GTC Workshop Japan 2011 (2011年7月22日）

Numerical Simulation on the Air Flow in an Urban City by Lattice Boltzmann Method using Multi-Node GPU Cluster

Wang Xian[1], Aoki Takayuki[1], Tsuzuki Satori[1]

1) Tokyo Institute of Technology GSIC (〒152-8550 Meguroku Oookayama 2-12-1 GSIC Collaboration Building 3F)

Key Words : *CUDA,LBM, GPU cluster, Performance*

1. Introduction

Recently, GPU has been further used for general purpose with the release of CUDA (Compute Unified Device Architecture) by NVIDIA. In computational fluid dynamics (CFD), Lattice Boltzmann method (LBM) is widely used currently as an alternative numerical scheme for simulating fluid flows governed by the Navier-Stokes equations[1]. Since GPU has many computational units and the weak correlations among adjacent computational points in solving the LB equation well match the data-parallel SIMT (Single-Instruction Multiple-Thread) characteristic of GPU, it is expected to obtain high efficiency of GPU on the LBM solver.

2. Governing Equations

The completely discretized form of Boltzmann equation with BGK model is

$$f_i(x+e_i\delta t, t+\delta t) - f_i(x,t) = -\frac{1}{\tau}\left[f_i(x,t) - f_i^{eq}(x,t)\right]$$

It is usually solved with its standard form by assuming $\delta t = 1$ according to the following two steps.

Collision step: $\bar{f}_i(x,t) = f_i(x,t) - \frac{1}{\tau}\left[f_i(x,t) - f_i^{eq}(x,t)\right]$

Streaming step: $f_i(x+e_i, t+1) = \bar{f}_i(x,t)$.

3. Model system

The model system is shown in Fig.1. The computational domain is 1536×768×768. The buildings were simulated by some rectangular blocks. The inlet velocity $U_{inlet}(z)=U_{top}(z/H_{max})^{0.2}$ and Re= $U_{top}\,H_{max}/\nu$. As the first step of this study, the turbulence model was not included and Re=5000. The bounce back boundary conditions were imposed for the solid boundaries of buildings and ground. As for other boundaries, zero flux was used for velocity and density.

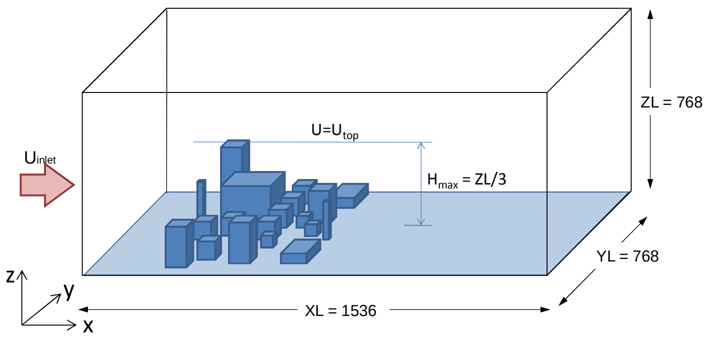
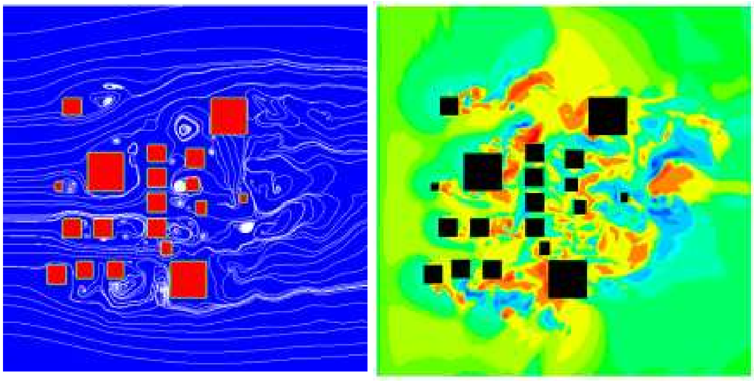


Fig. 1 Model system and solution domain.



(a) (b)

Fig. 2 (a) Stream lines and (b) w-contours at z=0.075ZL.

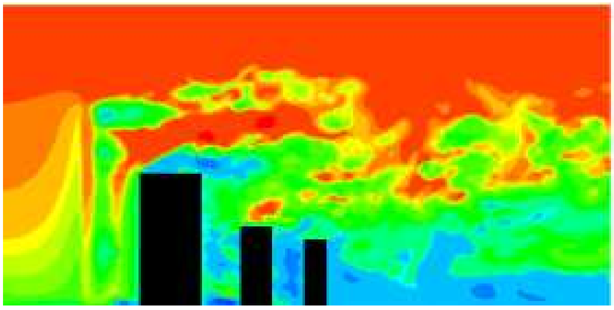


Fig.3 u-contours at cross section y=0.53YL.

4. Results

We use multi-node GPU cluster to simulate the flow field and the solution domain was decomposed in a 3D way to decrease the communication load [2]. Fig.2 shows (a) the stream lines and (b) w velocity contours in z=0.075ZL cross-section, which is near to ground. It can be seen from (a) many vortex appear behind buildings. From (b) we can see near the buildings, the air goes up, w (velocity in z direction) is large (red). Fig.3 shows the u velocity contours in y=0.53YL cross section. It can be seen u (velocity in x direction) is small (blue) behind the buildings, the flow almost does not move forward. Figures 2 and 3 show zone (XL/2, YL, ZL/2) only to avoid huge output data. For the computation, we used 96, 144 and 216 GPUs. The obtain performances are 3425, 4635 and 6445 GFLOPS, respectively. The ratios between computational time and communicational time are 3.7, 2.7 and 2.1 respectively.

5. Conclusions

LBM solver is very suitable to run on GPUs and high performance can be obtained.

Reference

[1] S.Y. Chen S.Y. and Doolen G.D.: Lattice Boltamann method for fluid flows, *Annual Review of Fluid Mechanics*, Vol. 30, pp. 329-364, 1998.

[2] Wang X. and Aoki T.: Multi-GPU performance of incompressible flow computation by lattice Boltzmann method on GPU cluster, *Parallel Computing*, 2011, doi:10.1016/j.parco.2011.02.007 (in press)