XIX International Conference on Water Resources
CMWR 2012
University of Illinois at Urbana-Champaign
June 17-22, 2012

SIMULATION OF FLOW FIELD AROUND AND INSIDE SCOUR PROTECTION WITH PHYSICAL AND REALISTIC PARTICLE CONFIGURATIONS

Xiaofeng Liu*, Yijiu Jiang† and Rusen Sinir††

*University of Texas at San Antonio
Department of Civil and Environmental Engineering
One UTSA Circle, San Antonio TX 78249 USA
e-mail: xiaofeng.liu@utsa.edu, web page: http://engineering.utsa.edu/~xiaofengliu/

†University of Texas at San Antonio
Department of Civil and Environmental Engineering
e-mail: jaq526@my.utsa.edu

††University of Texas at San Antonio
Department of Civil and Environmental Engineering
e-mail: rusensnr@gmail.com

Key words: Scour protection, computational modeling, rock arrangement, collision detection

Summary. We present a novel method toward physical and realistic simulations for porous scour protections, which is inspired by the collision detection and rigid body dynamics algorithms in computer graphics. In engineering practice, to protect hydraulic structures such as bridges and offshore platforms from scour damage, layers of stones with different sizes are put around these structures to reduce the sediment motion. These stones form a porous media layer and interact with the flow field around the structures. This model is comprised of two parts. The first is the tool to generate the configurations of the stones. The second is the immersed boundary method to simulate the flow field both outside and inside the scour protection.

1 INTRODUCTION

To protect the structures built in riverine and ocean environments from scour and foundation damage, scour protections are one of the key design elements. For rivers and streams, scour protections are also used in many places to shield the banks from being eroded by the turbulent flows. Scour protections are usually composed of stone and geomat covers.

The fundamental idea of scour protection is to create a barrier between the erodible sediment and the flow. However, the layers of stones in the scour protection are not watertight. In between the stones, there exist voids which permit outside turbulence to penetrate into the protection and possibly reach the sediment. The seeping flow to the sediment can be high enough to initiate and sustain its motion. To predict the effectiveness of porous scour protection and for better design, a detailed and realistic description of the 3D flow field both inside and around the protection is necessary. To consider the porous scour protection in numerical models, one could do pore-scale modeling. In this approach, there are two steps. The first step is the arrangement of the stones. The second step is how to incorporate these stones into the fluid dynamics model.

In this paper, we report the progress made in our group to deal with both steps. The structure of this paper is the following. First, the numerical model and our approach are presented. Then a preliminary simulation case for flow around and inside a scour protection for pile foundation is demonstrated. Then we will conclude with some discussions.

2 NUMERICAL MODELS

2.1 Generation of physical and realistic stone layers

For simple geometries such as spheres and boxes, it is relatively easy to arrange them in space. However, stones usually have irregular shapes in reality and it is not easy to give a physically correct configuration in space. These irregular stones are defined to be physically configured when they are stable in position and not penetrating into each other. To achieve this, we use the collision detection algorithm in computer graphics to mimic the construction process of these protection layers with the following steps:

- First, we scanned the stones using a 3D laser scanner and stored the surface geometries (Figure 1);
- Then, the stone surface geometries were randomly picked and dropped to the simulation domain around the hydraulic structures. The stone dropped will interact with those previously dropped and the environment such as the bed and the structures. In this research, we assumed that every object is a rigid body with certain properties such as density, friction, restitution, etc. When the desired number of stones have been dropped into the domain, the collision detection and rigid body dynamics algorithms continue until the system reaches an equilibrium[3];
- At the end, the spatial configurations of these stone particles (center of mass and principle axis) are recorded which uniquely defines their correct position and orientation.

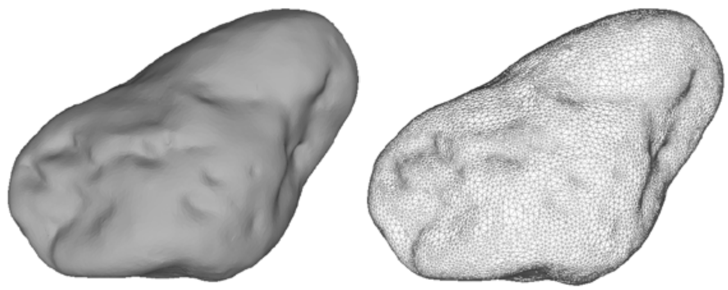

Figure 1. Example digitization of a cover stone

2.2 Fluid dynamics model

When the physically correct stone configurations are available, they are imported into the computational fluid dynamics (CFD) model. In general, there are two types of methods to represent the stones in the domain, namely Lagrangian and Eulerain (such as immersed boundary method). In a Lagrangian method, meshes are generated following the stones surfaces exactly. This seems impossible since the shapes and the arrangement of the stones are irregular. Therefore, we used an Eulerain approach. A new immersed boundary method (IBM) was proposed to simulate the effect of the stones. This new method has been implemented in the open source code OpenFOAM[1]. We also used the CFD solvers in OpenFOAM for the flow field simulations.

In the pioneering work of Peskin (1977), the immersed boundary method was first proposed to simulate the effect of complex geometry. Instead of using a body fitted mesh to describe the geometry, the simulation is carried out on a fixed Cartesian grid which does not conform to the geometry of objects. To impose the effect of the immersed boundary (IB) on the flow, which usually dictates no-slip at the boundary due to viscosity, novel techniques can be used to mimic this condition.

Most of the immersed boundary method is developed for structured meshes due to its simplicity. Another reason is that since any irregular geometry can be considered using IBM, there is no need to use unstructured meshes. The drawback of IBM is that the accuracy depends on the mesh resolution. On the other hand, unstructured mesh describes the boundary exactly. In many applications, unstructured meshes might be more desirable. For example, in scour simulations, there might have other features which can be better modelled using unstructured mesh, such as bridge piers and meandering channels.

As such, we have developed a new immersed boundary method based on unstructured meshes. Similar to regular IB method on structured meshes, computational cells in the domain are classified into different categories based on their locations relative to the objects. At each time step of the fluid simulation, the velocities of these cells are treated separately. In IBM, the key is the enforcement of solid boundary conditions based on interpolations. For unstructured mesh, this interpolation turns out to be more complicated than for structured meshes. We developed a novel interpolation method for unstructured mesh in our model. This model has been tested by simulating the classical flow past sphere case at *Re*=350 (Figure 2). The vortical structure behind the sphere is similar to those in the literature.

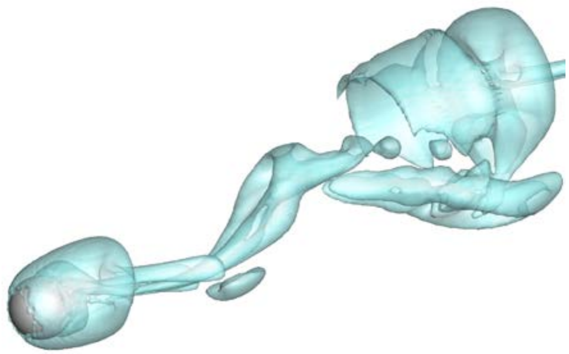

Figure 2. Demonstration of the new immersed boundary method by the classical flow over sphere case. Here the Reynolds number Re=350. The vortex shedding behind the sphere is shown.

3 PRILIMINARY SIMULATION RESULTS

In this section, we will present some of the preliminary results by using the new numerical model and method. The case we want to show is the 3D turbulent flow around a mono-pile surrounded by porous scour protections. Pile foundations are the most commonly used type of foundations to support structures in rivers, coastal, and offshore environments.

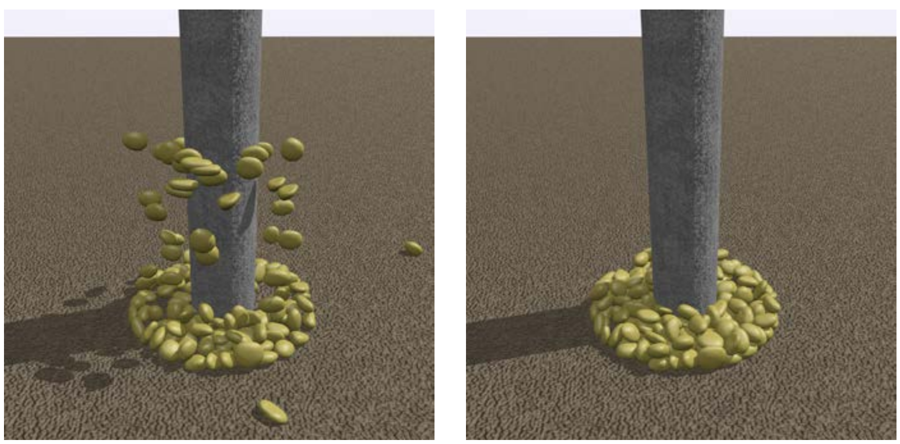

Figure 3. Generation of the physically correction and realistic configuration of the scour protection around a pile foundation (adapted from Liu, 2012). A total of 150 rocks were dropped around the pile to form a circular scour protection. The pile has a diameter of 1 m and the scour protection has a diameter of 3 m. The rocks have a mean diameter of 0.2 m. Only two snapshots are shown.

First, the physically correct and realistic configuration of the stones in the scour protection was generated (Figure 3). The diameter of the pile was 1 m and the diameter of the scour protection was 3 m. The pile and the scour protection were placed in a channel of 15 m long, 3 m wide and 3 m deep. Then the configuration of the stones and the pile were ported into our CFD code and a simulation was performed. An inflow of 0.1 m/s was imposed at the entrance of the channel. The outlet was modelled as a far field since it was located a significant distance from the pile. We did not use any turbulence model. The reason is that we found there were some compatibility issues remaining between our IBM method and wall treatment in many turbulence models. However, OpenFOAM has a variety of built-in turbulence models and it has the potential to resolve these issues.

In Figure 4, some of the simulations results are shown. In Figure 4 (a) and (b), the streamlines are shown. These streamlines originated from a line 10 cm above the bed and 4 m upstream of the pile. It is observed that the flow pattern both outside and inside the scour protection can be captured. In Figure 4 (c) and (d), the velocity and pressure distributions at the horizontal plane 10 cm above the bed are plotted. The velocity and associated shear stress are the driving force for the motion of sediment particles. More importantly, the velocity inside the porous scour protection is captured and it provides a good estimation for the potential of erosion under the scour protection.

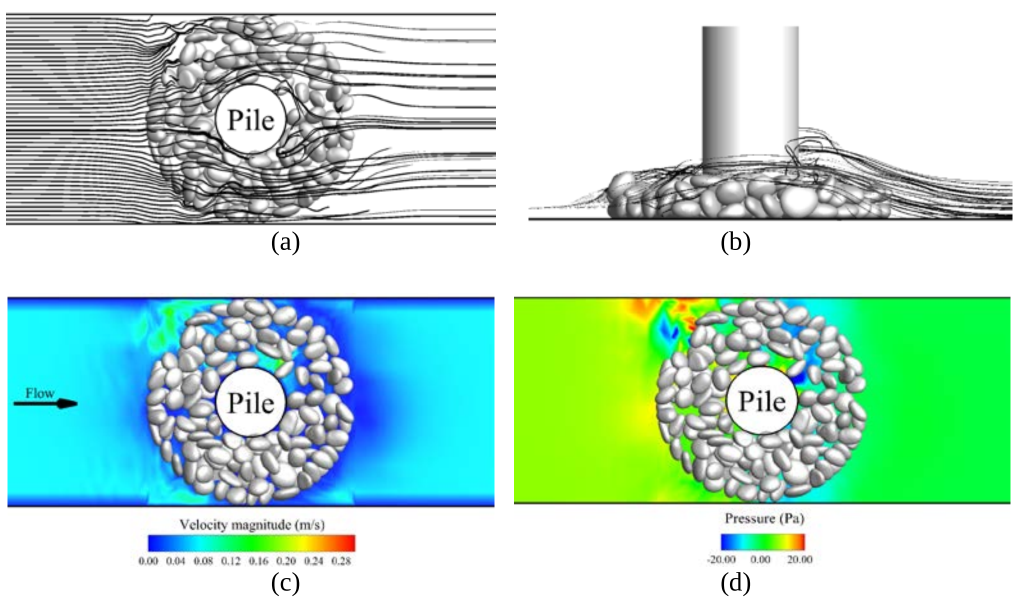

Figure 4. Simulation results for the flow around and inside the porous scour protection (adapted from Liu, 2012): (a) Top view of the stream lines, (b) Side view of the streamlines, (c) Velocity magnitude close to the bed, and (d) Pressure distribution close to the bed.

Although these results are very preliminary, they show the feasibility and the potential for more accurate description of the flow process inside the porous scour protection which was not seen before.

4 CONCLUSIONS

To understand the fluid dynamics around and inside a porous scour protection, numerical models with the capability to resolve the flow field at pore scale are very desirable. The internal flow information, in conjunction with the external flow, provides a good indicator for the effectiveness of scour protections. However, the issues of stone arrangement and representation should be addressed in this type of modelling. We presented a new method to generate the physically correct and realistic arrangement of the stones in the scour protection. This method is based on the collision detection and rigid body dynamics algorithms. To faithfully present the stones in the fluid dynamics code, a new immersed boundary method based on unstructured meshes has been developed. Preliminary simulation results from our models have shown that they are very promising.

5 ACKNOWLEDGEMENTS

The authors want to acknowledge the financial support through a Tenure-Track Research Award Competition (TRAC) award by the University of Texas at San Antonio (UTSA). The first author also wants to thank the traveling support provided by the College of Engineering at UTSA.

REFERENCES

[1] OpenCFD. OpenFOAM User's guide, version 2.0.1, http://www.openfoam.com/. (2011).
[2] X. Liu. Toward More Realistic 3D Flow Simulations around a Pile with Porous Scour Protection, In the proceedings of the 6th International Conference on Scour and Erosion, Paris, France, (2012).
[3] E. Coumans. Bullet 2.76 Physics SDK Manual, http://bulletphysics.com.(2011)
[4] C.S. Peskin. Numerical analysis of blood flow in the heart, Journal of Computational Physics. 25: 220–252. (1977)