Animation of Ice Melting Phenomenon Based on Thermodynamics with Thermal Radiation

Makoto Fujisawa*
Shizuoka University

Kenjiro T. Miura†
Shizuoka University

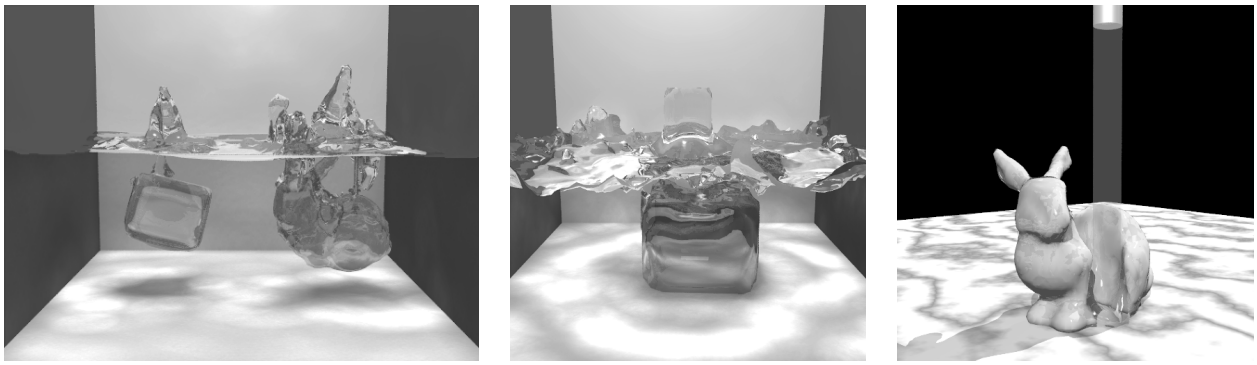


Figure 1: *Melting objects in hot water (left), with liquid flow (center), and thermal radiation (right)*

Abstract

This paper proposes a fast and efficient method for producing physically based animations of the ice melting phenomenon, including thermal radiation as well as thermal diffusion and convective thermal transfer. Our method adopts a simple color function called the VOF (Volume-of-Fluid) with advection to track the free surface, which enables straightforward simulation of the phase changes, such as ice melting. Although advection of functions that vary abruptly, such as the step function, causes numerical problems, we have solved these by the RCIP (Rational-Constrained Interpolation Profile) method. We present an improvement to control numerical diffusion and to render anti-aliased surfaces. The method also introduces a technique analogous to photon mapping for calculating thermal radiation. By the photon mapping method tuned for heat calculation, the thermal radiation phenomenon in a scene is solved efficiently by storing thermal energy in each photon. Here, we report the results of several ice melting simulations produced by our method.

CR Categories: I.3.7 [Computer Graphics]: Three-Dimensional Graphics and Realism—Animation;

Keywords: dynamic fluid simulation, multi-phase fluid, melting, thermodynamics, heat radiation

1 Introduction

There has been a great deal of research regarding physically based fluid simulation in the field of Computer Graphics, and production-quality animations can be generated. The research domain includes simulations still actively researched even in the field of Fluid Dynamics (e.g. multiple fluid and multi-phase fluid). This paper described simulations of ice melting phenomena based on thermodynamics, including thermal radiation. For accurate realization of the ice melting phenomenon, not only simulations of solids and liquids are necessary, but also those of the convection of gas, i.e., convective thermal transfer, and thermal radiation are required. The analysis of thermal radiation requires some of the most complicated calculations in thermodynamics. Moreover, it is necessary to simulate the fluid-dynamical behavior of both gas and liquid simultaneously, as well as the transitions among the solid, liquid, and gas phases. We propose a method to simulate the multi-phase fluid, including the phase changes, and take into account thermal diffusion, convective thermal transfer, and thermal radiation.

The main problem for fluid simulation is tracking the free surface. The level set methods are the most commonly used in CG [Osher and Fedkiw 2003]. These methods use a function (e.g., the signed distance function) smooth enough on the interface between a gas and a liquid and distinguish the liquid from the gas by the function values. The smoothness of the interface is maintained by the periodical reinitialization process. However, this process has a high computational cost, and some special treatment is necessary when we simulate the phase change between solid and liquid [Losasso et al. 2006a].

We represent gas-liquid two-phase fluids by a simple color function in which that the value is 0 for a gas and 1 for a liquid. This function is usually called the VOF (Volume-of-Fluid) that denotes volume fraction at each grid cell of the liquid. We have found that the VOF is more suitable than the level set (the signed distance function) to represent the liquid interface for simulation of ice melting, because it only has to increase and decrease the function value simply when we simulate the phase change from solid to liquid and vice versa. However, the function varies abruptly at the liquid interface and its advection causes problems. We adopted the CIP (constrained interpolation profile) method [Takewaki and Yabe 1987] to advect the VOF function. The CIP method can advect a function without losing its original shape even if it has various frequencies and it changes rapidly. When using the CIP method, a sharpness of the

*e-mail:r5545005@ipc.shizuoka.ac.jp
†e-mail:tmkmiur@ipc.shizuoka.ac.jp

GRAPHITE 2007, Perth, Western Australia, December 1–4, 2007.
© 2007 ACM 978-1-59593-912-8/07/0012 $5.00

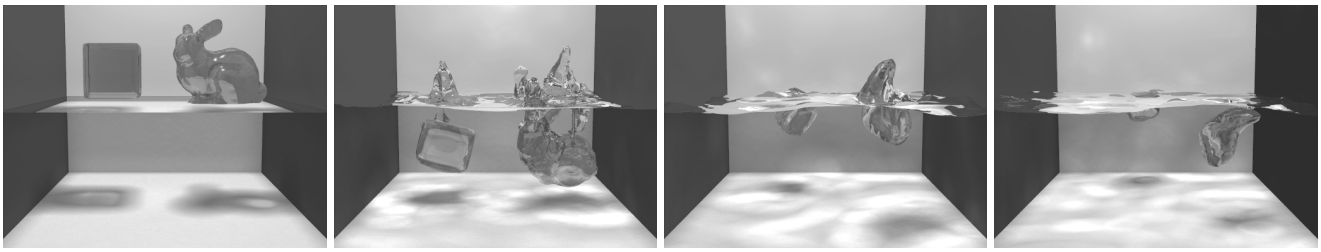


Figure 2: *An ice cube and an ice bunny falling into* $363[K]$ *hot water (*$64 \times 64 \times 64$ *grid).*

advected function at interface can be an important factor for the stable fluid simulation. In contrast, a smoother interface can be important factor for the CG applications. to avoid an aliasing of the interface rendering, In this paper, we improved the STAA (Surface Tracking by Artificial Anti-diffusion) method, which resolves the interface diffusion problem of the RCIP method and propose a new method suitable for CG, which retains the smoothness of interface and avoids excessive diffusion. Using this method, it is possible to achieve simulation stability, and improve smoothness of the interface, a drawback of the standard VOF method, at the same time.

The other key component to realize melting phenomena on the computer is thermodynamics simulation. Heat obeys the first law of thermodynamics, conducts through objects (thermal diffusion), and flows with fluids (convective thermal transfer). These laws are similar to those of fluid dynamics and almost the same codes to simulate the behavior of fluids can be utilized to develop thermodynamics simulators. Thermal radiation is an exceptional phenomenon associated with heat but not fluid. This is one of the heat transfer types where thermal energy is transmitted as electromagnetic waves, including infra-red and visible light irrelevant to the presence of a material medium. We propose a method using photon mapping to calculate thermal radiation. Thermal radiation can be described based on the wave theory, but it can be also done on photons - particles with the minimum light energy. The thermal radiation phenomenon can be analyzed efficiently by regarding a photon as an carrier of the thermal energy.

The rest of the paper is organized as follows. Section 2 describes previous work and Section 3 details our fluid simulation techniques for tracing the free surface of a fluid, gas-liquid two-phase flows, and surface tension. Section 4 discusses our thermal radiation simulation method by photon mapping. Section 5 presents experiments performed using our technique. Finally, we conclude the paper in section 6 with a discussion of future work.

2 Previous Work

A number of methods have been developed in the field of CG to simulate fluid flows. Harlow and Welch developed the MAC (Marker-and-Cell) method, which uses marker particles to track the free surface of a fluid [Harlow and Welch 1965]. Foster and Fedkiw created realistic fluid animations using the MAC method [Foster and Fedkiw 2001]. They adopted the semi-Lagrangian method [Stam 1999] to stably solve the 3-dimensional Navier-Stokes equations to calculate the advection term. Furthermore, they introduced a level set method to simulate fluid surfaces and successfully reproduced their complicated behavior.

Enright *et al.* developed the particle level set method to simulate the free surface of a fluid [Enright et al. 2002]. This method combines surfaces represented by implicit functions with marker particles, tracks the free surface, and compensates for conservation of volume, which is not guaranteed by the level set method. It can realistically simulate complicated free surfaces of fluids. For example, the method simulates interactions with rigid bodies (two-way interaction) [Carlson et al. 2004]. It was augmented by an octree data structure [Losasso et al. 2004]. It has been used to realise surface tension [Hong and Kim 2005], water drops calculated from contact angles [Wang et al. 2005], interactions with thin objects [Guendelman et al. 2005], and multiple fluids [Losasso et al. 2006b], bubbles in foam by a regional level set method [Zheng et al. 2006; Kim et al. 2007].

Takahashi *et al.* [Takahashi et al. 2003] represented splashing and foaming by combining the CIP [Takewaki and Yabe 1987] and VOF methods. Song *et al.* [Song et al. 2005] simulated multiphase fluids with droplets and bubbles by combining the CIP method and the level set method. Furthermore, Yabe *et al.* [Yabe et al. 2001b] discussed how to calculate surface tension and interactions with solid using the C-CUP method. We use a method based on these techniques. As movements smaller than the grid resolution can be reproduced by virtue of the excellent performance of the CIP method, satisfactory results can be obtained using a grid resolution half that used for the level set method. Furthermore, the VOF functions can deal with phase transitions directly by exchanging volume fraction between two phases by exchanging values between their VOF functions. One of the problems of the CIP method is overshoot. Xiao et al. developed the RCIP method [Xiao et al. 1996a], which uses rational functions to interpolate data to resolve this problem. However, in the simulation by this method, numerical diffusion at the fluid interface becomes problematic and the STAA method was devised to resolve this problem [Ikebata and Xiao 2002; Xiao et al. 2005]. Although the original STAA proposed by Ikebata and Xiao constructed signed distance fields to determine normal directions of the interface, we have reduced the processing time by calculating them with the smoothed VOF function. Furthermore, we propose a new RCIP advection method maintaining a certain interface width for rendering.

In the CG field, there have been a number of studies that simulating melting phenomena of solids, such as ice. Fujishiro and Aoki [Fujishiro and Aoki 2001] using morphology and form factors and Jones [Jones 2003] based on thermodynamics attempted to simulate deformation of melting solids for CG expressions. However, their methods were not complete; they did not take account of the generation and behavior of fluids resulting from phase changes and their methods had such limitations preventing them from handling specular reflection for thermal radiation calculations. In addition, their simulation required a large amount of processing time. To deal with fluids after the phase change, Carlson et al. simulated materials of high viscosity, such as wax, by calculating solid objects as fluids of high viscosity with a solver that yielded stable solutions even for highly-viscous fluids [Carlson et al. 2002]. Their solver can handle fluids of a wide range of viscosities and realizes the phase transitions by changing the viscosity of the fluids from high to low according to the temperature. However, for example, in the case of ice melting where resultant fluids have a low viscosity, it can

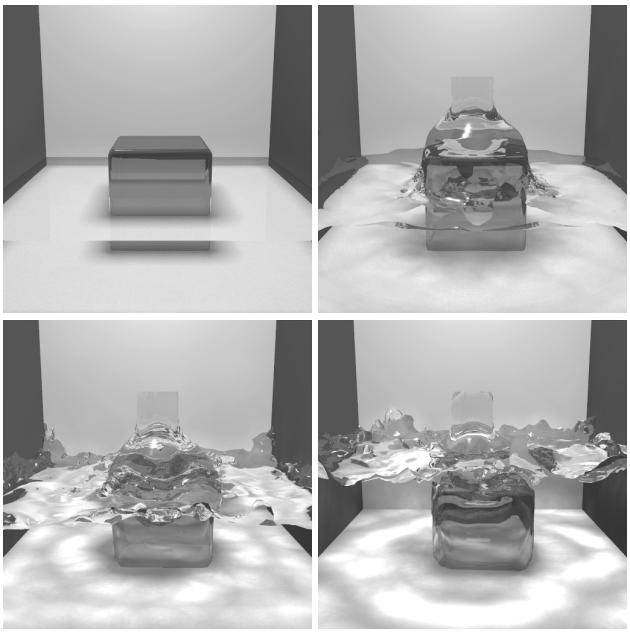


Figure 3: *Pouring* $363[K]$ *hot water onto an ice cube. The top of the ice cube is melting more rapidly because of forced convective heat transfer (*$64 \times 64 \times 64$ *grid).*

be expected to have numerical problems around solid boundaries, and it is difficult to obtain clear boundaries between solids and liquids, which may cause rendering problems. Furthermore, they did not account for either thermal radiation or convective thermal transfer. Losasso et al. simulated phase change from solids to fluids by the particle level set method by placing particles inside the solids [Losasso et al. 2006a]. They required special treatment using particles inside the solids because they used the signed distance function to represent the interface. We can take a much more straightforward approach due to the use of VOF functions representing interfaces.

On thermal radiation simulation, Fujishiro and Aoki [Fujishiro and Aoki 2001] calculated thermal radiation based on the principle of the form factors used in the radiosity method, one of the global illumination techniques. However, they can not take into consideration complex geometries, specular reflection, or heat collection as the radiosity method does not account for such properties. On the other hand, the photon mapping method [Jensen 1996] has become dominant over radiosity in the field of global illumination because it can solve these problems. In the filed of thermodynamics, Howell and Perlmutter developed a method to simulate thermal radiation with particles [Howell and Perlmutter 1964]. In the method reported here, we efficiently calculated thermal radiation by introducing the treatment of Howell et al. into the photon mapping technique.

3 Fluid Simulation

3.1 Navier-Stokes Equations

To simulate the behavior of fluids, it is necessary to solve the Navier-Stokes equations. The Navier-Stokes equations for an compressible fluid, including surface tension effects, are as follows:

$$\frac{\partial \rho}{\partial t} + (\mathbf{u} \cdot \nabla)\rho = -\rho \nabla \cdot \mathbf{u} \quad (1)$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\frac{1}{\rho}\nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{F}_{\mathrm{st}} + \mathbf{g} \quad (2)$$

$$\frac{\partial T}{\partial t} + (\mathbf{u} \cdot \nabla)T = a\nabla^2 T + \mathbf{S}_{\mathrm{r}} \quad (3)$$

where $\mathbf{u}$ is the fluid velocity, ν is the kinetic viscosity, ρ is the density of the fluid, T is the temperature, p is the pressure, $\mathbf{F}_{\mathrm{st}}$ is the surface tension, $\mathbf{g}$ is gravity, a is the thermal diffusivity, and $\mathbf{S}_{\mathrm{r}}$ is the radiation energy. Pressure-based schemes are generally used to solve the Navier-Stokes equations, the most common of which include the MAC [Harlow and Welch 1965] and SMAC [Amsden and Harlow 1970] methods.

We solve the Navier-Stokes equations by the C-CUP method [Yabe and Wang 1991], which is one of the pressure-based approaches. The C-CUP method solves the advection term by the CIP method and handles compressible and incompressible fluids in the same computational domain in a unified manner . We assume that the fluid is approximated to be incompressible and in Eq.(1) the density ρ remains the same value ($\partial\rho/\partial t = 0$) and we derive an equation on the pressure that forces $\nabla \cdot \mathbf{u} = 0$. We use the staggered MAC grid [Harlow and Welch 1965] for discretization and a preconditioned conjugate gradient method to solve the Poisson equation of the pressure. The temperature T is defined as a scalar value at the center of each grid and is advected and diffused by Eq.(3). The temperature is defined not only in fluids, but also in solids. In solids, heat is transferred by heat conduction and on their surfaces, heat energy is transferred to the fluids by convective thermal transfer. We use the solver developed by Carlson et al. [Carlson et al. 2002] to calculate viscous and heat diffusions.

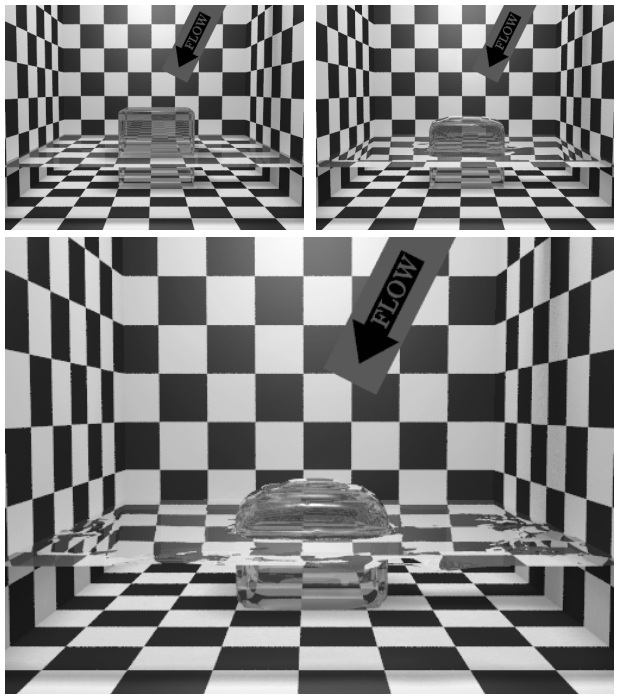


Figure 4: *Differences of the melting speeds with convective heat transfer. The duct labeled "FLOW" is blowing air and gas convection always exists in air. As water does not have much convection flow, the melting speeds of the ice cube above and below water are different (*$50 \times 50 \times 50$ *grid).*

3.2 CIP Advection Method

The advection equation should be solved to track the surface of the fluid. In the level set method, by guaranteeing that the necessary

smoothness of an implicit (level set) function is kept at the interface, the advection equation is discretized by a low-degree function and is solved. The CIP method adopts the reverse approach. That is, it uses an algorithm of cubic or greater accuracy similar to spline interpolation to discretize the advection equation and advect such a function that changes abruptly at the interface while maintaining its original shape without numerical oscillations. Suppose the function to be advected is $f(x,t)$, in the semi-Lagrangian method the function value at the current position is updated by $f(x-u\Delta t, t-\Delta t)$ at the back-traced position. The traditional methods interpolate $f(x-u\Delta t, t-\Delta t)$ with a piece-wise linear function, or use WENO similar to the method described by Enright *et al.* [Enright et al. 2003] These methods usually invoke numerical oscillations, and even if they suppress them, they can not restore the original profile and only maintain its smoothness. Hence, Yabe and collaborators proposed a method that preserves the profile between the grids called the CIP method [Takewaki and Yabe 1987]. The CIP method advects $f(x)$ and its positional derivative $g(x) = \partial f(x)/\partial x$ at the same time and in the same manner (on the assumption that $\partial u/\partial x = 0$). The profile between two grids i, $i+1$ is given by the following cubic interpolation function:

$$F_i(x) = a_i(x-x_i)^3 + b_i(x-x_i)^2 + c_i(x-x_i) + d_i. \quad (4)$$

The coefficients a_i, b_i, c_i, and d_i can be determined from the four constraints on $f_i, g_i, f_{i-1}, g_{i-1}$,

$$\begin{aligned} a_i &= \frac{g_i + g_{i-1}}{\Delta x^2} - \frac{2(f_i - f_{i-1})}{\Delta x^3} \\ b_i &= \frac{3(f_{i-1} - f_i)}{\Delta x^2} + \frac{2g_i + g_{i-1}}{\Delta x} \\ c_i &= g_i \\ d_i &= f_i \end{aligned}$$

Although the method seems very simple, its performance is excellent. Figure 5 shows an example of the flow of a 1-dimensional square wave. The advection velocity u is assumed to be constant and the profiles of the square wave after advection of a sufficient distance are drawn. In case (b) using the upwind difference, the profile is smoothed out, but in cases (c) and (d) using the CIP, the profiles can advect retaining the original shape. The Rational-CIP of (d) will be discussed in detail in section 3.3. Based on this excellent property of the CIP method, it is not necessary to use smooth functions as in the level set method, and it is possible to track the free surface of a fluid. No laborious reinitialization process is necessary. The surface tracking algorithm is described in the next subsection.

3.3 Surface Tracking

We introduce a color function ϕ called the VOF (Volume-of-Fluid) and advect it by the CIP method [Yabe et al. 2001b].

$$\phi(\mathbf{x},t) = \begin{cases} 1, & \mathbf{x} \in \text{water} \\ 0, & \text{otherwise} \end{cases} \quad (5)$$

The VOF function is governed by the following advection equation:

$$\frac{\partial \phi}{\partial t} + (\mathbf{u}\cdot\nabla)\phi = 0. \quad (6)$$

While the VOF function is advected, a small overshoot is generated. As the overshoot remains small during advection, it does not seem problematic on initial inspection. We determine physical quantities, such as the viscosity and the speed of sound, using the VOF function, or when the viscosities of water and air are ν_{water} and ν_{air}, respectively, and then the viscosity is given by

$$\nu(\mathbf{x}) = \phi(\mathbf{x})\nu_{\text{water}} + (1.0 - \phi(\mathbf{x}))\nu_{\text{air}}.$$

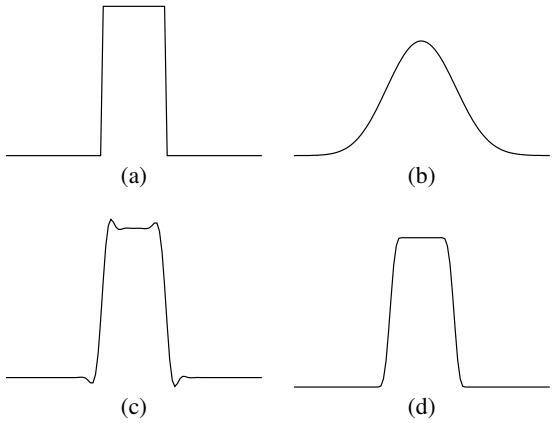


Figure 5: *Advection of 1-dimensional square wave: (a) Initial conditions; (b) Upwind difference; (c) CIP; (d) Rational-CIP profiles after advection.*

Hence, when the differences of the physical quantities of gas and fluid are very large, even a small overshoot is a significant problem. Therefore, we adopted the Rational-CIP (RCIP) method that interpolates the profile by a rational function [Xiao et al. 1996a; Xiao et al. 1996b].

Another technique was proposed that uses the tangent transformation to suppress the overshoot [Yabe et al. 2001b]. This method performs very elegantly for computational fluid mechanics applications, but for CG the surface of the fluid may not be smooth enough with some large grid resolutions. We employ the RCIP method because it generates a smoother surface around the interface because of the divergent tendency of the VOF function ϕ compared with that obtained by the tangent transformation. As the computation will diverge if this type of the numerical diffusion spreads out, we adopt the STAA method described in section 3.4.

3.4 Surface Tracking by Artificial Anti-diffusion

Advection of the VOF function by the RCIP method diffuses the interface regions the function values of which are changing from 0 to 1 and the slope of the function becomes gentle. Diffusion of the interface invokes numerical instability and if the physical quantities are calculated from the VOF function, the behavior of the interface deviates widely from reality. Xiao et al. developed the STAA method using volume transfer to suppress interface diffusion [Ikebata and Xiao 2002]. Their method generates a level set function (the signed distance function) and calculates normal vectors and application regions using the function, then determines the volume to be transferred and its direction.

We use the smoothed VOF function $\bar{\phi}$ for the calculation of surface tension instead of the signed distance function to avoid the computational cost required to generate the signed distance function. The accuracy becomes poorer than that obtained by the signed distance function, but it does not harm the quality of CG-oriented simulation results markedly.

After calculating the normal direction $\mathbf{n} = (n_x, n_y, n_z)$, we determine the displacement distance. As shown in Fig.6, the displacement distance r is given by

$$V = (\Delta x - r|n_x|)(\Delta y - r|n_y|)(\Delta z - r|n_z|). \quad (7)$$

If all volume is transferred, V is supposed to be 0, and hence r is

given by

$$r = \min(\Delta x/|n_x|, \Delta y/|n_y|, \Delta z/|n_z|). \quad (8)$$

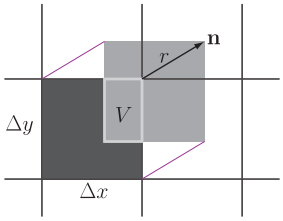
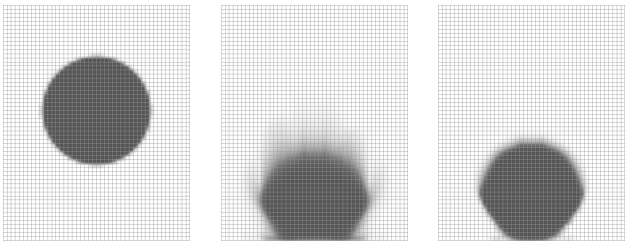


Figure 6: *Volume transfer*

This technique constructs sharp fluid interface, but for CG it is not appropriate to render it if it is too sharp. To resolve this problem, we use the smoothed VOF function mentioned above to control the displacement distance of the STAA. If the smoothed VOF function is given by $\bar{\phi}$, calculate $G = \alpha|\bar{\phi} - 0.5|$. Then, r is determined by

$$r = G^d \min(\Delta x/|n_x|, \Delta y/|n_y|, \Delta z/|n_z|). \quad (9)$$

By controlling the displacement distance by the two parameters α and d, we suppress excessive diffusion of the interfaces, while keeping their width. In our experiments, the method worked well when d was equal to the number of dimensions, i.e., $d = 2$ for 2D and $d = 3$ for 3D and $\alpha = 1.0$. As G is guaranteed to be $-0.5 \leq G \leq 0.5$ due to the nature of the RCIP advection, if $\phi = 0.5$, the original interface shape does not change because $r = 0$. If we use some power of G instead of G, the closer the volume is to the interface, the smaller the displacement distance becomes and we can advect the interface while retaining smoothness to some extent.



(a) (b) (c)

Figure 7: *2D simulation of falling droplets: (a) Initial conditions; (b) RCIP; (c) Our STAA method.*

Figure 7 shows an example of the fall of a 2-dimensional water droplet. Although the RCIP method invokes large numerical diffusion in the advection direction, our improved STAA method suppresses it and does not cause aliasing. Hence, we can obtain liquid surfaces smooth enough for high-quality rendering.

4 Thermal Radiation

Application of the Monte Carlo method to the thermal radiation calculation by Howell *et al.* [Howell and Perlmutter 1964] does not require basic equations or boundary conditions specific to individual situations, and their technique is very similar to the photon mapping technique [Jensen 1996]. Hence, in this research, we calculated thermal radiation efficiently by introducing the treatment of Howell *et al.* into the photon mapping method.

4.1 Thermal Radiation Calculation by Photon Mapping

Conventional analysis of radiation thermal transfer deals with radiation energy as a continuous quantity. When we apply the Monte Carlo simulation to radiation thermal transfer analysis, we assume that radiation energy is not a continuously variable quantity and is transferred by a set of particles (photons) having a constant energy. Here, we describe the treatment of these photons in relationship to the photon mapping method.

Radiating Thermal Energy from a Heat Source We assume that the heat source is not radiating thermal energy as a continuous quantity, but does so by photons with a constant discrete energy. As the energy radiated from the heat source per unit time and per unit area is given by σT^4 based on the Stefan-Boltzmann law, if the number of photons emitted from the heat source of area A per unit time is N, the energy of each photon is given by

$$E_p = \frac{\varepsilon \sigma T^4 A \Delta t}{N} \quad (10)$$

where Δt is the photon radiation time interval, $\sigma = 5.67 \times 10^{-8}[W/m^2 \cdot K^4]$ is the Stefan-Boltzmann constant, and ε is a radiative property of the surface called the emissivity. In this paper, we assume that a photon is emitted at every time interval and Δt is equal to the time step. We store the emissivity ε in the variables of diffuse and specular reflectivity parameters for rendering and can perform thermal radiation calculation without adding any new parameters to the existing photon mapping program.

Radiation Direction In the case that the surface of the heat source is assumed to be a scattering plane, the radiation direction of the photon (θ, ϕ) obeys the Lambertian reflectance law. In the spherical coordinate system, the radiation direction of the photon (θ, ϕ) is given by

$$\begin{aligned} \theta &= 2\phi\xi_\theta \\ \phi &= \cos^{-1}\sqrt{1-\xi_\phi} \end{aligned} \quad (11)$$

where ξ_θ and ξ_ϕ are uniform random numbers between $[0, 1]$.

Reflection and Absorption of Radiation Energy If a photon with the thermal energy arrives at an object, it is necessary to handle its reflection or absorption on the object surface. The incoming radiation energy on the surface is treated in two ways. Some energy corresponding to the surface absorptivity α is absorbed and the rest of the energy corresponding to $(1 - \alpha)$ is reflected. Similar to the original photon mapping, we determine the treatment of each photon using Eq.(12) with a uniform random number $\xi \in [0, 1]$ instead of modifying the heat energy of the photon.

$$\begin{aligned} \xi &\leq \alpha \quad \text{Absorption} \\ \xi &> \alpha \quad \text{Reflection} \end{aligned} \quad (12)$$

Due to the usage of the Russian roulette method mentioned above, all photons from the same heat source have the same thermal energy. Therefore, it is sufficient to count the number of photons inside the computation cell to sum up the incoming heat energy. Based on thermodynamics, the emissivity of the object ε should be equal to the absorptivity α, and we use the values of the diffuse and specular reflectance of the object surface for α in the same way as ε.

Increase in Temperature by Thermal Radiation At each time step, count the number of the photons retaining inside each computation cell and determine the increase in the temperature of the

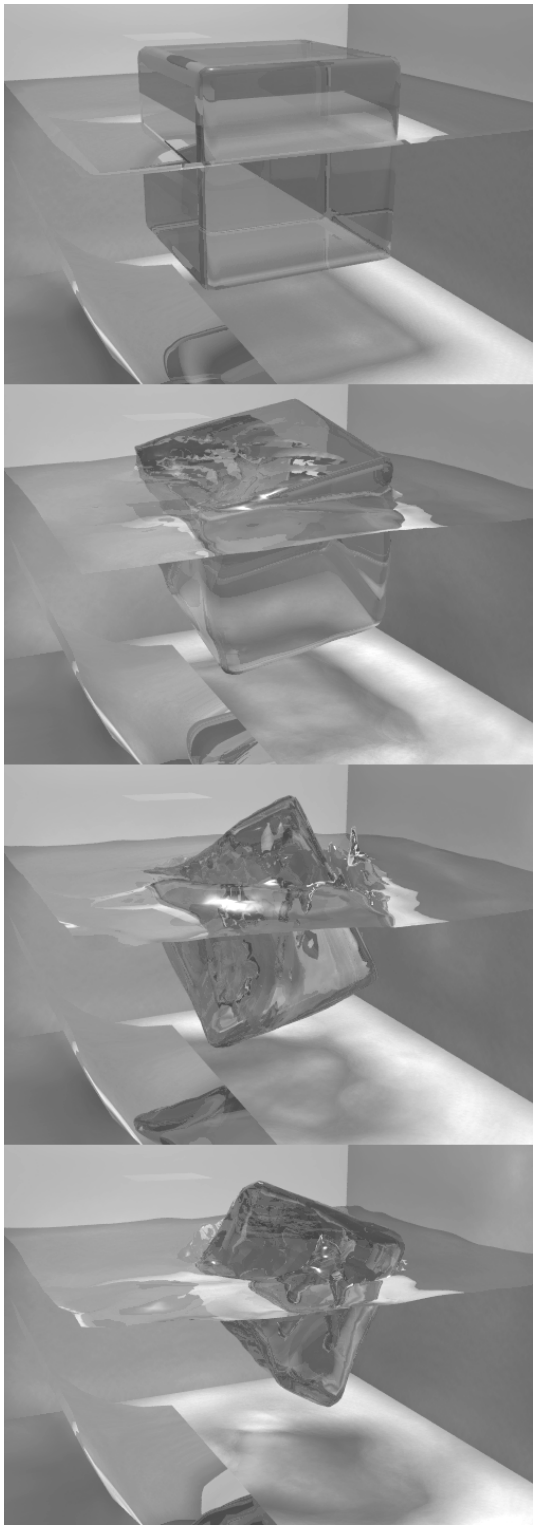


Figure 8: *Thermal radiation simulation by photon mapping. The yellow plate in the figure is a heat source, and The quarter of cylinder located on the lower left hand of the solid is a mirror. (*$64 \times 64 \times 64$ *grid).*

object using it. The increase in the temperature by one photon is calculated by

$$\Delta T_p = \frac{E_p}{c\rho h^3} \tag{13}$$

where E_p is the energy of each photon obtained by Eq.(10) and h is the grid width of the cell.

4.2 Phase Change

Matter shows different kinematic behavior depending on the macroscopic state, such as solid, liquid, and gas even if they consist of the same type of molecule at the microscopic level. These are called "phases". Generally, matter is classified into three phases. Each phase has a uniform boundary, and each is distinguished by a number of differences, such as the density. The transition from one phase to another is called a phase change and the change from solid to liquid and vice versa, such as ice melting discussed in this paper, is a typical example of a phase change (the melting point is the temperature when both melting and freezing occur). When a phase change from solid to liquid takes place, some energy is consumed to break down the molecular bonds in the solid. This energy is called the latent heat. If the latent heat is given by L, then the mass of phase change per unit time per unit volume is given by

$$M = \frac{H}{L} \tag{14}$$

where $H = \rho C_p \partial T/\partial t$ is the heat energy dependent on the temperature change. The mass of the phase change is calculated from the temperature change $\partial T/\partial t$ of each computation cell and the change in volume fraction dV is determined from the density of the melting material.

We use an additional VOF function to represent a solid and the interaction between solid and liquid is calculated by the method proposed previously [Yabe et al. 2001b]. The phase change from a solid to a liquid can be simulated by subtracting dV from the VOF function of the solid and adding it to that of the liquid.

5 Results

This section describes the animations generated using our method. The animations are created on a computer equipped with a 3.6 GHz Pentium 4 CPU and 2GB of main memory.

As an example of a melting solid in a liquid, Figure 2 shows the simulation result of an ice cube falling into hot water. The initial temperatures of the ice cube and the hot water are $263[K]$ and $363[K]$, respectively. The melting point is $273[K]$, the specific heat capacities $[kJ/(kg \cdot K)]$ and thermal conductivities[1] $[W/(m \cdot K)]$ are: ice $(8.86, 2.2)$, water $(4.17, 0.612)$, and air $(1.01, 0.0265)$. Heat is conducted from the hot water to the ice cube, and the ice cube melts from its corners by thermal diffusion. As the solid is defined by a VOF function, the phase change to liquid is accomplished smoothly . It is possible to define solid VOF functions of complex shapes from the implicit function fields generated from polygonal meshes.

Figure 3 shows the simulation result of pouring hot water onto a piece of ice. As the rate of melting becomes faster where flows are more rapid and convective thermal transfer is stronger, our simulator could reproduce the phenomenon in which the top region of the solid melts faster and a hole is carved in the ice. As we simulated gas-liquid two-phase flow, this raised the question of how

[1] The thermal diffusivity is given by the thermal conductivity subdivided by the specific heat capacity and density.

melting would occur depending on air convection. Figure 4 shows effects of gas convection for melting. The duct in the figure blows air in the left-downward direction. To compare the difference in melting between with and without convection, water is poured to submerge the bottom half of an ice cube and the same values for the thermal diffusivities of water and air are set up to avoid differences in the effects of thermal diffusion. As the density of water is 1000-fold larger than that of air and weak convection of air does not substantially affect flow in water, the top half of the ice cube melts faster than the bottom half. Although natural convection caused by temperature variations would melt ice, forced convection caused by external means results in more rapid melting.

Figure 8 shows the simulation results of thermal radiation by photon mapping. Heat is transferred to a solid from the heat source colored yellow by thermal radiation and the solid is melted into a shape depending on the radiation intensity of the heat source. One fourth of a cylindrical mirror is located on the lower left hand of the solid. The regions around the edge of the ice cube close to the mirror melt faster because of radiated heat reflected by the mirror. In this example, to clarify the effects of thermal radiation by photon mapping, we assume that the heat source does not conduct heat to the air directly and the water only refracts photons. (In reality, the refraction induces the medium to absorb some amount of thermal energy. The heat radiation simulated here is assumed to be similar to that caused by light from the sun.) By restricting the range of the direction of photon emissions in our method, we can produce animations of cutting a solid by a laser beam. Figure 9 shows an example of such simulation. In Figure 9, as mass conservation is not guaranteed, the volume of the resultant liquid after melting is rapidly decreased. In the melting phenomenon in air, a thin layer of the melted liquid covers the solid surface. In our simulation, it diffuses and the loss of mass occurs. To reproduce a variety of melting phenomena, this problem needs to be addressed and resolved in future work.

In these animations, the image of one frame is used after several time steps and the processing time was about $1 \sim 1.5$ minutes/step, and the memory usage was about 230MB in $64 \times 64 \times 64$ grid. The CIP method needs a lot of memory usage compared with the Level Set method because of the positional derivative. Most of the processing time is involved in pressure calculation and the CIP interpolation. In particular, the density difference on the gas-liquid interface have greatly decreased the convergence speed in the preconditioned conjugate gradient method when we solve the Poisson equation of the pressure in the solver of the Navier-Stokes equation. Although it is not necessary to use a fine grid for simulation because of the sub-cell resolution of our method, we may be able to adopt adaptive grid structures [Losasso et al. 2004] to reduce the computation time.

6 Conclusions and Future Work

We proposed a fast and efficient method for producing physically based animations of the ice melting phenomenon, and the method included thermal diffusion, convective thermal transfer, and thermal radiation. Our method adopts a simple step function called the VOF (Volume-of-Fluid) with advection to track the free surface, which enables straightforward simulation of the phase changes. This VOF function was advected by the improved RCIP-STAA method to suppress numerical diffusion and to render anti-aliased surfaces for CG. Furthermore, we proposed a technique to calculate thermal radiation efficiently by photon mapping and showed melting phenomena by laser beams as an example of its application.

In fluid simulation based on the VOF functions and the CIP method used in this research, the function value of each object represented its volume fraction in each computation cell and we could implement a phase change simulator in a straightforward manner by exchanging the volume fraction between two phases by exchanging values between their VOF functions. Because of this simplicity, we believe the VOF + CIP method is more suitable for phase change simulations than the level set method using the signed distance function. We expect that our method will be easily extended to deal with the ice freezing phenomenon, and boiling and condensation, which are phase transitions between solid and gas.

In future work, to achieve the conservative properties more accurately, we may use CIP-CSL2 [Yabe et al. 2001a], CIP-CSL4 [Tanaka et al. 2000], or VOF/interface markers [Aulisa et al. 2004].

Although the liquid representation by the VOF function can simulate phase changes easily, it is not possible to simulate a flow thinner than the grid width. To simulate a thin shell liquid on a solid formed by melting the solid, we can use an octree representation to handle such thin flows. However, the grid size of the smallest octree cells still restricts the thickness of the flows. To obtain a fundamental solution, it will be necessary to combine 3D simulation with 2D simulation techniques, e.g., such as that proposed by Stam [Stam 2003]. Other future research topics include simulations of ice freezing, boiling and condensation phenomena, as mentioned above.

References

AMSDEN, A., AND HARLOW, F. 1970. A simplified mac technique for incompressible fluid flow calculations. *Journal of Computational Physics 6*, 322–325.

AULISA, E., MANSERVISI, S., AND SCARDOVELLI, R. 2004. A surface marker algorithm coupled to an area-preserving marker redistribution method for three-dimensional interface tracking. *Journal of Computational Physics 197*, 2, 555–584.

CARLSON, M., MUCHA, P. J., VAN HORN III, R. B., AND TURK, G. 2002. Melting and flowing. In *ACM SIGGRAPH/Eurographics Symposium on Computer Animation*, 167–174.

CARLSON, M., MUCHA, P. J., AND TURK, G. 2004. Rigid fluid: animating the interplay between rigid bodies and fluid. In *Proceedings of SIGGRAPH 2004*, ACM Press, New York, NY, USA, 377–384.

ENRIGHT, D., MARSCHNER, S., AND FEDKIW, R. 2002. Animation and rendering of complex water surfaces. In *Proceedings of SIGGRAPH 2002*, ACM Press, New York, NY, USA, 736–744.

ENRIGHT, D., NGUYEN, D., GIBOU, F., AND FEDKIW, R. 2003. Using the particle level set method and a second order accurate pressure boundary condition for free surface flows. In *4th ASME-JSME Joint Fluids Eng. Conf.*, no. FEDSM2003-45144, ASME.

FOSTER, N., AND FEDKIW, R. 2001. Practical animation of liquids. In *Proceedings of SIGGRAPH 2001*, ACM Press, New York, NY, USA, 23–30.

FUJISHIRO, I., AND AOKI, E. 2001. Volume graphics modeling of ice thawing. In *Volume Graphics 2001*, Springer-Verlag, 69–80.

GUENDELMAN, E., SELLE, A., LOSASSO, F., AND FEDKIW, R. 2005. Coupling water and smoke to thin deformable and rigid shells. In *Proceedings of SIGGRAPH 2005*, ACM Press, New York, NY, USA, 973–981.

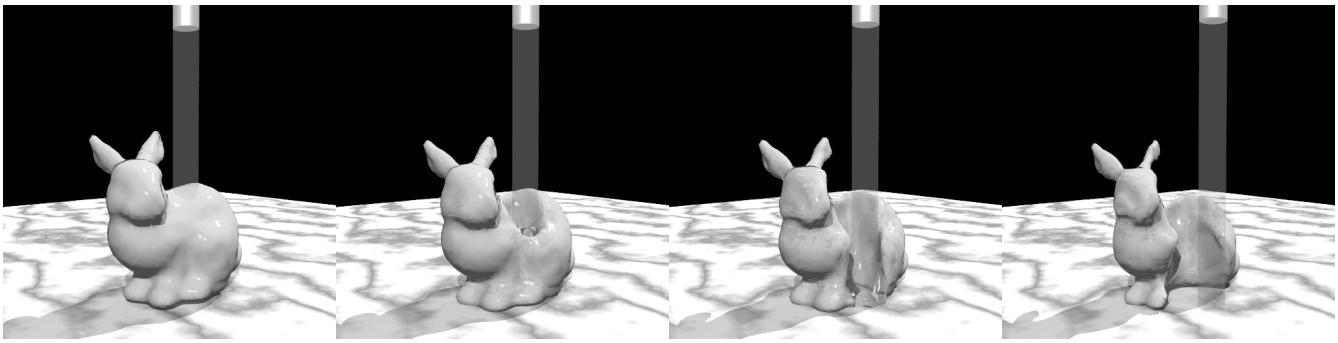


Figure 9: *Laser melting. Photons are emitted in only one direction from a heat source. A laser beam is used to radiate from a cylindrical source located at the top of the figure, moving slowly from above the bunny to its front right (*$64 \times 64 \times 64$ *grid).*

HARLOW, F., AND WELCH, J. 1965. Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface. *Physics Fluids 8*, 2182–2189.

HONG, J.-M., AND KIM, C.-H. 2005. Discontinuous fluids. In *Proceedings of SIGGRAPH 2005*, ACM Press, New York, NY, USA, 915–920.

HOWELL, J. R., AND PERLMUTTER, M. 1964. Monte carlo solution of thermal transfer through radiant media between gray walls. *Journal of Heat Transfer 86*, 116–122.

IKEBATA, A., AND XIAO, F. 2002. Development of conservative frontcapturing scheme and applications to multi-fluid simulations. In *Proceeding of Annual Conference (2002) of Japan Society of Mechanical Engineering*, 25–27. (in Japanese).

JENSEN, H. W. 1996. Global illumination using photon maps. In *Rendering Techniques '96 (Proceedings of the Seventh Eurographics Workshop on Rendering)*, 21–30.

JONES, M. W. 2003. Melting objects. *The Journal of WSCG 11*, 2, 247–254.

KIM, B., LIU, Y., LLAMAS, I., JIAO, X., AND ROSSIGNAC, J. 2007. Simulation of bubbles in foam with the volume control method. In *Proceedings of SIGGRAPH 2007*, ACM Press, New York, NY, USA, 98.

LOSASSO, F., GIBOU, F., AND FEDKIW, R. 2004. Simulating water and smoke with an octree data structure. In *Proceedings of SIGGRAPH 2004*, ACM Press, New York, NY, USA, 457–462.

LOSASSO, F., IRVING, G., GUENDELMAN, E., AND FEDKIW, R. 2006. Melting and burning solids into liquids and gases. *IEEE Trans. on Vis. and Comput. Graph 12*, 3, 343–352.

LOSASSO, F., SHINAR, T., SELLE, A., AND FEDKIW, R. 2006. Multiple interacting liquids. In *Proceedings of SIGGRAPH 2006*, ACM Press, New York, NY, USA, 812–819.

OSHER, S., AND FEDKIW, R. 2003. *Level Set Methods and Dynamic Implicit Surfaces*. Springer-Verlag.

SONG, O.-Y., SHIN, H., AND KO, H.-S. 2005. Stable but nondissipative water. *ACM Transactions on Graphics 24*, 1, 81–97.

STAM, J. 1999. Stable fluids. In *Proceedings of SIGGRAPH 1999*, ACM Press/Addison-Wesley Publishing Co., New York, NY, USA, 121–128.

STAM, J. 2003. Flows on surfaces of arbitrary topology. In *Proceedings of SIGGRAPH 2003*, ACM Press, New York, NY, USA, 724–731.

TAKAHASHI, T., FUJII, H., KUNIMATSU, A., HIWADA, K., SAITO, T., TANAKA, K., AND UEKI, H. 2003. Realistic animation of fluid with splash and foam. *Computer Graphics Forum (In Eurographics 2003 Proceedings) 22*, 3, 391–400.

TAKEWAKI, H., AND YABE, T. 1987. The cubic-interpolated pseudo particle (cip) method: application to nonlinear and multidimensional hyperbolic equations. *Journal of Computational Physics 70*, 355–372.

TANAKA, R., NAKAMURA, T., AND YABE, T. 2000. Constructing exactly conservative scheme in a non-conservative form. *Computer Physics Communications 126*, 232–243.

WANG, H., MUCHA, P. J., AND TURK, G. 2005. Water drops on surfaces. In *Proceedings of SIGGRAPH 2005*, ACM Press, New York, NY, USA, 921–929.

XIAO, F., YABE, T., AND ITO, T. 1996. Constructing oscillation preventing scheme for advection equation by rational function. *Computer Physics Communications 93*, 1–12.

XIAO, F., YABE, T., NIZAM, G., AND ITO, T. 1996. Constructing a multi-dimensional oscillation preventing scheme for the advection equation by a rational function. *Computer Physics Communications 94*, 103–118.

XIAO, F., IKEBATA, A., AND HASEGAWA, T. 2005. Numerical simulations of free-interface fluids by a multi-integrated moment method. *Computers and Structures 83*, 409–423.

YABE, T., AND WANG, P. Y. 1991. Unified numerical procedure for compressibleand incompressible fluid. *Journal of the Physical Society of Japan 60*, 2105–2108.

YABE, T., TANAKA, R., NAKAMURA, T., AND XIAO, F. 2001. An exactly conservative semi-lagrangian scheme (cip-csl) in one dimension. *Monthly Weather Review 129*, 332–344.

YABE, T., XIAO, F., AND UTSUMI, T. 2001. The constrained interpolation profile method for multiphase analysis. *Journal of Computational Physics 169*, 556–593.

ZHENG, W., YONG, J.-H., AND PAUL, J.-C. 2006. Simulation of bubbles. In *ACM SIGGRAPH/Eurographics Symposium on Computer Animation*, 325–333.

Animation of Ice Melting Phenomenon Based on Thermodynamics with Thermal Radiation

Makoto Fujisawa and Kenjiro T. Miura

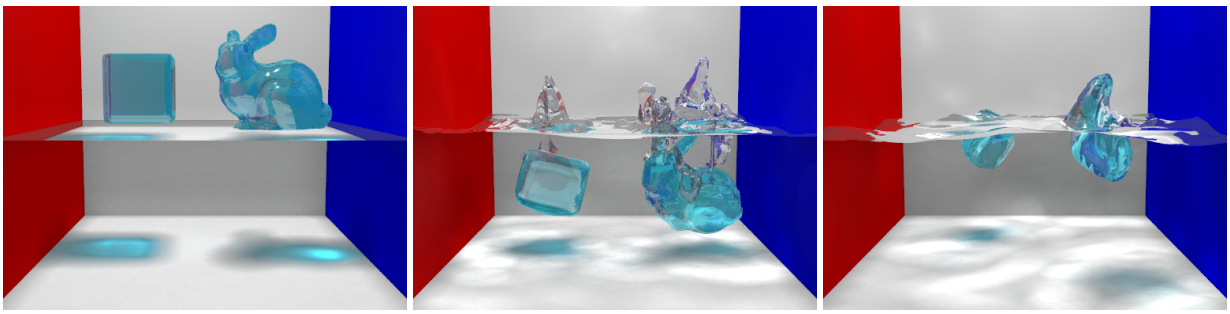


Figure 1: An ice cube and an ice bunny falling into 90 ℃ hot water

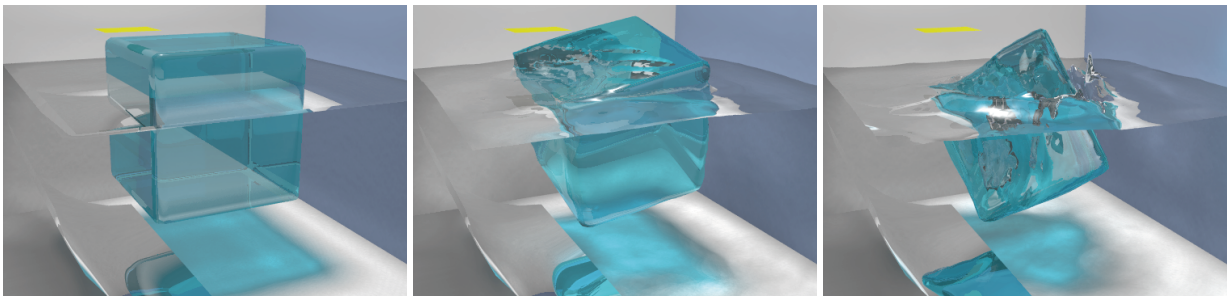


Figure 2: Thermal radiation simulation by photon mapping.

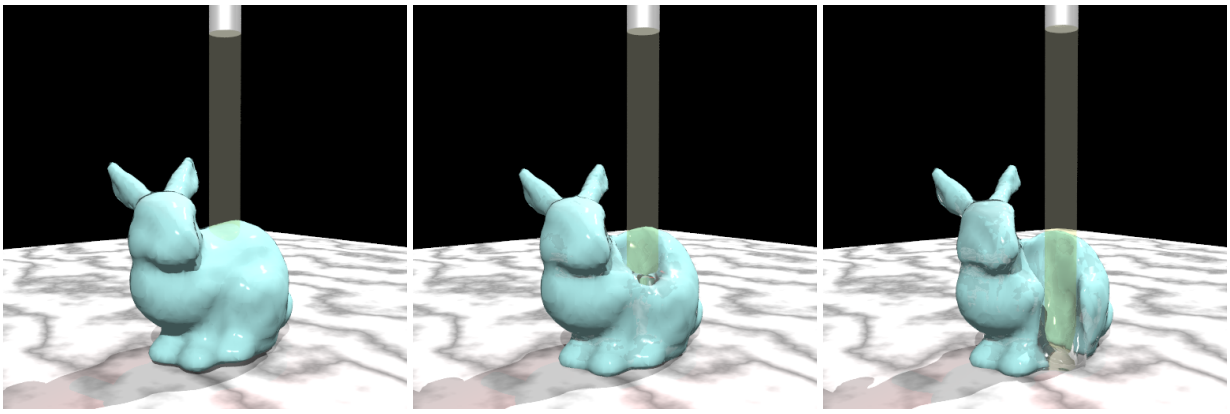


Figure 3: Laser melting.