氏名	むらやま みつひろ 村山 光宏
授与学位	博士（工学）
学位授与年月日	平成15年3月24日
学位授与の根拠法規	学位規則第4条第1項
研究科，専攻の名称	東北大学大学院工学研究科（博士課程）航空宇宙工学専攻
学位論文題目	Study of Numerical Method for Airplane Flight Simulation Using Unstructured Mesh CFD （航空機飛行模擬のための非構造格子 CFD を用いた数値計算法に関する研究）
指導教官	東北大学教授 中橋 和博
論文審査委員	主査 東北大学教授 中橋 和博 東北大学教授 澤田 恵介 東北大学教授 小濱 泰昭 東北大学助教授 松島 紀佐 東北大学助教授 大林 茂 （情報科学研究科）

論文内容要旨

Chapter 1 Introduction

Currently, CFD has been extensively used in the design process of airplanes, instead of the experimental approach. The next targets of CFD for aircraft developments will be analyses of more detailed aircraft configurations including small parts, more accurate analyses at off-design conditions, unsteady flow simulations with moving bodies and body deformations. Key research fields of CFD expected to be developed are as follows;

CFD for Accurate Prediction of Vortical Flows: Flow around a delta wing adopted for high-speed transports is governed by the leading-edge separation vortices at high incidence conditions in take-off and landing. Wing-tip vortices of airplanes during the take-off and landing are another serious problem. They will remain in the wake regions for a long time and affect following airplanes. In conventional CFD techniques, however, vortices are easily diffused by inherent numerical dissipations. Developments of effective methods to reduce these dissipations and accurately resolve vortices around a full airplane configuration are expected.

CFD for Moving Bodies: Numerical predictions of unsteady aerodynamic forces of moving bodies are still under study. For the safety of aircraft, flutter boundary, maneuver loads and gust loads should be estimated accurately. Since experiments for such problems involve high risk and cost, CFD is expected to play an important role. To treat moving body problems by CFD, a dynamic mesh method, where the computational mesh is moved/deformed accompanied by the body movement and deformation, is required. In the design fields, the dynamic mesh method is also very useful to evaluate modified configurations quickly. However, those dynamic mesh methods especially for the unstructured mesh may often lack robustness necessary for large deformation problems. Therefore, a robust and efficient dynamic mesh method is required for moving body problems.

CFD for Complete Flight Simulation of Airplane: Currently, the analysis of the maneuvering aircraft is obtained by expensive flight tests. If CFD can replace the flight test, the airplane development cost can be reduced greatly.

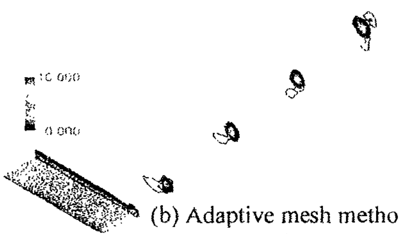
One of the most important issues for flight dynamics is the stability control of longitudinal motion of an airplane, especially trim. For trim, the aerodynamic forces are required as the function of control surface deflections at varieties of flight conditions. In addition, for the dynamic stability control, the dynamic response to the unsteady control surface deflection is required. Such wind tunnel tests will be extremely difficult and expensive. For these airplane maneuver simulations, aircraft responses to control surfaces must be computed. However, very few reports are available for the aircraft response to control surface deflections by CFD.

Objectives: The objective of this thesis is to develop the numerical methods that are required to simulate flows around an airplane in various flight conditions. The final target of this research is to realize a virtual flight test of an airplane in a computer. To achieve this goal, the numerical methods to accurately simulate vortical flows appearing at take-off and landing conditions, to handle moving bodies and to simulate unsteady flows of maneuvering airplane in response to control surface deflection have been developed. The Euler/Navier-Stokes unstructured mesh method is employed as the basic CFD approach because of the topological flexibility to generate high fidelity mesh around detailed and complex configuration of a complete aircraft. The validity of the present methods is discussed by examining the computational results. Finally, the possibility to achieve a complete airplane flight simulation based on high fidelity CFD is discussed.

Chapter 2 Unstructured Mesh CFD for Vortical Flows

In Chapter 2, two approaches to improve the accuracy of vortical flow simulations were discussed. Adaptive mesh refinement method coupled with the vortex center identification method was briefly reviewed. Then, the vorticity confinement method to reduce the diffusive property of the vortical flow simulations has been introduced on unstructured meshes. The effectiveness has been discussed by applying to simulations of four problems of vortical flows. In the case of a single vortex in a uniform flow, the effect of the vorticity confinement on unstructured mesh was validated and a difficulty in determining its coefficient because of the mesh dependency was pointed out. In the case of vortical flows around delta wings, the confinement was found more effective than the adaptive mesh refinement method. In the vortex breakdown case, it was demonstrated that the use of excessive confinement parameters could suppress the breakdown and thus destroy the flow physics. Finally, in the case of a wing tip vortex of NACA0012 wing shown in Fig. 1, it was proven that the confinement method could suppress the numerical diffusion of the vortex far away from the trailing edge by coupling with the adaptive mesh refinement method.

(a) Conventional method



(b) Adaptive mesh method & vorticity confinement method

Fig.1 Contours of the vorticity magnitude

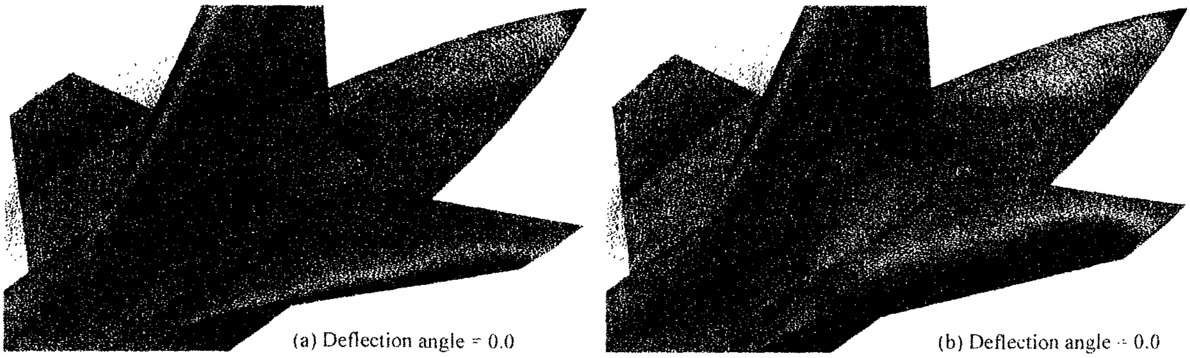
The results obtained in this chapter show that, although further studies will be required to reduce the dependency of the confinement coefficient on the mesh density, the vortex confinement method coupled with unstructured and adaptive refinement meshes may have a possibility of accurate simulations of vortical flows, especially in the simulation of the wing tip vortices. So far, however, if we can use computational resources enough, the adaptive mesh refinement method is more reliable approach. We must keep in mind the vorticity confinement method as a model to simulate vortical flows economically and must use the method carefully according to specific problems.

Chapter 3 Unstructured Dynamic Mesh Method

In Chapter 3, an unstructured dynamic mesh method has been developed to compute flow problems with moving and deforming body. The dynamic mesh is required for numerical simulations of flights of a maneuvering airplane including aircraft response to control surface deflection and aeroelasticity. A simple and robust unstructured dynamic mesh method using spring analogy concepts was proposed where a mechanism to keep each elemental shape of triangular, tetrahedral, and hybrid (tetrahedra, prisms, and pyramid) meshes was introduced. The method was applied to several problems and compared with the conventional dynamic mesh method. Using the conventional method, mesh movement failed in several cases. While, using the proposed method, the mesh could successfully be moved without a large penalty in CPU time. Therefore, it was demonstrated that the present method would improve the robustness for flow problems with large motions of bodies. These results showed a great possibility for the proposed dynamic mesh method to be applied to various kinds of fluid and design problems.

Chapter 4 Unstructured Dynamic Mesh Method with Surface Mesh Movement

In Chapter 4, an approach to handle surface mesh movement on a curved surface was proposed. To compute maneuver of an aircraft in response to control surface deflection, a special treatment of the computational mesh is required for the moving control surface, for example, a horizontal tail wing. Because of the change in location of a horizontal tail wing on a fuselage surface, the surface mesh of the tail wing–fuselage juncture should be modified. In general moving body problems, the dynamic mesh method of the three-dimensional volume mesh is required to treat the body movement. On the other hand, to treat the change of the deflection angle of the control surface, surface mesh points of the fuselage also have to be moved while maintaining the original curved surface of the fuselage due to the change of the intersection. However, it is difficult to directly move the surface mesh points along the curved surface without changing the original topology and geometry. Therefore, a mapping of a curved surface patch onto a two-dimensional parameter plane was utilized to efficiently and accurately move the surface mesh on a curved surface. The combination of the surface mesh movement method and the dynamic mesh method enabled to compute unsteady aerodynamic forces by changing both angle of attack of the aircraft and deflection angle of the horizontal tail wing as shown in Fig. 2. In addition, it was shown that the method would be applicable to the design optimization problems with the nacelle location/angle change.



(a) Deflection angle = 0.0

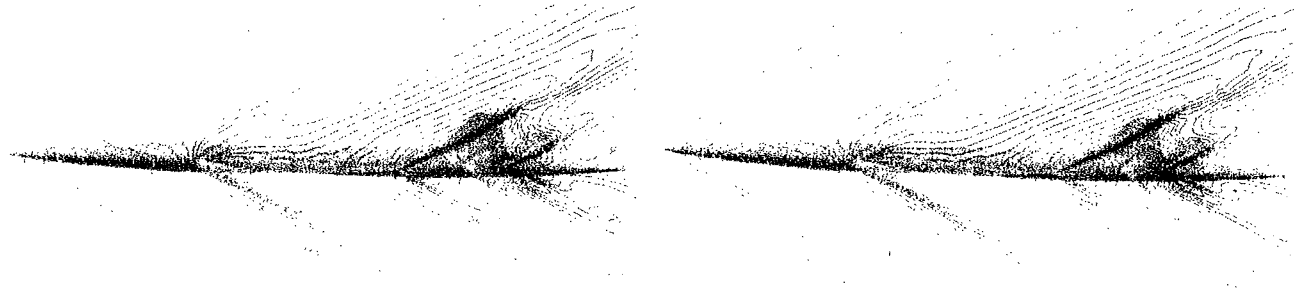
(b) Deflection angle = 0.0

Fig. 2 Close-up view near the horizontal tail wing after the surface and volume mesh movement

Chapter 5 Numerical Simulation of Airplane's Response to Control Surface Deflection

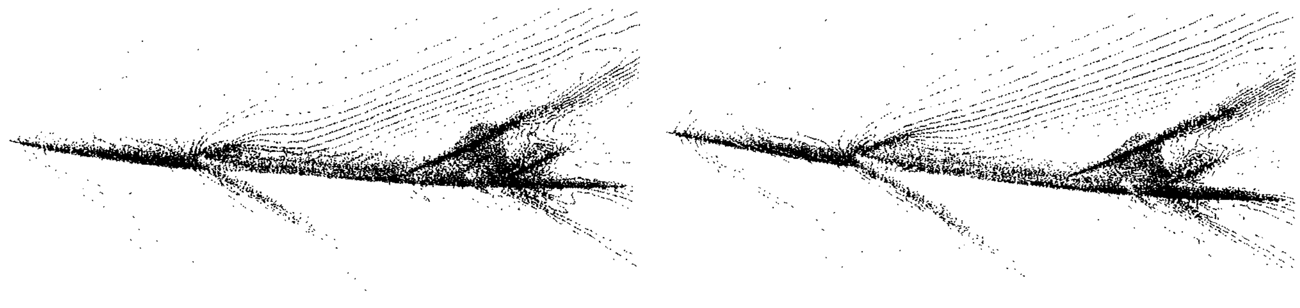
In Chapter 5, simulations of a maneuvering airplane in response to control surface deflection were discussed. The techniques described in the previous chapters were applied to the simulation of the longitudinal control surface response for an experimental supersonic airplane of the National Aerospace Laboratory of Japan. By the use of a three-dimensional unstructured dynamic mesh method with surface mesh movement, the computational mesh for a moving control surface was efficiently modified and numerical simulations of the static longitudinal stability control could be easily conducted. The computational results were validated by comparisons with experimental data. In addition, the computational results at supersonic and transonic speeds were compared and the differences of the flow characteristics between the two were discussed. Furthermore, computational results of the inviscid and viscous flows were compared.

Computation of the unsteady response of the airplane to the change of the deflection angle of the horizontal tail wing was also conducted. Equations of motion with regard to the airplane were solved at each computational time step with change of the deflection angle of the tail wing. The freestream Mach number M_∞ is 2.0. The initial angle of attack of the wing-fuselage is 2.0°. The horizontal tail wing is moved, so that the deflection angle is changed from 0.0° to -3.0° at 1200 time steps. Figure 3 shows the contours of the Mach number. It can be seen that the angle of attack of the wing-fuselage is increased according to the change of the deflection angle of the tail wing. This aerodynamic movement of the wing-fuselage is reasonable. As the deflection angle of the tail wing is decreased from 0.0°, the lift of the tail wing is decreased and the total pitching moment is increased. The nose-up movement of the airplane is caused by this moment. Although the accuracy of these results should be investigated carefully, these results show that efficient simulation of dynamic stability control is feasible by the methods herein presented. By means of unsteady simulation, it was shown that the method has a great potential for simulations of a dynamic flight control and more complicated flight test.



(a) $\beta = 0.0°$ (t = 0.0)

(b) $\beta = -1.0°$ (t = 0.2)



(c) $\beta = -2.0°$ (t = 0.4)

(d) $\beta = -3.0°$ (t = 0.6)

Fig. 3 Unsteady computational results: Contours of Mach number ($M_\infty = 2.0$)
β: deflection angle of the tail wing, t: time (sec.)

Chapter 6 Conclusions

In Chapter 6, the results and conclusions obtained in this thesis were summarized.

Consequently, it is demonstrated that the methods developed in this study are promising for airplane flight test in a computer. With further developments of computational resources, detailed analysis of real flight motions with response to the control surface deflections will be realized. However, for the perfect replace of the flight test by numerical simulations, the accuracy at off-design conditions where separations and complex vortical flows appear is still a critical problem. Further improvements of turbulence models for precise predictions of separations are required for the real flight tests in a computer.

論文審査結果の要旨

数値流体力学(CFD)は、アルゴリズム研究の進展および近年の計算機環境の発達に伴い、航空機巡航時の空力解析には不可欠な手段となっている。今後の航空分野CFDに期待されることは、より詳細な航空機実三次元形状での解析、非設計点における解析や非定常流解析の信頼性向上、そして最終的には飛行試験全体をもCFDにより解析することである。これらの要求に応えるには、航空機形状を計算空間において忠実に再現することのできる計算手法が必要となる。また離着陸時等の非設計点における解析には、渦を伴った複雑な流れ場に対する高精度数値計算法が必要となる。飛行運動の模擬には、操舵による空気力の変化と航空機の運動の連成問題を解かなくてはならない。操舵中の航空機のCFD空力解析はこれまでほとんど例がないが、計算機内での完全な飛行試験が実現すると、航空機開発時に必要な実飛行試験の回数を大幅に減らすことができ、低コストでより高性能な航空機開発を可能にする重要な研究課題である。

本論文は、CFDによる航空機飛行試験を実現するための計算アルゴリズムに関する研究成果についてまとめたもので、全編6章よりなる。

第1章は序論であり、本研究の背景及び目的を述べている。

第2章では、航空機離着陸時で問題となる翼端渦等を精度良く予測するための数値計算法として、渦中心同定法を用いた解適合格子法と非構造渦閉じ込め法を提案している。これら手法を高迎え角時におけるデルタ翼前縁剥離渦や航空機翼端渦の解析に適用して有効性を詳細に議論している。非構造格子上で渦閉じ込め法の検証を行ったのは世界で初めてのことであり、解適合格子と渦閉じこめ法の効果と問題点ついての重要な知見を示している。

第3章では、物体の移動や変形を伴った流れ場を解析する際に必要となる移動格子法に関するもので、計算格子の要素形状を適切に保つ新しい方法を提案している。このことにより、物体の大きな移動や変形を伴う流れ問題でも格子要素の重なりや歪みを防ぎ、移動格子法のロバスト性を大幅に改善した。この手法は、航空機の舵角応答や空力弾性解析に加え、様々な流体機械の設計問題にも適用でき、CFDの適用範囲を広めた非常に重要な成果である。

第4章では、航空機の舵角変化を計算空間上で取り扱うために、曲面上における表面格子の移動も含めた移動格子法を提案している。この手法により、これまで問題となっていた舵角変化時おける計算格子の取り扱いに関する問題を解決して、機体姿勢角の変化と共に操舵角の取り扱いも可能となったことが示されている。この方法はまた、エンジン取り付け位置の最適化等にも用いる事が可能であり、非常に有用な研究成果である。

第5章では、本研究で開発された計算手法を用いて航空宇宙技術研究所小型超音速実験機の姿勢角及び舵角変化に対する空力変化の解析を行い、その有効性を示している。更に、流体計算とともに機体の運動方程式も同時に解くことにより、舵角を動かしたことに対する機体の非定常的な応答解析も行い、計算手法の妥当性を検討している。操舵中の航空機空力解析、舵角応答や舵角のヒンジモーメント等の予測は、数値計算が発達した今日でも未だ実現されてなく、また実験による計測も非常に困難なものであり、CFDの新たな可能性を切り開いた重要な成果である。

第6章は結論である。

以上要するに本論文は、航空機の飛行試験を計算機内で実現するために必要となる数値計算手法を開発し、その精度及び有効性を実証したもので、数値流体力学および航空宇宙工学の発展に寄与するところが少なくない。

よって、本論文は博士(工学)の学位論文として合格と認める。