

It's well known that the aerodynamic concerned of the fan blade will change when a fan blade is in the different working conditions. In the design practice, considering the typical flight duty of the fan blade, the pressure ratio and efficiency in the cruise condition is evaluated for the aero performance, the maximum Von Mises stress on the pressure and suction surfaces of the fan blade at the red line rotation speed is computed for machanical performance consideration while the mass flow ratio of ingested water in bypass duct of the fan blade is assessed at approach condition for rain ingestion performance.

In this multi-point optimization problem, the aerodynamic performance of fan blade in cruise point, the mechanical performance at the approach condition and rain ingestion at climb condition are considered respectively.

The design parameters are the mentioned design parameters of the fan rotor, and the four different objective functions are :(1)maximize the pressure ratio and efficiency of fan blade in the cruise point.(2) minimize the maximum Von Mises stress on the pressure and suction surfaces of the fan blade at the red line rotation speed without sacrificing the fan rotor aero performance. (3) maximize the mass flow of ingested water in bypass duct without sacrificing the fan rotor aero performance.

Reasonable results are ensured by imposing constraints on the the pressure ratio(pr) and efficiency(eta) of the fan rotor. The objectives and constraints of designs are as follows:

(1)Maximize pr1 and eta1 in the cruise point

(2)Minimize the maximum Von Mises stress(maximumVonMisesstress2) on the pressure and suction surfaces

(3)Maximize the mass flow of ingested water(waterbypassmass3) in bypass duct

Subject to :

(1) eta1 $\geq$ $eta_{baseline1}$, pr1 $\geq$ $pr_{baseline1}$

(2)eta2 $\geq$ $eta_{baseline2}$, pr2 $\geq$ $pr_{baseline2}$

(3) eta3 $\geq$ $eta_{baseline3}$, pr3 $\geq$ $pr_{baseline3}$

2.3.2.2 Initial Search Space

In this work, the fan rotor design problems are formulated using the kriging-based method. The passage blade geometry of the transonic fan is generated by an in-house tool TFMAMA[27] which incorporates the through flow design, the 2D airfoil design and the radial stacking design of 2D airfoils. The design variables for the optimization are the swirl velocity (rVθ) distribution on the meridional channel of the blade and the spanwise distribution of the incidence angle, the maximum 2D airfoil thickness and the circumferential displacement of radial stacking. And the design variables of the swirl velocity distribution on the meridional channel of the blade are chord11, chord12, chord21, chord22, chord31, chord32, chord41 and chord42. And the design variables of the spanwise distribution of the incidence angle are span11, span12, span21, span22, span31, span32, span41 and span42. All those parameter range of design parameters can be seen in table 1. In the parameterization, the swirl velocity distribution is represented by the 3rd order Bezier surface function with 24 design freedoms. The radial distribution of incidence angle is described by the spline function with 8 freedoms. The final number of freedoms chosen in the optimization is 28.

Table 1 Parameter range of design parameters of the fan rotor

Design variables	Lower bound	Upper bound	Design variables	Lower bound	Upper bound
Chord11	0.2	0.5	span41	-0.2	0.0
Chord12	0.5	0.8	span42	0.8	1.2
Chord21	0.2	0.5	DC1	0.00744	0.01116
Chord22	0.5	0.8	DC2	0.0192	0.0288
Chord31	0.2	0.5	DC3	1.45E-4	2.17E-4
Chord32	0.5	0.8	DC4	-0.0516	-0.0344
Chord41	0.2	0.5	SPAN1	0.1312	0.1968
Chord42	0.5	0.8	SPAN2	0.4064	0.6096
span11	0.1	0.5	SPAN3	0.4752	0.7128
span12	0.8	1.3	TOB1	0.08976	0.1346
span21	0.1	0.5	TOB2	0.06	0.09
span22	0.8	1.3	TOB3	0.0344	0.0516
span31	0.1	0.5	TOB4	0.0168	0.0252
span32	0.8	1.3	TOB5	0.0176	0.0264

2.3.2.3 Search on Surrogate Models Using Genetic Algorithm

The multi-point optimization design process utilized in this paper is shown in Fig. 4. In this multi-point optimization, the design space sampling is performed first using latin hypercube method. Then aerodynamic and machanical computations are carried out at cruise condition and rain ingestion is computed at approach condition for all the sampling points. Base on these results, a initial surrogate model can be obtained for those four objectives. Then the optimization is executed within the prescribed design space based on the approximated results from the surrogate model. Once the optimum design is obtained, the aerodynamic, machanical and rain ingestion computation are performed to check the actual improvement of the design. Then, the surrogate model is updated with the new design, and the optimization is repeated. Every such cycle is a complete optimization run, and the cycle will be terminated once the pre-defined convergence criteria are met. In this study, such an optimization design process is carried out by using the commercial optimization software, ISIGHT 5.0. In order to obtain a more complete picture regarding the trade-off among pressure ratio, efficiency and water mass in bypass duct, the non-dominated sorting genetic algorithm (NSGA-II) [29]in ISIGHT 5.0 is implemented for the multi-point optimization.The population size, number of generations, crossover probability, crossover distribution index and mutation distribution index used in the optimization are 100, 50, 0.9, 10 and 20, respectively.

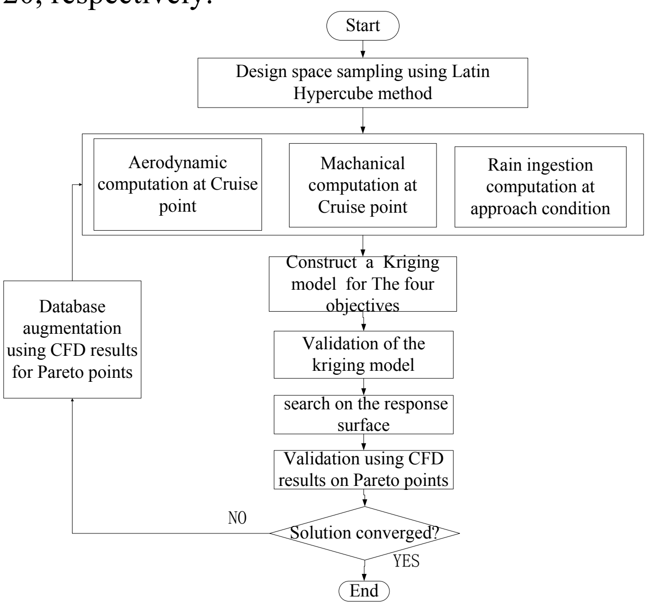


Fig.4 Procedure of multi-point optimization design

3 Results and Discussions

The optimization results from the medium grid computation are used for the detailed analyses.To understand how much potential aerodynamic, machanical and rain ingestion performance can be improved for the high bypass ratio fan, an multi-point optimization is carried out.

Considering the typical flight duty of the fan blade, the cruise condition is chosen as the evaluation point for the aero performance. Figure 5 shows predictions from the Kriging model, which illustrates the ability of the Kriging model and gives good agreement with CFD calculations. The pareto front of pressure ratio and efficiency in the multi-point optimization is marked in green circle in Fig. 6. One of the pareto front point in Fig. 6 is selected as the multi-point optimal design after the trade-off of the performance between the pressure ratio and efficiency. The blade geometry from the multi-point optimal design is compared with the baseline blade geometry, as shown in Fig. 7.

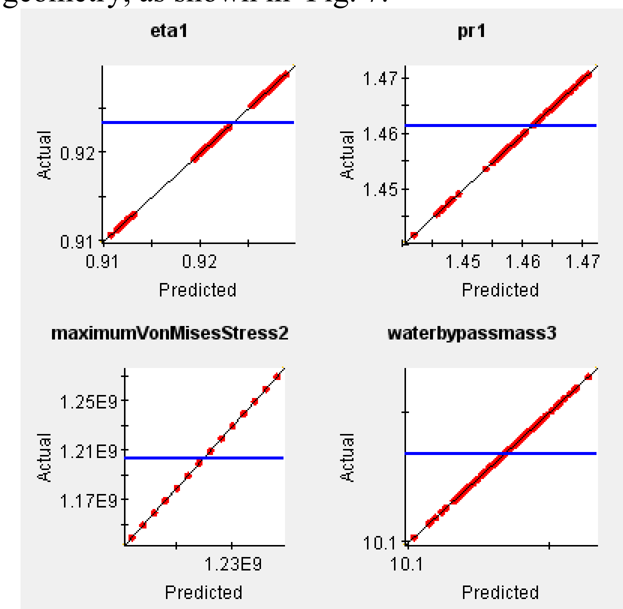


Fig. 5 Comparison of CFD values and predictions from the Kriging model on new points

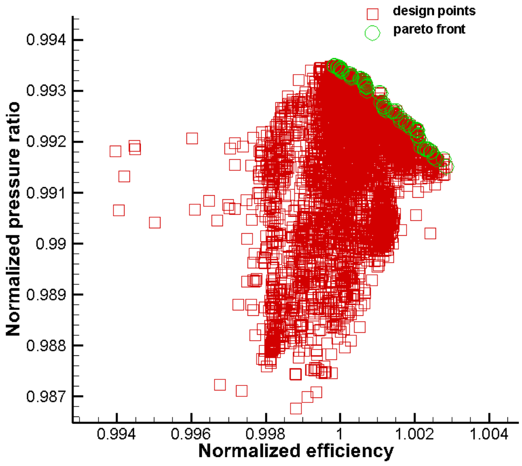


Fig. 6 Pareto front of the multi-point optimization

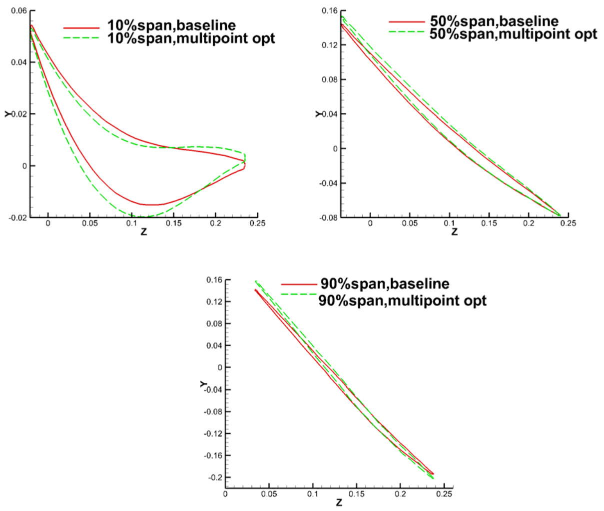
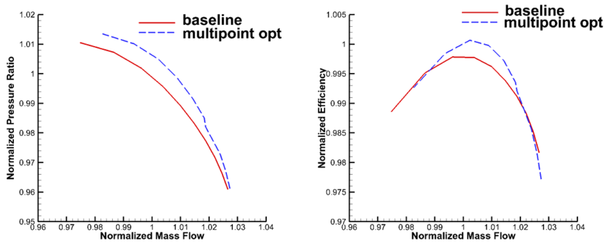


Fig. 7 Blade geometry for the baseline and multi-point optimal design

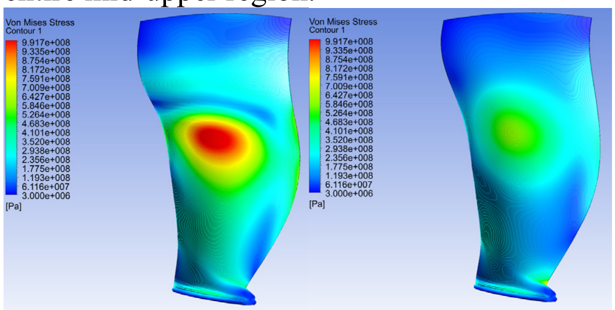
Compared with the baseline, the optimal geometry obtained through multi-point optimization has achievd a much better aerodynamic characteristics within the operation range between the 99% of the cruise mass flow rate point and the 102% of the cruise mass flow rate point. It can be seen from Fig.8 that the pressure ratio characteristic of the multi-point optimization is better than that of the baseline. With regard to the adiabatic efficiency characteristic, the multi-point optimization design improves the efficiency at the cruise operation point. About 0.73% pressure ratio improvement is gained at the cruise point, while about 0.1% efficiency improvement is gained at the cruise point. But at the same time, the optimized geometry has a worse efficiency characteristic ,namely the operation range between the stall point and the choke point shrinks for the multi-point design.



(a) Total pressure ratio (b) Adiabatic efficiency

Fig. 8 Comparison of the speedline between the baseline and the multi-point optimized fan blade at the cruise point

The static stress is evaluated at the red line rotation speed. Apart from the aero performance improvement, the multi-point optimization design also improves the mechanical performance by mitigating the maximum static stress on the fan blade, especially at the mid-upper region of the blade, as shown in Fig. 9. As the mechanical constraints are considered as an objective in the multi-point optimization design, Compared with the baseline design, the aero-mechanical optimal design not only reduces the maximum Von Mises stress from 995 MPa to 782 MPa, but also mitigates the stress level on the entire mid-upper region.

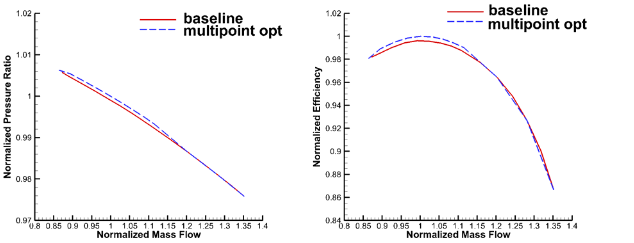


(a) suction surface of baseline

(b)suction surface of the multi-point design

Fig. 9 Distribution of Von Mises stress on the suction surfaces of the fan blade at the cruise point

The speedline of the multi-point optimal design at rain condition is also calculated on the medium grid and plotted in Fig. 10, together with the speedline of the baseline for comparison. Fig. 10a shows that the calculated pressure ratio of the multi-point optimal design at the rain condition is almost the same as that of the baseline. With the reduction of the flow rate, the pressure ratio of the multi-point optimization design and that of the baseline design both increase linearly. For the adiabatic efficiency characteristic in Fig. 10b, the multi-point optimal design shows higher efficiency than the baseline within the whole operation range. However, compared with the baseline design, the multi-point optimal design still achieves about 0.71% pressure ratio improvement is gained at the cruise point, and 0.15% pressure improvement at 95% of the cruise mass flow rate point, while about 0.08% efficiency improvement is gained at the cruise point, and 0.14% efficiency improvement at 95% of the cruise mass flow rate point.



(a) Total pressure ratio (b) Adiabatic efficiency

Fig. 10 Comparison of the speedline among the baseline, multi-point optimized fan blade at the rain condition

Figure 11 demonstrates the contour of the pressure gradient magnitude on the suction surface of the fan blade at the rain condition. It is clear to see that though the multi-point optimal design has similar shock structure to the baseline design, the strength of the shock wave still experiences a little bit decrease from 40% span to 80% span. No doubt, the multi-point optimal design exhibits the lowest shock wave strength within this range.

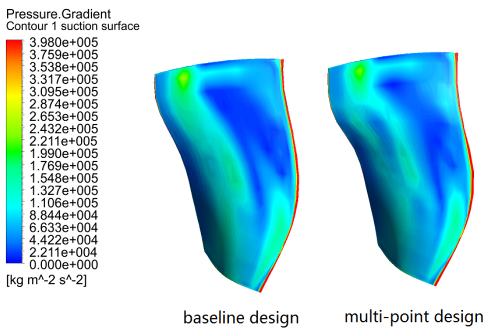


Fig. 11 Contour of pressure gradient magnitude on the suction surface of the fan blade at the rain condition

The aerodynamic, machanical and rain ingestion performance comparison of the initial and optimized fan blade is illustrated in Table 2. Due to the water mass objective function specifically designed for this operation range, the multi-point optimized geometry can achieve the most water mass ingestion in the bypass duct. It can be seen that about 2.1% improvement of water mass ingestion in bypass duct is obtained.

Table 2 performance comparison of the initial and optimized fan blade

Performance index	Aerodynamic performance at cruise point		Machanical performance at cruise point		Rain ingestion performance at approach condition	
Pr	baseline	1.455	/		/	
	Multi-point design	1.462				
Eta	baseline	0.921	/		/	
	Multi-point design	0.924				
Maximum Von Mises stress (pa)	/		baseline	0.995 E+09	/	
			Multi-point design	0.782 E+09		
the mass flow ratio of ingested water in bypass duct(kg/s)	/		/		baseline	89.1 %
					Multi-point design	91.2 %

5 Conclusion

A multi-point optimization method is proposed for the high bypass ratio fan blade in this paper. In addition to retaining the aerodynamic and machanical performance of the fan blade, maximizing the mass flow ratio of ingested water in bypass duct which represents the rain ingestion performance is also required in the optimization. With the introduction of all the 28 design variables in the design space, the multipal factors, i.e. pressure ratio,efficiency, Maximum Von Mises stress and the mass flow ratio of ingested water, are considered simultaneously, which effectively helps to obtain the satisfactory integrated performance regarding the trade-offs among 4 factors and thus evidently reduces the overall design cost of the multi-point design problem under different working conditions. According to the analysis of the results, several conclusions can be drawn as follows:

1. The multi-point optimization with the aerodynamic constraints improves the performance at both the cruise condition and lower mass flow rate condition. Compared with the baseline design, about 0.73% pressure ratio improvement is gained at the cruise point, while about 0.1% efficiency improvement is gained at the cruise point. But at the same time, the optimized geometry has a worse efficiency characteristic, namely the operation range between the stall point and the choke point shrinks for the multi-point design.

2. Apart from the aero performance improvement, the multi-point optimization design also improves the mechanical performance by mitigating the maximum static stress on the fan blade, especially at the mid-upper region of the blade. Compared with the baseline design, the multi-point optimal design not only reduces the maximum Von Mises stress from 995 MPa to 782 MPa, but also mitigates the stress level on the entire mid-upper region.

3. Due to the water mass objective function specifically designed for the multi-point optimization, 2.1% water mass can be ingested extraly in the bypass duct for the optimized fan blade.

Therefore, this method can be used to resolve the high-dimensional multi-point optimization problem in fan blade. It is also noticed that when high-bypass-ratio turbofans appeared with their transonic fans, the turbofan at takeoff and the helical speed is supersonic at the blade tip and there exists shock noise. Further numerical studies are investigated in that direction.

ACKNOWLEDGMENTS

This work is supported by National Science Foundation of China (Grants 51506189) and Science and Technology Commission of Shanghai Municipality under the project 13DZ2281200.

Copyright Statement

References

[1] Kissel G. Rain and Hail Extremes at Altitude, AIAA 79-0539, *15th AIAA Annual Meeting and Technical Display*, AIAA, Washington, DC,1979.

[2] Luers J. and Haines P. Heavy Rain Influence on Airplane Accidents, *Journal of Aircraft*, 20 (2), pp. 187-91,1983.

[3] Santa I. The Effect of Water Ingestion on the Operation of the Gas Turbine Engine, *22nd International Congress of Aeronautical Sciences*, Harrogate, United Kingdom,2000.

[4] Recommended practices for the assessment of the effects of atmospheric water ingestion on the performance and operability of gas turbine engines. *AGARD Advisory Report No. 332*.

[5] FAA.Water Ingestion Testing for Turbine Powered Airplanes, *FAA-AC-20-124*,1985.

[6] Williams J. and Young J. Movement of Deposited Water on Turbomachinery Rotor Blade Surfaces, GT2006-90792, *Proceedings of ASME Turbo Expo 2006 Power for Land, Sea and Air*, ASME, Barcelona, Spain,2006.

[7] Das K., Hamed A., and Basu D. Droplet Trajectories and Collection on Fan Rotor at Off-Design Conditions, GT-2006-91214, *Proceedings of ASME Turbo Expo 2006 Power for Land, Sea and Air*, Barcelona, Spain,2006.

[8] Williams, J C. *PhD Thesis*, University of Cambridge Department of Engineering.

[9] Day, I., Williams, J., Freeman, C. Rain ingestion in axial flow compressors at part speed. *ASME Turbo Expo 2005*. GT2005-68582，2005.

[10] Mund, F. C., and Pilidis, P. Effects of Spray Parameters and Operating Conditions on an Industrial Gas Turbine Washing System, *ASME Paper* N. GT2004-53551, 2004.

[11] Roumeliotis, I., Mathioudakis, K. Experimental Analysis of Wet Compression in Axial Compressor Stage, *European Turbomachinery Conference*, Lille, France, 2005.

[12] White, A.J., Meacock, A.J. An Evaluation of the Effects of Water Injection on Compressor Performance. *Proceedings of ASME Turbo Expo 2003*, GT2003-38237,2003.

[13] Jarrett, J. P., Daws, W. N., Clarkson, P. J. An Approach to Integrated Multi-disciplinary Turbomachinery Design, *ASME Journal of Turbomachinery*,127(3), pp.488-494, 2007.

[14] Luo, C., Song., L, Li, J., Feng, Z. Multiobjetive Optimization Approach to Multidisciplinary Design of a Three-dimensional Transonic Compressor Blade. *ASME* GT2009-59982, 2009.

[15] Siller, U., Aulich, M. Multidisciplinary 3D-Optimization of a Fan Stage Performance Map with Consideration of the Static and Dynamic Rotor Mechanics, *ASME* GT2010-22792, 2010.

[16] Aulich, M., Siller, U. High-Dimensional Constrained Multiobjective Optimization of a Fan Stage, *ASME* GT2011-45618, 2011.

[17] Astrua, P., Piola, S., Bonzani, Federico. Multi-objective Constrained Aero-Mechanical Optimization of an Axial Compressor Transonic Blade, *ASME* GT2012-68993, 2012.

[18] Kang, Y.,et al. Multi-Disciplinary Design Optimization and Performance Evaluation of a Single-Stage Transonic Axial Compressor, *ASME* GT2012-69252, 2012.

[19] Ioly, M., Verstraete, T., Paniagua, G. Full Design of a highly Loaded Fan by Multi-Objective Optimization of Through-Flow and High-Fidelity Aero-Mechanical Performance, *ASME* GT2012-69686, 2012.

[20] Li, H., He, L., Li, Y., Wells, R. Blading Aerodynamics Design Optimization with Mechanical and Aeromechanical constraints, *ASME* GT2006-90953, 2006.

[21] Wang, D. X., He, L. Concurrent Aerodynamic-Aeromechanic Design Optimization for Turbomachinery Blades Using Adjoint Method”, *ASME* 2009-59240, 2009.

[22] Qiu, S., Deng,X.Y., Zhang,X., Huang,T.H. Design optimization of a high bypass ratio fan rotor against water ingestion. *22nd AIAA Computational Fluid Dynamics Conference*, 22-26 June 2015, Dallas, TX. AIAA 2015-3063,2015.

[23] Jones, W. P., and Launder, B. E. The prediction of laminarization with a 2-equation model of turbulence, *Int. J., Heat Mass Transf.* 15:301, 1972.

[24] Bai, C. and Gosman, A.D. Prediction of spray wall impingement in reciprocating engines, *ILASS-Europe,* July 1999.

[25] Elsässer, G. Experimentelle Untersuchung und numerische Modellierung der freien Kraftstoffstrahlausbreitung und Wandinteraktion unter motorischen Randbedingungen, *Dissertation*, Logos Verlag, Berlin, 2001.

[26] Pilch, M. and Erdman, C.A. Use of Breakup Time Data and Velocity History Data to Predict the Maximum Size of Stable Fragments for Acceleration-Induced Breakup of a Liquid Drop, *Int. J. Multiphase Flow,* Vol. 13, No. 6, pp. 741-757, 1987.

[27] Deng, X.Y., Guo, F.S., Liu, Y.S., Han, P.L. Aero-mechanical optimization design of a transonic fan blade. *GT2013-95357*, 2013.

[28] Cressie, N. A. C. Statistics for Spatial Data, Revised Edition, *John Wiley & Sons*, New York, 1993.

[29] K. Deb, S., Agrawal, A., Pratab, T., Meyarivan. A Fast Elitist Non-Dominated Sorting Genetic Algorithm for Multi-Objective Optimization: NSGA-II. In KanGAL report 200001, *Indian Institute of Technology*, Kanpur, India, 2000.