1D-3D coupled simulation of surge waves in river-reservoir system by using the immersed boundary-lattice Boltzmann method

ZEHAO HUANG(1), YONGGUIANG CHENG(2), JIAYANG WU(3) & MINGDENG ZHAO(4)

(1,2,4) State Key Laboratory of Water Resources and Hydropower Engineering Science, Wuhan University, Wuhan, China, zehaohuang@outlook.com; ygcheng@whu.edu.cn; mdzhao@whu.edu.cn
(3) Changjiang Institute of Survey, Planning, Design and Research, Wuhan, China, 760816021@qq.com

ABSTRACT

Many run-of-river hydropower stations located across the rivers with multiple channels have several powerhouses, owing to the separation by islets. In the transient process caused by sudden load reduction of one powerhouse, significant surge waves will appear in the channels and cause negative impacts on the stability of the operating hydropower units in other powerhouses. To simulate such surge waves in a reservoir and its connected rivers, and to depict the three-dimensional (3D) motion of waves near powerhouses and ship locks, a one- and three-dimensional (1D-3D) coupled model based on the immersed boundary-lattice Boltzmann (IB-LB) method is proposed. The 1D shallow water LB (SWLB) model is used to simulate the waves in rivers, and the 3D free-surface LB (FSLB) model is used to simulate the waves in the reservoir. These two LB models are coupled by applying water stage prediction and correction (WSPC), and the topography of the reservoir is handled by the IB method. To demonstrate the applicability of this model, we report the simulation of a practical example. The transient process is generated by an emergent load rejection of all units in one powerhouse, while the units in the other powerhouse are working with rated power output. It is found that the surge waves travel between different channels in the reservoir with regular fluctuating patterns. The histories of water level fluctuations and frequency spectrum analysis reveal the spreading and oscillating rules of the waves. This work demonstrates that the proposed coupled model has good prospects in engineering applications.

Keywords: Surge wave; reservoir; lattice Boltzmann method; immersed boundary method; one- and three-dimensional coupling

1 INTRODUCTION

Run-of-river power stations are featured by low water head and large discharge. Therefore, the huge surge waves in the upstream reservoir caused by transient processes of some turbine units may greatly affect the stability of other operating units (Chen et al., 2016). For the power stations located across multiple channel rivers, such as the Gezhouba and the Changzhou hydraulic projects in China, the surge waves propagate between these channels, which causes water level fluctuations and units' output swing. Thus, it is necessary to study the propagating and fluctuating laws and control measures of such kind of surge waves.

At present, the breaking wave method (Wang and Liu, 2001) and shallow water equation based methods (Tseng, 2004; Delis and Skeels, 1998) are commonly used in simulating transient flows in open channels. The former is a simple and feasible way to calculate surge waves in channels; the latter are also accurate for calculating two-dimensional (2D) free surfaces. However, both of them have drawbacks when simulating flows of which the three-dimensional (3D) features are obvious. In addition, some assumptions in these methods are not satisfied if the topography is complex. These problems can be solved by using the 3D free-surface models based on the Navier-Stokes equations, but the expense is very large for practical problem simulations. One the other hand, the lattice Boltzmann (LB) method, based on the mesoscopic molecular dynamics, is newly developed due to many advantages, such as simple algorithm, intrinsic parallelism, and high efficiency (Li et al., 2004). Many researchers have proved that the 3D free-surface LB (FSLB) model is feasible and accurate to simulate flows with free surface (Sussman, 2003; Körner et al., 2005; Thürey et al., 2006). This model calculates the mass transfer through the migration of distribution functions, and it can be conveniently coupled with the immersed boundary (IB) method. The IB-LB coupled method was first proposed by Feng and Michaelides (2004), and developed by Cheng and Zhang (2010) and Diao et al. (2018). It uses a regular Eulerian grid for the flow domain and a Lagrangian grid for the boundaries. The IB-LB method is powerful in solving complex and moving geometries, because the fluid and the solid boundaries are coupled by simply adding a body forcing term into the LB equation. Though this IB-FSLB model is feasible and efficient (Diao et al., 2018), it is still unwise to use it in the whole computational domain, because the simulations of large-scale river and reservoir systems need huge computing resource and time. The dimensions of models should be adjusted according to the features of...
computational domains and flow structures. Therefore, the models that couple the 1D rivers with the 2D or 3D reservoirs should be applied.

This paper proposes a 1D-3D coupled IB-LB model to simulate the transient flows in 1D rivers and 3D reservoirs for the first time. In this model, the 1D shallow water LB (SWLB) method is used to calculate long and narrow open channels, and the 3D FSLB method is used to simulate reservoir areas where 3D flow features are obvious. The topography in the 3D computational domain is handled by the IB method, and the coupling process between the 1D and the 3D models is based on the water stage prediction and correction (WSPC) (Chen et al., 2012).

The rest of the paper is organized as follows. In Section 2, an overall description of the proposed model, including the 1D and the 3D LB models, the WSPC, and the IB model, are given. In Section 3, the accuracy of the present model is verified by a benchmark case. In Section 4, the surge waves caused by a sudden load rejection of units in a practical hydropower station are simulated to illustrate the model’s applicability and feasibility. Finally, a brief conclusion is drawn to end the paper.

2 The 1D-3D COUPLED IB-LB MODEL

2.1 The 1D shallow water LB method (1D SWLB)

The numerical solution for 1D shallow water equations is numerous and well known, such as method of characteristics (Stoker and Lindsay, 1958), finite difference method, and finite element method (Katopodes, 1984). However, in this study, the 1D shallow water LB method (Zhang et al., 2016) is used to approximate the shallow water flows. Dividing the river into segments with spacing of 1, and setting the discrete velocity space as \( \{e_1, e_2, e_3\} = \{-1, 0, 1\} \), the LB equation can be written as

\[
f_s(x + e_i, t + \Delta t) - f_s(x, t) = -\frac{1}{\tau} [f_s(x, t) - f_s^{eq}(x, t)] + R_s
\]

where, \( f_s \) is the distribution function; \( f_s^{eq} \) the equilibrium distribution function; \( x \) the coordinates of nodes; \( \tau \) the relaxation time; and \( R_s \) the external forcing term.

In Eq. [1], the equilibrium distribution functions are

\[
f_s^{eq} = \frac{-\rho u + \rho}{2}, \quad f_s^{eq} = 0, \quad f_s^{eq} = \frac{\rho u + \rho}{2}
\]

where, \( u = \sum_j f_j \) and \( \rho u = \sum_j e_j f_j \) are the macro velocity and macro momentum, respectively.

By correctly defining \( R_s \), this method can precisely approximate the shallow water equations. To get detailed information, readers can refer to Cheng and Suo (2000) and Zhang et al. (2016).

2.2 The 3D free-surface LB method (3D FSLB)

The simulation of free surfaces should distinguish fluid, air, and interface. In this method, the grid nodes in the computational domain are classified into three types, which are fluid, air, and interface nodes (Diao et al., 2018). The fluid grids are fulfilled with water, while the interface grids are partly filled with water, which completely separates the fluid and air regions (Figure 1). Therefore, the procedure of tracing the interface, namely the free surface, is: (a) tracing the motion of interface nodes, (b) handling the boundary conditions of interface nodes, and (c) updating the types of grid nodes around the interface.

![Figure 1. Schematic diagram for the three types of grids in FSLB.](image)

The tracing of the interface is realized by calculating the mass of fluid in each grid and the mass transfer between different grids. Therefore, we need to determine the volume fraction \( \varepsilon \) for each grid, which is the ratio
of fluid mass to fluid density \( \varepsilon = m / \rho \). In the LB model, the mass of fluid transfers along with the migration of distribution functions. For an interface node and its adjacent fluid nodes, the mass transfer can be written as

\[
\Delta m_\alpha(x, t + \Delta t) = f_\alpha(x + e_\alpha, t) - f_\alpha(x, t)
\]

where, \( \alpha \) and \( \bar{\alpha} \) are the positive and negative directions of distribution functions, respectively. The first term on the right side is the mass that flows into the node along the \( \bar{\alpha} \) direction, while the second term is the outflow from the same node along the \( \alpha \) direction.

For the mass transfer between interface nodes, considering volume fractions, the equation reads

\[
\Delta m_\alpha(x, t + \Delta t) = \left[f_\alpha(x + e_\alpha, t) - f_\alpha(x, t)\right] \frac{\varepsilon(x + e_\alpha, t) - \varepsilon(x, t)}{2}
\]

The structures of Eqs. [3] and [4] are symmetrical, and there is no mass transfer between air nodes and the other kinds of node. Therefore, these equations meet the requirement of mass conservation. For an interface node, its mass in the next time step can be calculated as

\[
m(x, t + \Delta t) = m(x, t) + \sum \Delta m_\alpha(x, t + \Delta t)
\]

The distribution functions and macro parameters on air nodes are neglected. Therefore, during the migration process, those distribution functions, which move from air nodes to interface nodes, should be reconstructed according to the interface boundary conditions. Assuming that the air is uniform distributed, its pressure is constant, and the viscosity is very small, the air can be regarded as moving along with the fluid. Thus, the reconstructed distribution function reads

\[
f_\alpha(x, t + \Delta t) = f_\alpha^{\text{eq}}(\rho_\alpha, u) + f_\alpha^{\text{eq}}(\rho_\alpha, u) - f_\alpha(x, t)
\]

where, \( u \) is the flow velocity on node \( x \) on time step \( t \), and \( \rho_\alpha \) shows the effect of air pressure on the interface. Now that all of the distribution functions on the interface nodes are specified, we can calculate the density of the interface nodes. Then by comparing the density to the corresponding mass, the interface nodes are handled based on the following rules:

(a) If \( m(x, t + \Delta t) > (1 + \kappa) \rho(x, t + \Delta t) \), the node turns into fluid node.

(b) If \( m(x, t + \Delta t) < (1 - \kappa) \rho(x, t + \Delta t) \), the node turns into air node.

here, \( \kappa \) equals to \( 10^{-3} \) to prevent the situation that the newly formed interface nodes change into other types in the next steps.

The nodes that turn into fluid nodes, their mass is larger than density; the nodes that turn into air nodes, their mass should equal to zero. Therefore, the mass of the newly turned fluid nodes and air nodes should be redistributed to ensure the mass conservation. Firstly, all of the air nodes near a newly turned fluid node should transform into interface nodes. Meanwhile, for the newly turned air node, all of its adjacent fluid nodes should also transfer into interface nodes. The distribution functions of these interface nodes are directed set as equilibrium distribution functions, while the pressure and velocity of them are the average value of their adjacent fluid nodes. Secondly, redistributing the fluid mass around the interface. For the newly turned fluid nodes and air nodes, the redistributed mass are respective \( m^\alpha = m(x, t + \Delta t) - \rho(x, t + \Delta t) \) and \( m^\alpha = -m(x, t + \Delta t) \). When the mass of these nodes are changed, the volume fraction of them should be synchronized. After this update, the treatment of free surface is over, and the simulation can continue to the next time step.

2.3 The water stage prediction and correction (WSPC) for 1D-3D coupling

The coupling method of the above 1D and 3D models is based on WSPC (Chen et al., 2012), of which the basic elements are shown in Figure 2. At the beginning, the external and internal boundaries should be determined. The external boundaries are the borders of the whole computational region, where the boundary conditions are known in advance. The internal boundaries connect to the coupled joints, which are the key points of coupling. On a coupled joint, both of the 1D and the 3D models should meet the continuity condition

\[
\sum_{i=1}^{M} Q_i = 0, \quad h_i = h_{\text{eq}}, \quad i = 1, 2, ..., M
\]
where, $M$ is the amount of internal boundaries which connect to the coupled joint; $Q_i$ is the discharge of outflow of the $i$ th internal boundary. The water depth $h_i$ on the $i$ th internal boundary equals to the water depth $h_{in}$ on the coupled joint.

In WSPC, water depths on all of the internal boundaries are fixed. If the water depth $h_i$ and discharge $Q_i$ on time step $t$ are known, we can assume that the water depth of a coupled joint on time step $t + \Delta t$ is $h'$. According to all of the external boundary conditions and the water depth $h'$, the flow field and the value of $\sum_{i=1}^{M} Q_i'$ on time step $t + \Delta t$ can be calculated. To satisfy the continuity condition, $\sum_{i=1}^{M} Q_i'$ should tend to zero. If we regard the coupled joint as a container, the net discharge can change the water depth in it

$$\Delta h' = \frac{\Delta t \sum_{i=1}^{M} Q_i'}{A_c}$$  \[8\]

where, $A_c$ is the basal area of this container. It can be calculated as

$$A_c = \alpha B \Delta t \sqrt{gh'}$$  \[9\]

where, $\alpha$ ranges from 0 to 1; $h'$ is the water depth in $k$ th iteration; $B$ is the sum of channel width. $\Delta h'$ can be used to correct the water depth, and $\sum_{i=1}^{M} Q_i'$ finally tends to zero by iteration. Chen et al. (2012) has proved that the iteration process is convergent.

In summary, the steps of the coupling process on time step $t + \Delta t$ are: ($k$ is the iterative step and $\lambda$ is a small constant)

(a) Evolving the whole flow field to $k$ th iterative step, and calculating the $\sum_{i=1}^{M} Q_i'$.

(b) If $|\sum_{i=1}^{M} Q_i'| < \lambda$, stop the iteration. Otherwise, go to step (c).

(c) Calculating the corrected value $\Delta h'$ according to Eq. [8].

(d) Correcting the water depth of coupled joints, $h^{k+1} = h' + \Delta h'$.

2.4 The immersed boundary (IB) method for treating topography

In this study, we used the GIS software LocaSpace Viewer to extract the point cloud data of elevations of the watershed topography. These data were transferred into Lagrangian grids, and then the topography boundaries could be handled by the IB method. The effects of boundaries on the flow field are realized by adding an external forcing term into the LB equation (Feng and Michaelides, 2004). The IB dynamic equations are

$$U(X, t) = \int_{\Omega_i} u(x, t) \delta(x - X) dx$$  \[10\]

$$F(X, t) = S_i X$$  \[11\]

$$f(x, t) = \int_{\Gamma_b} F(X, t) \delta(x - X) dX$$  \[12\]

where $X$, $F$, $U$ are position, force and moving speed of the boundary nodes, respectively; $x$, $f$, $u$ are the Eulerian coordinate, external force and flow velocity; $S_i$ is the boundary force generation operator. The interaction of the flow field $\Omega_i$ and the boundary $\Gamma_b$ is realized by the Dirac delta function $\delta(r)$. It spreads the boundary force to the nearby fluid nodes and imposes the flow velocity onto the boundary.
Unlike those immersed boundaries that are surrounded by fluid, in this occasion, only part of the topography boundaries are immersed by the flow field owing to the free surface. To handle the non-slip boundary conditions when free surface exist, we chose the IB method with multi-direct forcing term and velocity correction, which can be found in Diao et al. (2018).

3 MODEL VERIFICATION

To verify the accuracy and feasibility of the 1D-3D coupled model for open channel flow, the transient process in a tail water channel was simulated. The length and width of this channel are respective 100 m and 10 m, and the bottom of it is horizontal. The upstream boundary is discharge boundary, and the downstream water depth is fixed at 15 m. We divided this channel into two parts. The upstream part is calculated by the 1D shallow water LB method, and the downstream part is simulated by the 3D IB-FSLB method (Figure 3). The size of grid are respective 200 and 250×50×100, and the corresponding spacing are 0.25 m and 0.2 m. The time steps for both 1D and 3D models are set as 0.01 s.

We simulated the transient processes caused by output increase and load rejection, and compared the results to the data from Delft3D. The rated discharge of units is 600 m$^3$/s; the guide vanes are opened and are closed linearly both in eight seconds. Initially, water in the channel is stationary. At the end of the output increase, flow in the channel becomes steady, which is the original condition for the load rejection. Figure 4 shows the good agreements between the results from the present model and Delft3D. Moreover, this model is more accurate at the inflection points of the curves, which demonstrates its fine numerical convergence. Because the Delft3D uses roughness coefficient to represent the riverbed friction, which is different from the non-slip boundary condition in the IB-LB method, the results are not completely the same.

4 SIMULATION OF SURGE WAVES IN A RESERVOIR

4.1 Engineering data

This simulation was based on the engineering data and geographical parameters of a run-of-river power station built on the Yangtze River. Two islets near the dam site divided the river into main, second, and third channels. The normal water level in this reservoir is 66.0 m.

There are twenty-one hydropower units in this hydraulic project. Among them, fourteen units with rated discharge of 825 m$^3$/s are installed in the main channel powerhouse; two units with rated discharge of 1130 m$^3$/s and five units with rated discharge of 825 m$^3$/s are placed in the second channel powerhouse. The third channel only has functions of sand sluicing and shipping instead of power generation.

Figure 5 shows the locations of six monitoring points:

(a) Point A is in front of the main channel scouring sluice;
(b) Point B is in front of the main channel water inlet;
(c) Point C is in front of the second channel water inlet;
(d) Point D is in front of the third channel scouring sluice;
(e) Point E is located at the flood bank in the main channel;
(f) Point F is located at the estuary of tributary.

Figure 5. Locations of six monitoring points.

4.2 Computational conditions of the transient process

The transient process simulation is based on the situation that load rejections of all units in the main channel powerhouse occur simultaneously, while the seven units in the second channel powerhouse operate with rated discharge.

The computational domain is the 40-kilometers reservoir and river system from this power station to the upstream hydraulic project (Figure 6). The reservoir, which is 5 km upstream from the power station, is simulated by the 3D IB-FSLB method. The other 35 km river is calculated by the 1D SWLB method. The river is divided into 3000 segments, and the reservoir is divided by a uniform lattice of 500 × 400 × 50. The corresponding spacing of grids are 11.67 m and 8.64 m, respectively. The upstream boundary of the 1D river is fixed discharge boundary, and the downstream boundary of the 3D reservoir is discharge-water head boundary. The roughness coefficient of the river is 0.012.

Figure 6. Schematic diagram for the computational domain.

Flow field in the reservoir should be steady before the simulation of the transient process. The upstream water level is 67.0 m and the water level in front of the powerhouses is near 66.0 m when all the units operate with the rated discharges. Therefore, the upstream water level for steady flow calculation is set as 67.0 m. The total rated discharges of the main and the second channel powerhouses are 11550 m$^3$/s and 6385 m$^3$/s, respectively. The discharge-water head boundary for the steady flow calculation refers to Chen (2017). In addition, flow through the scouring sluices and ship locks are neglected.

The duration of steady flow simulation is 14.4 hours. Figure 7 shows the contour of water level and flow velocity vectors when the flow field is steady. The velocity decreases quickly after the flow enters the reservoir. The water levels in front of the main and the second channel powerhouses are lower than the upstream water level. Because the boundaries at the scouring sluices and the upstream of tributary are solid walls, water in these areas are generally static, and the water levels are little higher than those in front of the powerhouses.
4.3 Results of the surge wave simulation

4.3.1 Surge wave fluctuating and propagating phenomena

The duration of load rejections of the main channel units is 33 seconds, and the discharge of the units is assumed to decrease linearly. From the beginning of the load rejection, the water levels and velocity at four typical moments are shown in Figure 8. To illustrate the fluctuations of water surface in a better way, water levels in these figures are magnified 100 times along the vertical direction.

After the load rejection, surge waves immediately propagate from the load rejected units ($t=0.5$ min). At $t=1.0$ min, the surge waves reach the third channel flood bank for the first time. Then the surge waves are reflected, which causes the rise of water level in front of the second channel powerhouse. In the following period, the surge waves propagate back and forth between the flood banks. This phenomenon is particularly obvious on the preliminary stages of this simulation ($t=2.0$ min and $t=5.0$ min), for the water levels near the two banks fluctuate alternately.

The surge waves also propagate upstream. About $t=2.0$ min, it affects the main channel scouring sluice for the first time, which cause an obvious increase of water level. Meanwhile, it also reaches the upstream river and the tributary ($t=3.5$ min).
4.3.2 Water level fluctuating histories at different locations

To analyze the water level fluctuations at each important building of the hydraulic project, Figure 9 shows the histories of water level at each monitoring point. We can get many information from these figures, such as the first moment affected by the surge waves, the maximum and the minimum heights of water level, and the attenuation duration of surge waves. For instance, the maximum and the minimum heights of water levels at point D are respective 68.0 m and 65.7 m, which occur around 1.2 hours, and 2.55 hours, respectively.
To study the spreading and oscillating rules of the surge waves, we overlaid the histories of water levels in front of the two powerhouses (Figure 10). It is clear that the peaks or valleys of water level in the main channel occur when the water level in second channel reach valleys or peaks, which is the result of the surge waves propagation between the two flood banks.

**Figure 9.** Histories of water level at each monitoring point.

**Figure 10.** Propagation of surge waves between two flood banks causes water fluctuation.

### 4.3.3 Water level fluctuating frequency spectrums

The surge waves are composed of low frequency fundamental wave and high frequency wave. The former is due to the propagation of surge waves between water inlets and upstream boundaries, and the latter is because of the wave propagation between flood banks. We used the fast Fourier transformation (FFT) to divide these frequencies. Dominant frequencies and spectrum analysis diagrams of waves at monitoring points A, B, C, and D are shown in Table 1 and Figure 11, respectively.

In this simulation, the water depth ranges from 28 m to 31 m, thus the speed of surge waves is about 16 m/s according to the wave speed formula. Firstly, the distances from the water inlets to the upstream river boundary and to the upstream tributary boundary are 40 km and 9 km, respectively. The corresponding periods of surge waves are $40000 \times 4/16=10000$ s and $9000 \times 4/16=2250$ s, and the corresponding frequencies are $10 \times 10^{-5}$ Hz and $4.44 \times 10^{-4}$ Hz, which are close to the frequencies 1 and 2 in Table 1. Secondly, the distance between the main channel water inlet and the third channel scouring sluice is 3.55 km. The period is $3550 \times 2/16=443.7$ s, and the corresponding frequency is $2.25 \times 10^{-3}$ Hz, which approximates to the frequency 3. Finally, the period...
of surge waves between the flood banks is $1050 \times 2/16 = 131.25$ s, and its frequency $7.62 \times 10^{-3}$ HZ is close to the frequency 4. In summary, the dominant frequencies represent the frequencies of water oscillation between different channels, which means the results of this simulation agrees with the analyses according to physical rules.

<table>
<thead>
<tr>
<th>Point</th>
<th>Frequency 1 / Hz</th>
<th>Frequency 2 / Hz</th>
<th>Frequency 3 / Hz</th>
<th>Frequency 4 / Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>$9.86 \times 10^{-5}$</td>
<td>$4.16 \times 10^{-4}$</td>
<td>$2.5 \times 10^{-3}$</td>
<td>$7.78 \times 10^{-3}$</td>
</tr>
<tr>
<td>B</td>
<td>$9.86 \times 10^{-5}$</td>
<td>$4.16 \times 10^{-4}$</td>
<td>$2.5 \times 10^{-3}$</td>
<td>$7.78 \times 10^{-3}$</td>
</tr>
<tr>
<td>C</td>
<td>$9.86 \times 10^{-5}$</td>
<td>$4.16 \times 10^{-4}$</td>
<td>$2.5 \times 10^{-3}$</td>
<td>$7.78 \times 10^{-3}$</td>
</tr>
<tr>
<td>D</td>
<td>$9.86 \times 10^{-5}$</td>
<td>$4.16 \times 10^{-4}$</td>
<td>/</td>
<td>/</td>
</tr>
</tbody>
</table>

**Table 1.** Dominant frequencies of water level fluctuations.

Figure 11. Spectrum analysis diagrams of water level fluctuations.

5 CONCLUSIONS

The proposed 1D-3D coupled model combines the advantages of 1D shallow water LB method and 3D free-surface LB method. With the utilization of WSPC and the IB method, the simulation of surge wave fluctuations and propagations in a river-reservoir system after a load rejection transient process is successful. From the results of water levels, flow velocities, and frequency spectrums, it can be concluded that the proposed 1D-3D coupled model is feasible and accurate. The simulated dominant frequencies are in accord with the physical laws, which also proves that this model has good applicability and prospects in engineering researches.

ACKNOWLEDGEMENTS

This work was supported by the National Natural Science Foundation of China (NSFC, Grant Nos. 51839008, 51579187 and 11172219) and the Specialized Research Fund for the Doctoral Program of Higher Education of China (Grant No. 20130141110013).

REFERENCES


