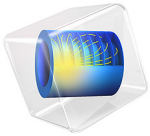

Tin Melting Front

Introduction

This example demonstrates how to model phase transition by a moving boundary interface according to the Stefan problem. It is adapted from the benchmark study in Ref. 1.

A square cavity containing both solid and liquid tin is submitted to a temperature difference between left and right boundaries. Fluid and solid parts are solved in separate domains sharing a moving melting front (see Figure 1). The position of this boundary through time is calculated according to the Stefan energy balance condition.

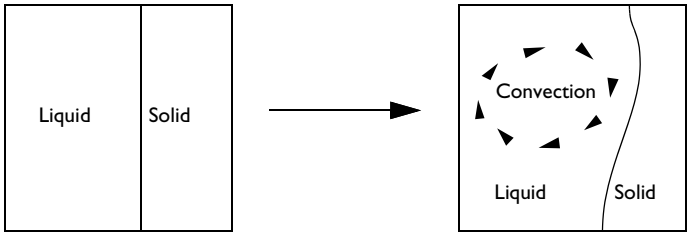

Figure 1: Square cavity with moving phase interface.

In the molten tin domain, motion generated by natural convection is expected due to the temperature gradient. This motion, in turn, influences the front displacement.

Model Definition

The geometry presented in Figure 2 shows a square of side length 10 cm filled with pure tin. The left and right boundaries are maintained at 508 K and 503 K, respectively.

Because the fusion temperature of pure tin is 505 K, both liquid and solid phases co-exist in the square.

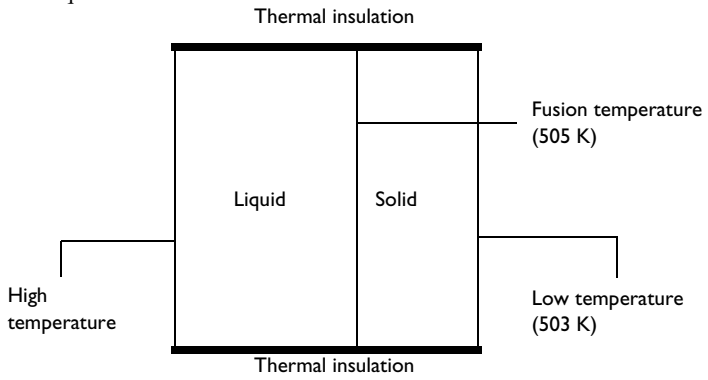

Figure 2: Geometry and boundary conditions at initial time.

The initial temperature distribution is assumed to vary linearly in the horizontal direction as shown in Figure 3.

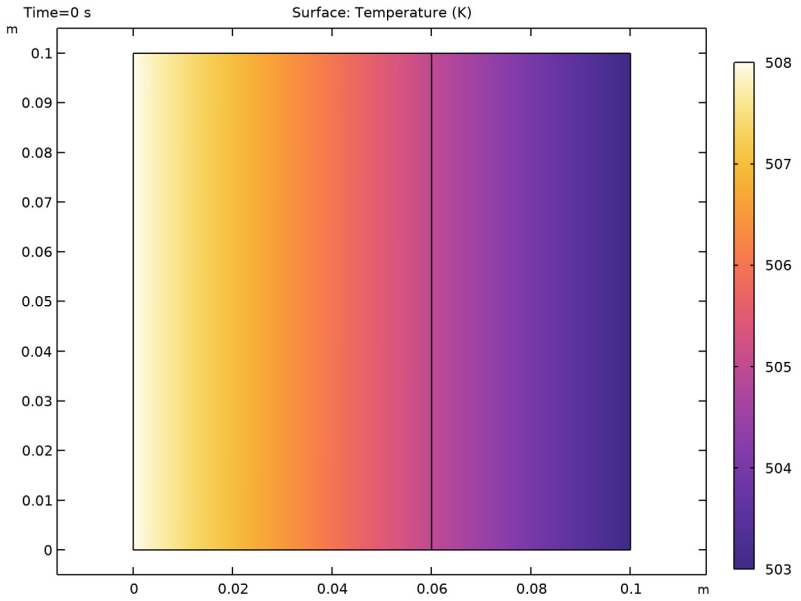

Figure 3: Initial temperature profile.

The melting front is the vertical line initially located at $x = 6$ cm where the temperature is 505 K.

The fluid motion in the liquid part on the left is modeled by the incompressible Navier–Stokes equations with Boussinesq approximation, as described in *Gravity* and *The Boussinesq Approximation* sections in the *CFD Module User's Guide*, with $T_{\text{ref}} = T_{\text{f}}$ the fusion temperature of tin. The reduced pressure formulation is used to enhance the accuracy of the buoyancy forces since they are relatively small compared to the other terms in the momentum balance.

As the metal melts, the solid-liquid interface moves toward the solid side. The energy balance at this front is expressed by the Stefan condition:

$$\rho_0 \Delta H \mathbf{v} \cdot \mathbf{n} = (\Phi_l - \Phi_s) \cdot \mathbf{n} \tag{1}$$

where ΔH is the latent heat of fusion, equal to 60 kJ/kg, $\mathbf{v}$ (SI unit: m/s) is the front velocity vector, $\mathbf{n}$ is the normal vector at the front, and Φ_l and Φ_s (SI unit: W/m^2) are the heat fluxes coming from the liquid and solid sides, respectively (see Figure 4).

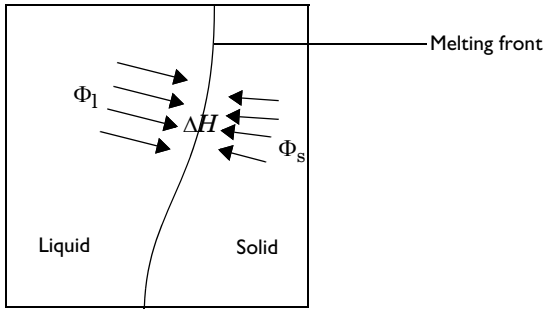

Figure 4: Heat fluxes at the melting front.

Table 1 reviews the material properties of tin (Ref. 2) used in this model.

TABLE 1: MATERIAL PROPERTIES OF TIN.

PARAMETER	DESCRIPTION	VALUE
ρ_0	Density	7500 kg/m^3
C_p	Heat capacity	200 J/(kg·K)
k	Thermal conductivity	60 W/(m·K)
α_p	Coefficient of thermal expansion	2.67·10^{-4} K^{-1}
μ	Dynamic viscosity	6.0·10^{-3} Pa·s

TABLE 1: MATERIAL PROPERTIES OF TIN.

PARAMETER	DESCRIPTION	VALUE
T_f	Fusion temperature	505 K
ΔH	Latent heat of fusion	60 kJ/kg

Results and Discussion

Figure 5 shows the velocity profile in the fluid domain. The convective cell due to buoyancy increases the melting speed at the upper part of the cavity.

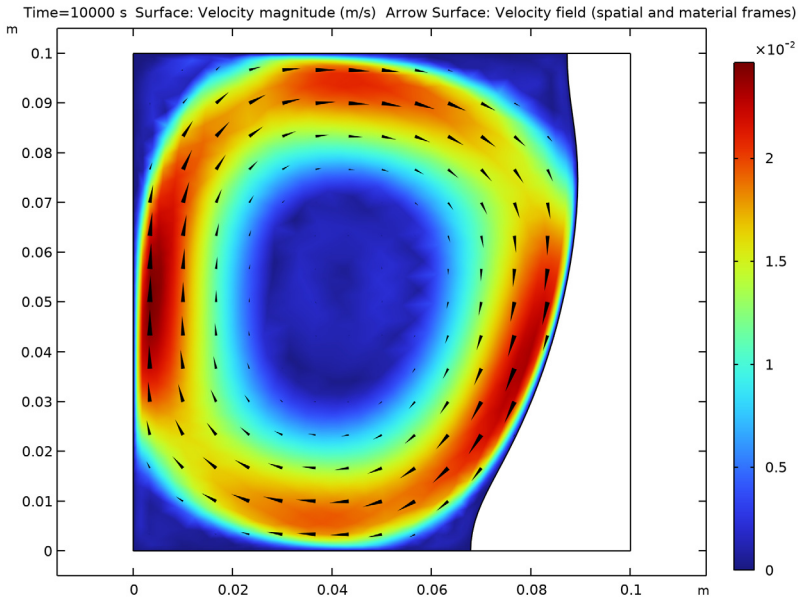

Figure 5: Velocity profile in the fluid at the end of the simulation.

At the end of the simulation, the melting front does not move anymore because balance between left and right adjacent fluxes has been reached.

In Figure 6, the temperature profile is represented jointly by a surface plot for temperature distribution and a velocity arrow plot.

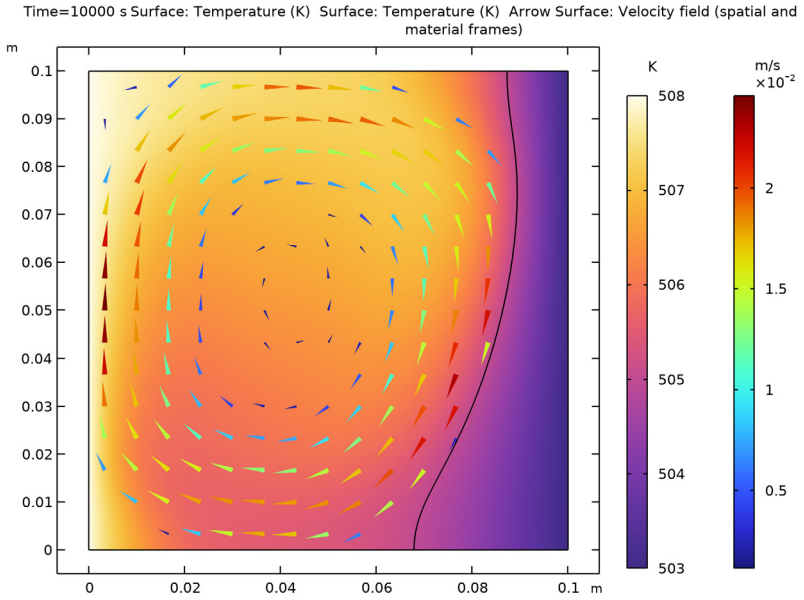

Figure 6: Temperature profile at the end of the simulation.

Notes About the COMSOL Implementation

The quantities Φ_l and Φ_s, illustrated in Figure 4, can be computed on each side of the melting front using the `up` and `down` operators. The components of $\Phi_l - \Phi_s$ are then given by `up(ht.tfluxx)-down(ht.tfluxx)` and `up(ht.tfluxy)-down(ht.tfluxy)`. However, this method evaluates the temperature gradient which may lead to imprecisions due to the mesh discretization. Instead, the quantity $(\Phi_l - \Phi_s) \cdot \mathbf{n}$, involved in Equation 1, is more precisely evaluated through the Lagrange multiplier for temperature, `T_lm`. This variable is available when weak constraints are enabled in the region of interest, as it is the case here with the fixed temperature constraint at the melting front. For more information about weak constraints, refer to the *Weak Constraint* section in the *COMSOL Multiphysics Reference Manual*.

To handle the melting front movement, a mesh deformation is necessary. During such a transformation, matter from solid tin is removed while the same amount of liquid tin is added to the fluid. The appropriate tool for deforming the mesh without reflecting any

expansion or contraction effects to the material properties is the **Deformed Geometry** node under **Definitions**.

References

1. F. Wolff and R. Viskanta, "Solidification of a Pure Metal at a Vertical Wall in the Presence of Liquid Superheat," *Int.J. Heat and Mass Transfer*, vol. 31, no. 8, pp. 1735–1744, 1988.

2. V. Alexiades, N. Hannoun, and T.Z. Mai, "Tin Melting: Effect of Grid Size and Scheme on the Numerical Solution," *Proc. 5th Mississippi State Conf. Differential Equations and Computational Simulations*, pp. 55–69, 2003.

Application Library path: `Heat_Transfer_Module/Phase_Change/tin_melting_front`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
k_Sn	60[W/(m*K)]	60 W/(m·K)	Thermal conductivity
Cp_Sn	200[J/(kg*K)]	200 J/(kg·K)	Specific heat capacity
alpha_Sn	2.67e-4[1/K]	2.67E-4 1/K	Coefficient of thermal expansion
mu_Sn	6e-3[Pa*s]	0.006 Pa·s	Kinematic viscosity
rho_Sn	7500[kg/m^3]	7500 kg/m³	Density
DelH	60[kJ/kg]	60000 J/kg	Latent heat of fusion
Tf	505[K]	505 K	Melting point
Th	508[K]	508 K	Hot wall temperature
Tc	503[K]	503 K	Cold wall temperature

GEOMETRY 1

Square 1 (sq1)

1 In the **Geometry** toolbar, click **Square**.

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `0.1`.

4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.06

5 Select the **Layers to the left** check box.

6 Clear the **Layers on bottom** check box.

7 In the **Geometry** toolbar, click **Build All**.

8 Click the **Zoom Extents** button in the **Graphics** toolbar.

MATERIALS

Tin (Solid)

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Tin (Solid)` in the **Label** text field.

3 Select Domain 2 only.

Before defining the material properties, set the solid and fluid domains in the physics interfaces to let COMSOL Multiphysics flag what properties you need to specify.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 Select Domain 1 only.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids and Fluids (ht)**.

2 In the **Settings** window for **Heat Transfer in Solids and Fluids**, locate the **Physical Model** section.

3 In the T_{ref} text field, type `Tf`.

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Heat Transfer in Solids and Fluids (ht)** click **Fluid 1**.

2 Select Domain 1 only.

MATERIALS

Tin (Solid) (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Tin (Solid) (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`k_Sn`	W/(m·K)	Basic
Density	rho	`rho_Sn`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`Cp_Sn`	J/(kg·K)	Basic

Duplicate this material: selecting the fluid domain will make COMSOL Multiphysics enquire for the missing needed parameters.

4 Right-click **Component 1 (comp1)>Materials>Tin (Solid) (mat1)** and choose **Duplicate**.

Tin (Liquid)

1 In the **Model Builder** window, expand the **Tin (Solid) (mat1)** node, then click **Component 1 (comp1)>Materials>Tin (Solid) 1 (mat2)**.

2 In the **Settings** window for **Material**, type `Tin (Liquid)` in the **Label** text field.

3 Select Domain 1 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Dynamic viscosity	mu	`mu_Sn`	Pa·s	Basic

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Initial Values 1

Define the initial temperature as a function of `Xg`, the first coordinate on the undeformed geometry.

1 In the **Model Builder** window, under **Component 1 (comp1)> Heat Transfer in Solids and Fluids (ht)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T text field, type `Th-Xg/0.1[m]*(Th-Tc)`.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Incompressible flow**.

4 Select the **Include gravity** check box.

5 Select the **Use reduced pressure** check box.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids and Fluids (ht)**.

Temperature 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `Th`.

Temperature 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `Tc`.

Phase Change Interface 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Phase Change Interface**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Phase Change Interface**, locate the **Phase Change Interface** section.

4 In the T_{pc} text field, type `Tf`.

5 In the $L_{s \to f}$ text field, type `DelH`.

6 From the **Solid side** list, choose **Downside**.

LAMINAR FLOW (SPF)

In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 1 only.

The model only contains information about the pressure gradient and estimates the pressure field up to a constant. To define this constant, you arbitrarily fix the pressure at a point.

MULTIPHYSICS

Nonisothermal Flow 1 (nitf1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Nonisothermal Flow 1 (nitf1)**.

2 In the **Settings** window for **Nonisothermal Flow**, locate the **Material Properties** section.

3 Select the **Boussinesq approximation** check box.

4 From the **Specify density** list, choose **Custom, linearized density**.

5 From the ρ_{ref} list, choose **From material**.

6 In the $\alpha_{p,0}$ text field, type `alpha_Sn`.

COMPONENT 1 (COMP1)

Deforming Domain 1

1 In the **Physics** toolbar, click **Deformed Geometry** and choose **Free Deformation**.

2 In the **Settings** window for **Deforming Domain**, locate the **Smoothing** section.

3 From the **Mesh smoothing type** list, choose **Laplace**.

Prescribed Normal Mesh Displacement 1

1 In the **Deformed Geometry** toolbar, click **Prescribed Normal Mesh Displacement**.

2 Select Boundaries 1–3 and 5–7 only.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

4 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog box, type 100 in the **Step** text field.

5 In the **Stop** text field, type 10000.

6 Click **Replace**.

For more robust convergence, tighten the relative tolerance, which controls the size of the time steps taken by the solver.

7 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

8 From the **Tolerance** list, choose **User controlled**.

9 In the **Relative tolerance** text field, type 1e-4.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Dependent Variables 1** node, then click **Material mesh displacement (comp1.material.disp)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 From the **Method** list, choose **From parent**.

6 In the **Study** toolbar, click **Compute**.

RESULTS

Temperature (ht)

This default plot shows the temperature profile.

Temperature and Fluid Flow (nitf1)

This default plot shows the temperature profile with the fluid flow. To reproduce Figure 6, modify the properties of the arrow plot.

Fluid Flow

1 In the **Model Builder** window, expand the **Temperature and Fluid Flow (nitf1)** node, then click **Fluid Flow**.

2 In the **Settings** window for **Arrow Surface**, locate the **Arrow Positioning** section.

3 Find the **x grid points** subsection. In the **Points** text field, type `15`.

4 Find the **y grid points** subsection. In the **Points** text field, type `15`.

Filter 1

1 In the **Model Builder** window, expand the **Fluid Flow** node, then click **Filter 1**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `spf.U>0.1*nitf1.Uave`.

4 In the **Temperature and Fluid Flow (nitf1)** toolbar, click **Plot**.

Velocity (spf)

This default plot shows the velocity profile in the fluid region. To reproduce Figure 5, add arrows of the velocity field to visualize the convective flow direction.

1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.

Arrow Surface 1

1 In the **Velocity (spf)** toolbar, click **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Laminar Flow>Velocity and pressure>u,v - Velocity field (spatial and material frames)**.

3 Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.

4 From the **Color** list, choose **Black**.

5 In the **Velocity (spf)** toolbar, click **Plot**.

Deformed Geometry

The last default plot shows the mesh deformation and mesh quality.

1 In the **Model Builder** window, under **Results** click **Deformed Geometry**.