 Proceedings of ICCES'08, pp.585-585

Numerical Computation about a Suspended Body by a Gas Flow Levitation Furnace

R. Toda, S. Oshima and I. Hagiwara[1]

Summary

A gas flow levitation furnace is a device making high-purity materials without touching a container for melting a solid body suspended by gas flow. Previously, almost all of the researches were the consideration about process of melting by experiments. So, there are few examples of reserches about an influence of the shape of a gas flow levitation furnace. So we calculate this flow changing velocity of jet flow, the shape of the furnace and the distance between the furnace and the suspended body by using Cartesian Grid. In this paper, we assume the shape of a suspended body is a column and flow is two-dimension. To calculate this flow field, derive the drag and lift coefficients, Strouhal number, and so on. Comparison the drag on the column, consider about influence of some factors.

Introduction

A finite difference method for the simulation using Cartesian grid is developed. The calculation grid is not necessarily remade despite the shape of the furnace. In this paper, the distance from the adjacent stencils to the body boundary is integrated into the finite difference formulation to satisfy the body boundary conditions. The central term in the diffusional term is constructed implicitly to avoid the severe diffusion number condition. The pressure boundary condition is given to solve the least squares method to satisfy Neumann boundary condition around the stencil. The code is developed based on Fractional Step Method. In time integrate, the second order explicit Adams-Bashforth method is used for advective terms.

Results

In this section, we show one result derived by using the above sheme, and it is proved that this scheme is almost correct. Figure 1 shows this calculational model that is two-dimensional computation of flow around a circular cylinder. Table 1 shows a caculational result (C_D, C_L : Drag and lift coefficient , St : Strouhal number).

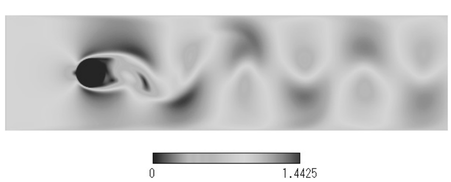

Figure 1: Flow field, Re = 200

Table 1: Results, Re = 200

	C_D	C_L	St
Proposal method	1.38	± 0.68	0.20

[1]Tokyo Inst. of Tech.Department of Mechanical Sciences and Engneering,Tokyo, Japan