INTRODUCTION TO UNSTRUCTURED HYBRID MESH BASED FLOW SIMULATION TECHNIQUE

H.T. Ahn*1

비정렬 혼합격자 기반 유동해석 기법 소개

안 형 택*1

In this paper, flow simulation algorithms for utilizing unstructured hybrid meshes are introduced. First, various types of meshes are introduced. Advantages and disadvantages of each type of meshes are discussed. Unstructured hybrid mesh approach, that is best suited for high speed viscous flow simulation, is presented. Lastly, various types of flow simulations using unstructured hybrid meshes are introduced.

Key Words : 비정렬 혼합격자(Unstructured Hybrid Mesh), 고속 점성유동 (High Speed Viscous Flow)

1. INTRODUCTION

For viscous flow simulations, the superiority of the hybrid meshes over the conventional structured or unstructured meshes with simplexes is advocated by many researchers [1,2]. The hybrid meshes can combine good viscous layer resolving capability obtained from their structured elements, with the geometric flexibility of the simplicial unstructured meshes. The merits of the hybrid meshes can be further enhanced by introducing additional element types into the conventional hybrid meshes which typically consist of prisms and tetrahedrons. For example, placing local hexahedra on the viscous and wake regions can result in significant savings in the number of elements. The inclusion of hexahedra necessitates use of pyramids as buffer elements between hexahedra and tetrahedra which complicates application of the numerical solution methods.

In this paper, we present the solution method for using hybrid meshes for high speed viscous flow.

2. HYBRID MESH DISCRETIZATION

Conservative, finite-volume discretization scheme is used for solving the incompressible Navier–Stokes equations. A node-centered median dual volume is used for spatial discretization. An edge-based algorithm is used for the computation of the numerical fluxes [3,4]. For the viscous flux evaluation at an edge, another conceptual finite volume composed of its neighbor cells is used for the velocity gradient computation. A new computationally efficient algorithm is presented for the computation of the velocity gradients. This algorithm is composed of the first step of a face-wise loop to evaluate the surface integrals of edge-duals. The second step is an edge-wise operation for the final computation of velocity gradients and viscous fluxes.

The employed node-centered median dual control volume is shown in Fig. 1. The region indicated by dashed lines around node i represents node-duals in two dimensions. The node-dual is constructed by connecting lines defined by edge midpoints and centroids of the cells sharing the center node i. The L and R in Fig. 1 represent left and right sides of the node-dual boundary assuming that the edge is directed outward with respect to the node-dual i. In three dimensions as depicted in Fig. 2,

---

1 정회원, 울산대학교 조선해양공학부
* E-mail: htahn@ulsan.ac.kr

the node dual is constructed by connecting faces (instead of lines in two dimensions) defined by edge midpoints, cell centers and face centers sharing the common node n.

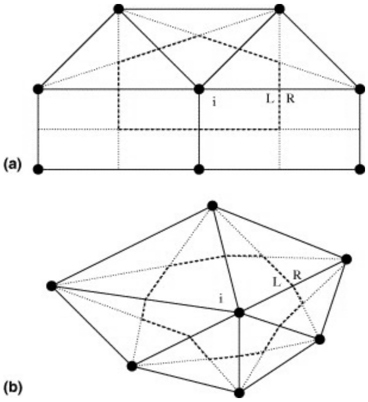

Fig. 1 Node-duals in two dimensions: (a) node-dual with mixed cells, (b) node-dual with triangular cells.

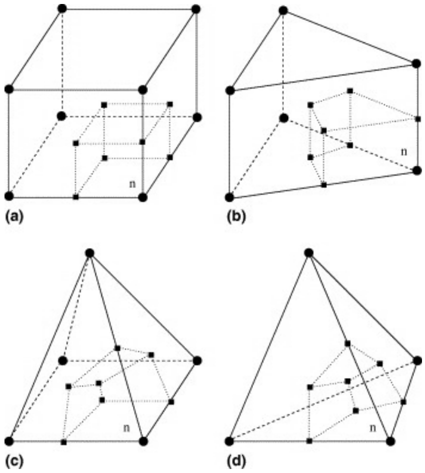

Fig. 2 Contributions to the dual volume of node n from different types of elements in three dimensions: (a) hexahedron, (b) prism, (c) pyramid, and (d) tetrahedron.

In order to evaluate the viscous fluxes through the control volume boundaries, the gradients of velocity components are needed to be pre-computed at each edge. For this velocity gradient computation, another conceptual finite volume called edge-dual is constructed. The edge-dual is composed of the neighbor cells sharing a common edge. Various kinds of edge-duals encountered in hybrid meshes are delineated in Fig. 3 for two dimensions and in Fig. 4 for three dimensions.

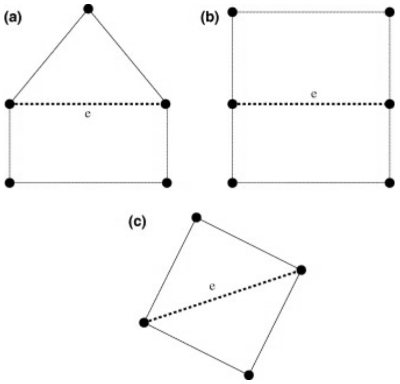

Fig. 3 Edge-duals in two dimensions for computation of the first order spatial derivatives.

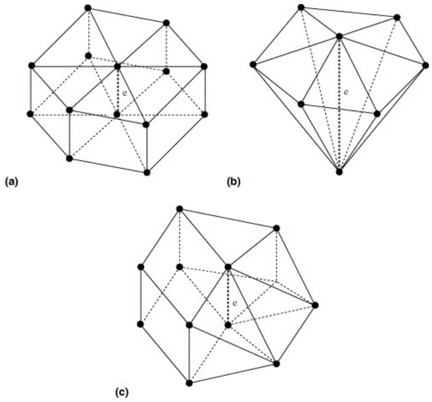

Fig. 4 Edge-duals in three dimensions for computation of the first order spatial derivatives: (a) edge-dual composed of prism, (b) edge-dual composed of tetrahedra, (c) edge-dual composed of mixed elements.

3. RESULT

The general hybrid mesh used for the current study is displayed in Fig. 5. It includes local hexahedra on the frontal viscous region of the cylinder, prisms in its rear half, prisms at the interface between the hexahedral and tetrahedral regions, and tetrahedrons for the rest.

To take advantage of the merits of local hexahedra, the general hybrid mesh 1 as shown in Fig. 5 is modified with inclusion of local hexahedra in the wake region of the cylinder, as well. The resulting mesh is shown in Fig. 6. This mesh will be termed general hybrid mesh-2 (GHM-2) in the following. The first mesh of Fig. 5

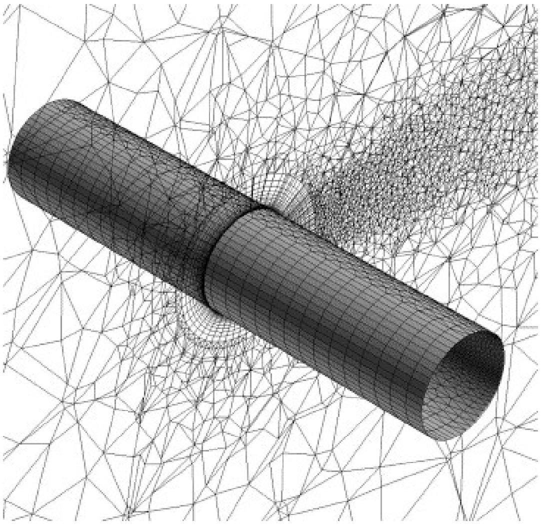

Fig. 5 General hybrid mesh-1 (GHM-1) containing hexahedra in the frontal region of the cylinder.

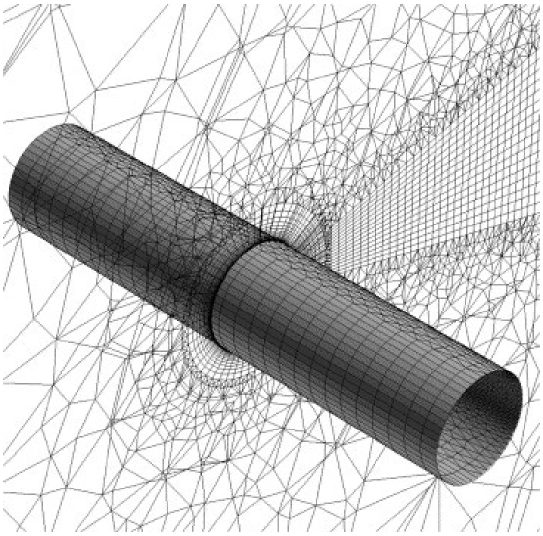

Fig. 6 General hybrid mesh-2 (GHM-2) containing hexahedra in the wake region as well as in the frontal region of the cylinder. L/D = 5.

contains local hexahedra only in the frontal area of the cylinder.

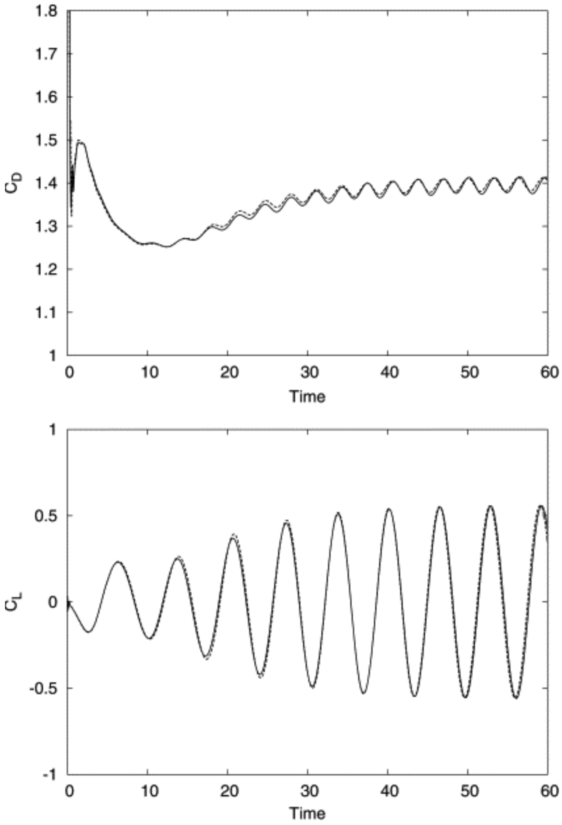

Fig. 7 Local hexahedra effect on C_D and C_L histories. Solid lines are for the GHM-1 with tetrahedra in the wake region, and the dashed lines are for the GHM-2 with hexahedra in the wake region (Re = 150).

The C_D and CL histories from the flows around a circular cylinder by using the two general hybrid meshes are presented in Fig. 7. The results using the two hybrid meshes are almost identical in terms of the hydrodynamics forces exerted on the cylinder. This is important as it provides an indication that the accuracy of the developed scheme is not affected by the presence of extra interfaces between hexahedra, pyramids and tetrahedra, as well as extra pyramids in mesh GHM-2.

4. CONCLUSION

Applicability of general hybrid mesh for viscous flow simulation is illustrated.

ACKNOWLEDGEMENT

This work was partly supported by the SOTOP Porject of the School of Naval Architecture and Ocean Engineering at the University of Ulsan.

REFERENCES

[1] 1999, Kallinderis, Y., "Hybrid grids and their applications. In: J.F. Thompson, B.K. Soni and N.P. Weatherill, Editors," *Handbook of grid generation*, CRC Press, Boca Raton, FL.

[2] 1997, Mavriplis, D.J., "Unstructured grid techniques," *Annual Review of Fluid Mechanics 29*, pp.473-514.

[3] 1992, T.J. Barth, "Aspects of unstructured grids and finite-volume solvers for the Euler and Navier-Stokes equations," *Technical Report AGARD Report 787 on unstructured grid methods for advection dominated flows*, AGARD, pp.6.1-6.60.

[4] 2005, Kallinderis, Y. and Ahn, H.T., "Incompressible Navier-Stokes method with general hybrid meshes," *Journal of Computational Physics*, Vol.210, pp.75-108.

[5] 2006, Ahn, H.T. and Kallinderis, Y., "Strongly coupled flow/structure interactions with a geometrically conservative ALE scheme on general hybrid meshes," *Journal of Computational Physics*, Vol.219, pp.671-696.