氏　　　　名	澤田　惠介（さわだ　けいすけ）
授与学位	博士（工学）
学位授与年月日	平成5年6月10日
学位授与の根拠法規	学位規則第5条第2項
最終学歴	昭和57年3月 京都大学大学院工学研究科航空工学専攻修士課程修了
学位論文題目	Developments of Computational Fluid Dynamics Methods with Applications to Astrophysics and Aeronautical Engineering （数値流体力学解法の開発と天文物理学および航空工学への応用）
論文審査委員	東北大学教授　大宮司久明　　東北大学教授　高山和喜 東北大学教授　南部　健一

論　文　内　容　要　旨

Outline of the Present Paper

The modern numerical methods for solving the governing equations of fluid flows extended the reliability and applicability of Computational Fluid Dynamics (CFD) methods for various practical problems in a unified manner. In the present paper, we consider the developments and applications of such numerical methods for solving the compressible gas flows arising in various physical and engineering problems. We also consier a development of a numerical algorithm which may contribute to the construction of a rigorous numerical scheme for the simulation of the multidimensional flow field. Accordingly, the present paper deals with following three different issues :

(1) The first part of the present paper is devoted to the numerical simulations of gas flows in a semi-detached binary system. In this study, we try to introduce the latest available numerical methods to solve the flow field with a high spatial resolution, which has not been possible by the classical first order scheme nor by the conventional central difference scheme with a linear artificial dissipation due to the limited stability bound for large spatial gradient of the physical variables intrinsic to the problem. A higher order accurate solution obtained here successfully suggests the existence of spiral shock waves in the

accretion disk which implies a new mechanism of the accretion phenomena.

(2) The second part of the paper is devoted to the numerical simulations of the flow field over various complete aircraft configurations. The flow field over a complete aircraft has been a long pacing item of CFD in aeronautical engineering. To this end, various numerical approaches have been proposed, such as the zonal approach or the use of the unstructured grid systems. The present works, however, show that even with the conventional structured approach, we can treat very complicated configurations in a simple but versatile manner by employing a special class of transformation. Navier-Stokes simulations for different configurations of complete aircrafts are successfully conducted to show the ability of the present approach.

(3) The last part of this paper considers an algorithm development of numerical scheme. Although many numerical methods have been proposed which have really extended the horizon of CFD methods, it is still true that there are many issues which must be resolved for the construction of truly reliable numerical schemes. One such remaining problem is how to attain the rigorous multidimensional property of a numerical scheme. We here consider the multidimensional extension of the conventional MUSCL approach for a finite volume scheme.

Now let us explain the contents of the present paper in turn. We at first show the CFD applications to the astrophysical problem in sections 2 and 3. We consider the accretion flow problem which deals with the gas flows near the semi-detached compact binary stars. We assume that the gas is ejected from one component of the binary stars (hereafter we call this component as the mass-losing star). If the temperature of the ejected gas is sufficiently low, then the gas cannot escape from the binary system but is bounded by the combined gravitational field of two stars and it will be attracted to the other component (hereafter we call this as the mass-accreting star). However, since these two stars are rotating each other, the ejected gas cannot be reached directly to the mass-accreting star because the gas particle possesses the angular momentum to the mass-accreting star. Gas particles attracted by the gravitational force then form a disk structure around the mass-accreting component due to the excessive angular momentum and this disk structure is called the accretion disk.

Of various modes of accretion, the accretion of gas flow onto a compact gravitating object via the accretion disk is of great interest since it is believed to be the energy source of high-energy radiating objects such as X-ray stars and quasars. The standard accretion theory assumes the existence of an axi-symmetric thin Kepler disk around a compact mass-accreting object. Gas flows in this accretion disk is believed to be highly turbulent. The Reynolds stress is expected to play the central role in transferring the angular momentum of gas particles from the inner circular orbit to the outer one. With this angular momentum transfer, the gas particles in the accretion disk can move toward the central compact object.

Despite its high Reynolds number ($R_e \sim 10^{14}$), the gas flow in the accretion disk is not yet proved to be turbulent because it is a rotating supersonic flow. Even if the flow is assumed to be turbulent, the angular momentum could be transferred inward if fluid elements exchange the angular momentum rather than the angular velocity between nearby orbits. In this case, a thin ring structure will be formed in stead of the accretion disk. Thus the standard theory of the accretion disk still has many ambiguous problems which should be clarified.

In the course of the present study, we at first conduct numerical simulations of the gas flows in the semi-detached binary system. A high spatial resolution as well as the required numerical stability are both provided by using the Osher upwind scheme along with the preprocessing approach proposed by van Albada. The calculated results show that the ejected gas from the mass-losing star forms the expected accretion disk around the mass-accreting star. Moreover, the results indicate that there appear two or three spiral shaped shock waves in the accretion disk.

Corresponding to these shock waves, a new scenario is proposed for the accretion mechanism where gas can be accreted through the interactions with spiral shaped shock waves. These spiral shaped shock waves can strip off the angular momentum of the gas particles and let the particles move inward. This angular momentum transfer due to the shock waves can operate even if the flow in the accretion disk is not turbulent. With these results, we pose some questions regarding the validity of the fundamental assumptions of the standard accretion disk model.

Section 4 and section 5 are devoted to show the CFD applications in aeronautical engineering where the numerical simulations of gas flows around complete aircraft configurations are concerned. As mentioned before, a flow simulation of a complete aircraft configuration has been the long pacing item of CFD methods in the aeronautical engineering. We employ a versatile transformation technique in generating structured grid systems around complete aircraft configurations and also a finite volume upwind scheme, to realize this goal.

At first, an algebraic grid generation method using the multi-block transformation technique and an explicit upwind finite volume scheme to solve the Euler equations on this grid system are developed. They are applied to solve the compressible inviscid flow over a short take-off and landing aircraft. Subsequently, the grid generation method is replaced by an interactive one to increase the geometrical flexibility, and the scheme is extended to solve the viscous flows by adopting a planar Gauss-Seidel relaxation technique for achieving a rapid convergence. This implicit scheme is applied to various problems such as the viscous supersonic flow over a reentry vehicle and the viscous transonic flow over a short take-off and landing aircraft.

Since the Reynolds number of flows over aircraft configurations is generally very high (R_e

$\sim 10^7$), so that the flow fields are expected to be turbulent. The treatment of turbulence is also very important in the aeronautical engineering because a significant amount of the total drag exerted on the aircraft is due to the surface skin friction which is critically affected by whether the boundary layer is turbulent or not. Usually, some kind of turbulence model is adopted in the computation of high-Reynolds number flows because the direct simulation of turbulent flow field is quite time consuming and is still out of the practical engineering scope.

In the aeronautical engineering, the zero-equation algebraic turbulence model of Baldwin-Lomax is very popular due to its simplicity. This model has been applied to various problems and the applicability of the model to various flow conditions has been well defined. However, the multi-block structured grid system employed in the present computation does not allow us to use this simple turbulence model. This is because the zero-equation model changes its form in the boundary layer and in the wake region, while it is very difficult to identify whether a fluid element is inside the boundary layer of one block or in the wake of another. Therefore, we decided to adopt a more general formulation of the turbulence model, *i.e.* the $q-\omega$ two-equation turbulence model, in the present scheme at the sacrifice of computational cost.

It is shown in these sections that the combination of the present multi-block structured grid system and the explicit／implicit finite volume schemes to solve the Euler and Navier-Stokes equations can really provide a practical means to simulate the flow fields over complete aircraft configurations of real engineering interests within the framework of the structured appr oach.

In section 6, we present an algorithm development for attaining a higher order spatial accuracy valid in the multidimensional numerical scheme. We consider the preprocessing part of a higher order accurate cell-centered finite volume scheme in the multidimensional space. In the cell-centered finite volume scheme, the so-called MUSCL (Monotone Upwind Scheme for Conservation Law) approach is usually adopted to attain the higher order spatial accuracy. In this approach, the local distribution of the dependent variable is fitted by a polynomial function of an appropriate degree whose coefficients are determined by the cell averaged values in the nearby cells. Based on this distribution, the dependent variable at the cell interface is found which is then used to provide the numerical flux function necessary for the finite volume integration. The spatial accuracy of the scheme is related to the degree of the assumed polynomial function. There is no ambiguity in this formulation so far as one dimensional problem on the equally spaced grid is cosidered.

In applying this approach to the multidimensional problems, a dimension by dimension splitting approach is often adopted. In this approach, the unidimensional operator is sequentially and independently applied to each spatial direction in the computational space instead of the physical space. Obviously, the obtained solution critically depends on the nonuniformity of grid spacing and also on the skewness of grid line. In the numerical method presented in section 4, the nonuniformity of grid spacing was explicitly considered. However, the method still ignored the skewness of grid lines.

Here we try to improve the accuracy of the numerical scheme on a generalized curvilinear coordinate system by considering the nonuniformity of grid spacing and also the skewness of grid line, simultaneously in the preprocessing stage. For this end, the MUSCL approach is directly extended to the multimensional space by taking a multi-variable polynomial function to fit the local distribution of the dependent variable. A piecewise planar and a piecewise parabolic distributions of the dependent variable, corresponding to the second order and the third order schemes, are considered. In fitting the local distribution, the polynomial function cannot be determined uniquely even if its degree is prescribed. How to select the best one out of the possible candidates is discussed in view of maintaining the stability of the scheme.

Test calculations concerning the unsteady shock propagations over a single wedge are conducted to verify the improved spatial resolution as well as the numerical stability of the proposed method. We also compare the obtained flow fields with the results calculated by the existing TVD or ENO schemes.

Finally in section 7, we give the comprehensive discussions of the problems treated in the present paper and try to give some conclusions relating to the previous sections and also to the perspective views gained during the present investigation.

審 査 結 果 の 要 旨

数値流体力学は，近年急速に発展し，流れ現象の解明に，また工学分野で流れに関する諸問題を解決する有力な手段として利用されている。本論文は，数値流体力学の天文物理学と航空工学への応用と，圧縮性流れの一つの数値解法の提案について述べたもので，全編7章よりなる。

第1章は序論である。

第2章では，半分離型2重星のロッシュ溢れ流れの数値シミュレーションを行い，コンパクト星まわりに降着円盤が発達する過程を説明している。ここでは2次元の連続体流れを仮定し，一般曲線座標系の圧縮性オイラー方程式を2次精度風上差分法で解いている。降着円盤内に渦状衝撃波の発生することを予測し，また粘性が働かなくてもこの衝撃波によりガスがコンパクト星に降着することを見出している。

第3章は第2章の議論を発展させたもので，計算格子を変えて降着円盤内流れ場の高精度解析を行うと共に，比熱比が重要なパラメータであることを示している。比熱比小の場合には定常的降着円盤が得られるが，比熱比大の場合には流れ場が振動することなどの知見を得ている。

第4章では，短距離離着陸機「飛鳥」の遷音速非粘性流れ解析を行っている。マルチブロック格子を用い，圧縮性オイラー方程式を陽的風上有限体積法で解き，ジェット排気効果を含めて全機まわりの流れを求めている。また風洞試験の結果と比較し，翼表面圧力分布がだいたい合うことを示している。

第5章では，前章の解析法を，遷音速粘性流れに適用できるように拡張している。すなわち，会話型手法によってより適切なマルチブロック格子を形成し，$q-\omega$ 2方程式乱流モデルを導入し，また平面対称ガウスザイデル緩和法で時間積分を行うことによって解の収束性を改善している。この解析法を用い，宇宙往還機まわりの超音速流れを解析した結果は，縦3分力とも実験データと非常に良く一致し，また「飛鳥」まわりの遷音速流れ解析でも実験とかなり良い一致が得られている。

第6章では，圧縮性流れのMUSCLアプローチを2次元に拡張している。その特徴は，2次元の安定な区分的連続な再構成関数を用いることと，有限体積法の積分計算を時空間で正確に行っていることである。くさびを過ぎる衝撃波のひき起こす非定常流れを計算し，ここで提案した3次精度のものが良好な結果を与えることを示している。

第7章は総括である。

以上要するに本論文は，非定常圧縮性流れの高精度有限体積法を提案し，また数値流体力学による天体現象の説明と，機体まわりの複雑な流れ場のシミュレーションを行ったもので，流体工学とりわけ数値流体力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。