J. Marine. Sci. Appl. (2009) 8: 338-342
DOI: 10.1007/s11804-009-8092-2

Three-dimensional hydrodynamic model of concrete tetrahedral frame revetments

GAO Zhu[1*], LI Xing[2], TANG Hong-wu[3], GU Zheng-hua[4]

1. College of Civil & Hydroelectric Engineering, China Three Gorges University, Yichang 443002, China
2. Ningbo Huitong Engineering Construction Co.Ltd, Ningbo 315000, China
3. College of Water Conservancy and Hydropower Engineering, Hohai University, Nanjing 210098, China
4. College of Civil Engineering and Architecture, Zhejiang University, Hangzhou 310058, China

Abstract: Revetments of concrete frame tetrahedrons are being used more and more in river engineering in China. Due to their complex geometry, it is difficult to measure the velocity fields inside them using traditional measurement methods. This limits understanding of their mechanics, potentially leading to suboptimal solutions. A 3-D hydrodynamic model based on the commercial computational fluid dynamics (CFD) code, Fluent, was developed to predict velocity fields and drags. The realizable k-ε model was adopted for turbulent closure of the Reynolds averaged Navier Stokes (RANS) equations. The study demonstrates that the numerical model can effectively supplement experimental studies in understanding the complex flow fields and mechanics of concrete frame tetrahedron revetments. Graphs showing the drag coefficient C_D versus Reynolds number Re and lift coefficient C_L versus Reynolds number Re were produced.

Keywords: concrete frame tetrahedrons; numerical simulation; drag; lift
CLC number:TV 85 **Document code:** A **Article ID:** 1671-9433(2009)04-0338-05

1 Introduction

Tetrahedral frame revetments (TFR) protection method has been more and more widely used in river engineering over the last two decades in China. This technique utilizes the scattered poles of tetrahedral frame to alter the flow patterns and increase the water turbulence of the inflow in the surrounding of each pole, thus TFR can reduce flow velocity, consume lots of flow energy in this mild way to achieve the scour protection purpose. The successful application cases are bank-protection project in Jiu Jiang reach of the Yangtze River, stilling basin protection project in Shang Qing Dam, etc[1-6]. However, due to the complex geometry of TFR, it is difficult to measure the velocity fields in the TFR using the traditional measurement methods, this causes the major research work still stay at the stage of experiments and field study, which limit further understanding about the complex flow field and the mechanism of TFR protection method.

For the reasons stated above, a three-dimensional numerical hydrodynamic model based on CFD code Fluent is developed, as an alternative method, to predict the velocity fields and drags for TFR. The configurations of the numerical model have been verified in another similar study about single wall sticking tetrahedral frame (TF), in which the numerical results agreed well with the experimental results. So in this study, by using the periodic conditions, research objects have been expanded from single TF to TFR based on this modified numerical model, then the velocity fields and plots of drag coefficient C_D and lift coefficient C_L versus Reynolds number Re for a periodic module are also computed and presented.

2 Numerical model

2.1 Governing equations of fluid Flow

The equations of mass and momentum conservation solved by Fluent for incompressible fluid can be written in Cartesian tensor form as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\rho\frac{\partial u_i}{\partial t} + \rho u_j\frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \mu\frac{\partial^2 u_i}{\partial x_j \partial x_j} + \frac{\partial}{\partial x_j}(-\rho\overline{u_i'u_j'}) + \rho g_i \tag{2}$$

where, u_i, p, ρ and μ are the mean fluid velocity

Received date: 2008-12-30.
Foundation item: Supported by the Science Foundation of China Three Gorges University (No. 0620070016), Opening Foundation of the Environmental Engineering Key Discipline from Zhejiang University of Technology (No. 20080218), NSFC (No.50779014, No.50879019), Ph.D. Discipline Foundation of the Ministry of Education of China (No.200802940001), Jiangsu "333" Program for High Level Talent, "Six Talent Peak" Project Foundation of Jiangsu Province (No.2007006), and "11th Five-year Plan" (2008BAB29B09).
***Corresponding author Email:** gaozhu.cn@gmail.com

components, pressure, density and molecular viscosity of the fluid flow, respectively; $-\rho\overline{u_i'u_j'}$ is known as the Reynolds stress tensor, which represents the effects of turbulent fluctuation in the fluid flow.

2.2 Turbulence closure

In order to approximate the governing equations, a turbulence model has to be introduced for the modeling of the Reynolds stress tensor. The commonly used model is the standard k-ε turbulence model, which is based on the assumption of isotropic turbulence. The relatively newly developed realizable k-ε turbulence model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physical characteristics of turbulent flows. It has some improvements on the standard k-ε model and renormalization group (RNG) k-ε model in simulating three dimensional fluid flows involving in rotation, boundary layers under strong adverse pressure gradients, separation and recirculation[7]. Considering the available computational resources and time, the realizable k-ε turbulence model is used to close the governing equations in this work.

2.3 Geometry, boundary conditions and solution strategy for CFD model

TFR consists of uniformly spaced TFs, a photo of TFR sticking on the bottom of flume is shown in Fig.1.

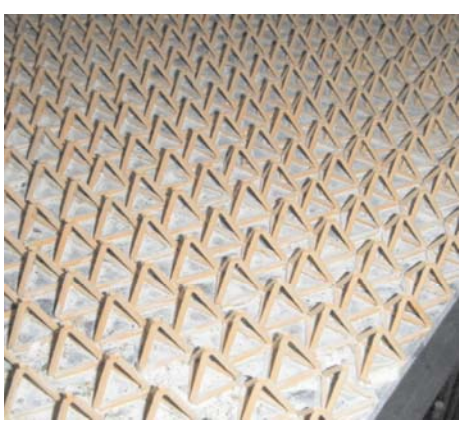

Fig.1 Single-layer TFR sticking on the bottom of flume in experiments

In this paper, the hydraulic characteristics of TFR in free stream is studied, however, due to the symmetry of the TFR geometry, only a portion of the domain needs to be modeled. The computational domain is 44 mm long (x direction), 48 mm wide (y direction), and 44 mm high (z direction) as shown in Fig.2. The length L of the square poles comprising TF is 40 mm, and edge length a of the pole's square cross section is 3 mm.The flow direction is along the x-axis, the angle of attack is kept at zero, the top and the bottom boundaries are specified as the slip wall with a specified zero shear stress and zero roughness height on them. Different mass flow rates, normal to the boundary, are applied to the inlet boundary of periodic module for different cases. The front and back faces are specified as the symmetry boundary. The surfaces of TF are specified as wall boundary with a roughness height 0.25 mm for the wood wall. The finite volume method is used to solve the governing equations. Computational domain is meshed by unstructured grids shown in Fig.3.

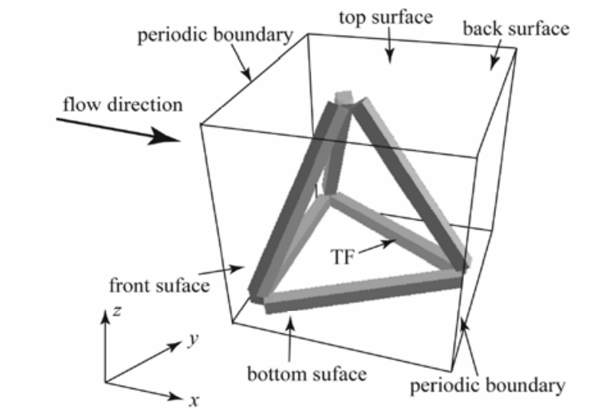

Fig.2 Geometry and boundaries of the periodic module of TFR

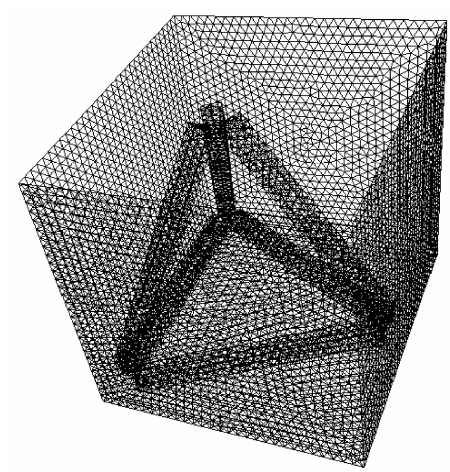

Fig.3 Unstructured grids

Calculation is initiated using the default under-relaxation coefficients, first order upwind for convective fluxes and the steady solver. The SIMPLEC algorithm is used for velocity-pressure correction. A value of 10^{-6} has been employed as the convergence criteria.

3 Results and discussion

3.1 Hydraulics of periodic module of TFR

Fig.4 is the streamline plot of TF. For every numerical test in the periodic module of TFR, it is found that the patterns of distribution of velocity are similar. The regions of the maximum velocities, denoted by colored clouds, appear in the top corner where is far away from the solid wall of TF, as shown in Fig.5, and the minimum velocities appear at the surface of the wall of TF because of the no-slip conditions, as shown in Fig.6.

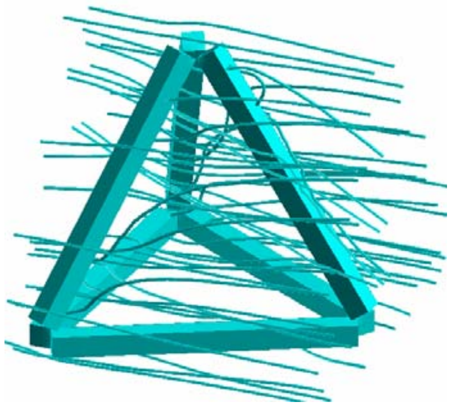

Fig.4 Streamlines of the periodic module of TFR

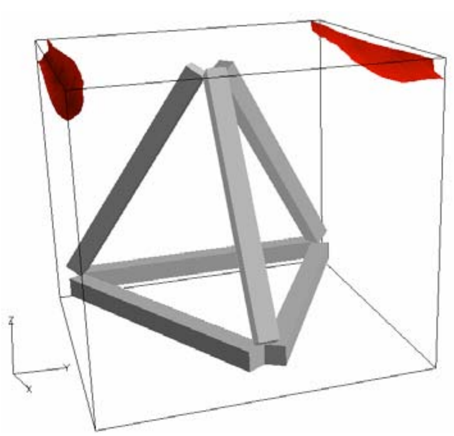

Fig.5 Clouds denoting the positions of the maximum velocities occurrence

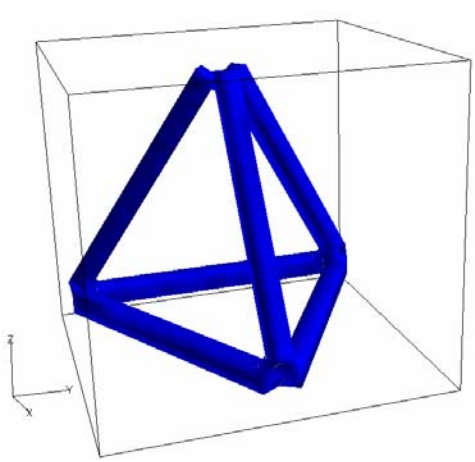

Fig.6 Clouds denoting the positions of the minimum velocities occurrence

Although the magnitude of turbulence intensity for different cases is different, it is also found that the patterns of distribution of turbulence intensity for different cases are similar. The maximum turbulence occurs in the rear of the two upstream tilt poles, as shown in Fig.7, while the minimum values occur at the least disturbed positions, as shown in Fig.8.

Using the periodic module of TFR, more complicated forms of TFR can be obtained by duplicating the periodic modules in three dimensions. For example, the contour of velocity magnitude of the flow fields of TFR (4×4×1) at the z=0.02 m is shown in Fig.9.

3.2 Drag coefficient C_D and lift coefficient C_L versus Reynolds Number *Re*

In various engineering practices, in order to predict the magnitude of drag and lift for any given TF size, both the complete relations of drag coefficient C_D versus Reynolds number Re, and the lift coefficient C_L versus Reynolds number Re are needed. They are defined as follows:

$$C_D = \frac{F_D}{\frac{1}{2}\rho U^2 A_f} \qquad C_L = \frac{F_L}{\frac{1}{2}\rho U^2 A_f} \qquad Re = \frac{3Ua}{\nu} \tag{3}$$

where A_f is the frontal area, defined as $4aL$ in this paper, that is 480 mm^2, U the cross section average velocity, and ν the kinematic viscosity of the water.

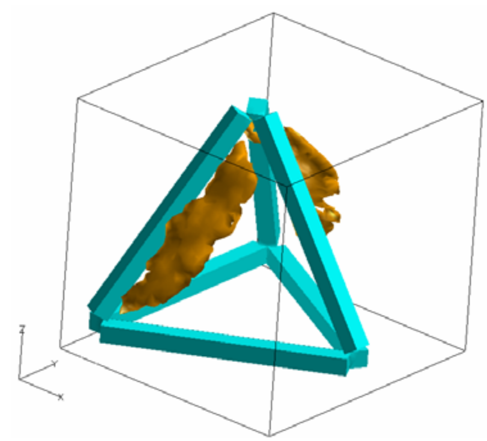

Fig.7 Clouds denoting the positions of the maximum turbulence intensity occurrence

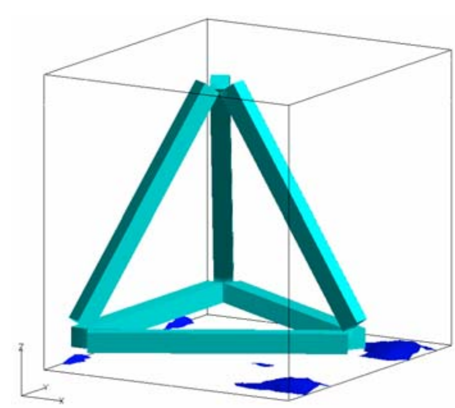

Fig.8 Clouds denoting the positions of the minimum turbulence intensity occurrence

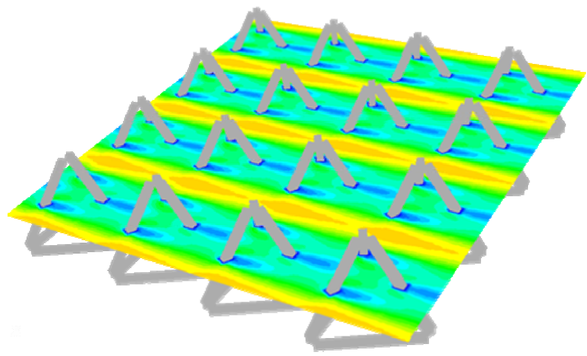

Fig.9 Contour of velocity magnitude of the flow fields of TFR

Changing the hydraulic parameters, the two plots can be obtained, as shown in Fig.10 and Fig.11. The Reynolds numbers vary from 1 060 to 160 000 in the computation series. In Fig.10, C_D have the minimum value at about R_e=10 000, although the points are not exactly on a line, it is acceptable that drag coefficients C_D are approximately kept at a constant value of 1.43 for a periodic module of TFR. This study is defined as the second study in order to narrate more clearly in the following.

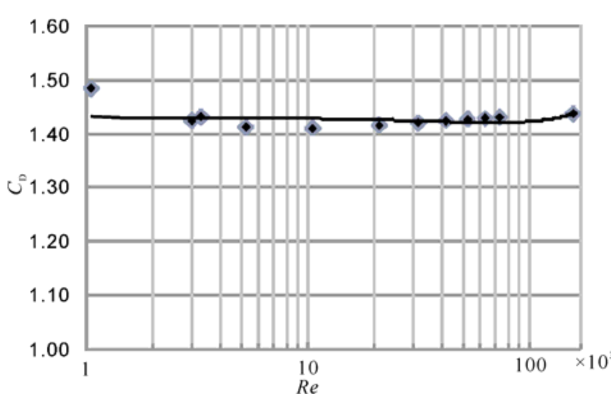

Fig.10 Drag coefficient CD versus Reynolds number *Re*

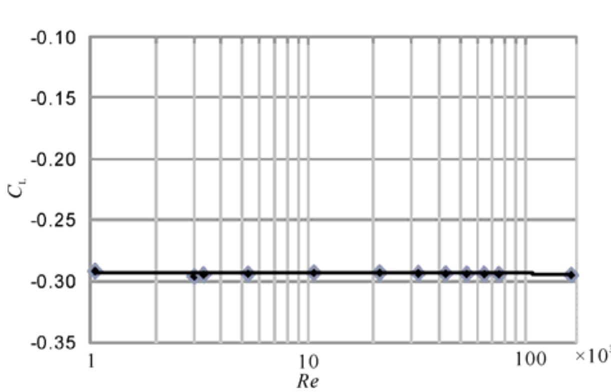

Fig.11 Lift coefficient CL versus Reynolds number *Re*

The above computational results for the drag of TF also have been validated indirectly by the other two studies. One study used a simple integral method (data obtained by ADV) to calculate the drag F_D of single TF, which was sticking on the bottom of flume[8], the other one is a numerical study as a supplement for the former experimental study. The coefficients C_D originated from the above two studies are approximately kept a constant value about 1.56 when the Reynolds numbers are larger than 10^5. These two studies are defined as the first study.

Major difference of these two kinds of studies is that: whether the flow is affected by the wall of flume? Obviously the drag of TF for the case (i.e. the first study) in which TF is sticking on the bottom of the flume[8], will be different from that computed in this paper (i.e. the second study), in which periodic module of TFR is, in fact, placed in the freestream without the flume wall confine. At the same time, the geometry of single TF in both kinds of studies are same, so the order of magnitude of the drag of the two kinds of studies should be the same one, the values of drag coefficients C_D (1.43 for the second study and 1.56 for the first study) just validate these features, thus the present computational model has been validated indirectly in this way.

Furthermore, in Fig.11, because the computed lift points to the negative z-axis, the lift coefficients C_L are all negative, which indicate that: when the angle of attack is kept at zero, just like the situations in this study, the direction of the force exerting on TF along the vertical direction is the same as gravity, so the lift will increase the stabilization of TF, and its value is a constant 0.29 in the range of Reynolds numbers used in this study.

4 Conclusions

A 3-D hydrodynamic model based on CFD code Fluent is developed to predict the velocity fields and drags for TFR. The realizable k-ε model is adopted for turbulent closure of the RANS equations. The study demonstrates that the numerical model can effectively supplement experimental studies in understanding the complex flow fields and the mechanism of TFR protection method.

The configurations of this numerical model have been verified in another similar study about single tetrahedral frame, which is sticking on the bottom of flume. In that study, the numerical results agreed well with the experimental results. So in this study, by using the periodic conditions, research objects have been expanded from single TF to TFR based on the modified numerical model. At last, the computed results were validated again by comparing the order of magnitude of the drag got from different studies. However, the direct experimental results are expected to validate current study in the future.

Both the diagrams of drag coefficient C_D versus Reynolds number Re and lift coefficient C_L versus Reynolds number Re are presented. In the range of Reynolds number included in this study, drag coefficient C_D is approximately kept at a constant value of 1.43, and lift coefficients C_L has a constant value 0.29 for a periodic module of TFR. Using the periodic module of TFR, more complicated forms of TFR can be obtained by duplicating the periodic modules in three dimensions.

References

[1] FANG Shilong, TANG Hongwu, ZHOU Yilin. Experimental study on effect of local scour at piers and protection by tetrahedron frame[J]. Advances in Water Science, 2006, 17(3): 354-358.
[2] LI Ruohua, ZHOU Chuntian, YAN Zhongmin. Optimization study on the velocity reducing effects of tetrahedron frames[J]. Express Water Resources & Hydropower Information, 2003, 24(11): 13-15.
[3] LU Taishan, HAN Yinguan, XU Qiuning. Experiment study on regulation engineering wandering channel in sediment-laden river[J]. Journal of Water Resources and Water Engineering, 1997, 8(2): 17-24, 29.
[4] TANG Hongwu, LI Futian, XIAO Yang. Experimental study on effect of scour prevention and sedimentation promotion of bank protection of tetrahedron penetrating frame groups[J]. Port & Waterway Engineering, 2002, 344(9): 25-28.
[5] WANG Nanhai, ZHANG Wenjie, WANG Bin. Application of a new technology of bank protection employing tetrahedron like penetrating frame groups in Yangtze river[J]. Journal of Yangtze River Scientific Research Institute, 1999, 16(2): 12-17.
[6] XU Guobin, ZHANG Yaozhe. Application of tetrahedron-like concrete penetrating frames in river improvement bank protection and emergency work[J]. Journal of Tianjin University, 2006, 39(12): 1465-1469.
[7] Fluent Inc. FLUENT 6.2 User's Guide[M]. 2005.
[8] GAO Zhu, TANG Hongwu, XIAO Yang, GU Zhenghua. Hydraulics of tetrahedron-like penetrating frame[C]// 9th International Symposium on River Sedimentation. Beijing: Tsinghua University Press, 2004: 1827-1832.

GAO Zhu was born in 1978. He is a lecturer of China Three Gorges University. His current research interests include hydraulics and river mechanics, etc.

LI Xing was born in 1978. He is an engineer of Ningbo Huitong Engineering Construction Co.,Ltd. His current research interests include hydraulics, river and seawall construction, etc.

TANG Hong-wu was born in 1966. He is a professor of Hohai University. His current research interests include hydraulics and river mechanics, etc.

GU Zheng-hua was born in 1974. He is a associate professor of Zhejiang University. His current research interests include hydraulics and river mechanics, etc.

四面体框架群三维水动力模型

高柱[1]，李星[2]，唐洪武[3]，顾正华[4]

（1. 三峡大学 土木水电学院，湖北 宜昌 443002；2. 宁波市汇通工程建设有限公司，浙江 宁波 315000；
3. 河海大学 水利水电工程学院，江苏 南京 210098；4. 浙江大学 建筑工程学院，浙江 杭州 310058）

摘 要：四面体框架群（TFR）防护方法在我国的河流工程中正得到越来越多的应用，然而由于其复杂的几何形状，采用传统的量测手段准确量测四面体框架群内的流场遇到极大障碍，这限制了研究者对四面体防护方法的深入理解. 本文利用 CFD 通用代码 Fluent，建立了用于计算四面体框架群流速场和阻力大小的三维水动力数值模型. 采用可实现 k-ε 紊流模型封闭雷诺时均方程. 研究表明该数值模型可以有效补充物理试验研究工作，有助于理解四面体框架复杂的三维流场和防护机理，分别给出了阻力系数 C_D 与雷诺数 Re 以及升力系数 C_L 与雷诺数 Re 的关系图.

关键词：四面体框架；数值模拟；阻力；升力