Two-Fluid Heat Transfer from Single or Multiple Droplets Using a One Field VOF Model

V. Mehdi-Nejad, J. Mostaghimi and S. Chandra

Centre for Advanced Coating Technologies
Department of Mechanical & Industrial Engineering
University of Toronto

This paper presents a new method to simulate heat transfer across the interface separating immiscible liquids. One-field volume tracking is used which involves solving one set of equations for conservation of mass, momentum and energy. The original idea behind volume tracking methods has been used not only to advect mass and momentum but also energy across cell boundaries. Moreover, the Van Leer method is used to approximate advection temperatures across the sharp temperature gradients existing at fluid-fluid interface. As examples to this method, the fall of a single and multiple hot liquid droplets in a cool surrounding are simulated. The effect of changing the surrounding thermal diffusivity is studied. Results show a region of high temperature in the wake region behind the droplet and a temperature distribution along the droplet surface. Also in the case of two successive droplets simulation, the heat dissipation from the downstream droplet cause a dramatic change in temperature distribution of the upstream droplet

1. Introduction

Heat transfer across an interface separating two immiscible fluids is the basis of many common phenomena surrounding us such as cooling of a cup of coffee on a table or the heat transfer between the ocean and the atmosphere. Other examples the industry include heat exchangers, spray cooling, spray atomizers, and combustors.

To calculate the exact amount of heat transfer across the interface separating two fluids, one should take into the account the deformability and motion of the interface. However, the complexities of the geometry of the interface can be neglected in many cases. For example during heat transfer from a stationary fluid in an open container, the interface can be approximated as a solid plane or when fuel droplets are spreading in the combustion chambers, they are usually approximated as ideal spheres. In some other instances interface deformations are so large that they cannot be neglected, for example when fuel droplets impact on a solid surface and splash. In these situations, the geometry of the interface plays an important role in determining the amount of the heat transfer.

The fluid mechanics of the last group falls under the category of interfacial, incompressible flow problems, in which immiscible fluids are separated by interfaces having arbitrary locations and very complex geometries. Rider and Kothe [1] have reviewed techniques used to describe the motion of such interfaces. Two approaches have been

proposed: either "capturing" or "tracking" the interface. In the first approach, the interface is treated as a region having a large density gradient. This method is easy to implement, but for complex topologies the exact location of the interface becomes difficult to pinpoint. It may diffuse over several computational grids, reducing the accuracy of the results [2].

In a "tracking" method, the interface is considered to be a discontinuity moving through the computational domain with the fluids. There are different ways of tracking the interface: it can be approximated by a piecewise polynomial function (front tracking method [3]); the interface can also be considered to be the zero level set of a smooth function (level set methods [4,5]); or the volume of fluid in each cell through which the interface passes is calculated at each time step (volume of fluid (VOF) method). In most early efforts to simulate interfacial flow the presence of the lighter fluid was ignored to simplify the problem. Only in recent years have some codes been developed that include both fluids [6, 7, 8].

There have been few studies regarding heat transfer across the interface between two fluids which includes problems such as: droplet impact, spread and solidification [9, 10, 11]; Spread and solidification of the melt in a cast; Heat transfer through the vapor-liquid interface during boiling[3,12]; and natural convection in two layer immiscible liquid systems [13,14].

We have developed a technique based on the volume tracking method of Youngs to calculate the heat transfer in a two fluid flow with immiscible interface. We are convinced that using VOF methods in combination with a mixed property approach [15] will offer an alternative solution to these types of problems. In this method we have separated the diffusion and convection terms of the energy equation and treat them separately. Our advection temperature is based on the Van Leer method [16]. We have studied the fall of hot tin droplets in an oil tank. Heat transfer in both liquids is considered. The fluid-fluid interface has an initial temperature jump and is deformable.

2. Numerical Method

In the present study we used a one-field volume of fluid tracking method based on the model developed by Youngs [17], modified to include multiple fluid phases [7]. It is assumed that only one flow field governs the motion of all the present fluids, which is a reasonable assumption when dealing with incompressible, immiscible fluids with homogenous properties. Our numerical algorithm is based on RIPPLE [18]; however the original volume tracking method (Hirt-Nichols algorithm [19) was replaced by Youngs' two-dimensional method [17]. Further modification of the code can be found in [15,20]. The numerical formulation is summarized below:

If there exist *k* fluids in the computational domain, the velocity field for all of them can be expressed as:

$$\vec{V} = \vec{V}_k \tag{1}$$

An Eulerian scheme was used to model the problem, so the continuity and momentum equations are:

$$\nabla \cdot \vec{V} = 0 \tag{2}$$

$$\frac{\partial(\rho\vec{V})}{\partial t} + \vec{\nabla}\cdot(\rho\vec{V}\vec{V}) = -\vec{\nabla}P + \vec{\nabla}\cdot(\bar{\bar{\tau}}) + \vec{F}_b \tag{3}$$

where $\vec{V}$ is the velocity vector field, P is the pressure, and $\vec{F}_b$ contains represents all the body forces acting on the fluid. The energy equation can also be written as:

$$\frac{\partial(\rho h)}{\partial t} + \vec{\nabla}\cdot\left(\rho\vec{V}h\right) = \vec{\nabla}\cdot(\kappa\vec{\nabla}T) \tag{4}$$

where h is the specific enthalpy, T is the temperature and κ is the thermal conductivity. Due to the incompressibility of the problem, the pressure work is neglected. Moreover due to the nature of the problem, viscous dissipation is ignored.

The continuum surface force (CFS) method [21] is used to model surface as a body force that acts only on interfacial cells. In the one-field method, the concept of mixed properties is used to interpret the cells containing multiple fluids. If M_k is the mass of material k in a cell and V is the cell volume, then:

$$\rho_k = \frac{M_k}{V} = \frac{V_k}{V}\frac{M_k}{V_k} = g_k \rho_k^0 \tag{5}$$

where ρ_k^0 is the bulk density of fluid k and g_k is the volume fraction. Values of g_k lie between 0 and 1 and are defined as follows for the fluid k:

$$g_k = \frac{V_k}{V} \quad \& \quad \sum_k g_k = 1 \tag{6}$$

For problems involving just two fluids, only one volume fraction (f) is necessary to define the location of the interface between them. The mixed properties used in equations (3) & (4) can then be defined as:

$$\rho = f\rho_1^0 + (1-f)\rho_2^0 \tag{7}$$

$$\mu = f\mu_1^0 + (1-f)\mu_2^0 \tag{8}$$

$$\kappa^{-1} = f/\kappa_1^0 + (1-f)/\kappa_2^0 \tag{9}$$

$$(\rho h) = f\rho_1^0 h_1^0 + (1-f)\rho_2^0 h_2^0 \tag{10}$$

$$\left[\rho C_p\right] = f\rho_1^0 \left(C_p\right)_1^0 + (1-f)\rho_2^0 \left(C_p\right)_2^0 \tag{11}$$

The superscript "0" refers to pure substances.

In equation (4), to find the relationship between specific enthalpy *(h)* and temperature *(T)*, the following equation is used:

$$h_i^0 = h_{o_i}^0 + \int_{T_o}^{T} \left(C_p\right)_i^0 dT \tag{12}$$

where $\left(h_{o_i}^0\right)$ is a reference enthalpy for the substance *"i"* at the reference temperature T_o.

Since the fluids were assumed incompressible, an advection equation was derived for the volume fractions [17] by using the concept of mixed properties:

$$\frac{\partial f}{\partial t} + \vec{V} \cdot \vec{\nabla} f = 0 \tag{13}$$

The above equation was used to track the location of the interface. It was solved using the Youngs' method [17] which is a geometrical technique and is based on computing the volume of fluid transferred from one cell to another the next in one each time step. Further details on the continuity and momentum equations can be found in [15, 22]. Here we will focus on the energy equation (4).

The finite element approach [23] is used to discretize equation (4) where volume averaged quantities are used to approximate cell centered quantities. To determine the exact value of energy transfer across the cell face, one should first know the exact amount of the mass and momentum transfer across the same face. In the present code a volume tracking algorithm, the "Youngs method" [17] (Figure 1), is used to determine this value.

The next step is to associate energy with these masses. To do this, one should know their temperature. Yet, finding the advection temperature can be challenging in the presence of a fluid-fluid interface. Fluids on either side of this interface can have different temperature and thermal properties. In this case, finding the advection temperature is not trivial and

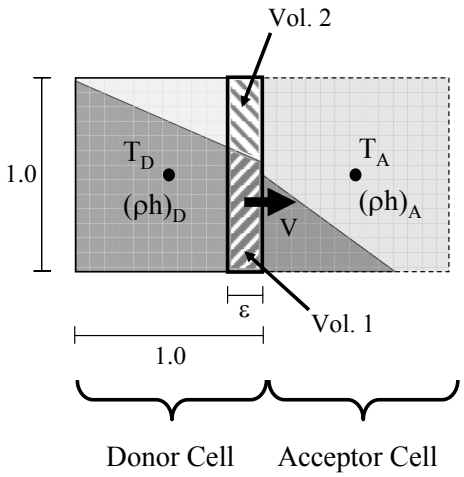

Figure 1: Momentum and energy advection method based on Youngs

depends on the diffusivities of fluids, the interface velocity, and temperature gradient across the interface. This problem falls into the category of advecting momentum or energy through a discontinuity in the space [24, 25]. In our case, the only variable that crosses the fluid/fluid interface is the thermal energy and the only discontinuity that can occur is the temperature jump across the interface. Here a piecewise linear reconstruction based on the Van Leer method is used to determine the advection temperature.

3. Validation and results

As an example, we have studied the heat transfer from a single or multiple droplets to their surroundings. This problem has many application in understanding heat and mass transfer from fuel particles and direct contact heat exchangers. Literature reviews regarding heat transfer from droplets can be found in [26] and [27]; experimental and analytical data can be found in [28, 29]. In these studies it is usually assumed that the interface deformation is negligible and that droplets remain spherical, which is a questionable assumption for large droplets.

1.1 The fall of a single liquid tin droplet in oil

A tin droplet with an initial diameter of 2.0 mm, a downward velocity of 0.5 m/s and a uniform temperature of 800°C is released in a tank filled with oil at rest having a uniform temperature of 250°C. Properties used for oil and tin are presented in table 1.

Table 1: *Fluid properties used during the numerical simulation*

	$\rho\ [kg/m^3]$	$\mu\ [N.s/m^2]$	$\kappa[W/m.K]$	$\kappa[W/m.K]$	$\sigma[N/m]$
Tin	6970.0	1.92×10^{-3}	33.6	244	0.5
Oil	1066.7	2.28×10^{-3}	0.26 - 22.6	2700.0	-

The domain is an 8.0×8.0 mm cylinder. Due to symmetry, only half of the domain was solved on a 120×240 grid (Δr=Δz=33.33 μm).

Figure 2 shows temperature contours for this setting. There are three sets of simulations which differ only by the value of the thermal conductivity in oil. This value has been increased from 2.6 W/m.K in case (a) to 12.6 W/m.K and 22.6 W/m.K in cases (b) and (c) respectively.

At t=1.8 ms, the formation of a high temperature region around the droplet is observed. This region is asymmetric and expands in the wake region behind the droplet. The thickness of this region depends on the oil diffusivity and grows as diffusivity increases in oil from case (a) to case (c). At t=5.4 ms, for all the cases the high temperature region behind the droplet takes the form of a wake. However, a distinct difference in temperature contours is observed between the three compared cases. For case (a) where the oil diffusivity is low, a noticeable increase in heat transfer is observed at the droplet surface at around 120^o after the attack angle. This increase of heat transfer leads to an increase in temperature behind the droplet in the same angle. However, at higher oil diffusivity this region is considerably larger and the high temperature region gets the typical egg form in case (c). At t=9.0 ms, the same general pattern described in t= 5.4 ms is still observed. However, the size of this high temperature region has considerably expanded in cases (b) and (c), and it is observed that the core temperature in case (a) is around 75^oC warmer than the other cases.

This temperature distribution can be explained if one can follow the mechanism of heat transfer during the droplet fall. As the droplet moves inside the oil tank, the temperature of oil particles that are in contact with the front of the droplet increases. These particles continue their path toward the back of droplet during which they continuously gain energy. As these droplets reach the wake region, due to circulation in this area, they either leave the droplet surface and follow a straight path or enter the circulation region. This is the reason why in this region, the heat transfer is maximum [28].

Most of the particles in the wake region do not leave this area. So as time goes on, their temperature increases by the diffusion from the droplet. So we should expect that the temperature near the centre of the circulation to be the maximum temperature. This is confirmed incase (a) at t=9.0 ms. However, another diffusion-convection mechanism exists between this wake region and the neighboring oil particles and depending on the amount of diffusivity in oil, the temperature distribution in this region can dramatically change (cases (a) to (c) at t= 9.0 ms).

1.2 The fall of two successive liquid tin droplets in oil

In many industrial applications such as spray cooling, spray atomizers, and combustors, the interaction and heat transfer between droplets is as important as the heat transfer between the droplet and the surrounding. To study this interaction we have simulated the fall of two successive droplets in an oil tank. Both droplets have the same initial conditions (initial diameter of *2.0 mm*, initial temperature of 800^oC and initial downward velocity of *0.5 m/s*); the spacing between them is *1.0 mm*.

Figure 3 shows this simulation. At t= 3.0 ms after the release, the similar temperature contours are observed around both droplets. At t= 6.0 ms, the high temperature region of the lower droplet reaches the top droplet and as a consequence, the high temperature region around the top droplet enlarges. This effect becomes stronger as both droplets continue their path towards the bottom of the tank so that at t=14.0 ms the top droplet is considerably warmer than the lower one.

4. Conclusion and discussion

We have successfully developed a two-fluid heat transfer model. We showed that by using only one set of governing equation, we can capture the characteristics of the heat transfer trough a liquid interface, even in the presence of a large temperature gradient. As examples to our method, we successfully simulated the heat transfer from single and multiple moving hot tin droplets in an oil tank at rest and showed the effect of thermal conductivity and vorticities behind droplets on the heat dissipation.

5. References

[1]. W.J. Rider and D. B. Kothe 1998, J. Comp. Physics, Vol. 141, pp. 112-152
[2]. H. Haj-Hairi and Q. Shi 1994, Phys. Fluids 6 (8), pp. 2555-2557
[3]. G. Tryggvason, *et al* 1998, In: 29th CFD Lecture Series 1998-03. *Von Karman Institute for Fluid Dynamics*
[4]. M. Sussman, P. Smereka, S. Osher 1994, J. Compt. Physics, Vol 114, No.1, pp. 146-159
[5]. X. L. Li 1993, Phys. Fluids A 5 (8), pp. 1904-1913
[6]. M. Rudman 1998, Int. J. for Numerical. Methods in Fluids, Vol. 28, pp. 357-378
[7]. D. B. Kothe 1998, In *Free Surface Flows* (Springer-Verlag, New York, NY, ISBN 3-211-83140) H.C. Kuhlmann, H-J Rath, Editors, pages 267-331
[8]. M. Sussman and E. G. Puckett 2000, J. of Computational Physics, Vol. 162, pp. 301-337
[9]. M. Pasandideh-Fard, M. Bussmann, S. Chandra and J. Mostaghimi 2001, Atomization and Sprays, Vol. 11, pp. 397-414
[10]. J. Mostaghimi, M. Pasandideh-Fard, and S. Chandra 2002, Plasma Chemistry and Plasma Processing, Vol.22, No.1, pp. 85-104
[11]. D. Poulikakos, J. Fukai, Z. Zhao 1996 , Int. J. Heat & Mass Transfer, Vol: 39, Issue: 13 pp. 2771-2789
[12]. S. W. J. Welch and J. Wilson 2000, J. Comp. Phys. 160, pp 662-682
[13]. J. N. Koster 1996, Int. J. Heat & Mass Transfer, Vol.39, No.3, pp. 467-478
[14]. A. Prakash and J. N. Koster 1997, Int. J. Heat & Mass Transfer, Vol.40, No.12, pp.2799-2812
[15]. V. Mehdi-Nejad, S. Chandra and J. Mostaghimi 2003, Physics of Fluids, Vol.15, No.1, pp.173-183
[16]. B. Van Leer 1979, J. of. Comp. Phys. Vol 32, pp. 101-136
[17]. D. L. Youngs 1982, in Numerical Methods for Fluid Dynamics, edited by k. W. Morton and R. C. Mjolsness , and M. J. Baines, Academic Press, pp. 273-285
[18]. D. B. Kothe, R. C. Mjolsness and M. D. Torrey 1991, " RIPPLE: A computer program for incompressible flows with free surfaces," Technical Report LA-12007-MS, LANL
[19]. C. W. Hirt and B. D. Nichols 1981, J. Comp. Phys. **3**9, 201
[20]. M. Bussmann, J. Mostaghimi and S. Chandra, "On a three-dimensional volume tracking model of droplet impact", Physics of Fluids, Vol. 11, pp. 1406-1417 (1999)
[21]. J. U. Brackbill, *et al* 1992, J. Comp Physics, pp. 100- 335
[22]. M. Bussmann, S. Chandra and J. Mostaghimi 2000, Physics of Fluids, Vol. 12, pp. 3121-3132
[23]. S. V. Patankar 1980, *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing Corp., New York
[24]. M. Y. Hussaini and B. Van Leer 1997, *Upwind and high-resolution schemes*, New York, Springer
[25]. B. Van Leer 1999, J. Comp. Phys. Vol 153, pp.1-5
[26]. M. Kaviany 1994*, Principles of Convective Heat Transfer*, Chapter 6, Springer-Verlag, New York, 1994
[27]. W. A. Sirignao 1999, *Fluid Dynamics and Transport of Droplets and Sprays*, Cambridge University Press, Cambridge
[28]. R. Clift, J. R. Grace and M. E. Weber 1978, *Bubbles, Drop, and Particles*, Academic Press, New York, pp.97-138, 1978
[29]. Z. Feng and E. Michaelides 2000, Int. J. Heat & Mass Transfer, Vol. 43, pp. 219-229

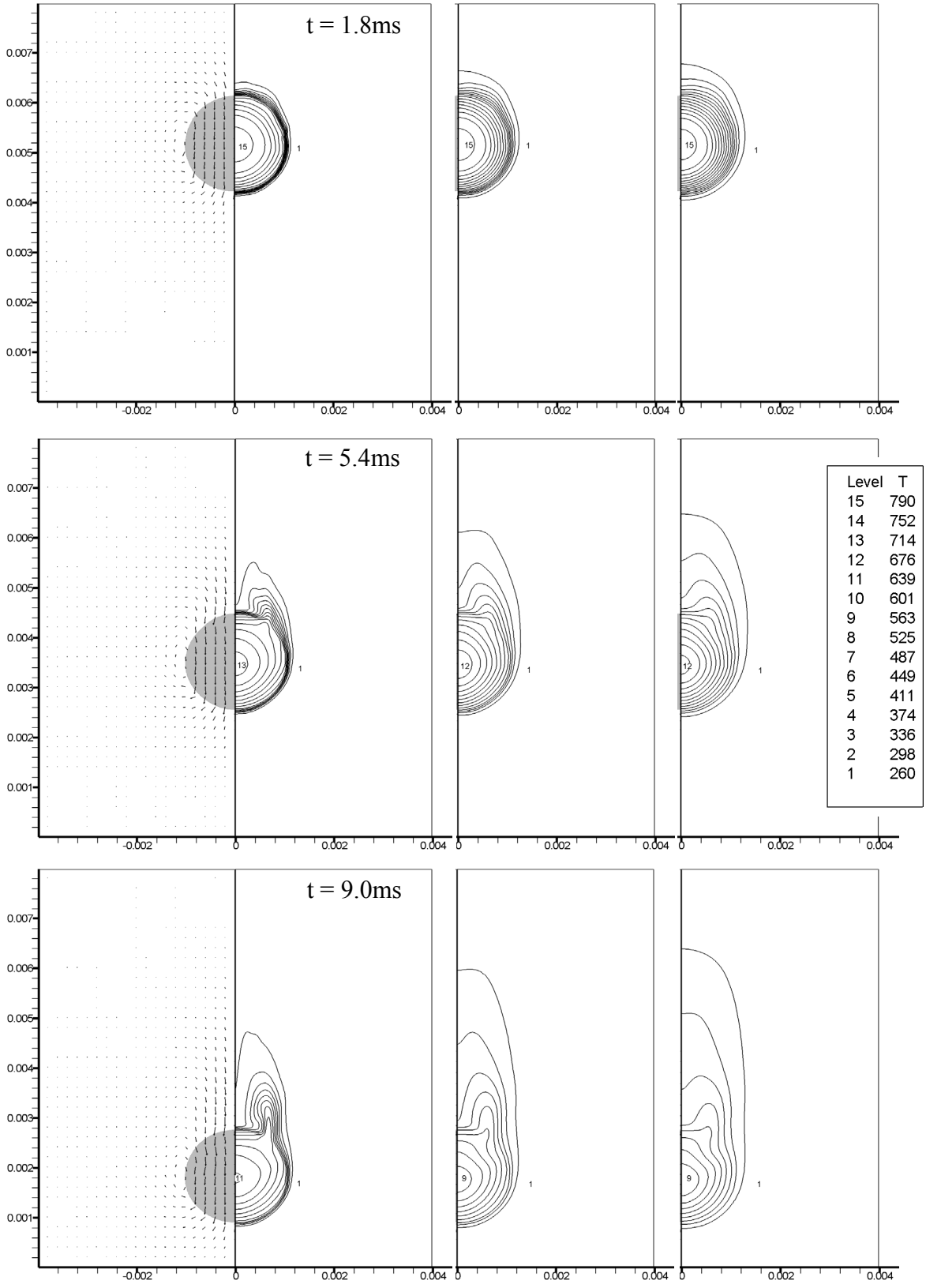

Figure 2 - Comparison between sequences of the fall of a tin droplet ($T_{initial}$=800^{o}C, α = 1.976×10^{-5}) in an oil tank ($T_{initial}$=250^{o}C) having different diffusivities (a) K_{oil}= 2.6W/mK, (b) K_{oil}= 12.6W/mK (c) K_{oil}= 22.6W/mK

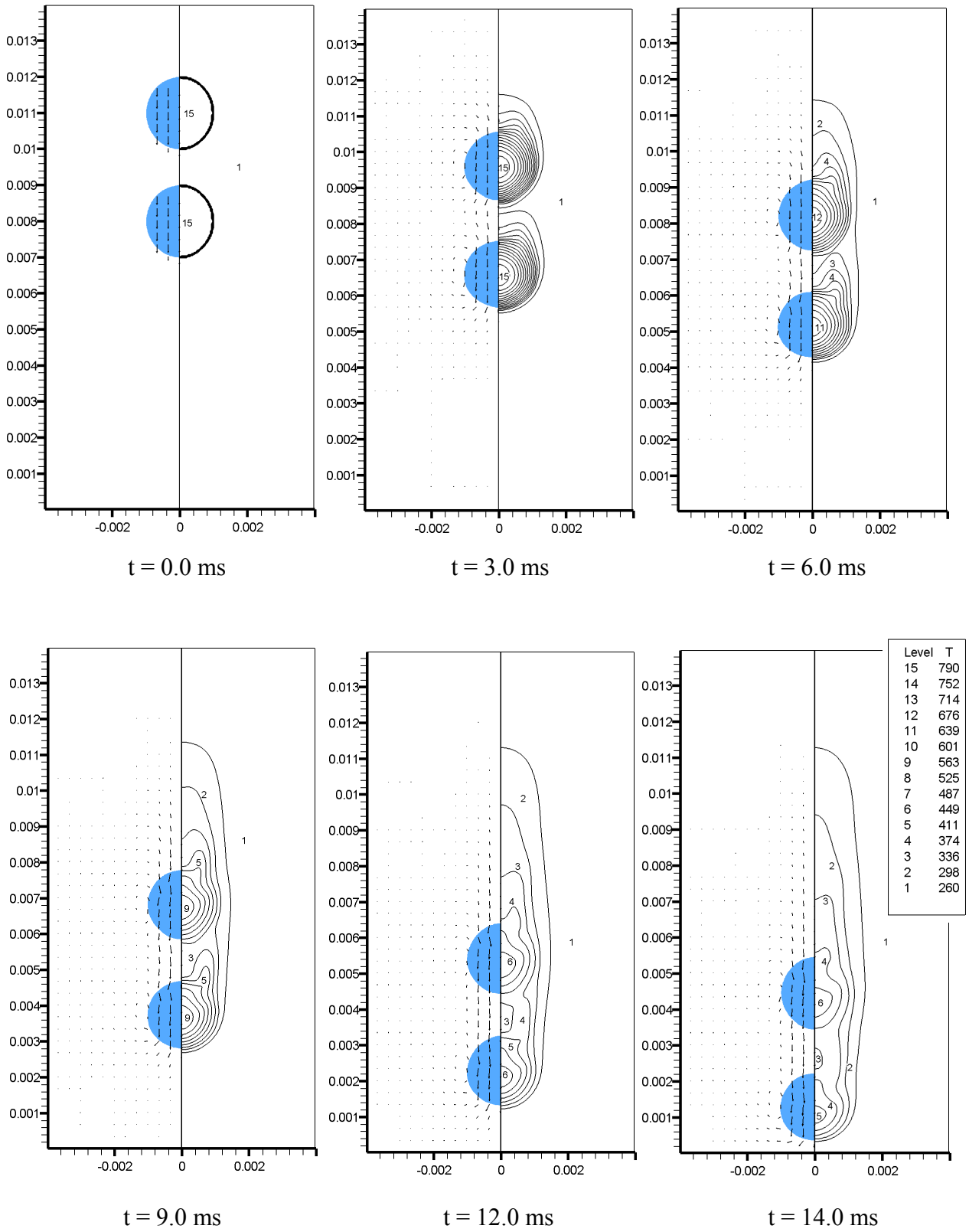

Figure 3 - Sequence of the fall of two successive tin droplets ($T_{initial}$=800°C, $\alpha = 1.976\times10^{-5}$) in an oil tank ($T_{initial}$=250°C, K_{oil}= 22.6W/mK)