8th. World Congress on Computational Mechanics (WCCM8)
5th. European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 – July 5, 2008
Venice, Italy

Convergence Acceleration Method for Computational Fluid Dynamics using Immersed Boundary Cartesian Grid Method

Takahiro Fukushige[1]*,Toshihiro Kamatsuchi[1], Toshiyuki Arima[1] and Seiji Fujino[2]

[1] Honda R&D Co., Ltd.
Fundamental Technology Research Center
1-4-1 Chuo, Wako-shi, 351-0193, Japan
fukushige@f.rd.honda.co.jp

[2] Kyushu University
6-10-1 Hakozaki, Higashi-ku, Fukuoka-shi,
812-8581, Japan
fujino@cc.kyushu-u.ac.jp

Key Words: *CFD, Immersed Boundary, BCM, Convergence Acceleration, IRS.*

ABSTRACT

A new numerical method for convergence acceleration on Computational Fluid Dynamics(CFD) is proposed. The proposed method is to apply Implicit Residual Smoothing (IRS) to CFD with 'multi-cube'. The good convergence performance has been obtained by our proposed method.

Introduction

The computational time of CFD continues to increase, while progress of computer performance has been made. One of the reasons is considered that applications of CFD have become more complex. For example, CFD is employed to estimate aerodynamic performance for a complex shaped object such as formula one car [1] (Fig. 1) . Concerning complex shape, however, the problem of grid generation still remains. It requires large amount of time and labor. To overcome the problem in meshing for complex-shaped object, we have already proposed an algorithm [2]. This algorithm consists of two approaches. One is Immersed Boundary method [3], and the other is Building-Cube Method (BCM) [4]. These approaches have several advantages except for solution convergence. In this paper, Implicit Residual Smoothing (IRS) [5] is proposed for improvements of solution convergence.

Figure 1. Formula 1 race car (Honda Racing F1 Team RA106)[1]

Numerical scheme

・Computational grid

The computational grid used in this study is based on the BCM. The BCM mesh generation consists of two steps: The first is to generate cube in various sizes to fill the flow field as shown in Fig. 2. The second step is to generate Cartesian grid in each cube (Fig. 3). Two cells overlap between adjacent cubes to exchange the flow information at the boundaries.

・Implicit Residual Smoothing (IRS)

The governing equations discretized by FVM are solved by LU-SGS [6] scheme. To improve solution convergence, the IRS is adopted within each cube. In this study the explicit residual R_i is modified as,

$$\overline{R}_i = R_i + \varepsilon \sum (\overline{R}_j - \overline{R}_i) \tag{1}$$

where $\overline{R}_i$ is the smoothed residual, $\overline{R}_j$ is residual of cell j adjoining cell i and ε is a positive parameter used to control the smoothing.

Equation (1) can be solved by Jacobi iteration described as follows,

$$\overline{R}_i^m = \frac{R_i + \varepsilon \sum \overline{R}_j^{m-1}}{1 + \varepsilon \sum 1} \tag{2}$$

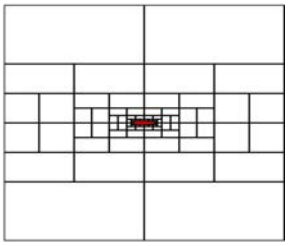

Figure 2. Cube boundary around airfoil

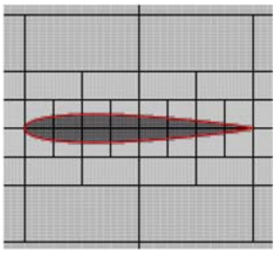

Figure 3. Cartesian mesh around airfoil

Result and discussions

IRS is applied to flow simulation around 3D airfoil. The surface geometry is shown in Fig. 4. Computational grid in which cubes of heterogeneous size is shown Fig. 5. In this case, cell size at interface is different which, in general, makes convergence worse. Good solution convergence is obtained as shown in Fig. 6. Possible reason is that IRS promotes propagation and attenuation of residual error across cube interface. Regardless of the size of cube, significant improvement for convergence was obtained.

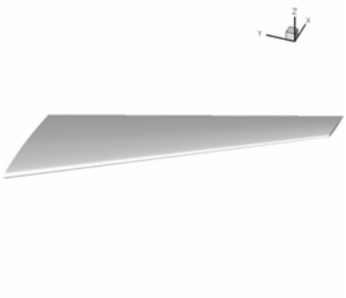

Figure 4 Surface data for 3D airfoil

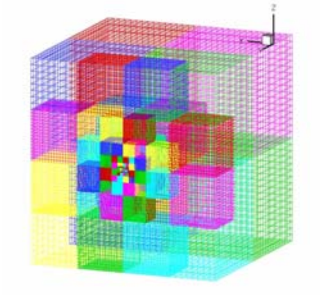

Figure 5. Computational grid for actual problem

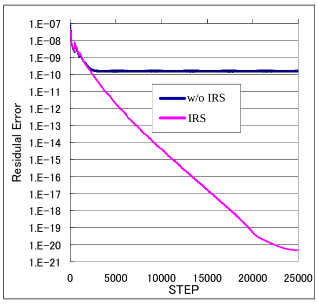

Figure 6. Convergence history for actual problem with and w/o IRS

REFERENCES

[1] URL: http://www.honda.co.jp/F1/race2006/formation/machine-ra106/

[2] Kamatsuchi, T., *Turbulent Flow Simulation around Complex Geometries with Cartesian Grid Method*, J. Sci. Stat. Comput., Vol.13, No.2, pp.631-644,(2007).

[3] Peskin,C. S.,, *Flow pattern around Heart Valves*, J.C.P , Vol.10, pp252-275, (1972).

[4] Nakahashi, K., *Building-Cube method for Flow Problems with Broadband Characteristic Length*, Computational Fluid Dynamics, Springer, pp77-81, (2003).

[5] Jameson A., *The evolution of computational methods in aerodynamics*, Journal of Applied Mechanics Review, 50(4b), pp1052-1070, (1983)

[6] Yoon, S., and Jameson, A., *Lower-upper Symmetric-Gauss-Seidel Method for Euler and Navier-Stokes Equations*, AIAA Journal, No. 26, pp.1025, 1026,(1988)