11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

A HIGH-ORDER UNSTRUCTURED MIXED MESH METHOD FOR ROTOR AERODYNAMIC PREDICTION

Min Kyu Jung[1], Je Young Hwang[2] and Oh Joon Kwon[3]*

[1] KAIST, Daejeon 305-701, Korea, mercury_mk@kaist.ac.kr
[2] KAIST, Daejeon 305-701, Korea, oiorrrr@kaist.ac.kr
[3] KAIST, Daejeon 305-701, Korea, ojkwon@kaist.ac.kr

Key Words: *Unstructured Mixed Mesh, High-Order WENO Scheme, Rotor Aerodynamics.*

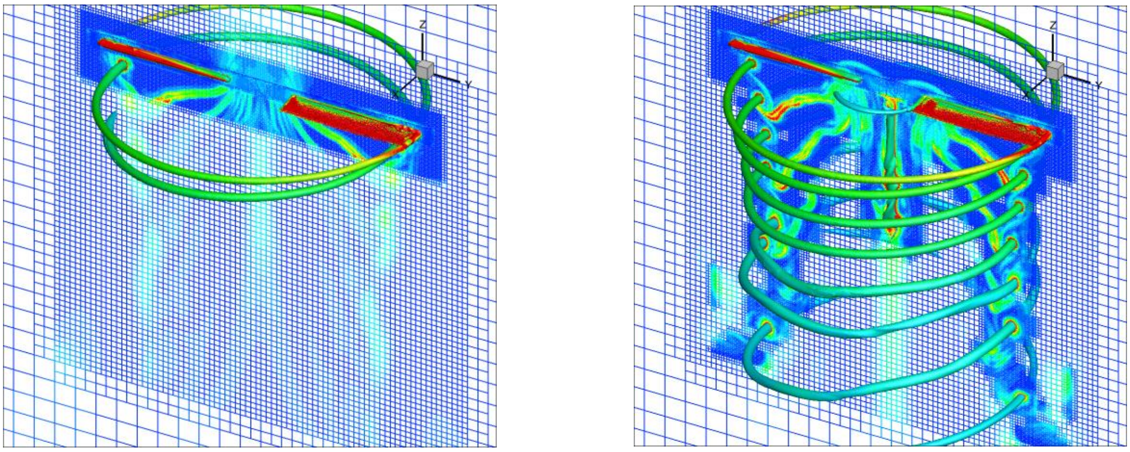
Accurate prediction of the flow fields around rotorcrafts is one of the most challenging problems in the field of applied aerodynamics. The key aspect in the rotorcraft flow simulations is the accurate capturing of rotor wake that has significant impacts on the blade aerodynamic loading, overall vehicle performance, vibration, and noise. The main difficulty involved in the rotorcraft aerodynamic simulations is in the fact that the trailed tip vortices have very small core size and convect over a relatively long distance. To capture these tip vortices accurately with less numerical dissipation for given order of accuracy of the scheme, dense cells need to be distributed along their trajectories. Solution-adaptive mesh refinement methods based on feature-detection techniques are also adopted to locally enhance the density of mesh. Another approach of accurately capturing the tip vortices and avoiding excessive numerical dissipation is to use high-order accurate numerical schemes.

To overcome the shortcomings of using single mesh topology and to achieve high-order accurate solution, dual-mesh solution methods have been proposed. The OVERFLOW solver was developed by adopting structured body-fitted grids in the near-body region, and multi-level Cartesian grids in the off-body domain [1]. The two grid zones are coupled with an overset mesh method by using a domain connectivity module. Recently, the Helios flow solver has been developed based on a hybrid grid paradigm to allow more flexible modeling of complex geometries [2]. HELIOS employs a parallel unstructured mesh flow solver for near-body and the serial high-order structured flow solver for the Cartesian grids at off-body. However, the file-based data exchange between the two independent heterogeneous mesh solvers is typically known to degrade the solution efficiency and the parallel scalability of the coupled code.

In the present study, a new unstructured mixed mesh flow solver was developed for simulating the flows around rotors more accurately and efficiently. In the present mixed mesh topology, the near-body flow domain is modeled by using a body-fitted prismatic/tetrahedral mesh, and the remaining region away from the body is filled with a Cartesian-based mesh. The near-body domain is discretized using a second-order node-based finite-volume method, while the outer domain is discretized with a cell-centered finite-volume method based on a high-order accurate WENO scheme. To further enhance the accuracy of the solution, a solution-adaptive mesh refinement technique is also applied for the off-body Cartesian mesh region.

For validation, the Caradonna-Tung rotor in hovering flight, which was experimentally tested [3], was simulated. The rotor has two blades, and the blades are made of NACA0012 airfoil sections with an untwisted and non-tapered planform of an aspect ratio of six. The flow simulation is conducted at a tip Mach number of 0.877, a Reynolds number of 3.93 million, and a collective pitch angle of eight degrees. Mesh adaptation was conducted up to the second refinement level after the flow solution is fully converged.

Figure 1 shows comparison of the wake structures represented by the iso-λ_2 surface for the initial and adaptive meshes with the seventh-order WENO scheme in the off-body Cartesian mesh region. The typical minimum cell size at the initial off-body Cartesian mesh is $\Delta x = 0.05c$. In the case of the initial mesh solution, the wake vortices are emanated from the rotor blade tips, but are quickly diffused, even at seventh-order accuracy. After applying the adaptive mesh refinement scheme, it is observed the wake structures are much better resolved further downstream in the far wake by maintaining the shape and the strength along the adaptively refine cells.



(a) Initial mesh solution (b) Adaptive mesh solution

Figure 1. Predicted wake structures for Caradonna-Tung rotor with 7th-order accurate method. (a) Initial mesh solution; (b) Adaptive mesh solution.

ACKNOWLEDGMENTS

This work was supported by the National Research Foundation of Korea(NRF) grant No. 2009-0083510 funded by the Korea government(MSIP) through Multi-phenomena CFD Engineering Research Center. This work was also supported by National Research Foundation of Korea(NRF) Grant funded No. 2011-0029094 by the Korean Government.

REFERENCES

[1] Buning, P. G., et al, "OVERFLOW User's Manual," NASA Langley Research Center, Jul, 2003.
[2] Wissink, A. M, Sitaraman, J., Sankaran, V., Mavriplis, D. J., and Pulliam, T. H., "A Multi-Code Python-Based Infrastructure for Overset CFD with Adaptive Cartesian Grids," 46th AIAA Aerospace Science Meeting, AIAA 2008-927, 2008.
[3] Caradonna, F. X. and Tung, C., "Experimental and Analytical Studies of Model Helicopter Rotor in Hover", NASA TM 81232, Sep. 1981.