A fully Lagrangian mesh-free numerical model for river ice dynamic

Rubens Augusto Amaro Junior\textsuperscript{1,2}, Andrea Mellado-Cusicahua\textsuperscript{1}, Ahmad Shakibaeinia\textsuperscript{1} and Liang-Yee Cheng\textsuperscript{2}

\textsuperscript{1}Department of Civil, Geological and Mining Engineering Polytechnique Montreal, Montreal Canada
rubens.amaro@polymtl.ca
andrea.mellado-cusicahua@polymtl.ca
ahmad.shakibaeinia@polymtl.ca

\textsuperscript{2}Department of Construction Engineering, Polytechnic School of the University of São Paulo, São Paulo, Brazil
cheng.vee@usp.br

This paper presents a three-dimensional fully Lagrangian numerical model, based on the hybrid discrete element method (DEM) and moving particle semi-implicit (MPS) mesh-free techniques, for modelling the dynamics of river ice floes. The model considers the ice-water dynamics as a multiphase discrete-continuum system. The DEM uses a Hertzian contact dynamic model to simulate the ice floes interaction. It predicts the motion and collision of ice floes as well as their interaction with water, boundaries, and any obstacle in their way. Considering the mesh-free Lagrangian nature of the DEM and MPS, the developed model has an inherent ability to predict the highly dynamic movement of the ice floes (sliding, rolling, colliding, and piling-up) in violent free surface flow. The model is validated and evaluated with a small-scale experiment based on dam-break with floating block floes, which mimics the characteristics of a jam release. The results show the ability of the model to accurately reproduce the three-dimensional dynamic behavior of ice floes as well as the complex hydrodynamics, for example, the water surface motions in the horizontal and vertical directions. The model can be extended for simulations of more complex large-scale ice dynamic problems such as ice-jam formation and release and interaction of ice floes with hydraulic structures.
1. Introduction

Every year in the spring, triggered by the temperature and flow increase, the ice cover breaks up and the resulting ice floes are carried downstream. Along the way, they slide, roll, collide, and jam, which impact river morphology and ecosystem and may pose a major threat to riverside communities and infrastructures. Understanding the processes and mechanisms involved in the dynamics of river ice has a key role in the assessments of the potential impacts, which in turn, will promote adequate and reliable management decisions. Due to the intrinsic complexity, simplified theoretical models of river ice dynamics are not able to describe the complete problem. The experimental and field studies of river ice dynamics have also been restricted, due to the common issues such as instrumental limitations, lack of conventional access to the field, and cost effectiveness (Gerard and Davar 1995). Compared to theoretical solutions and experiments, numerical simulations have caught the researchers’ attention due to its convenience and efficiency. The Québec Dam Safety Act, for instance, requires that dam owners perform numerical simulations to determine the water level increase and the surge speed for different dam-break scenarios (Nolin et al. 2017), which is of great importance to evaluate the potential dangers of a dam-break and its impact. A range of numerical formulations using mesh-based methods can be adopted for solving water-ice interaction problems. However, the handle of solids in free surface flows is still a challenge due to the presence of splashing, fragmentations, merging or multiple bodies interactions. Boundary tracking or remeshing techniques are required, increasing the computational complexity. Recent efforts on accurate simulation of ice dynamics have resulted in two classes of numerical techniques, based on either discrete description or continuum description of ice parcels. While the accuracy of the continuum description largely depends on the accuracy of the ice rheological models, the discrete models have shown to be able to describe the underlying physics in detail. Among discrete techniques, the discrete element method (DEM) (Cundall and Strack 1979), coupled with the 2D depth-averaged Eulerian hydrodynamics, have been extensively used to study sea ice (and more recently river ice) processes (e.g. in Hopkins and Tuhill (2002); Stockstill et al. (2009)). These models have shown to be promising for simulation of sea ice, however they have not been able to capture the 3D highly-dynamic ice-water interaction, which is characteristic of river ice.

Due to the their ability to deal with large deformations and fragmentations in the continuum, the mesh-free particle-based (Lagrangian) methods, such as smoothed particle hydrodynamics (SPH) (Gingold and Monagham 1977; Lucy 1977) and moving particle semi-implicit (MPS) (Koshizuka and Oka 1996), opened new perspectives for the modelling of the fluid-solid interaction problems in recent years. In a particle-based method, each particle, containing a set of continuum field variables such as mass and momentum, move in the Lagrangian system and satisfy the equations of mass, momentum and energy conservation (Shakibaenjia and Jin 2010).

This paper is a first attempt to develop a three-dimensional fully Lagrangian model for water-ice interaction which couples MPS and DEM, for continuum (water) and discrete (ice) phases, respectively. The fluid domain is modelled by the MPS method, in which the differential operators are replaced by weighted average discrete operators on irregular nodes, and a semi-implicit algorithm is applied to solve the governing equation of the continuum. Furthermore, it is very effective for the simulations involving large deformation of free surfaces, fragmentation and merging, or involving complex shaped bodies, large deformation or motion of boundaries, multi-bodies, multi-phase flows and multi-physic problems. The DEM is coupled with the MPS method and is used to simulate the solid ice phase. Usually used to represent granular material (Liu and Ji
here the DEM is adopted following the original idea of Koshizuka et al. (1998), in which the shell of a rigid body is represented by a cluster of particles whose relative positions remain unchanged, similar to the multi-sphere technique (Favier et al. 1999). In order to address numerical instabilities due to non-smooth plane modeled by particles, an approach based on the faces of the bodies and contact force computed using the normal vectors of solid walls, is adopted here (Amaro Jr et al. 2019).

This study also performs experiments on a dam-break with floating block floes and the results are used to validate and evaluate the developed numerical model. Detailed descriptions of the experimental setup and the theoretical background of the numerical models are briefly summarized in sections 2 to 4. Comparisons between the numerical results and the experimental data are presented in section 5. The conclusions are summarized in the final section.

2. Experimental setup

The experiments are performed in a dam-break setup with floating block floes. The test case has been selected based on its highly-dynamic nature, simple geometry, and the quality data that can be obtained for validation of the model. The setup also has the characteristics similar to jam releases and breach of ice-covered reservoirs.

2.1 Experimental apparatus and material

The experiments were performed at the École Polytechnique de Montréal using a plexiglass prismatic tank of 70 cm length, 15 cm width and 30 cm height, as shown in Figure 1(a). A schematic drawing of the setup is shown in Figure 1(b). For the analysis, the reference frame with origin at the point A, as shown in Figure 1(b), was considered. A reservoir is created by placing a removable gate 15 cm from the origin A. The reservoir is filled with water of height $h = 15$ cm. Flat and flexible vinyl was used to seal the gate and reduce water leakage.

Nine equal-size synthetic ice blocks made of polyethylene (with a density similar to that of natural ice) were added on the surface of the water reservoir to simulate the ice floes. The blocks have dimensions of $4.8 \text{ cm} \times 4.8 \text{ cm} \times 2.0 \text{ cm}$ and are identified by different symbols. The center of each block was marked with diagonal lines as depicted in Figure 2(a). Figure 2(b) shows how the blocks were numbered for the analysis.

2.3 Experimental procedure

The experiments are started by near-instantaneous removal of the gate of weight 4751.4 grams. The gate is opened using a pulley and weight mechanism as depicted in Figure 1(b). A horizontal bar retains the weight such that the gate does not move before the bar is removed, as illustrated by the red arrow in Figure 3. The experiments show that the system opens the gate within 0.2 seconds.

2.4 Data acquisition

Two cameras were used to track the horizontal (x axis) and vertical (y axis) motions of each block. One was placed above the tank and the other in front of the tank as shown in Figure 1(b). The one placed at the top is a FASTCAM Mini WX100, high-speed camera (Figure 4(a)) capable of taking 1080 frames per second (fps) with a $2048 \times 2048$ resolution. The frontal camera is a Canon Vixia HF S200, normal camera (Figure 4(b)) with a $1920 \times 1080$ resolution (full HD) and a frequency of about 32 fps. The images are extracted, from the time at which the gate opens, $t = 0$, up to $t = 1.5657$ seconds. Furthermore, the water surface profile was captured by the side camera and its evolution was determined.
Figure 1. (a) Tank used in the experimental setup of the dam-break. (b) Schematic drawing of the tank, water level, solid blocks, and cameras and light source disposition.

Figure 2. (a) The nine blocks used in the experiment (top view). (b) A sketch of the nine blocks with the numbers used for the analysis.
Figure 3. Gate opening system using pulleys and a weight.

Figure 4. (a) The FASTCAM Mini WX100 placed above the tank. (b) The Canon Vixia HF S200 camera placed in front of the tank.

3. Governing equations

The governing equations for incompressible viscous flow are expressed by the conservation laws of mass and momentum

\[
\frac{D\rho}{Dt} = \rho \nabla \cdot \mathbf{u} = 0, \quad [1]
\]

\[
\frac{D\mathbf{u}}{Dt} = -\frac{\nabla P}{\rho} + \nu_f \nabla^2 \mathbf{u} + \mathbf{f}, \quad [2]
\]

where \(\rho\) is the fluid density, \(\mathbf{u}\) is the velocity vector, \(P\) is the pressure, \(\nu_f\) is the kinematic viscosity and \(\mathbf{f}\) is the external body force per unit mass vector.

For the rigid body dynamics, the governing equations of motion are those of translational motion and rotational motion expressed as
where $m$ is the total mass of the rigid body, $\mathbf{a}$ represents the acceleration vector at the center of the rigid body, $\mathbf{f}_{\text{ext}}$ denotes external forces, $\mathbf{I}$ is the inertia matrix, $\mathbf{\omega}$ stands for the angular velocity vector about the principal axes of the rigid body and $\mathbf{M}_{\text{ext}}$ is the external moment. The external forces are calculated considering the effect of gravity $\mathbf{f}_g$, hydrodynamic forces on the rigid surface $\mathbf{f}_h$ and contact forces between the rigid bodies $\mathbf{f}_c$, which is explained in the next sections. The contribution of the hydrodynamic force added to the gravity force and the hydrodynamic moment $\mathbf{M}_h$ acting on the rigid body are calculated as

$$\mathbf{f}_h + \mathbf{f}_g = -\iint_S P \, ds + m \mathbf{g},$$  \hspace{1cm} \text{(5)}$$

$$\mathbf{M}_h = -\iint_S \mathbf{r} \times P \, ds,$$  \hspace{1cm} \text{(6)}$$

where vector $\mathbf{r}$ denotes the position vector from the center of the rigid body and $ds$ is the area multiplied by the normal vector on the rigid body surface. Focusing the impulsive hydrodynamic loads on the rigid solid, shear forces were neglected in Eq. (6) and only the normal force due to pressure was considered.

### 4. Numerical methods

#### 4.1 Moving Particle Semi-implicit (MPS)

The MPS method solves the governing equations of continuum by replacing the differential operators by discrete operators derived based on a weight function ($\omega_{ij}$)

$$\omega_{ij} = \begin{cases} \frac{r_e}{|\mathbf{r}_{ij}|} - 1 & |\mathbf{r}_{ij}| \leq r_e \\ 0 & |\mathbf{r}_{ij}| > r_e \end{cases},$$  \hspace{1cm} \text{(7)}$$

where $r_e$ is the effective radius that limits the range of influence and $|\mathbf{r}_{ij}| = |\mathbf{r}_j - \mathbf{r}_i|$ is the distance between the particles $i$ and $j$.

The summation of the weight of all the particles in the neighborhood of the particle $i$ is defined as its particle number density $n_i = \sum_{j \neq i} \omega_{ij}$, which is proportional to the fluid density.

For a scalar function $\phi$ and a vector function $\mathbf{\phi}$, the gradient, divergence and Laplacian operators are defined as

$$\langle \nabla \phi \rangle_i = \frac{d}{n_0} \sum_{j \neq i} \frac{\phi_j - \phi_i}{|\mathbf{r}_{ij}|^2} \mathbf{r}_{ij} \omega_{ij},$$  \hspace{1cm} \text{(8)}$$
\begin{align}
  \langle \nabla \cdot \phi \rangle_i &= \frac{d}{n^0} \sum_{j \neq i} \frac{\Phi_j - \Phi_i}{|\mathbf{r}_{ij}|^2} \cdot \mathbf{r}_{ij} \omega_{ij}, \\
  \langle \nabla^2 \phi \rangle_i &= \frac{2d}{\lambda_i n^0} \sum_{j \neq i} (\phi_j - \phi_i) \omega_{ij},
\end{align}

where \( d \) is the number of spatial dimensions and \( n^0 \) is the particle number density for a fully supported region. \( \lambda_i = \sum_{j \neq i} |\omega_{ij}| |\mathbf{r}_{ij}|^2 / \sum_{j \neq i} \omega_{ij} \) is a correction parameter.

To solve the incompressible viscous flow, a semi-implicit algorithm is used in the MPS. At first, predictions of the particles velocity and position are carried out explicitly by using viscosity and external forces terms of the momentum conservation. Then the pressure of fluid and wall particles is obtained implicitly by solving a linear system of pressure Poisson equation (PPE) (Tanaka and Masunaga 2010)

\[
  \langle \nabla^2 P \rangle_{i,t+\Delta t} = -\frac{\rho}{\Delta t^2} \alpha_c P_{i,t+\Delta t} = \gamma \frac{\rho}{\Delta t^2} \left( \frac{n_0 - n_i^t}{n_0} \right) + (1 - \gamma) \frac{\rho}{\Delta t} \nabla \cdot \mathbf{u}^*,
\]

where \( \Delta t \) is the time step, \( n_i^t \) is the current particle number density, \( \alpha_c \) is the coefficient of artificial compressibility and \( \gamma \) is the relaxation coefficient. Both \( \alpha_c \) and \( \gamma \) are used to improve the stability of the computation method. Also, to prevent instability issue induced by attractive pressure and to reduce the effect of nonuniform particle distribution, the first order pressure gradient was adopted here as (Wang et al. 2017)

\[
  \langle \nabla P \rangle_i = \left[ \sum_{j \neq i} \omega_{ij} \frac{\mathbf{r}_{ij}}{|\mathbf{r}_{ij}|} \otimes \frac{\mathbf{r}_{ij}^T}{|\mathbf{r}_{ij}|} \right]^{-1} \sum_{j \neq i} \frac{P_j - \hat{P}_i}{|\mathbf{r}_{ij}|^2} \mathbf{r}_{ij} \omega_{ij},
\]

where \( \hat{P}_i \) is the minimum pressure between the neighborhood of the particle \( i \).

Finally, the velocity of the fluid particles is updated by using the pressure gradient term of the momentum conservation and the updated positions of the particles are obtained.

Solid wall is modelled by using three layers of fixed particles. The particles that form the layer in contact with the fluid are denominated wall particles, of which the pressure is computed by solving PPE, together with the fluid particles. The particles that form two other layers are denominated dummy particles, which are used to assure the correct calculation of the particle number density of the wall particles. Pressure is not calculated in the dummy particles. As boundary condition of rigid walls, a relative velocity of the fluid and the wall of zero is imposed. The Dirichlet pressure boundary condition is imposed to the particles identified as free surface and is considered during the implicit step of the method. For the present MPS, the Neighborhood Particles Centroid Deviation (NPCD) method (Tsukamoto et al. 2016) is adopted. By identifying accurately the free surface particles, the NPCD method improves the stability and accuracy of the pressure computation, by eliminating spurious oscillations due to misdetection of free surface particles.

4.2 Discrete Element Method (DEM)

The shell of a rigid body is represented by a collection of particles whose relative positions remain unchanged (Koshizuka et al. 1998). The overall contact force on the rigid body is calculated by
adding the contributions due to the normal $f_n$ and tangential $f_t$ contact forces between particles belonging to different bodies, similar to the multi-sphere DEM approach (Favier et al. 1999). The normal forces between the pair of closest particles $k$ and $l$ belonging to different bodies can be described following a non-linear Hertz’s elastic contact theory (Johnson 1985)

$$ f_{n,kl} = k_{n,kl} \frac{3}{2} \delta_{kl}^{3/2} \mathbf{n}_c + c_{n,kl} \delta_{kl}^{1/4} \hat{\delta}_{kl} \mathbf{n}_c, \quad [13] $$

where $k_{n,kl} = \frac{4}{3}E_{kl}\sqrt{l_{0,kl}}$ is the normal stiffness constant of pair $kl$, $\delta_{kl}$ is the overlap (penetration) between two wall particles belonging to two different bodies, $\hat{\delta}_{kl}$ is the rate of normal penetration, $c_{n,kl} = \xi_n \sqrt{6 m_{kl} E_{kl} \sqrt{l_{0,kl}}}$ is the normal damping constant and $\mathbf{n}_c$ is the contact normal vector defined as follows:

1. If particle $l$ is face, the normal of $l$ is used as $\mathbf{n}_c$.
2. If particle $l$ is not face, but $k$ is face, the reverse of the normal of $k$ is used as $\mathbf{n}_c$.
3. Neither particles are face, the reverse of the distance vector $\mathbf{r}_{kl}/|\mathbf{r}_{kl}|$ is used as $\mathbf{n}_c$.

As the geometry of a cuboid is enough to model all rigid bodies considered in the present work, the number of neighbor particles is used as the criterion to identify if a particle is face. The contact ratio $\xi_n$ must be assigned for each case, whereas $E_{kl}$, $m_{kl}$ and $l_{0,kl}$ are obtained as

$$ E_{kl} = \frac{E_k E_l}{(1 - v_k^2)E_k + (1 - v_l^2)E_l}, \quad m_{kl} = \frac{m_k m_l}{m_k + m_l}, \quad l_{0,kl} = \frac{l_{0,k} l_{0,l}}{l_{0,k} + l_{0,l}}, \quad [14] $$

where $E_k$, $E_l$, and $v_k$, $v_l$ as the Young’s modulus and the Poisson’s ratio of particles $k$ and $l$, respectively. In the present work, only one initial distance between two adjacent particles $l_0$ is used for all domain, thus leading to $l_{0,kl} = l_0/2$. Moreover, if $l$ is a particle belonging to a fixed rigid wall, then $m_l \to \infty$, implying that $m_{kl} = m_k$.

Tangential forces are given by a linear dash-pot following by the Coulomb friction law or repulsive and damped forces by assuming a linear model. The force with smaller absolute value is used during the simulation. The linear spring dashpot model is given by:

$$ f_{n,kl} = k_{t,kl} \delta_{kl}^t \mathbf{t}_c + c_{t,kl} \hat{\delta}_{kl}^t \mathbf{t}_c, \quad [15] $$

where $k_{t,kl} = \frac{2}{7}k_{n,kl}\sqrt{l_0}$, $c_{t,kl} = \frac{2}{7}c_{n,kl}$ (Hoomans 2000), $\delta_{kl}^t$ is the tangential deformation, $\hat{\delta}_{kl}^t$ is the rate of tangential deformation, and $\mathbf{t}_c$ is the tangential contact vector.

The Coulomb law is modified with a sigmoidal function in order to make it continuous in the origin regarding the tangential velocity (Vetsch 2011)

$$ f_{t,kl}^c = \mu_{kl} |f_n| \tanh(8 \hat{\delta}_{kl}^t) \mathbf{t}_c, \quad [16] $$

where $\mu_{kl}$ is the kinetic friction coefficient for the pair of particles $k$ and $l$. In the present work, when the neighbor particle $l$ belongs to a forced or fixed solid, the kinetic friction coefficient $\mu_t$ of the neighbor particle $l$ is adopted for $\mu_{kl}$, otherwise $\mu_{kl}$ is obtained as the mean value.
\[ \mu_{kl} = \left( \frac{\mu_k^0 + \mu_l^0}{2} \right)^\theta, \]

similar to the interaction viscosity between two particles in Shakibaeinia and Jin (2012). In this work, \( \theta = 1 \) is adopted for all simulations.

**5. Results and discussion**

In order to illustrate the ability of the proposed model to accurately reproduce the complex dynamic behavior of water-ice interaction phenomenon, the in-house developed code was applied to the solid-fluid dam-break case, previously detailed in section 2. For all simulations, we adopted the density of the fluid as \( \rho = 1000 \) kg/m\(^3\), kinematic viscosity \( \nu_f = 10^{-6} \) m\(^2\)/s and the gravity acceleration \( g = 9.81 \) m/s\(^2\). The physical properties of the solids and simulation parameters are presented in Table 1 and Table 2, respectively.

<table>
<thead>
<tr>
<th>Table 1. Physical properties of the solids.</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \rho_s ) (kg/m(^3))</td>
</tr>
<tr>
<td>-----------------</td>
</tr>
<tr>
<td>tank</td>
</tr>
<tr>
<td>block</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 2. Simulation parameters.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
</tr>
<tr>
<td>---------------------------------</td>
</tr>
<tr>
<td>Particle distance ( l_0 ) (m)</td>
</tr>
<tr>
<td>Time step ( \Delta t ) (s)</td>
</tr>
<tr>
<td>Effective radius ( r_e ) (m)</td>
</tr>
<tr>
<td>Surface threshold ( \beta_1 )</td>
</tr>
</tbody>
</table>

Figures 5 and 6 demonstrate a qualitative evaluation, showing a sequence of frames from the experiment and the simulation carried out in the present study. The colors on the free surface of the fluid in the numerical simulation are related to its dimensionless velocity field \( \nu'^* = |\nu|/(gh)^{-0.5} \), where \( h = 0.15 \) m is the water column height. After the collapse of the water column, the water transports the solids downstream, with a good agreement between experimental and numerical solid positions at the instant \( t = 0.307 \) s. However, the numerical dissipation inherent of particle-based methods leads to a slight delay of the computed wave front compared to the experiment. At the instant \( t = 0.645 \) s, the numerical solid motions agree well with the experimental ones, even though the computed water splash is slightly underestimated. Between the instants \( t = 0.952 \) s and 1.197 s, the merging of the collapsed upward water flow and reflected wave dramatically disturbs the water surface and initiates a more chaotic motion of the solids, especially for the solids downstream. The computed positions of the three upstream blocks are still in a good agreement with the measured ones, while the computed positions of the remaining six blocks deviate from the experiment.
Figure 5. Snapshots of the experimental and numerical dam-break case (front view).

(a) $t = 0.307\, s$  \hspace{1cm}  (b) $t = 0.645\, s$

(c) $t = 0.952\, s$  \hspace{1cm}  (d) $t = 1.197\, s$
Figure 6. Snapshots of the experimental and numerical dam-break case (top view).

Figure 7 illustrates the propagation of the experimental (red dots) and numerical (blue dots) water surface profiles. The numerical water surface profile reproduces the experimental one very well until the instant $t = 0.737$ s. Qualitative differences are observed only at the instant $t = 0.921$ s, where the experimental surface elevation downstream is higher than the numerical result. These differences are reasonable, since an intense splash with air-water mixture is created approximately at this instant, after the fall of the vertical run-up jet onto the underlying fluid.
Figure 7. Evolution of the experimental (red dots) and numerical (blue dots) wave profiles.

(a) $t = 0.000 \, s$
(b) $t = 0.184 \, s$
(c) $t = 0.368 \, s$
(d) $t = 0.553 \, s$
(e) $t = 0.737 \, s$
(f) $t = 0.921 \, s$

Figure 8. Measured (red cross) and computed (blue line) block motions along the horizontal direction ($x$ axis). Blocks (a) 1, (b) 2, (c) 3, (d) 4, (e) 5, (f) 6, (g) 7, (h) 8 and (i) 9.

Figure 8 shows the experimental and numerical horizontal motions ($x$ axis) of the 9 blocks. From Fig. 8, the numerical simulation provides consistent results for all horizontal block motions compared with the experimental ones until the instant $t = 0.90 \, s$, approximately. After that, the
block motions behavior become more chaotic due to the merging of the collapsed upward water flow and reflected wave, as previously observed. Therefore, the discrepancies between the computed and measured motions occur after \( t = 0.90 \) s, although the computed motions of the blocks 2, 3, and 9 match well with the experiment.

Experimentally measured and numerically computed vertical motions (y axis) are presented in Figure 9. The numerical and experimental values present an initial discrepancy at \( t = 0 \) due to the slight water height increase after the addition of the blocks in the dam. The measured vertical motions are well reproduced by the numerical simulation before \( t = 0.80 \) s. While the discrepancies begin to increase for the upstream blocks (1, 4 and 7), the differences between computed and measured motions of the downstream blocks tend to reduce after \( t = 1.20 \) s, indicating a more stable free surface motion for this region.

The overall good agreement between experimental and computed motions demonstrates that the present model is able to reproduce the main behaviors of the complex fluid-solid interaction phenomenon.

![Figure 9](image-url)

**Figure 9.** Measured (red cross) and computed (blue line) block motions along the vertical direction (y axis). Blocks (a) 1, (b) 2, (c) 3, (d) 4, (e) 5, (f) 6, (g) 7, (h) 8 and (i) 9.
6. Concluding remarks

A three-dimensional fully Lagrangian numerical model, based on a hybrid DEM-MPS technique, for modelling of the dynamic of water-ice interaction was investigated in the present work. A dam-break experiment with floating block floes has been designed and conducted to investigate the dynamic behavior of ice floes as well as the complex hydrodynamics. Qualitative and quantitative comparisons between the experimental and numerical results were carried out. The positions of the blocks obtained by the present model and experiment were compared, and an overall good agreement showed the effectiveness of the present approach to reproduce the main features of the fluid-solid interaction phenomenon. Some differences between computed and measured motions were observed when complex flow patterns, due to the merging of the collapsed upward water flow and reflected wave, occur. The results suggest that the present model can be extended for simulation of real-scale ice dynamic problems, such as, ice-jam formation and breakup and interaction of ice floes with hydraulic structures. Furthermore, additional experimental efforts are desirable in providing experimental data for validating the model.

Acknowledgments

Authors would like to acknowledge the support of the Coordenação de Aperfeiçoamento de Pessoal de Nível Superior - Brasil (CAPES) and the Natural Sciences and Engineering Research Council of Canada (NSERC) for providing scholarship support for the first and second authors, respectively. This research was also partially supported by Canada Research Chair Program. Authors would like to also thank Mr. Étienne Bélanger for his assistance in setting up the experiments. The first and fourth authors are also grateful to Petrobras for financial support on the development of the MPS/TPN-USP simulation system based on MPS method.

References


