Vortex Particle Method for Aerodynamic Analysis: Parallel Scalability and Efficiency

Khaled Ibrahim*, Guido Morgenthal

Bauhaus-Universität Weimar, 99423 Weimar, Germany
E-mail: khaled.ibrahim.tolba@uni-weimar.de
Web page: http://www.uni-weimar.de/Bauing/MSK/19.html

ABSTRACT

The Vortex Particle Methods (VPM) uses a grid free formulation in order to solve the Navier-Stokes equations of an in compressible fluid flow. The VPM models vorticial flows typical for aerodynamic applications especially around bluff bodies i.e. moderate to high Reynolds number via solving the vorticity transport equations in Lagrangian form. Vortex sheets are modeled as particles with invariant strength which are transported downstream by convection and diffusion due to the induced velocity field and the free stream velocity. The vorticity field can be computed as superposition of the local vorticity created by each particle. Based on the Bio-Savart law the induced velocity field is computed and accordingly the particles are convected through a suitable time integration scheme. Singularities may arise when two particles approach each other at infinitesimally small separation; this is avoided by convolution of the Bio-Savart integral with a suitable mollification kernel e.g. Gauss kernel.

Different algorithms are available in literature for solving the Bio-Savart integral. For N particles discretization of the domain, the "naive" particle-particle interaction method scales as $O(N^2)$. Alternatively within the current implementation of the VPM method used for this work, an enhanced algorithm termed the Particle-Particle—Particle-Mesh (P3M) algorithm is used, which requires an intermediate correction step that can be efficiently computed in parallel. Graphical Processing Units (GPUs) where used in a thread-per-particle implementation. Here, in parallel fashion each thread calculates the contributions on its local particle due to all the other particles at a given point in time.

A standard procedure for simulation of fluid structure interactions with a line-like bluff body is a two-dimensional sectional analysis model. Here, the aerodynamic effects perpendicular to the flow direction are disregarded. An extension to this approach is modeling the inherently three-dimensional problem as a coupled set of two-dimensional simulation slices, which accounts for the characteristics of the bluff body as well as the aerodynamics. Since each slice could be simulated independently at each time step, OpenMP parallelization is used to compute each slice in parallel, a synchronization point at the end of the time step is essential for coherence between the CFD solutions and the structural dynamics solution. For the local particles interaction computations at each slice the GPU resources are managed in a serial manner.

The focus of this paper is the analysis of the parallel efficiency and scalability of the method being applied to an engineering test case, namely the buffeting response of a long-span bridge girder. The target is to assess the optimal configuration and the required computer architecture, such that it becomes feasible to efficiently utilize the method within the computational resources available for a regular engineering office e.g. the simulations are performed on a gaming-type computer.

REFERENCES

[1] Cottet, G.H., Koumoutsakos, P., 2000. Vortex Methods: Theory and Practice. Cambridge University Press.
[2] Morgenthal, G., 2002. Aerodynamic Analysis of Structures Using High resolution Vortex Particle Methods. Ph.D. thesis. University of Cambridge.
[3] Rossinelli, D., Bergdorf, M., Cottet, G.H., Koumoutsakos, P., 2010. GPU accelerated simulations of bluff body flows using vortex particle methods. J.Comput. Phys. 229, 3316–3333.
[4] Morgenthal, G., Corriols, A.S., Bendig, B., 2014. A gpu-accelerated pseudo-3d vortex method for aerodynamic analysis. Journal of Wind Engineering and Industrial Aerodynamics 125, 69 – 80