ADAPTIVE HYBRID CARTESIAN GRID METHOD FOR VORTEX-DOMINATED FLOWS

Hu Ou(胡偶)[1], *Zhao Ning*(赵宁)[1], *Liu Jianming*(刘剑明)[2], *Wu Jie*(吴杰)[1]

(1. College of Aerospace Engineering, Nanjing University of Aeronautics and Astronautics, Nanjing, 210016, P. R. China; 2. Department of Mathematics, Xuzhou Normal University, Xuzhou, 221116, P. R. China)

Abstract: An efficient compressible Euler equation solver for vortex-dominated flows is presented based on the adaptive hybrid Cartesian mesh and vortex identifying method. For most traditional grid-based Euler solvers, the excessive numerical dissipation is the great obstruction for vortex capturing or tracking problems. A vortex identifying method based on the curl of velocity is used to identify the vortex in flow field. Moreover, a dynamic adaptive mesh refinement (DAMR) process for hybrid Cartesian gird system is employed to track and preserve vortex. To validate the proposed method, a single compressible vortex convection flow is involved to test the accuracy and efficiency of DAMR process. Additionally, the vortex-dominated flow is investigated by the method. The obtained results are shown as a good agreement with the previous published data.

Key words: hybrid Cartesian grid; adaptive mesh; vortex identification; vortex-dominated compressible flows

CLC number: V211 **Document code**: A **Article ID**: 1005-1120(2013)03-0221-06

INTRODUCTION

The accurate simulation of vortex-dominated flow problems is a challenging task in computational fluid dynamics (CFD) field, like the computation of the wakes of aircraft or rotorcraft. The vortex-dominated flow problems are always governed by the unsteady vortex and the interaction of vortical structures, which requires an accurate and efficient numerical method to capture and preserve vortices.

Several methods have been developed to simulate the vortex-dominated flows. One approach often considered to reduce the numerical dissipation of vortices is to use the higher-order scheme to compute the fluxes across the cell interfaces, such as WENO scheme[1] or discontinuous Galerkin (DG) method[2]. Another approach proposed by Morton and Roe[3] is the vorticity-preserving scheme in pure acoustics. The third choice is developed by Harris, et al[4]. They developed an Eulerian vorticity transport solver with adaptive mesh refinement method for rotorcraft flow field analysis.

An efficient compressible Euler equation solver for vortex-dominated flows is described based on the adaptive hybrid Cartesian mesh and vortex identifying method. The framework of the solver allows the dynamic adaptive mesh refinement (DAMR) coupled with vortex identifying method to capture and preserve vortices. The use of finer mesh with the improved resolution in vorticity region can efficiently reduce the numerical dissipation.

1 FINITE VOLUME FORMULATION ON ADAPTIVE HYBRID CARTESIAN GRID

1.1 Adaptive hybrid Cartesian grid

For CFD simulation, the method of mesh generation is a critical technique. There are three main ways to generate the computational mesh: Structured grid, unstructured grid and adaptive Cartesian grid. The ultimate objective of each method for mesh generation is to apply the gener-

Foundation item: Supported by the National Natural Science Foundation of China(11102179).

Received date: 2013-04-15; **revision received date**: 2013-07-07

Corresponding author: Zhao Ning, Professor, E-mail: Zhaoam@nuaa.edu.cn.

ated mesh for the numerical simulation which can effectively reveal the flow field characteristics and accurately simulate the flow phenomena. For the unsteady flows, it will be much better if the mesh can dynamically be generated and adaptively capture flow characteristics. Therefore, the adaptive Cartesian grid method is the suitable choice. When using Cartesian grid method, however, it is difficult to treat the immersed boundary, especially for complex geometry and high Reynold number turbulence flow problems. To overcome this difficulty, an adaptive hybrid Cartesian grid method[5-6] is used for vortex dominated flows. As shown in Fig. 1, the hybrid Cartesian grid consists of a body-fitted structured mesh in near-body region and an adaptive Cartesian mesh which matches the outer boundary of body-fitted grid.

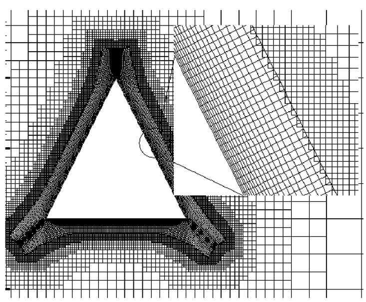


Fig. 1 Hybrid Cartesian grid

In the adaptive hybrid Cartesian grid method, the immersed boundary problem is in a single Cartesian grid and can be avoided by using the body-fitted mesh in near-body region. Thus boundary layer can be resolved much better for viscous problem. Meanwhile, the adaptive Cartesian mesh can also be used in other computational domain with the advantages like automatic mesh generation, and dynamically capturing flow characteristics. Therefore, combining these two meshes will be a good choice for simulating flow problems, especially for vortex-dominated flow problems like wing-tip vortex problem. Moreover, the off-body vortex can be more accurately resolved by dynamic adaptive mesh refinement method as shown bellow.

For CFD simulations, the accuracy of flow field can generally be improved by using either high-order scheme to compute the fluxes across the cell interfaces (p-refinement) or mesh refinement in the concern part of flow field (h-refinement). In this paper, the latter approach is used to improve the accuracy of vortex-dominated flow-fields.

For adaptive Cartesian grids, a quadtree-based structure[7] is used to store the mesh information of adaptive Cartesian grid. The appealing advantage of tree-based data structure is easy implementing refinement or coarsening cells, which leads to facile solution adaptations. For vortex-dominated flows, the vorticity is a remarkable flow feature, represented by the curl of velocity. Therefore, calculating the curl of velocity in each cell can identify the vortex in the flow field. To capture and identify the dynamic vortex in the flow field, it is essential to perform solution-based grid adaptations. As mentioned above, the curl of velocity is used as the adaptation criteria which is described as

$$\tau_{c_i} = | \nabla \times U | h_i^{\frac{r+1}{r}} \tag{1}$$

where $i=1, 2, \cdots, N$, N is the total number of cells and h_i the length scale of cell, $\nabla \times U$ is the curl of velocity and r a constant set as $r=2$. The standard deviation of the parameter is computed as

$$\sigma_c = \sqrt{\frac{\sum_{i=1}^{N} \tau_{c_i}^2}{N}} \tag{2}$$

Then the following conditions are used for grid adaptation:

(1) If $\tau_{c_i} > \sigma_c$, cell i is flagged for refinement;

(2) If $\tau_{c_i} < 0.3\sigma_c$, cell i is flagged for coarsening.

1.2 Numerical method

Consider the 2-D unsteady compressible flows governed by Euler equations

$$\frac{\partial}{\partial t}\int_{\Omega} \boldsymbol{W} \mathrm{d}\Omega + \oint_{\partial\Omega} \boldsymbol{F}_c \mathrm{d}S = 0 \tag{3}$$

where $\boldsymbol{W}$ is the state vector and $\boldsymbol{F}_c$ the convective flux. If an arbitrary control volume Ω_I is considered, the equation can be spatially discretized as

$$\frac{\mathrm{d}\boldsymbol{W}}{\mathrm{d}t} = -\frac{1}{\Omega_I}\sum_{m=1}^{N_F} (\boldsymbol{F}_c)_m \Delta S_m \tag{4}$$

where m is the interface between cell I and J, N_F the number of control volume faces and ΔS_m the area of face m. The solution is updated using a Runge-Kutta explicit time integration procedure.

In the present work, a cell-centered finite volume solver is developed to solve compressible Euler equations. The AUSM+[8] upwind scheme is used for computing inviscid fluxes. To achieve a second-order accuracy, a reconstruction of the assumed solution variation becomes necessary. Green-Gauss theorem based linear reconstruction procedure[9] is employed to compute gradient information at the cell centroids and Venkatakrishnan's limiter[9] is used as a limiter function in order to prevent the generation of oscillations and spurious solutions in the regions of high gradients.

The adaptive hybrid Cartesian grid consists of near body region body-fitted grids and Cartesian meshes. To keep data structure unique, two issues should be settled. First, a stencil for a hierarchical cell-based quadtree data structure is shown in Fig. 2. When computing cell I, if we treat the left edge j as one edge, it may cause problem of conservation. Therefore, a cell finite volume method is used to treat the edge j as two edges. While there are first interface j_1 between cell I and its lower level neighboring cell J_1 and the second interface j_2 between cell I and its lower level neighboring cell J_2. By using this approach the "hanging node" problem is avoided and the conservation problem is secured. Secondly, as shown in Fig. 3, the required connectivity information between two meshes should be established. An approach similar to that in Ref. [5] is used to treat the overset region, rather than using the "cutting cell" method established in Ref. [6]. In Fig. 3, interface j and cell I belong to the Cartesian grid and cell J belongs to body-fitted grid, $\boldsymbol{n}$ is the unit normal vector of interface j. To com-

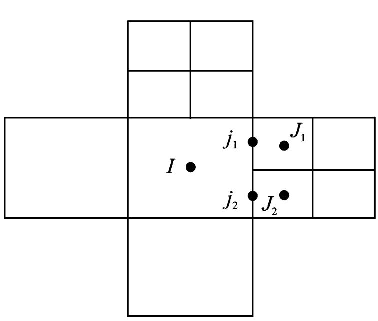


Fig. 2 Stencil of cells in adaptive Cartesian grid

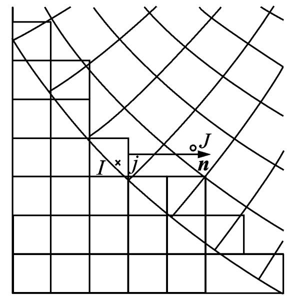


Fig. 3 Donor cell identification

put the fluxes across interface j needs to take cells I and J as the left and the right side cells, respectively. For interface j, cell I is given in advance. However, cell J, defined as a "donor cell", should be obtained by using some search method. When we use the traditional search method, which tests every cell to find one cell required, it will be time-consuming, especially for 3-D problems. Therefore, a "donor cell" search method based on alternating digital tree (ADT) technique[10] is used, which greatly shortened the search time.

2 RESULTS

2.1 Accuracy and efficiency study with vortex propagation problem

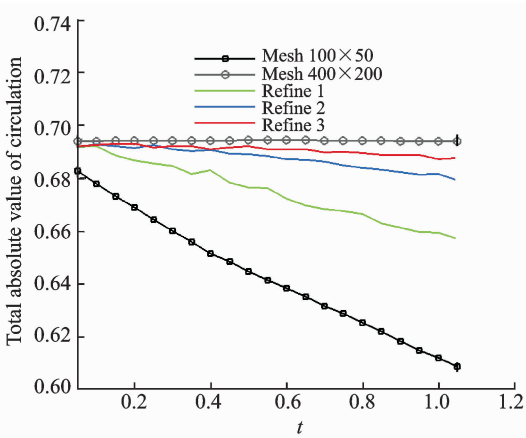
To verify the vorticity preserving capability of the DAMR process, we consider the problem of vortex convection by a uniform flow. The vortex was proposed by Jiang, et al[1]. The computational domain is taken as $[0,2]\times[0,1]$, and a vortex is superposed to the uniform flow and centers at $(x_c, y_c)=(0.25, 0.5)$. The vortex is described as a perturbation to the velocity (u, v), the temperature $(T=p/\rho)$ and the entropy $(S=\ln(P/\rho^\gamma))$ of the mean flow and we denote it by the tilde values

$$(\tilde{u},\tilde{v})=\varepsilon\tau \mathrm{e}^{\alpha(1-\tau^2)}(\sin\theta, -\cos\theta)$$
$$(\tilde{T},\tilde{S})=\left(-\frac{(\gamma-1)\varepsilon^2 \mathrm{e}^{2\alpha(1-\tau^2)}}{4\alpha\gamma}, 0\right) \tag{5}$$

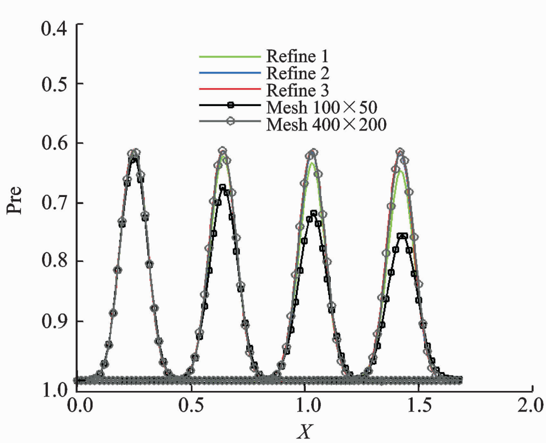
where $\tau=r/r_c$ and $r=\sqrt{(x-x_c)^2+(y-y_c)^2}$, ε indicates the strength of the vortex, α controls the decay rate of the vortex, and r_c is the critical radius of the vortex, p and ρ refer to the pressure and the density, respectively, γ is the ratio of specific heats, and set as $\gamma=1.4$. We chose $\varepsilon=0.5$, $r_c=$

0.05, α=0.204 and Ma=1.1 (supersonic flow).

Three different regular Cartesian meshes are chosen: 100 × 50 (h = 0.02), 200 × 100 (h = 0.01), 400 × 200 (h = 0.005), where h is the length scale of the mesh, and three different mesh refine times based regular Cartesian mesh 100×50. Fig. 4(a) shows the total absolute value of circulation changing with non-dimensional time (t). Fig. 4(b) refers to the pressure distributions (Pre) at different time on the centre line y=0.5, X is the non-dimensional coordinate along x direction. Table 1 presents the loss of total absolute value of circulation at different mesh scale and mesh refining times. The given results show that finer mesh scale will be more accurate to capture the vorticity and the diffusion of vortex will be decreased. At the same time, if we use the dynamic adaptive mesh refinement method by increasing refining times, the diffusion of the vortex can also be decreased.



(a) Time history of total absolute value of circulation



(b) Pressure distributions of different time on y=5

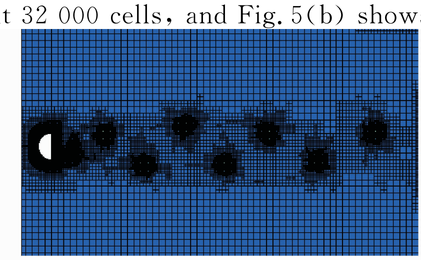
Fig. 4 Vortex propagation problem

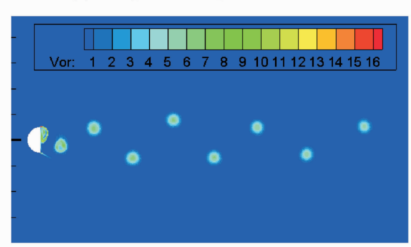
Table 1 Circulation loss and grid number

h	Circulation loss/%	Grid number
0.02	10.83	5 000
0.01	2.19	20 000
0.005	0.16	80 000
Refine Time	Circulation loss/%	Grid number
1	4.70	6 300
2	1.51	11 500
3	0.47	32 000

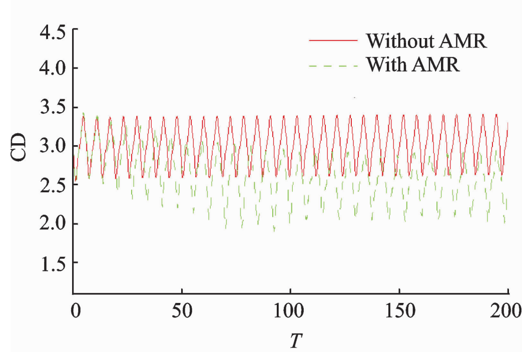
2.2 Flow over half cylinder

Now, consider a subsonic flow at Mach number Ma=0.3 through a half cylinder with non-dimensional radius r=0.5. Two singular points will cause unsteady vortex shedding. We use the test case to verify the capability of the dynamic adaptive hybrid Cartesian grid mesh to capture and track the unsteady vortex shedding. The initial mesh contains 18 149 cells, and three levels of mesh refinement are conducted based on the initial mesh. Fig. 5(a) shows the instantaneous adaptive computational mesh, which contains about 32 000 cells, and Fig. 5(b) shows the corre-


(a) Adaptive computational mesh



(b) Vorticity distribution



(c) Comparison of drag coefficients

Fig. 5 Flow over half cylinder