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

DOMAIN FREE DISCRETIZATION METHOD FOR INCOMPRESSIBLE NAVIER-STOKES EQUATIONS AND ITS APPLICATION TO SIMULATE VORTEX SHEDDING FROM AN OSCILLATING CIRCULAR CYLINDER

***Chun-Hua Zhou[1] and Ke-Ming Cheng[1]**

[1] Department of Aerodynamics, Nanjing University of Aeronautics and Astronautics,
Nanjing 210016, P. R. China
chzhou@nuaa.edu.cn

Key Words: *Domain-Free Discretization, Vortex Shedding, Boundary Condition, Moving bodies.*

ABSTRACT

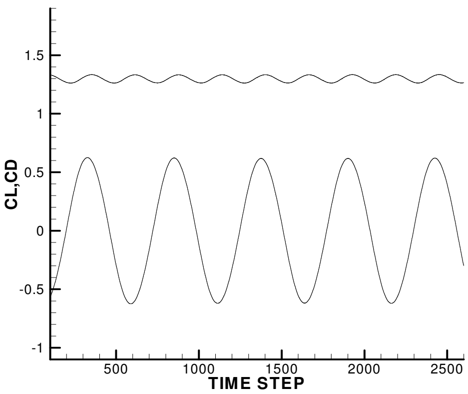
The basic idea of the domain-free discretization (DFD) method is the extension of PDE(s) solution, and is inspired from the analytical method [1-2]. In this paper, the local DFD method is presented to solve the incompressible Navier-Stokes equations. Being different from classical numerical methods, the discrete form of governing equations at an interior point may involve some points outside the solution domain. The functional values at the exterior dependent points are updated at each time step to impose the wall boundary condition by the approximate form of solution in the normal direction to the wall boundary. Some points inside the solution domain are constructed for the approximate form of solution, and the flow variables at the constructed points are evaluated by linear interpolation over triangles. The method of artificial compressibility in [3] is used to solve the flow equations. The numerical schemes used in DFD are the finite-element Galerkin approximation for spatial discretization and the dual-time scheme for temporal discretization.

In this local DFD method, wall boundaries can be superimposed upon computational meshes. The mesh can stay fixed while the body is moving, and there is no need to generate a new mesh or deform the previous mesh at each time step. In principle, mesh generation for complex geometry problems becomes quite easy.

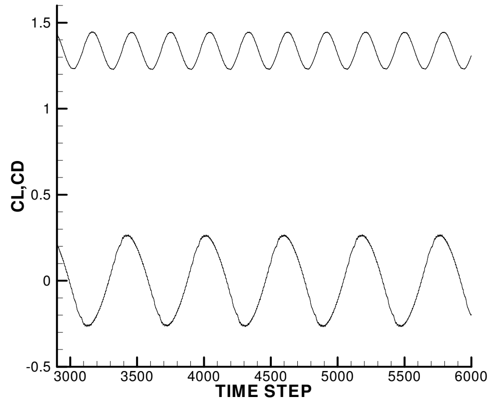
The simulation of flows past a fixed cylinder at different Reynolds numbers (85<Re<190) is included as a basis of comparison with other numerical results and experimental data for a complete validation of the DFD method.

Two phenomena are investigated in the simulation of flows around moving boundaries. In both cases, the cylinder oscillation is forced. The first is the flow induced by the harmonic in-line oscillation of cylinder in fluid at rest. The Reynolds number is equal to 100 and the Keulegan-Carpenter number is equal to 5. The second is the flow induced by the transverse oscillation of a cylinder in uniform flow at fixed Reynolds number equal to 185. The cylinder oscillation frequency ranges from 0.8 and 1.20 of the natural vortex shedding frequency, and the oscillation amplitude is 20% of the cylinder

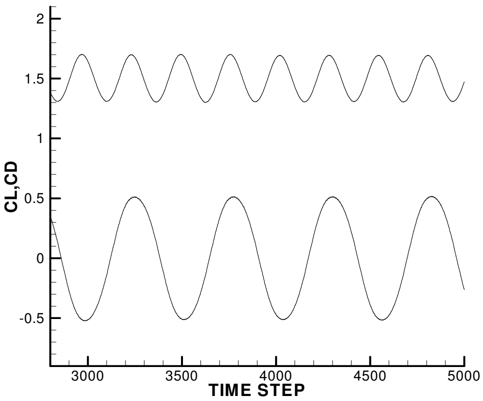
diameter. Again for validation of the DFD method, comparisons with measurements and predictions are made in both cases. Some of results for case 2 are presented below.



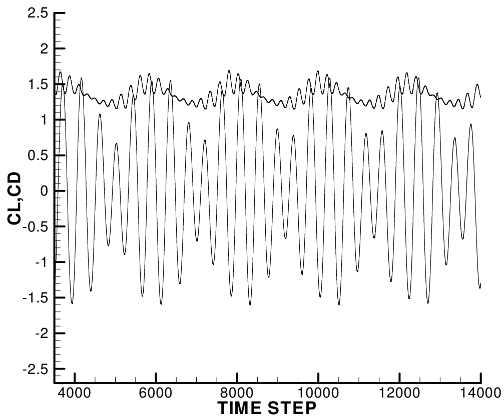
(a) $f / f_e = 0.0$



(b) $f / f_e = 0.9$



(c) $f / f_e = 1.0$



(d) $f / f_e = 1.2$

Lift and drag coefficients versus time Re=185

REFERENCES

[1] C. Shu and Y. L. Wu, Domain-free discretization method for doubly connected domain and its application to simulate natural convection in eccentric annuli, *Comput. Methods Appl. Mech. Engrg.* 191, 1827-1841 (2002).

[2] C. H. Zhou, C. Shu and Y. Z. Wu, Extension of Domain-Free Discretization Method to Simulate Compressible Flows over Fixed and Moving Bodies, *International Journal for Numerical Methods in Fluids*. 53: 175-199 (2007).

[3] P. Lin, L. Martinelli and T. J. Baker, Two-dimensional implicit time dependent calculations for incompressible flows on adaptive unstructured meshes. *15th AIAA Computational Fluid Dynamics Conference*, 11-14 June. Anaheim, CA. AIAA 2001-2655.