氏　　　　　名	阿部　陽介
学位（専攻分野）	博士（理学）
学位記番号	総研大甲第838号
学位授与の日付	平成１７年３月２４日
学位授与の要件	物理科学研究科　核融合科学専攻 学位規則第６条第１項該当
学位論文題目	Numerical simulation of momentum transport processes through gas-liquid interface
論文審査員	主　査　教授　林　隆也 助教授　渡邉　智彦 助教授　三浦　英昭 教授　小森　彰夫 グループリーダー　高橋　桂子（海洋研究開発機構） 教授　矢部　孝（東京工業大学）

論文内容の要旨

Free-surface turbulent flows are ubiquitously found in the nature as well as in various industrial devices such as a chemical plant, not to mention the turbulence in rivers and oceans. A two-phase flow for a gas and a liquid with a distorted interface is one of the most common issues for free-surface turbulent flows, and, recently, has been much paid attention to in the field of environmental science, specifically, in the climate projection study. Interaction between Atmosphere and Ocean is a key issue in the climate projection because their interaction affects the global circulation as well as the local weather. At the Earth Simulator Center, a high-resolution (100m - 10km for horizontal) non-hydrodynamic Atmosphere-Ocean coupling model has been developed in order to simulate not only a global phenomenon, but also a local one, such as a typhoon which may bring a terrible disaster. In this high-resolution regime, however, no reliable method of evaluating physical quantities, such as the momentum, heat, and vapor, exchanged through the interface has been established. Therefore, it is necessary to investigate fundamental physical processes in gas-liquid flows with a free-surface.

In previous studies on the gas-liquid flows with a free-surface, mainly explored was the liquid side turbulence near the interface on which no or very low shear flow condition was imposed. A direct numerical simulation (DNS) of free-surface flows with the coupled gas-liquid dynamics should be useful for understanding a gas-liquid interaction mechanism through a detailed analysis of turbulent statistics which can not be obtained by experiments. Nevertheless, only a few trials have been made because of the numerical difficulties in tracking a free-surface, such as the numerical diffusion and instability accompanied with the interface deformation. Since, in a high velocity range commonly seen in realistic free-surface flows, turbulent characteristics near a free-surface could be influenced by deformation of the free-surface due to wind shear, it is necessary to take into account the effects of the interface distortion in the DNS of the gas-liquid flow simulation.

Various kinds of two-phase flow models have been developed such as the level-set method and the volume-of-fluid (VOF) method. Recently, Kunugi (1997) has developed a precise free-surface tracking algorithm for two-phase flows, that is, Multi-interfaces Advection and Reconstruction Solver (MARS). Yabe and Wang (1991) proposed the CIP-combined, unified procedure (C-CUP method) which can treat the compressible and incompressible fluids, simultaneously, in the same code. Even in cases that an interface is largely deformed by a strong wind shear, the C-CUP method can resolve the complicated interfaces on the Cartesian coordinates. Since it is not necessary to reconstruct the numerical grid in the C-CUP method, the numerical cost and errors can be reduced in comparison to MARS. Using the C-CUP method, Mutsuda et al. (1998, 1999) succeeded in performing a DNS of wave-breaking and entrained air bubble in a gas-liquid turbulent layer. However, there has been no quantitative evaluation of the friction stress by means of the two-phase flow simulation.

The objective of this study is to show the validity of the numerical method based on the C-CUP scheme, by quantitatively comparing the simulation results with experimental data, towards construction of a reliable method for evaluating the friction stress between Atmosphere and Ocean. The reasons for adopting the C-CUP method are summarized as follows. First, applicability of the C-CUP method to the compressible fluid has an advantage in simulation of severe phenomena such as a typhoon. Second, the C-CUP method can keep thickness of a transition region near the sea-surface compact, which is important in long-term integration. Moreover, it is a more direct way to solve the same equations for both Atmosphere and Ocean, as is done in the C-CUP method, than separately treating them through boundary conditions on the sea-surface.

In this study, the two-phase flow simulations for a gas and a liquid are performed with a numerical setting as given bellow. Setting of the numerical domain is relevant to tank-experiments. Turbulent Reynolds number Re_τ is changed from 150 to 300 in order to find the wind velocity dependence of the friction. An effect of the water depth on evaluation of the friction velocity is also studied. In the deeper case, the water depth is relevant to that of the tank-experiments. The wind velocity is in a regime that the free-surface deformation accompanied with ripples can not be ignored, and is comparable to the representative velocity range in the tank-experiments. The continuum surface force (CSF) model, which is often used in two-phase flow simulations, proposed by Brackbill et al. (1992) is applied to the surface tension. In a transition region near the free-surface, the viscous coefficient is modeled by an inversely proportional relation which shows a better property than the linear relation and agrees with experimental data for two-phase mixing fluids by McAdams et al. (1942). In addition to the C-CUP method, the present simulation code is implemented with the following schemes in order to improve the numerical stability and accuracy in the high velocity range. The rational CIP method proposed by Xiao et al. (1996) can avoid negative values of the density and pressure in the transition region near the free-surface. The tangent transformed density function, which defines the transition region, proposed by Yabe and Xiao (1993) can reduce the numerical diffusion of the transition region keeping the compactness.

From the statistical analysis of turbulence in the simulation results, we have quantitatively discussed the turbulent structures near the free-surface and the mechanism of the momentum transfer through the gas-liquid interface. The results obtained from the present simulation study on the two-phase flow, are summarized below.

It is shown that the vertical velocity fluctuations normal to the free-surface are induced on the liquid side near the interface. To explain this behavior, we have made analysis of the pressure-strain correlation term that remains finite only near the free-surface on the liquid side. The surface-normal component of the pressure-strain correlation term has a positive value while the streamwise component is negative. This behavior indicates that the turbulent energy on the gas side is redistributed mainly into the vertical component of the velocity fluctuations on the liquid side through the free-surface deformations which are excited by the shear stress. Moreover, the result that all vorticity components increase near the free-surface shows the presence of three-dimensional turbulent structures. Thus, it is necessary to take into account the effects of the free-surface deformation on the momentum transfer process through the gas-liquid interface when a large shear stress exists. It is also suggested that the vertical velocity component should be accurately calculated, although the conventional numerical simulations have treated the velocity fields near the free-surface as almost two-dimensional.

The turbulent structures are similar to those of the experiment, in which the representative wind velocity is almost the same as that of the present study. The Reynolds stress distribution is correlated with the free-surface structure. This result reflects that the momentum transfer through the gas-liquid interface is related to the Reynolds stress produced by the vertical velocity fluctuations near the free-surface. In order to verify the correlation, we have investigated a phase relation between a time series of the Reynolds stress on the gas side and that of the free-surface position. According to this analysis, large positive spikes of the Reynolds stress, which enhance the downward momentum transfer, often appear at positions where the 'burst' and the 'sweep' are observed. It is also consistent with the experiments. In addition, bursts of the Reynolds stress are observed in close to the crest over the windward side, as reported in the experiment. This agreement suggests that the present simulation captures the bursting phenomena produced by the Reynolds stress in relation to ordered motions such as the wind flow separation and the reattachment over wind waves. Therefore, it is considered that the momentum transfer process with the free-surface deformation can be well reproduced by the present DNS of the two-phase flow.

The friction velocities are evaluated from the peak values of the friction stresses near the free-surface on the liquid side. The relation between the friction velocity and the mean velocity on the gas side is in good agreement with the tank-experimental data for both cases of the Reynolds numbers of 150 and 300. Especially, a better agreement is found for the deeper water condition. In the velocity range considered here, the free-surface deformation can not be ignored in contrast to the conditions for the conventional DNS studies, where the liquid turbulence is decoupled from the gas dynamics by supposing a flat interface and applying the free-slip condition. Therefore, the present study presents the first quantitative evaluation of the friction stresses on the free-surface, which is validated by a comparison with the tank experimental data.

The three-second power law and the spectrum form of wind waves proposed by Toba (1972) are also examined for the case of the present numerical simulation. The form of the energy spectrum of wind waves is derived from a combination of the three-second power law and the similarity of the spectral form of wind waves. These are well substantiated by data from a wind-wave tunnel experiment. The three-second power law is obtained from the spectrum of wind waves based on the local balance between the wind waves and both the turbulent structures of the air and liquid flows. The period and the wave height of significant waves in dimensionless forms, which are considered to correspond to the peak frequency and the energy level, respectively, are used as representative quantities of wind waves. From the results of the present study, it is confirmed that the relation of the period and the height of significant waves is consistent to the three-second power law within the error coming from the wind-wave tunnel experiment and the field observation. In the gravity wave range, the spectral form on the high frequency side is proportional to the -4 power of the angular frequency of wind waves. The wind waves grow in a way that on the logarithmic diagram of the spectral density versus the angular frequency, the spectrum slides up along the line of the form, keeping its similar form. It is confirmed that the spectral density obtained by the present simulations has the -4 power of the angular frequency. The spectrum level shows a better agreement with the data of the wind-wave tunnel experiment than that of the field observation. The result suggests that the present numerical simulation can accurately represent the developing phase of wind waves in wind-wave tunnel experiments. As the wave number becomes large, the effect of surface tension is incorporated. Thus, the -4 power line found in the gravity wave range gradually approaches the -8/3 power line for the capillary wave range, which is also reproduced by the present simulations.

The obtained results confirm that the present simulation method has a possibility of extension to a larger gas-liquid flow system towards construction of a reliable scheme for evaluating the friction stress between Atmosphere and Ocean.

論文の審査結果の要旨

液相・気相や高密度相・低密度相など、二相間の不連続面を有する状況での物理現象は自然界に多く見られる。核融合ではプラズマ閉じ込め領域とその周辺領域などがそれにあたる。不連続面を含む系における非線形ダイナミクスを直接の対象とした計算機シミュレーション研究は、まだその手法において未開拓の部分が多く残されている。本研究では、このような複雑現象のシミュレーション手法確立を念頭において、地球流体分野に研究課題を求め、風波を含む大気海洋界の液相・気相不連続面を有する運動量輸送現象のシミュレーション研究をおこなっている。研究の背景として、地球シミュレータのスーパーコンピュータ開発によって、大気と海洋を結合しその相互作用を取り入れた地球規模気象予報シミュレーションが構想される時代となってきたことが挙げられる。しかしながら現状では大気海洋の相互作用は経験的な物理モデルしか存在せず、運動量輸送などを定量的に評価するための基礎的シミュレーション実現が必須な研究課題となっている。本研究の具体的目標はこの点におかれている。

これまで自由界面の変形が無視できない乱流場での気液二相流計算の研究は少なく、特に気液間の摩擦応力の定量的評価を行った例はなかった。このために本研究では先進的計算スキームであるC-CUP (CIP and Combined, Unified Procedure) 法を用いたシミュレーションコードを新規に開発し、水槽実験と同様の体系における定量的評価を行い、実験との比較によってその有効性の実証を試みている。計算コードの開発にあたっては、気液界面における物理量の単調性を維持するために有理関数CIP法(Cubic-Interpolated Propagation method)、気液界面の形状をシャープに保ち数値拡散を防ぐために正接関数変換CIP法を用いるなど、計算の充分な精度を確保するための工夫を行っている。

その結果、次のような成果を得ている。

1.界面変形の影響が無視できない流れ場では、自由界面近傍において3次元性が卓越し、乱流強度の鉛直成分が増大することから、いままで考慮されていなかった速度の鉛直成分が鉛直混合に影響を与えうることを明らかにした。これらの結果は、渦度3成分の分布、速度変動の二乗平均値分布、圧力－歪分布などから裏付けられている。

2.風速の速い風波の峯の後面においてReynolds応力の生成頻度が高いことから、水槽実験に見られる現実の自由界面における運動量輸送機構を再現できることを示した。このことは、風波の位相に対するReynolds応力の生成を伴うバースト発生率のヒストグラムによって示されている

3.自由界面における摩擦速度が、水槽実験により得られた結果と良く対応することから、本数値計算は、自由界面における摩擦速度を定量的に再現できることを示した。平均風速値と摩擦速度の関係のグラフにこのことが示されている。また、液相の水深が結果に影響を与えることも示した。

4.波高と周期の関係に関して、風波を特徴付ける3/2乗則が十分な精度で成り立つことが示されたことから、風波の特性としてのスケールによらない普遍的な発達過程を表現できることを示した。

これらの結果は、水槽実験との対応が定量的評価及び普遍的な相似則の存在から裏付けられており、本シミュレーションモデルによる運動量輸送量の評価方法が、大気海洋結合

モデルへ拡張できる可能性が十分あることを示していると考えられる。

以上の成果は、気液二相間の運動量輸送について物理機構・定量的評価に関して新しい知見を与えており、また不連続面を有する物理現象のシミュレーション手法が適切に機能することを具体的応用例で示している。成果は、プラズマ閉じ込め周辺領域物理現象やペレット方式粒子補給過程など核融合プラズマの非線形ダイナミクス計算手法にも示唆を与えるものである。これらにより、学位論文として充分な価値のある内容を含むと判断した。