STUDY ON AERODYNAMIC PERFORMANCE IN TURBOMACHINES BASED ON FLUID-STRUCTURE COUPLING METHOD

Yongqiang Shi[1,2]*, Haolong Liu[3] and Qingzhen Yang[1]
*Author for correspondence
[1]School of Power and Energy/ National Key Laboratory of Aerodynamic Design and Research, Northwestern Polytechnical University, Xi’an, 710072, China
[2] Faculty of Engineering, Lund University, SE-22100 Lund, Sweden
[3] College of Energy & Power Engineering, Nanjing University of Aeronautics and Astronautics, 210016, China
E-mail: yqshi@nwpu.edu.cn

ABSTRACT

With the development of the modern turbomachinery design technology, the effects of fluid-structure coupling on aerodynamic performance and structural dynamic characteristics cannot be ignored. Therefore, the structural dynamic characteristics and aerodynamic performance is numerically simulated and studied by using the computational fluid dynamics and computational structural dynamics methods in present work. The aerodynamic data and deformation displacement data are transferred between CFD and CSD by linear interpolation and Constant-Volume Tetrahedron transformation respectively. Through comparing the aerodynamic force contour and displacement contour level on aerodynamic grids and structural grids, the data transfer approach is verified. Then, the fluid-structure coupling characteristics in a fan rotor are analyzed. The results show that, the stress, deformation displacement is affected by fluid-structure interaction, but vibration frequency is changed slightly under the action of aerodynamic force; the mass flow, total pressure ratio and adiabatic efficiency at design rotation speed are affected by fluid-structure interaction remarkably. More interesting, the choking flow rate of coupling computation is greater than that obtained by uncoupled analysis, but it will be slightly decrease with the total pressure ratio decreasing. Moreover, as the rotate speed decreasing, the coupling effect on mass flow, total pressure ratio and adiabatic efficiency are reduced.

INTRODUCTION

The fan rotor of aero-engine withstands aerodynamic loads, but it also withstands the centrifugal force from high rotation speed. Those forces certainly make blades deformed and affect the shape of the flow channel. The shape shifting will cause the loads to redistributing. So, the blades shape will be balanced by aerodynamic and structural forces.

The deformation of blades by the coaction of aerodynamic and structural forces cannot be ignored in modern turbomachinery design. From the perspective of structural dynamics, the blades would be distorted by the action of aerodynamic force, therefore, it should be considered in fatigue strength analysis and prediction of fatigue life. Meanwhile, from the perspective of aerodynamics, because of blades distortion, the shape of flow channel will be changed, which cause the flow field parameters changed, and lead to differences between design and actual performance. Therefore, analysis of fan characteristics based on coupling of CFD and CSD, research on the influence of aeroelasticity on structural dynamics and aerodynamic performance, which is of significance for renovation of modern turbomachinery design technique.

One of the key techniques for fluid-structure interaction is data exchange between CFD and CSD. Due to the different objects dealt by CFD and CSD in coupling calculation, the meshes adopted by them differ greatly. In the fluid-structure coupling analysis, while the flow field around the solid surface is concerned in CFD, the loads on the solid surface and the influence of them on internal structure are paid close attention in CSD. Hence, the grid density of the solid structure and flow field will be different on solid surface, which requires information exchange on solid surface between them in the calculation.

The data exchange between the structural domain and the flow domain is usually performed by interpolation. The interpolation method is one of the factors restricting the accuracy of aeroelastic simulation [1]. The quality of data exchange directly affects the accuracy of fluid-structure coupling calculation, which is one of the technical problems in development of the fluid-structure interaction. Therefore, the fluid-structure interfaces were studied, and linear interpolation method, and Constant-Volume Tetrahedron(CVT) approach for data transfer were developed in present work. And then, the solver for three-dimensional flow field and the software for structural dynamics analysis were combined to develop an analysis system for fluid-structure interaction in turbomachinery, and a fan rotor was calculated by it. At last, the aerodynamic and structural performance of the fan by coupling and uncoupling approach was compared.

ANALYSIS METHODOLOGY

In order to predict the response of a flexible structure in a fluid flow, the structural motion equations and the fluid equations must be interacted. However, in this paper, merely the structural and aerodynamic characteristics of fan at the ultimate equilibrium positions which are caused by the centrifugal force and aerodynamic force are concerned. So, the procedure of fluid-structure coupling is advanced with interaction by solving the steady flow filed and the statics balance equations respectively which exchange information on the blade surface. In present work, the results of flow field are obtained by a steady Navier-Stokes solver and the structural computation is performed by commercial software.

Flow Field Computation

The conservative differential form of the three-dimensional Navier-Stokes equations in arbitrary curvilinear coordinate system can be written as

$$\frac{\partial \mathbf{Q}}{\partial t}+\frac{\partial \mathbf{E}}{\partial \xi}+\frac{\partial \mathbf{F}}{\partial \eta}+\frac{\partial \mathbf{G}}{\partial \zeta}=\frac{\partial \mathbf{E}_v}{\partial \xi}+\frac{\partial \mathbf{F}_v}{\partial \eta}+\frac{\partial \mathbf{G}_v}{\partial \zeta}+\mathbf{K} \tag{1}$$

where, $\mathbf{Q}=J^{-1}\left[r\rho \quad r\rho u \quad r\rho v \quad r\rho w \quad re\right]^T$,E , F and G are inviscid flux vectors, Ev, Fv, and Gv are viscous flux vectors; **K** is source term. **J** is Jacobian matrix, ρ is density, u , v and w are absolute velocity components in direction φ 、 r and z, respectively, and e denotes the total internal energy, and for details see [2].

The solution of Eq.1 is obtained using time-marching method, and the 3rd order accuracy NND scheme [3] is applied to pursuing numerical approximation of the spatial derivative, and implicit LU-SGS scheme [4] is used for temporal discretization. Besides, the Badwin-Lomax turbulence model [5], which is widely used for numerical solution of turbulent flow in turbomachinery, is adopted in this work.

Transformation Methodology

In the process of fluid-structure coupling, the solution of flow filed provides aerodynamic loads to the structure computation for the structure solver to calculate the structural deformation. In return, the solution of statics balance equations provides the surface deformation to the flow solver in order to reform the mesh and to change the flow filed. Hence, the pressure distribution on blade surface which is obtained by Computational Fluid Dynamics need to be transferred from fluid grids to structural grids. Meanwhile, the deformation displacements of blades which are obtained by structural computation need to be interpolated from structural grids to fluid grids.

In order to perform a coupled multidisciplinary analysis, a series of data exchange methods for fluid-structure interaction have been proposed by scholars all over the world, such as surface spline interpolation [6], the multiquadric-biharmonic method [7] , boundary element method [8], the Constant-Volume Tetrahedron(CVT) approach [9], and so on. In 2004, a detailed comparison research was done among these data exchange methods mentioned above by M. Sadeghi and F. Liu et al [10]. In order to exchange data between CFD and CSD, two interpolation techniques are adopted in this work: when transferring the pressure distribution on the fluid surface mesh to a corresponding nodal load distribution on the structure, linear interpolation methods(LIM) are employed; and when transferring the displacements of the structural nodes to corresponding perturbations of the fluid mesh, the improved CVT methods [11] are adopted.

In CVT method, each node $\vec{x}_a$ on aerodynamic grid is associated with the three closest non- colinear structural nodes $\vec{x}_{s,1},\vec{x}_{s,2},\vec{x}_{s,3}$ as shown in 错误!未找到引用源。. To calculate the displacement of the aerodynamic node, the three closest structural nodes should be identified firstly. Once the structural nodes associated with the aerodynamic node are identified, the position of the aerodynamic node can be given by the expression

$$\mathbf{c}=\alpha\mathbf{a}+\beta\mathbf{b}+\gamma\mathbf{d} \tag{2}$$

where, $\mathbf{a}=\vec{x}_{s,2}\text{-}\vec{x}_{s,1}$, $\mathbf{b}=\vec{x}_{s,3}\text{-}\vec{x}_{s,1}$, $\mathbf{c}=\vec{x}_a\text{-}\vec{x}_{s,1}$, $\mathbf{d}=\mathbf{a}\times\mathbf{b}$.

α, β, γ are constant and can be calculated as

$$\alpha=\frac{(\mathbf{b}\cdot\mathbf{b})(\mathbf{a}\cdot\mathbf{c})\text{-}(\mathbf{a}\cdot\mathbf{b})(\mathbf{b}\cdot\mathbf{c})}{(\mathbf{a}\cdot\mathbf{a})(\mathbf{b}\cdot\mathbf{b})\text{-}(\mathbf{a}\cdot\mathbf{b})(\mathbf{a}\cdot\mathbf{b})} \tag{3}$$

$$\beta=\frac{(\mathbf{a}\cdot\mathbf{a})(\mathbf{b}\cdot\mathbf{c})\text{-}(\mathbf{a}\cdot\mathbf{b})(\mathbf{a}\cdot\mathbf{c})}{(\mathbf{a}\cdot\mathbf{a})(\mathbf{b}\cdot\mathbf{b})\text{-}(\mathbf{a}\cdot\mathbf{b})(\mathbf{a}\cdot\mathbf{b})} \tag{4}$$

$$\gamma=\frac{\mathbf{c}\cdot\mathbf{d}}{\mathbf{d}\cdot\mathbf{d}} \tag{5}$$

And then, coordinate position of the aerodynamic node can be written as

$$\vec{x}_a=\vec{x}_{s,1}+\alpha\mathbf{a}+\beta\mathbf{b}=\sum_{i=1}^{3}\phi_i\vec{x}_{s,i} \tag{6}$$

where $\phi_1=1-\alpha-\beta$, $\phi_2=\alpha$, $\phi_3=\beta$.

The volume of the tetrahedron which is formed by the structural nodes and the aerodynamic node is given by

$$V=\frac{(\mathbf{a}\cdot\mathbf{b})\times\mathbf{c}}{6} \tag{7}$$

As the structure deforms, the values of the structural nodes change. The structural vectors **a**, **b** and therefore **d** are known, but the new aerodynamic node has to be calculated from equation (2). with the following approach. The projection of the aerodynamic node is forced to move linearly in the structural triangle by fixing α and β at their initial values. The value of γ can be calculated by a particular extrapolation approach

$$\gamma=\frac{\mathbf{d}_0\cdot\mathbf{d}_0}{\mathbf{d}\cdot\mathbf{d}}\gamma_0 \tag{8}$$

which ensures a constant volume of the tetrahedron spanned by **a**, **b** and **c**.

This is the primary principle of Constant-Volume Tetrahedron method [12].

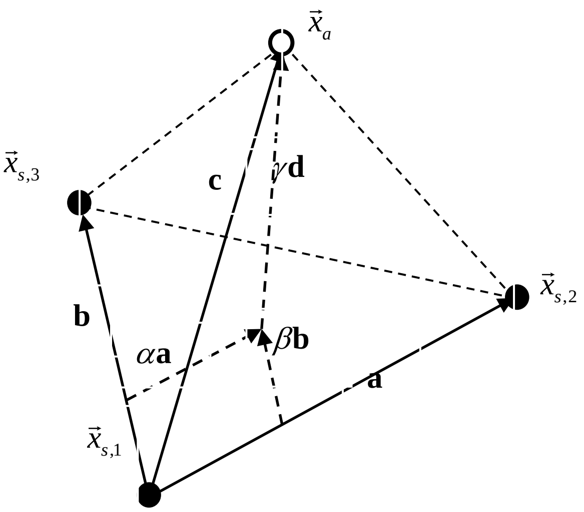

Figure 1 Constant-volume tetrahedron

RESULTS AND ANALYSIS

In this study, the aerodynamic performance and structural characteristics of a transonic fan rotor were numerical analysed by using above mentioned methods.

Calculation Condition

The rotor design pressure ratio is 1.787 at a mass flow of 83.55kg/s. The design rotation velocity is 1122.596rad/s; inlet total pressure P_0=101325Pa; total temperature T_0=288.16K, and Pb=94700.0Pa. The fan blade is made of titanium alloy elastic modulus E=102GPa and Density ρ=4850 kg/m^3. The performance was numerical simulated as relative rotating speed of 1.0, 0.95, 0.85, 0.7 and 0.6 respectively in this work. The convergence condition is that the error of the maximum displacement on the blade surface obtained by adjacent iterations is less than 10^{-6}m.

Validating the Data Transfer Method

The structural analysis is done with 9102 nodes, 5840 elements and 41×74×3 grids in chordwise, radial directions and thickness directions respectively in blade finite element model. The aerodynamic mesh density is 31×31×81 in pitchwise, radial directions, streamwise, respectively. the finite element mesh and the aerodynamic grids are show in Figure 2. We can find that the mesh adopted by CSD and by CFD is quite different. Thus, if the effect of interpolation is not improved, the numerical simulation precision will be directly impacted.

In present paper, the linear interpolation methods(LIM) and Constant-Volume Tetrahedron(CVT) approach are adopted to exchange information between CSD and CFD. From Figure 3 and Figure 4, we can know that, LIM and CVT have high precision to meet the information exchange demand respectively.

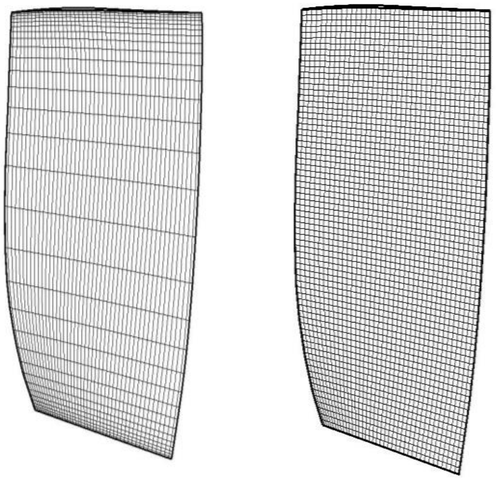

Figure 2 Surface mesh for aerodynamic computation and structural computation

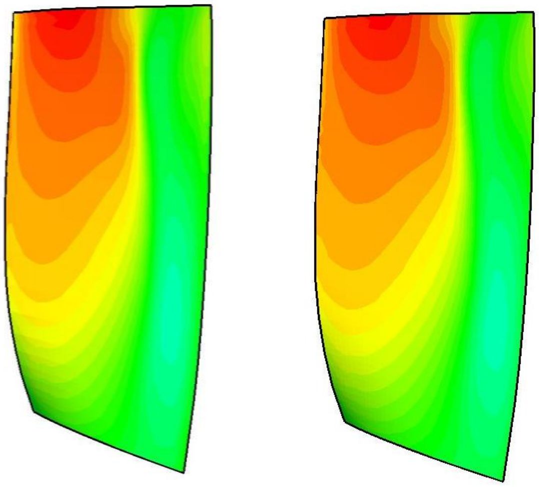

Figure 3 Contours of the pressure on pressure and suction surface of a fan blade before and after interpolation

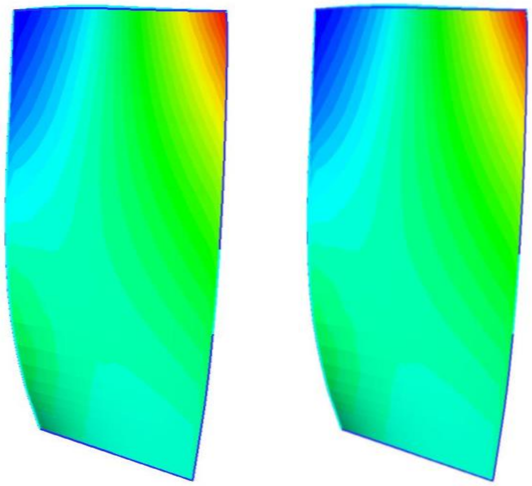

Figure 4 Contours of the deformation displacements of a fan blade before and after interpolation

Computation Results

The coupled aerodynamic performance and structural dynamic characteristics of the fan are obtained by coupling CFD and CSD; also the uncoupled results were obtained.

The maximum values of the deformation displacement and stress of CEFO (the uncoupled results merely considering the effects of centrifugal force on deformation) and STAE (the coupled results considering the coaction of aerodynamic force and centrifugal force) at various rotation speed are shown in Table 1 and Table 2, respectively.

The Tables shows us that, the numerical results are different between CEFO and STAE. and the deformation displacements should reduce more than 10% when considering the effects of aerodynamic force, while the maximum stress reduce about 4%. Obviously, the effects of aerodynamic force on structural characteristics are considered in the fan design, resulting in reduction of the deformation displacement and stress, improving the blade mechanics properties. In addition, the centrifugal force will reduce along with the decrease of rotation speed, which leads to the weakness of predominance for centrifugal force in the influence on the blade mechanics properties.

Table 1 The maximum deformation displacement on blade surface at various rotation speed

$\overline{n}$	DCEFO (mm)	DSTAE (mm)	Relative Variation (%)
1.0	9.126	8.128	-10.94
0.95	8.652	7.685	-11.18
0.85	7.688	6.826	-11.21
0.7	6.414	5.532	-13.75
0.6	5.551	4.778	-13.92

Table 2 The maximum stress at various rotation speed

$\overline{n}$	FCEFO (GPa)	FSTAE (GPa)	Relative Variation (%)
1.0	0.93533	0.89867	-3.92
0.95	0.85679	0.82087	-4.19
0.85	0.70775	0.67460	-4.68
0.7	0.50430	0.48039	-4.74
0.6	0.38455	0.36577	-4.88

Table 3 The vibration frequency at various rotation speed for coupled and uncoupled computation

$\overline{n}$		1ST (Hz)	2ND (Hz)	3RD (Hz)
1.0	CEFO	334.99	661.59	836.49
	STAE	335.45	662.84	838.12
0.95	CEFO	322.79	647.73	834.22
	STAE	323.18	648.81	835.75
0.85	CEFO	298.71	620.74	830.01
	STAE	298.93	621.24	830.39
0.7	CEFO	263.76	582.76	824.74
	STAE	263.76	582.80	824.12
0.6	CEFO	241.62	559.75	821.33
	STAE	241.51	559.67	820.80

The blade dynamic frequency of first three mode at various speed is given in the Table 3. As is shown, the values of frequency obtained by uncoupled and coupled computation are nearly equal. Besides, the coupled results are greater than uncoupled ones at higher rotation speed, while these are slightly less than uncoupled ones at lower rotation speed, which also reveals that the effects of aerodynamic force on blade mechanics properties are different at various rotation speed.

In **Figure 5**, the fan characteristic curves of RIGID (the uncoupled results with regarding blade as rigid), CEFO and STAE at design rotation speed are demonstrated.

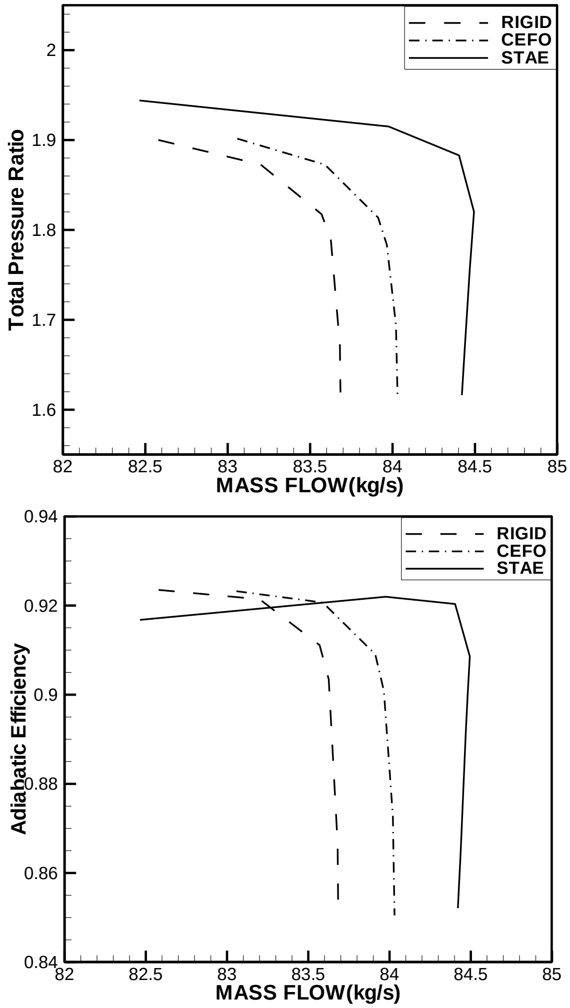

Figure 5 Characteristic curves of the fan at design rotation speed

From the figures they can be observed that the choking flow rate from coupled calculation is higher than those from the two uncoupled calculations. The choking flow rate from STAE is about 0.879% higher than those from RIGID. It is worth noting that, the choking flow rate is not a constant value but a slight decrease in coupled calculation along with the decrease of pressure ratio.

The fan characteristic curves at various rotation speed are present in Figure 6. Consistent with the above conclusion, the choking flow rate from STAE is higher than those from RIGID, but it is nearly equal in lower rotation speed.

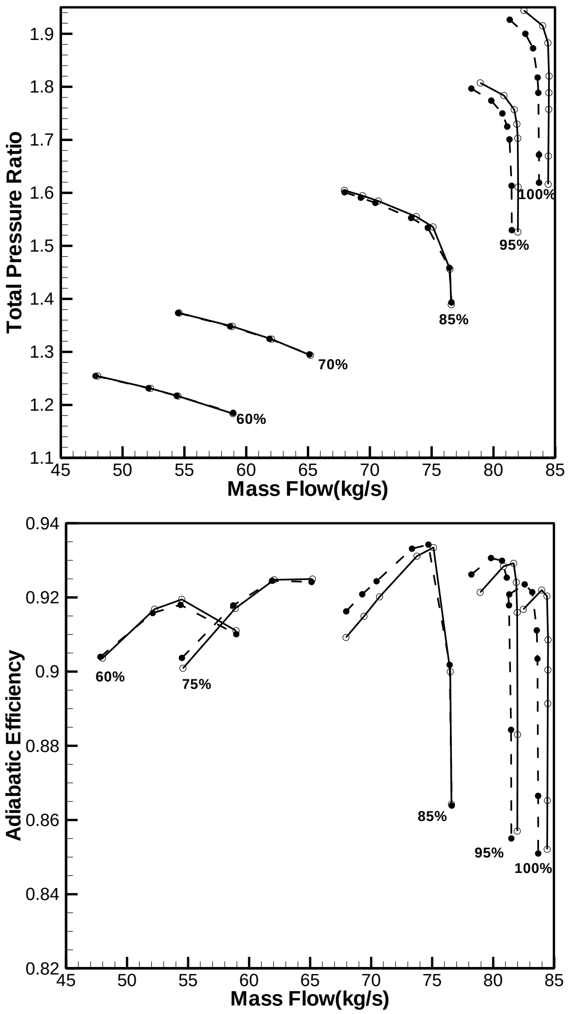

Figure 6 Characteristic curves of the fan (solid line for STAE, and dotted line for RIGID)

CONCLUSION

In this paper, a three-dimensional N-S solver and a commercial software were used for aerodynamic and structural analysis to represent flow field and structure deformation accurately; while a linear interpolation method and Constant-Volume Tetrahedron approach were developed to exchange data between CFD and CSD in a high-fidelity way; and then a computation system for fluid-structure interaction is developed to analyse aerodynamic performance and structural characteristics of a fan rotor.

Numerical results indicate that the blade deformation and stress are affected by fluid-structure interaction, and the effects of aerodynamic force on deformation and stress will be strengthen along with decrease of the rotation speed, but the vibration frequency is hardly affected by FSI. Hence, the effect of fluid-structure interaction should be considered in the design of blade with high aspect ratio and poor rigidity.

The mass flow at design rotation speed obtained by coupled computation is higher than that from uncoupled computation, but the coupling effects will reduce with the decrease of rotation speed. The pressure ratio and the maximum of adiabatic efficiency from coupled results are higher than uncoupled ones, especially at high rotation speed, and the point of maximum efficiency is shifted in coupled calculation. Therefore, the effect of fluid-structure interaction on aerodynamic performance should be considered in the fan design.

ACKNOWLEDGEMENT

This work was supported by the Ph.D. Programs Foundation of Ministry of Education of China (No.20126102120042), National Natural Science Foundation of China (No.11302179) and National Key Laboratory Fund.

REFERENCES

[1] Xu. M., Chen, S. L., Study of Date Exchange Metod for Coupling Computational CFD/CSD, *Chinese Journal of Applied Mechanics*, Vol.21, 2004, pp.33-36.

[2] Yang, Q.Z., Shi, Y.Q., etc., Numerical analysis of three-dimensional unsteady flow around oscillating cascades, *Acta Aerodynamica Sinic*, Vol.24, 2006, pp.62-66.

[3] Zhang, H. X., Nonoscillatory, Non-free-parameter Dissipation Scheme, *Acta Aerodynamica Sinica*, Vol. 6, 1988, pp143-165.

[4] Yoon, S., Lower-Upper Symmetric-Gauss-Seidel Method for the Euler and Navier-Stokes Equations, *AIAA Journal*, Vol. 26, 1988, pp.1025-1026.

[5] Baldwin, B. S., and Lomax, H., Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows. *AIAA paper 78-257* ,1978.

[6] Harder, R. L. and Desmarais, R. N., Interpolation using surface splines, *Journal of Aircraft*, Vol. 9, 1972, pp.189-191.

[7] Hardy, R. L., Theory and applications of the multiquadric-biharmonic method, *Computers & Mathematics with Applications*, Vol. 19 ,1990, pp.163-208.

[8] Chen, P. C., Jadic, I., Interfacing of fluid and structural models via innovative structural boundary element method, *AIAA Journal*. Vol. 36,1998, pp.282-287.

[9] Goura, G. S. L., Badcock K. J. etc., A data exchange method for fluid-structure interaction problems, *The Aeronautical Journal*, Vol. 105, 2001, pp.215-221.

[10] Sadeghi, M., Liu, F., Lai, K. L., Tsai, H. M., Application of Three-Dimensional Interfaces for Data Transfer in Aeroelastic Computations, *22nd Applied Aerodynamics Conference and Exhibit*, Providence, Rhode Island, August 2004.

[11] Shi, Y. Q., Wang, Q. Q., and Yang, Q. Z., Analysis of Fluid-Structure Coupling Characteristics in Fan/compressor rotor, *Computer Simulation*, Vol. 28, 2011, pp.115-119.

[12] Badcock, K. J., Rampurawala, A. M., and Richards, B. E., Intergrid Transformation for Aircraft Aeroelastic Simulations, *21st Applied Aerodynamics Conference*, Orlando, Florida, June 2003.