Prediction of Ice Formation Using Smoothed Particle Hydrodynamics (SPH) Method

M. M. Kamyabi
Department of Chemical Engineering, Vali-e-Asr University of Rafsanjan, Rafsanjan, Iran
mm.kamyabi@vru.ac.ir

Abstract
Smoothed Particle Hydrodynamics (SPH) was applied to simulate the three-phase phenomenon of ice formation on a solid surface exposed to air flow in three dimensions. The air flow carried subcooled water droplets. The method was verified by comparing its prediction of the rate of ice formed with the literature data. The effect of air flow velocity on the amount and rate of ice formation was investigated. It was seen that with increasing the air flow velocity the amount of formed ice was also increased. Moreover, two defrosting schedules of constant heat flux on the surface were examined. According to the results, the reduction percent of ice formation was increased by increasing the heat flux and the reductions were more significant at lower air velocities.

Keywords: Ice formation, Meshless method, SPH, Phase change

Introduction
Ice formation on solid surfaces is a common phenomenon that happens in many situations from daily life to industry. Ice formation is a phase change process that may have a useful or harmful effect in different situations. For example, ice formation in the refrigerator is a useful process but its formation on the surface of airplanes is a dangerous problem that decreases the lift force and increases the drag force [1]. However, the experimental study of ice formation in some situations (such as inside the wind tunnel) is very difficult [2,3]. This matter in addition to the intrinsic benefits of simulation, makes the simulation a substantial way for the prediction of ice formation.

At least three phases are involved when ice is forming on a solid surface exposed to humid air flow: The solid phase, the ice phase, and the humid air phase. Due to the various existed phases, CFD simulation of such systems is complicated. More complexity comes from the moving solid-fluid boundary existed during icing.

Usual methods for solving multi-phase flows are based on the Eulerian viewpoint. It means that the governing equations of each phase are solved on a Eulerian network. In these methods, it is necessary to re-calculate the location of the interface (solid-fluid boundary) at each time step and modify the mesh subsequently. Although these methods have been widely improved and benefit from high efficiency, they suffer from many difficulties especially in the face of moving boundaries. These difficulties include numerical unrealistic diffusion and high computational costs consumed for adapting mesh [4].
Some other methods which are based on the Lagrangian viewpoint, are dealing better with the moving boundary problems. These methods are based on a flexible mesh (such as arbitrary Eulerian-Lagrangian method) or a group of computational particles (nodes). Smoothed particle hydrodynamics (SPH) is a member of the second class. SPH as the oldest meshless Lagrangian method was developed firstly by Lucy [5] and Monaghan and Gingold [6]. This method’s applicability has been proved in many problems.

**Problem Definition**

An air flow containing sub-cooled water droplets (named carrying flow) hits a solid surface from the bottom side as shown in Figure 1. As a result, some water droplets stick to the surface and form the ice.

![Air Flow](image)

**Fig.1- Problem Geometry**

Hydrodynamics of the carrying flow was supposed similar to a continuum single-phase flow because of the low concentration of water droplets [7]. Therefore, the governing equations for the fluid are conservation of mass and momentum for single-phase flow.

As discussed, it was not demanded to apply separate mass and momentum conservation equations for the sub-cooled droplets but the energy conservation equation was considered during heat transfer between the air and the droplets. The aforesaid equation is as below:

\[
\frac{dE}{dt} = \rho C_p \frac{dT}{dt} = K_a \left( \frac{d^2 T_a}{dx^2} + \frac{d^2 T_d}{dx^2} \right)
\]  

(1)

Where \(K\), \(\rho\), and \(C_p\) are heat conductivity, density and specific heat capacity and indexes of \(d\) and \(a\) are for droplets and air phases respectively. The energy conservation model of Monahan et al. [8] was used in the current work. According to this model for the solid-fluid boundary movements \((dx/dt)\) due to the phase change, the following equation was applied:

\[
K_s \frac{dT_s}{dx} - K_i \frac{dT_i}{dx} = \rho h_{ls} \frac{dx}{dt}
\]

(2)

Where \(h_{ls}\) is the latent heat of fusion and \(l_s\), and \(i\) stand for solid, liquid and formed ice respectively. This equation declares that a part of the energy received by the boundary is added to the released heat of icing and transfers from the other side. For the present problem, the heat
transfer is by convection from the air side. Moreover, the kinetic energy of droplets should also be considered. Therefore, the revised equation was:

\[ h_{ad} (T_a - T_d) + (\beta VG) \frac{V_j^2}{2} - \rho_i K_i \frac{dT}{dx} = \rho_i h_{es} \frac{dx}{dt} \]  

In the above equation, the location of the boundary (x) is still an unknown. The SPH method allows determining the location of the boundary exactly in every moment. For this purpose, thermal equations of both phases were solved. Ice formation was defined as a state that the enthalpy of the sub-cooled droplet became equal to the enthalpy of the ice. Then if the ice particle was close enough to the surface and its velocity was low enough the particle sticks to the solid surface.

**Numerical Method**

Using SPH, particles are computational points and carry fluid quantities such as mass, velocity, and pressure. SPH method is built on the concept of interpolation where interpolated value \( U \) at position \( r \) is computed as:

\[ U(r) = \sum \frac{m_j}{\rho_j} U(\tilde{r}_j) W(\tilde{r}_j - r, h) \]  

Where \( U \) is each arbitrary field function and \( \frac{m_j}{\rho_j} \) is the volume of the neighboring particle \( j \). Particle \( j \) is the neighbor if it is within the circular area around a point at \( r \) with radius \( h \). \( W \) is a positive smoothing or kernel function representing the Dirac delta function. In the present work, the suggestion of Müller et al. [9] for the kernel function was used as:

\[ W(r, h) = \begin{cases} \frac{315}{64 \pi h^3} (h^2 - ||r||^2)^3 & 0 < ||r|| < h \\ 0 & ||r|| > h \end{cases} \]  

This kernel function is not suitable for calculating pressure forces. The Kernel function suggested by Spikey [9] was used for calculating pressure terms:

\[ W(r, h) = \begin{cases} \frac{15}{\pi h^3} (h - ||r||)^3 & 0 < ||r|| < h \\ 0 & ||r|| > h \end{cases} \]  

Different derivative schemes have been developed. In the present work the scheme suggested by Kelager [9] was used because of its high accuracy in calculating first derivatives:

\[ \nabla U = \rho \sum_j \left( \frac{U_j}{\rho_j} + \frac{U}{\rho^2} \right) m_j \nabla W_{ij} \]  

Only for the first derivative of energy, the following scheme was used [8]:

\[ \frac{dE_i}{dt} = \sum_j 4m \frac{1}{\rho_i \rho_j} \frac{K_i K_j}{K_i + K_j} \tau_{ij} \cdot \nabla W_{ij} (T_i - T_j) \]
The second derivatives were estimated by this equation [11].

$$\nabla^2 U = \sum_j \frac{m_j}{p_j} (U_j - U_i) \nabla^2 W_{ij}$$

(9)

For time integration the semi-implicit Euler method was used.

Equations 1, 2, … was solved by the SPH method which its code was developed in the Fluidix software.

**Results and discussion**

Rare experimental data are available for the icing phenomenon. Moreover, matching all input parameters among experiments and present work is impossible. However, the average mass rates of formed ice in two experimental works were compared with the prediction of the present work in Table 1.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Average mass flow rate (Kg ice/s)</td>
<td>$1.1 \times 10^{-5}$</td>
<td>$5 \times 10^{-6}$</td>
<td>$5.6 \times 10^{-5}$</td>
</tr>
</tbody>
</table>

Although the agreement in the order of magnitude of the ice mass rate is satisfying, the differences come from the steady-state assumption took by [12,13] and higher droplet temperatures supposed in the present work.

Figure 2, shows the 2-D screenshots of contours of velocity and temperature of the air flow at some arbitrary time steps. According to the figure, as long as the flow passes on the solid surface, the ice particles are formed and the flow is adopted.

Varying air velocity, the simulations were repeated. In Figure 3, the percent of ice formation is shown in three different air velocities of 10, 20, and 50 m/s.
According to this figure, the percent of ice formation is increased by increasing the air velocity. The rate of ice formation is also shown in Figure 4. According to this figure, the rate of ice formation is also increased by increasing the flow velocity. Actually, when the velocity is increased both the number and intensity of droplet-solid collisions is also increased. As a consequence, more ice particles form and stick to the surface. For considering an anti-ice system, a constant heat source (Q) was added to the surface and the simulations were repeated.

Figure 5 shows the amount of ice formation on the surface at different heat source (Q) and air velocities. Table 1 compares Figures 3 and 5 and shows the ability of different anti-ice systems in the reduction of ice formation. It is seen that the effectiveness of such an anti-ice system is more highlighted at low air velocities.
Fig 5. Effect of heat per mass on the amount of ice formation

Table 2 - Percent of reduction in the ice formation due to the anti ice system

<table>
<thead>
<tr>
<th>Q (W/Kg)</th>
<th>10(^6) W/Kg</th>
<th>10(^7) W/Kg</th>
</tr>
</thead>
<tbody>
<tr>
<td>V (m/s)</td>
<td>10 20 50</td>
<td>10 20 50</td>
</tr>
<tr>
<td>% Ice formation reduction</td>
<td>87.5 84 64</td>
<td>90 85 72</td>
</tr>
</tbody>
</table>

Figure 6, shows the effect of the same conditions on the rate of ice formation. The simultaneous effects of air velocity and heat source are clear in Figures 5 and 6. According to these figures, with increasing heat per mass of particles, the ice formation is decreased and the reduction is more significant in low air flow velocities.

**Conclusions**

The ice formation on a solid surface was simulated. The effects of air flow velocity on the amount and rate of ice formation were investigated in the usual conditions and in the presence of and anti-ice system. It was seen that more ice is formed in high velocities and the formation of ice was decreased with increasing the heat flux. This reduction was more significant at low air velocities.

**References**


