29th Nordic Seminar on Computational Mechanics
NSCM-29
R. Larsson (Ed.)
2016

URBAN CFD-SIMULATION USING POINT CLOUD DATA

SIMON INGELSTEN*, ANDREAS MARK*, FREDRIK EDELVIK*, ANDERS LOGG† AND MAGNUS ÖSTERBRING⋆

* Fraunhofer-Chalmers Centre for Industrial Mathematics (FCC)
Gothenburg, Sweden
e-mail: simon.ingelsten@fcc.chalmers.se, web page: http://www.fcc.chalmers.se/

† Mathematical Sciences, Chalmers University of Technology
Gothenburg, Sweden

⋆ NCC and Civil Engineering, Chalmers University of Technology
Gothenburg, Sweden

Key words: Computational Fluid Dynamics, Immersed Boundary methods, Point Cloud

Introduction

Accurate point cloud representations of complex geometries are today relatively easy to obtain at a low cost with a 3D-scanner. The majority of present CFD simulation tools require a body-fitted mesh in order to impose the correct boundary conditions on geometrical objects. Further, the generation of the mesh is generally based on some form of surface representation of the geometrical object itself. This means that a high-quality surface representation must first be generated from the point cloud which is a challenging and time-consuming task. Examples of studies where CFD simulations combined with point cloud data have been used are simulation of thermal environment[1] and simulation of flooding risk[2]. In both these cases a 3D model had to be extracted from the data for the point cloud to be used in the numerical model. In this study the fluid flow solver IPS IBOFlow[3], developed at the Fraunhofer-Chalmers Research Centre for Industrial Mathematics, is used for CFD simulation directly on a point cloud.

Method

In IPS IBOFlow[3] the incompressible fluid momentum and continuity equations are discretized and solved on a Cartesian octree grid and the pressure-velocity coupling is solved in a decoupled way using the iterative SIMPLEC algorithm. All variables are stored co-located in the cell centers and Rhie-Chow interpolation is used to avoid pressure oscillations. Interior objects are treated using a novel hybrid immersed boundary (IB) method[4,5], where the solution is mirrored over the solid boundary in such a way that the no-slip condition is fulfilled at the surface. Mass conservation is ensured by excluding the fictitious velocity field that occurs inside the solid from the continuity equation. Objects of arbitrary shape represented by a triangulated geometry may be included in the simulations. A conjugated heat transfer solver including convection, conduction and radiation has recently been developed[6]. The immersed boundary techniques are also perfectly suitable for fluid-structure interaction applications[7].

In the current work the solver is further developed for simulation directly on scanned point clouds through a voxelization technique. Each point of the cloud defines a voxel of certain size and cells intersecting at least one voxel are marked as solid cells that define the immersed boundary. The octree grid is automatically refined around the voxelization IB. Fluid cells that are not connected to an outlet are automatically converted to solid cells to ensure a physical solution for the pressure field.

A case study is performed by simulating air flow on a part of the open scanned point cloud of the Johanneberg campus at Chalmers University of Technology, Gothenburg, Sweden. The point cloud is shown in Figure 1, and in Figure 2 the resulting voxelization and example of grid refinements around an object in the point cloud are shown.

Figure 1: Part of the open point cloud of the Johanneberg Campus at Chalmers University of Technology as seen from above (left), from the main entrance (middle) and from the technology garden (right).

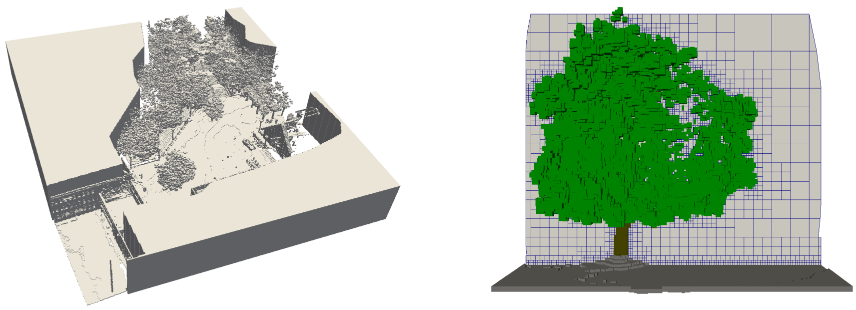

Figure 2: Immersed boundary cells in the point cloud discretization (left) and grid refinements around one of the trees in the technology garden (right).

The simulation setup is shown in Figure 3 together with an overview of the resulting fluid velocity field. At the Chalmers main entrance, to the bottom left in the figure, an inlet velocity of 5m/s is defined. Two outlets are also used. The flow is solved to steady state, which is reached in only 3 minutes for a grid with 1.7 million fluid cells and in 30 minutes for a grid with 7.7 million cells on a workstation computer.

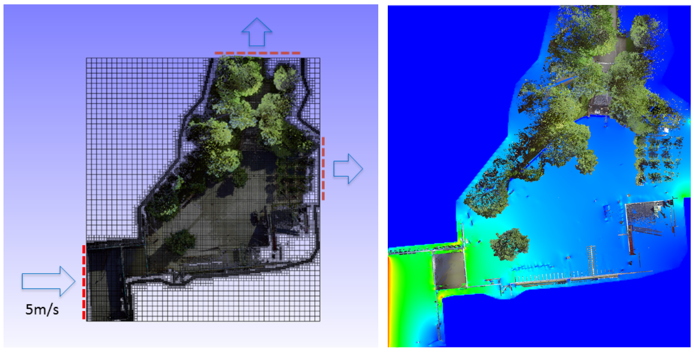

Figure 3: Simulation setup with the point cloud inlet, outlets and octree grid, seen from above (left) and resulting velocity field (right).

Simulation results

In Figure 4 the resulting flow field is shown through a single tree in the yard and around the trees outside the student union building. From the figures it can clearly be seen that the immersed boundary method imposes the no-slip wall condition on the fluid at the surface of the scanned geometries.

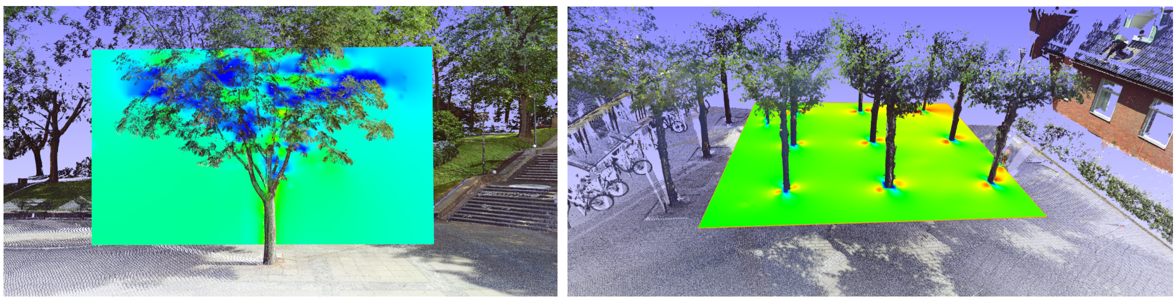

Figure 4: Computed velocity field through one of the trees in the point cloud (left) and at the trees next to the student union (right).

Summary

The proposed method combines immersed boundary techniques with point cloud data in a robust way. In addition to the very efficient preprocessing, the simulation itself is fast and a steady state solution is obtained in a few minutes. In addition to the scanned point cloud, it also possible to include CAD-objects or even additional point clouds in the same simulation, such that the effect of the placement of an object in a scanned environment can be analyzed. A similar analysis in software using body fitted meshes would require the complex manual preprocessing to be executed again for each new geometrical setup. With the immersed boundary method, on the other hand, arbitrary combinations may be simulated with minimal extra effort from the user. For planning and design of buildings and investigation of urban environments, this type of CFD simulations can therefore be used to, for example, predict wind loads, temperatures or spreading of pollution in a highly effective way.

Acknowledgement

This work was supported by Building Futures, an Area of Advance at Chalmers.

REFERENCES

[1] Maragkogiannis, K., Kolokotsa, D., Maravelakis, E., Konstantaras, A., *Combining Terrestrial Laser Scanning and Computational Fluid Dynamics for the Study of the Urban Thermal Environment*, Sustainable Cities and Society, 13, 207-216, 2014

[2] Meouche, R.E., Rezoug, M., Vandevenne, W., Hamzaoui, R., *Using a Laser Scanning to Construct a 3D Numerical Model to Stody the Flooding Risk in Urban Area*, Journal of Engineering and Technology,5(3), 2013

[3] IPS IBOFlow <http://ipsiboflow.com/>

[4] Mark, A., van Wachem, B. G. M., *Derivation and validation of a novel implicit second-order accurate immersed boundary method.* J. of Comput. Physics 227, 6660–6680 (2008)

[5] Mark, A., Rundqvist, R., Edelvik, F., *Comparison Between Different Immersed Boundary Conditions for Simulation of Complex Fluid Flows.* Fluid Dyn. Mater. Proc. 7, 241– 258 (2011)

[6] Mark, A., Svenning, E., Edelvik, F. *An Immersed Boundary Method for Simulation of Flow With Heat Transfer*, International Journal of Mass and Heat Transfer, 56(1-2), 424-435, 2013

[7] Svenning, E., Mark, A., Edelvik, F., *Simulation of a Rubber Beam Interacting with a Two-Phase Flow in a Rolling Tank*, European Conference on Mathematics for Industry, Lund, Sweden, 2012