NUMERICAL SIMULATION OF FLOW AROUND BLUFF BODIES BASED ON VIRTUAL BOUNDARY METHOD

Qing Yang\textsuperscript{1}, Shuyang Cao\textsuperscript{2}

\textsuperscript{1}Ph.D candidate of State Key Laboratory of disaster reduction in Civil Engineering, TONGJI University, Shanghai, China, 811benson@tongji.edu.cn

\textsuperscript{2}Professor of State Key Laboratory of disaster reduction in Civil Engineering, TONGJI University, Shanghai, China, shuyang@tongji.edu.cn

ABSTRACT

The Immersed Boundary Method for rigid boundary (Virtual Boundary Method) is selected and integrated with staggered grid and fractional time-step method to simulate the flow around bluff bodies. A bilinear interpolation procedure is combined to realize the data transmit between grid points and boundary points to form the boundary. The selection of constant values in the forcing term is discussed based on the error of velocity at the boundary. Then the method is used to simulate the flow around bluff bodies. The qualitative description about this simulation is firstly presented by providing the current simulation’s streamlines and pressure contours. The phenomena illustrated in these sketches are the same as the other available literatures. At the end of this phase, the virtual boundary method is applied to the preliminary simulation of flow around two-dimensional bridge section to demonstrate Virtual Boundary Method can cope with complex geometry. Under this basis, the quantitative analysis of this simulation is carried out to investigate the influence of additional boundary treatment used in virtual boundary method on the accuracy of the aero-dynamic coefficients. The feasibility of Virtual Boundary Method is verified by the good agreement between this simulation’s results and other references’ outcomes.

Keywords: Virtual Boundary Method, Bluff bodies, Aero-dynamic coefficients, Additional boundary treatment

Introduction

Virtual boundary method is a new type of mathematic formulation and numerical scheme based on the immersed boundary idea which was proposed by Peskin (1972) to simulate the interaction between heart valve and blood. The core conception of immersed boundary idea is that a set of shear force is exerted on grid points nearby the boundary to build the body shape. Based on this conception, immersed boundary algorithm has two characteristics. First, the numerical simulation of flow over body, which is conducted by this model, can be solved on the regular Cartesian coordinate. It can avoid the use of coordinate transformations and mapping techniques which require highly accurate solution and a lot of time to accomplish the Jacobians transformation. Second, when immersed boundary method is applied to simulate moving body in the flow field, it is unnecessary to regenerate the grid to grab the updated position of boundary at every time step which is commonly used in conventional CFD method and creates large computational overhead. These above two features of immersed boundary method indicate that the problems of complex geometry and moving boundary, which are difficult to deal with in traditional numerical simulation, could be easily analyzed by immersed boundary method.

Originally the immersed boundary model was only adopted to study flow patterns and fluid-structure interaction of elastic boundary, such as the deformation of red blood cells in a shear flow [Eggleton and Popel (1998)], flow in elastic blood vessels [Vesier and Yoganathan (1992)], the swimming of sperm and bacteria [Fauci and McDonald (1994), Dillon Fauci and Gaver (1994)]. The classical immersed boundary algorithm model for elastic boundary
problems was summarized by Peskin (2002). And then it was successfully employed to biology research to modeling arthropod filiform hair motion in the air by Heys et al. (2008).

After previous comprehensive researches on the elastic border, the immersed boundary algorithm was applied by some researchers to study the rigid boundary problems. However, when initial immersed boundary model referred in above papers, which is suitable to elastic boundary, was used to solve rigid boundary problems, it existed problem, including sever stability constraints and spurious elastic effects, because of the discrepancy of constitutive laws between elastic border and rigid surface.

Under this background, virtual boundary method was introduced by Goldstein et al. (1993) to simulate flow over rigid bodies. In this method, forcing term take advantage of feedback loop of the fluid velocity on the surface to achieve the required rigid boundary conditions: the fluid velocity is equivalent to the surface velocity. In Saiki’s paper [Saiki and Biringen (1996)], virtual boundary method was combined with high-order finite difference and bilinear interpolation to simulate stationary, rotating, oscillating cylinders in uniform flow at low Reynolds numbers (Re<400). Recently virtual boundary method was applied by Deng et al. (2005) to show the three dimensional flow structures of two spheres in tandem arrangement.

In the last papers about virtual boundary method, the simulations were focused on the simple body shape: circular cylinder or other theoretical researches. Other bluff body shapes were not studied in the numerical simulations. The local continuity for the mesh containing the immersed border, which has been discussed in detail in other immersed boundary models [Li et al. (2004), Kim et al. (2001)], was not presented in virtual boundary method.

In the present work, virtual boundary method is combined with staggered grid and finite difference to make simulation. Firstly the mechanism of forming border is discussed in detail to demonstrate the function of feedback loop in the model. Secondly the simulation of the flow over stationary bluff bodies, including square cylinder, circular cylinder and other bluff shape, is performed. The emphasis of second stage in this paper is to verify the feasibility of virtual boundary method under regular Cartesian grid. The advantage of immersed boundary idea for coping with moving body isn’t presented in this work. Finally the local conservation is considered in the simulation to make comparison under traditional Cartesian grid.

**Numerical Method**

**Governing Equation**

Basic equations of virtual boundary method can be expressed as below:

\[ \rho \left( \frac{\partial u}{\partial t} + (u \cdot \nabla) u \right) = -\nabla P + \mu \nabla^2 u + F \]  
\[ \nabla \cdot U = 0 \]

In this equation, \( F \) is forcing term to form the rigid immersed boundary by feedback loop. Its expression is defined by the reference of Goldstein et al. (1993):

\[ F = \alpha \int_0^t (u - V) dt' + \beta [u - V] \]

Here \( \alpha \) and \( \beta \) are the negative constants, which are critical factors of the feedback loop; \( u \) is the fluid velocity on the boundary; \( V \) is the velocity of body border.

The above equation is the feedback to velocity difference \( u - V \) and acts to enforce the classic no-slip boundary condition (\( u = V \)) on the rigid immersed boundary. The first term of the equation is inclined to annihilate any velocity difference between \( u \) and \( V \). It can be
viewed as the restoring force to bring the fluid velocity close to the interface velocity. The second term is considered as the energy dissipation caused by surrounding fluid cell.

In order to illustrate distribution of the boundary points and virtual boundary method, Fig.1 and Fig.2 are presented as below:

![Fig.1 Distribution of boundary points](image1)

![Fig.2 Virtual boundary illustration](image2)

In Fig.1, the hollow points are boundary points, which can also be called Lagrangian points. The other grid points are regarded as Euler mesh points. The dotted line stands for the embedded border. The forcing terms are exerted on these boundary points to form the desired body shape. Fig.2 indicates that after forming the boundary, the fluid field is divided into two parts: inner flow field and outer flow field. The flow inside the body is independent from the outer fluid. As the time increase, the inner fluid velocities tend to decay. The outer flow field is undisturbed by internal flow field.

**Fractional Step Method**

Fractional step method is used to discretize the governing equations. This method is described as follows:

The first step: the initial fluid components including \( u \) and \( v \) are set to calculate the estimated velocities \( u^* \) and \( v^* \).

\[
\frac{u^* - u^n}{\Delta t} = \frac{1}{\gamma} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) + F
\]

\[
\frac{v^* - v^n}{\Delta t} = \frac{1}{\gamma} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) + F
\]

The second step: Poisson equation is solved to get the correction pressure.

\[
\frac{P_{i+1} - 2P_{i,j} + P_{i-1}}{(\Delta x)^2} + \frac{P_{j+1} - 2P_{i,j} + P_{j-1}}{(\Delta y)^2} = D_{i,j}
\]

\[
D_{i,j} = \Delta t \left( \frac{\partial u^*}{\partial x} + \frac{\partial v^*}{\partial y} \right)
\]

The third step: the updated velocities are gotten by using equation (8)–(9)

\[
u_{u+1}^* = u^* - \Delta t \frac{\partial P}{\partial x}
\]

\[
u_{v+1}^* = v^* - \Delta t \frac{\partial P}{\partial y}
\]
**Interpolation Procedures**

In virtual boundary method, the ideal condition is that boundary points (Lagrangian points) coincide with fluid grid points (Euler mesh points). The frontier fluid velocity can be directly gotten by solving governing equation. The forcing term can be easily calculated to impose on the boundary. But body shape is generally complex. The embedded border can cut through the underlying Cartesian grid in an arbitrary manner. In particular, when the staggered mesh is adopted to avoid the numerical oscillation caused by collocated grid, all fluid components can’t coincide with the grid points consistently. Considering the above conditions, the interpolation procedure is necessary to distribute the effect of forcing term to the Euler mesh points nearby the boundary and extrapolate the fluid velocity on the border from surrounding Euler grid points. It’s a data transfer process.

In this process, the bilinear interpolation is well understood and widely applied in immersed boundary model to realize the data transfer [Saiki and Biringen (1996), Li et al (2004)]. Compared with spectral interpolation utilized by Goldstein et al. (1993), the bilinear interpolation is more accurate and convenient to be applied in the Virtual Boundary Method. The bilinear interpolation expression, as illustrated in Fig.3, is summarized as below:

In Fig.3, \( P \) is the boundary point \((x_s, y_s)\); \((i, j)\), \((i+1, j)\), \((i, j+1)\), \((i+1, j+1)\) are the four surrounding Euler velocity grid points.

\[
U(x_s) = \sum_{i,j} D_{i,j}(x_s)U_{i,j}
\]

\[
D_{i,j}(x_s) = d(x_s - x_i)d(y_s - y_j)
\]

In equation (10), the fluid velocities of virtual boundary points \((x_s, y_s)\) are interpolated from the surrounding grid points by bilinear function \(D_{i,j}\) (see equation (11)) to calculate the forcing term on the border. And then the impact of immersed border is extrapolated back to the Euler mesh points by the equation (12). It is put into the governing equations to solve fluid velocities at new time-step.

\[
F_{i,j} = \frac{1}{N_b} \sum_{n=1}^{N_b} D_{i,j}(x_s)F_n(X_s)
\]

Where \(N_b\) is the number of boundary points which affects the \((i, j)\)th Euler grid point; \(F_n(X_s)\) is the forcing term of the border.

**Additional Boundary Treatment**

Additional boundary treatment is the characteristic of immersed model. It is employed to reinforce local mass conservative law of the embedded border. Li et al. (2004) postulated...
that if no treatment is applied, the local continuity of the flow adjacent to the immersed boundary may not be satisfied, especially at the beginning of the simulation when incorrect initial condition is used.

The mass source introduced as a treatment measure [Kim et al. (2001)] to ensure the local conservation is regarded as adverse to the global mass conservative within the entire calculation domain. Although the velocities can satisfy the no-slip boundary condition, the pressure distribution near the boundary may be incorrect by using the mass source term. The imposition of zero normal gradient condition of pressure is another additional boundary treatment measure proposed by Li et al. (2004) in order to obtain the realistic solution and increase the accuracy of the solution near the boundary. This alternative approach has been tested that it can alleviate the above problems. It is used to make sure that there is no flow across the border. The local continuity of the boundary can be satisfied automatically.

![Fig.4 Implementation of zero normal gradient pressure](image)

As demonstrated in Fig.4, the zero normal gradient condition of pressure can be imposed by setting $P_A=P_B$ (Fig.4a) and $P_A=P_E$ (Fig.4b) to achieve the desired parameter distribution. In this paper, simulations of flow around square cylinder with and without zero normal gradient condition are tested. The analysis is stated in following section below.

### Calculation domain

The traditional equidistance Cartesian grid is applied in the simulation. The grid size $DX=0.025$. The time step size $\Delta t=1.5 \times 10^{-4}$. The two dimensional bluff body with diameter $D$ is centered inside calculation domain (height $H$). When the body is cylinder, the blockage ratio is fixed at 1/8. As for the 2D bridge deck, the blockage ratio is set at 1/20. Frictionless Boundary Condition is used on the lateral boundary: $\frac{\partial u}{\partial y}=0, \frac{\partial P}{\partial y}=0, \frac{\partial v}{\partial y}=0$. The boundary condition of inlet is set as: $u=1.0, v=0$. The Neumann boundary condition is applied on the outlet of the calculation grid.

![Fig.5 Calculation domain and boundary condition](image)
Simulation Results

Mechanism analysis of forming virtual border

The equation (1) can be simplified as follows:

\[
\frac{dq}{dt} = f = \alpha_f \int q dt + \beta_f q
\]

(13)

Where \( q = u - V \) is velocities differences as discussed in the above sections. The first term including \( \alpha_f \) can be viewed as the spring connecting between the fluid cell on the boundary and virtual border. This equation indicates that when the fluid velocity \( u \) on the boundary deviates from the setting value of border velocity \( V \), the forcing term \( f \) compels \( u \) back to \( V \).

In unsteady flow field, the value of \( f_{\alpha} \) must large enough to trace the changing flow field.

At the same time, the time restriction in this method can be expressed as below:

\[
\Delta t < \frac{-\beta - \sqrt{\beta^2 - 2\alpha k}}{\alpha}
\]

(14)

Where \( k \) is a dependent constant of order one. The time step size is determined by constants: \( \alpha_f, \beta_f \). Hence, the selection of constant value is needed to be discussed.

The different combination of constants are applied to simulate the no-slip wall boundary. The proper constant value is selected based on the error of velocity at the boundary (\( l_2 \)-norm) to launch simulation.

\[
l_2 - \text{norm} = \frac{1}{n_b} \sum_{i}^n u_i
\]

(15)

Where \( n_b \) is the number of boundary points, \( u_i \) is the velocity of boundary points.

![Fig.6 l2-norm of boundary points with different constant combinations](image)

In Fig.6, as the constants increase, the time required to achieve desirable boundary condition (\( U = 0 \)) lessen. It indicates the larger constant value could realize immersed boundary quickly, and the combination c and d could both act more efficiently in reinforcing boundary condition, in which the time required is almost same. Furthermore, the combination c tends to retain the entire calculation efficiency based on the time restriction. After the above analysis, the combination c is considered as the most appropriate selection for the simulation.

![Fig.7 Visualization of immersed boundary](image)

![Fig.8 The variation of x-direction velocity \( u \) of internal points with time](image)

![Fig.9 The variation of y-direction velocity \( v \) of internal points with time](image)
As shown in the Fig.7, the streamline bypass the immersed boundary to verify the success of simulation for physical border. Furthermore the inner fluid field is analyzed. The variations of velocities of random internal points with time are drawn in the Fig.8 and Fig.9. The variation of inner velocities is at low amplitude within the entire calculation time. And the values are very small in the range of $10^{-2}$~$10^{-3}$. These phenomena indicate that inner flow field is completely isolated from the outer flow field.

**Qualitative Description of simulation of flow around bluff bodies**

The simulation of flow around two dimensional bluff bodies conducted by virtual boundary method is carried out to verify feasibility of this program. Considering the vortex structure become three dimension in high Reynolds numbers and the blockage ratio (1/8) adopted in this paper [Guo et al. (2003)], the range of Reynolds numbers of present two dimensional simulation is 10~500.

Fig.10 is the streamline contour of flow around square cylinder at Re=10~60. Fig.10a reveal that two steady symmetrical vortices appear in the downstream region. As Re increases, the length of separated bulb becomes longer (shown in Fig.10b). When Re is 60, the steady and closed near-wake becomes unstable. The transverse oscillation starts at the end of the near-wake and initiates a wave along the trail. The critical value of Re$_{crit}$ for this calculation is 60. It is the same as determined by Guo et al. (2003).

This phenomenon for square cylinder can also be visualized by vortex contour in Fig.11a (Re=60). When Re reaches 100, the free shear layers begin to roll up and form eddies as seen in Fig11.b. This feature is well known as the von Karman vortex street. As Re comes to the upper limit (Re=500), the flow pattern change into turbulence flow. The regularity disappears in the wake region (seen in Fig.11c).

These above description about this simulation coincide with phenomena observed by Breuer et al. (2000) and Liu et al. (2008).

**Fig.10 Streamline contour of Flow around square at different Re**

| a. Re=10 | b. Re=30 | c. Re=60 |

**Fig.11 Vortex contour of Flow around square at different Re**

| a. Re=60 | b. Re=100 | c. Re=500 |

Fig.12 is the streamline sketch of flow around square with attack angle at different Re. The constant values in this simulation are the same as the square cylinder ($\alpha_f=1.6\times10^5$, $\beta_f=-600$). For circular cylinder, $\beta_f$ is different from the square cylinder ($\beta_f=-400$). The attached, recirculating, symmetric bubble is revealed in the wake at Re=40 (Fig.13a). The vortex shedding from the circular cylinder occurs at Re=50 (Fig.13b). The same situation was obtained in the Saiki et al.’s (1996) simulation.
The virtual boundary method is directly applied to simulate the flow around two-dimensional bridge section. Fig.14 is the preliminary simulation under Re=2000 to demonstrate that virtual boundary method can cope with the complicated geometry.

Quantitative Analysis of simulation of flow around bluff bodies

Considering the adoption of boundary treatment in virtual boundary program, two cases (case1 and case2), which are differentiated by zero normal gradient condition, are proposed to analyze the influence on the numerical accuracy of aero-dynamic coefficients for square cylinder.

As shown in Fig.15, it is observed that although the drag coefficient $C_d$ of square cylinder without zero normal gradient condition (case1) is smaller than others, the drag coefficient trend depending on Re of case1 is in line with other reference’s variation law [Breuer et al. (2000), Guo et al. (2003), Liu et al. (2008)]. On the other hand, the case2’s drag coefficient has good agreement with other’s results [Breuer et al. (2000), Guo et al. (2003), Liu et al. (2008)]. The accuracy of numerical simulation is improved by using the boundary treatment.

Fig.16 reflects data difference variation of $C_d$ for square cylinder depending on $Re$. In this diagram, the data difference means the discrepancy between the current simulation (case1 and case2) and other literatures. The discrepancy of case2 is smaller than the case1 within the whole calculation time zone. Although the conclusion can be summarized the boundary treatment measures is helpful to annihilate the deviation of simulation based on the above
investigation, the zero normal gradient condition can’t be view the only sole factor to influence the accuracy.

Because from the Fig.16, it can be obviously watched that as the Reynolds number increases, the discrepancy of two cases declines. The discrepancy may be caused by the coarse equidistance Cartesian grid in the vicinity of virtual boundary, which isn’t good enough to capture the velocity difference nearby the body surface. [Breuer et al. (2000), Deng et al. (2005)].

![Figure 17 Strouhal number depending on Re](image1)

![Figure 18 Lift variation at different Re](image2)

The variation of $St$ at different $Re$ is presented in Fig.17. In the range 60<$Re<$300, the case2 fits other literature [Breuer et al. (2000)] well, showing an increase in the $St$ with increasing Reynolds numbers. The case1 show a little deviation in $St$. This phenomenon coincide with the above conclusion.

In Fig.18, lift coefficient of this computation (Max(Cl)-Min(Cl) ) are plotted. As shown in the figures, the numerical results of this paper have good agreement with previous literature [Breuer et al. (2000)].

Conclusions

In this paper, the proper constant value selection of forcing term in the simulation is discussed. Considering the time restriction, the appropriate combination is chosen to launch simulation. And then the mechanism of forming border is visualized. The streamline bypass the immersed boundary to verify the success of simulation for physical border. The inner flow field is isolated from the outer flow field and velocities decay with time. The qualitative description of this simulation is presented by providing sketches of flow around bluff bodies. Then the 2D bridge deck is simulated to show that Virtual Boundary Method can treat with the complicated geometry situation. Under this basis, the quantitative analysis is carried out to investigate the influence of boundary treatment measures on the numerical accuracy of simulation. Although the boundary treatment measure is helpful to annihilate the deviation of simulation, the zero normal gradient condition can’t be view the only sole factor to influence the accuracy.

Acknowledgement

This work was funded by the Ministry of Education 20100072110051 and SLDRCE09-B-04.

References


