Abstract of Dissertation

Title of Dissertation:
Influence of Cell Topology on the Numerical Accuracy of CFD based on FVM

(FVM に基づく CFD におけるセル形状が数値予測精度に及ぼす影響)

氏　　名　**ヘフニー　モハマッド　モハマッド**

Abstract:

With the advent of modern computers, detailed simulations of flow field around complicated geometries using Computational Fluid Dynamics (CFD) are now routinely employed in many fields of industrial and environment relevance. This has greatly stimulated the development of general numerical algorithms, which based on Finite Volume Method (FVM), and unstructured meshing approaches as credible design aids for these complex turbulent flows for several reasons. Foremost, unstructured meshes are capable to map complex domains and it turned out to be the largest flexibility method in the treatment of complex geometries. Also, the discretization practice of numerical algorithms are based on arbitrary control volumes and sequentially gives considerable freedom in meshing complex geometrical configurations in three spatial dimensions. However, the cell topology of the unstructured mesh determines the ultimate geometry of the control volume regardless of the discretization scheme type being cell- or vertex-centered scheme and hence affects the overall numerical accuracy. The objective of this study is to better establish quantitative and qualitative assessments of the influence of cell topology in the computational mesh on the CFD results in order to help modelers to choose the most effective mesh type for their applications.

In quantitative assessment part, numerical experiments are performed with different meshing approaches under similar numerical and mesh resolution conditions in order to lucid a physical picture of cell topology effect. It include two- and three-dimensional problems of engineering interest, for which either exact or benchmark numerical solutions are available or reliable experimental data exist, including line source in cross-flow, natural convection in thermal cavity, turbulent flow around single building, gas diffusion near single building, and turbulent flow over backward facing step. The comparative results and the grid convergence assessment showed that the cell topology strongly affects the overall accuracy of the numerical predictions for all cases considered. The hexahedral-based meshes were shown to provide the best computational solution based on the lowest numerical error values and high levels of agreement with experimental data while other mesh styles including tetrahedral-based meshes resulted in much higher numerical error values.

The qualitative assessment showed that using cells rather than orthogonal hexahedral cells violates the order of discretization and it effectively reduces the accuracy of face integrals to first order. The non-orthogonality treatment and the defer correction formula are checked showing that the size of the computational molecule (stencil) in case of tetrahedral-based meshes is larger than that of the hexahedral-based mesh. Although discretizations of both meshing approaches are second order accurate, the increase in the computational module implies a larger leading term in the Taylor series expansion and higher error. Moreover, the influence of cell aspect ratio is shown to be very pronounced in determining the quality of the CFD simulations. Even if the mesh is very fine, its effect still exists in the computational solution especially in the case of tetrahedral- or triangular-based meshes. Finally, the study gave some recommendations to the CFD modelers to be followed when constructing the unstructured mesh for their applications.