Heat Pipe with Accurate Liquid and Gas Properties

Introduction

Heat pipes are designed to transfer heat efficiently through vaporization, mass transfer, and condensation of a working fluid. They are found in a wide variety of applications where thermal control is of importance, with cooling of electronics being a prominent example.

Inside a heat pipe, the temperature difference between the hot and cold sides together with the temperature dependence of the vapor pressure, induce a pressure difference across the vapor chamber. The pressure difference, in turn, drives the vapor from the hot to the cold side. The vaporization acts as a heat sink at the vapor–wick interface at the hot side, and conversely, the condensation as a heat source, at the cold side. This model demonstrates how the laminar flow in the vapor chamber of the heat pipe can be coupled to the liquid phase transport through the porous wick, and how thermodynamic properties of water can be obtained from COMSOL's built-in thermodynamics database. The importance of vapor transport is compared to the conductive heat transfer in the pipe wall. The former dominates the latter by several orders of magnitude.

Model Definition

Heat pipes exits in variety of different shapes. Tubular ones are probably the most common type however. Here we will look at an axisymmetric model of a copper tube with a porous copper wick and a vapor chamber. The heat pipe has a contact surface at the bottom, which is to be connected to the source of the heat to be removed. At the top of the pipe, a similar contact surface for a heat sink is used. The latter often corresponds to a finned metal structure that can easily be cooled by a fan. The geometry used including the different parts is visualized in Figure 1 below.

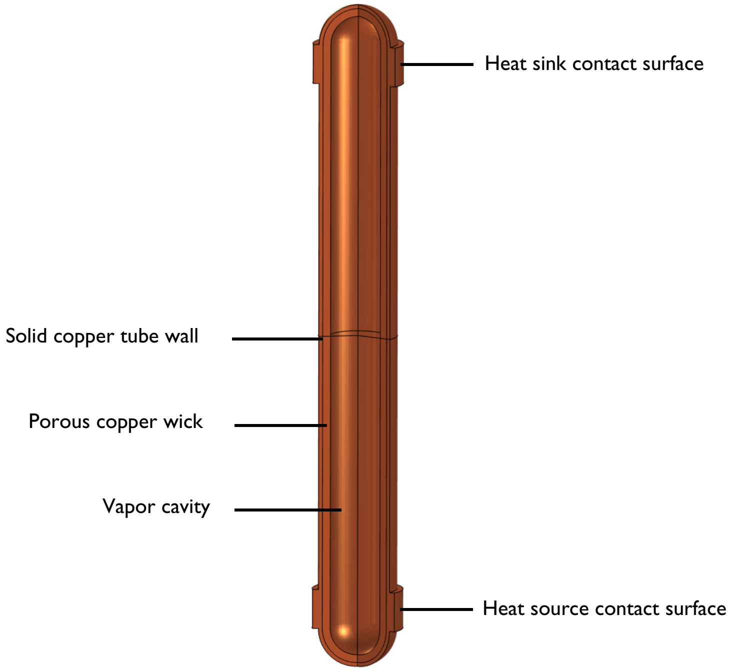

Figure 1: Overview of the heat pipe geometry.

Before setting up the model, we will investigate under what conditions our assumption of a saturated wick holds.

For heat pipes operating near ambient conditions, the capillary pressure, Δp_c, is usually the limiting factor (Ref. 1):

$$\Delta p_c = 2\frac{\sigma}{r_c} \tag{1}$$

Here, σ is the surface tension and r_c is the capillary radius. At the capillary limit, this pressure equals those pressure needed to drive the vapor, the static pressure due to gravity, and the pressure needed to drive liquid through the wick in the manner of

$$\Delta p_c = \Delta p_v + \Delta p_g + \Delta p_l \tag{2}$$

For most applications, we can neglect all but the liquid term, which can be obtained from Darcy's law:

$$\Delta p_l = \frac{\mu_l L_{eff}}{K A_w} \dot{V} \tag{3}$$

where μ_l is the dynamic viscosity of the liquid, L_{eff} is the effective length of the heat pipe, K is the permeability of the wick, A_w is the cross-sectional area of the wick, and $\dot{V}$ is the volumetric flow rate. The latter is governed by the rate of evaporation:

$$\dot{V} = \frac{\dot{Q}}{\Delta H_{vap} \rho} \tag{4}$$

where, $\dot{Q}$ is the heat transfer rate, ρ is the density of the liquid, ΔH_{vap} is the latent heat of vaporization (with dimensionality of energy per mass). Inserting Equation 2-4 into Equation 1, and neglecting Δp_v and Δp_g yields

$$\dot{Q} = 2\frac{K A_w \Delta H_{vap} \rho \sigma}{\mu_l L_{eff} r_c} \tag{5}$$

Evaluating Equation 5 with $K = 1 \cdot 10^{-9}\ m^2$, $A_w = 1 \cdot 10^{-4}\ m^2$, $\Delta H_{vap} = 2.5 \cdot 10^6$ J/kg, $\rho = 1 \cdot 10^3\ kg/m^3$, $\sigma = 7 \cdot 10^{-2}$ N/m, $\mu = 1 \cdot 10^{-3}$ Pa·s, L = 0.15 m, and $r_c = 3.1 \cdot 10^{-5}$ m, gives a value of 7.5 kW. In the model, a modest heat transfer rate of 30 W will be used, thus far from the capillary limit. For comparison, a CPU of a typical consumer PC produces on the order of 10–100 W.

PHYSICS SETUP

A **Laminar Flow** interface is used to solve for the laminar flow in the vapor cavity. It is subjected to single boundary condition, apart from the axial symmetry line. The pressure is prescribed to equal the saturated vapor pressure at the cavity–wick interface.

$$p = p_{H_2O,sat}(T) \tag{6}$$

This implies that the water and vapor phase are assumed to be in equilibrium at this position. The vapor pressure increases with temperature, which is what drives the vapor from the high temperature region to the low temperature region. For the liquid flow in the porous wick, a **Brinkman Equations** interface is used. The velocity in the wick at cavity–wick interface is computed from the vapor flow rate on the cavity side

$$\mathbf{u}_l = \frac{\mathbf{u}_v \rho_v}{\rho_l} \tag{7}$$

The pressure level is fixed using a pressure point constraint on the solid wall in the middle of the geometry.

For heat transfer in all parts of the geometry, the tube wall, the wick, and the vapor cavity, a **Heat Transfer in Porous Media interface** is used. It includes domain features for each domain type.

MATERIAL PROPERTIES

Material properties are created using the **Thermodynamics** node. A vapor system using the ideal gas law is setup for the vapor phase, while a liquid system using IAPWS models (Ref. 2) is created for the liquid phase in the wick. To describe the saturation pressure a vapor pressure function is created for the liquid system. In order to easily apply properties in the model, two materials are created using the **Generate Material** option available for thermodynamic systems. Copper from the material library is used for the properties in the tube wall.

Results and Discussion

As a first step, analyze the temperature when conduction is the only means of energy transport in the pipe. This corresponds to using a dry wick, with no liquid water, and negligible natural convection in the vapor. The resulting temperature is seen in Figure 2 below.

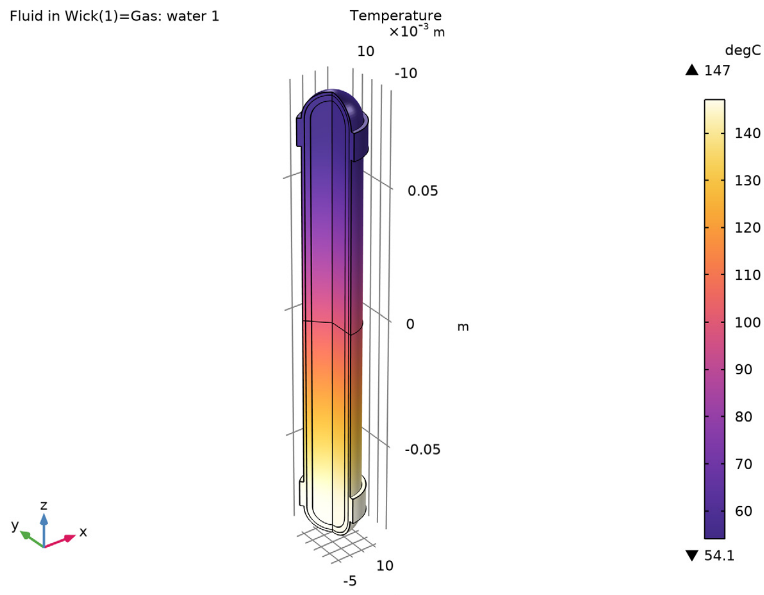

Figure 2: Temperature of heat pipe with dry wick.

Note that the temperature at the heat source is almost 100°C higher than at the heat sink. In an application where temperature sensitive parts are present (electronics, plastics, and so on) such a high temperature would be detrimental.

In the second simulation, the wick is assumed to be saturated with liquid water, corresponding to a heat pipe running at its design point. The resulting temperature profile, seen in Figure 3, looks dramatically different.

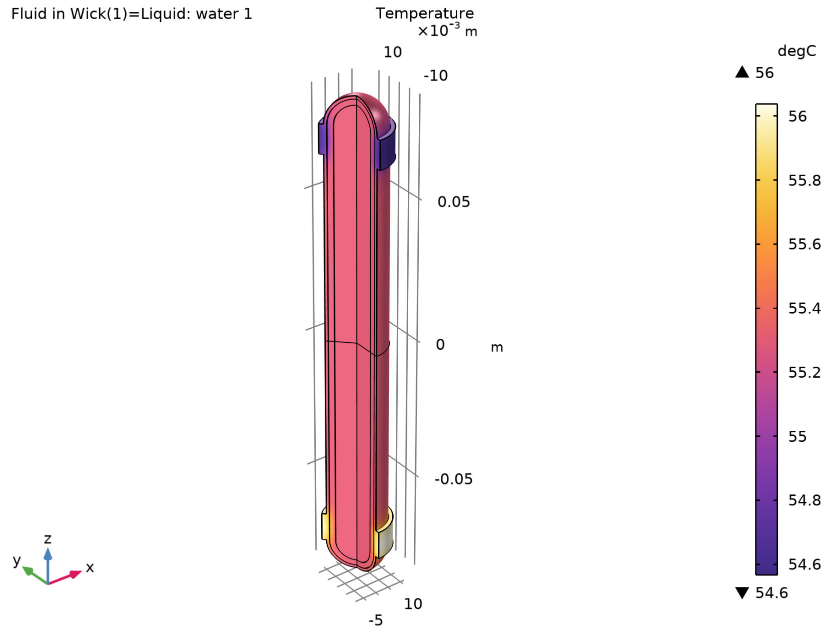

Figure 3: Temperature of heat pipe with saturated wick.

The predicted temperature difference between heat sink and heat source is now less than 2°C. And the surface of the heat pipe outside of the contact areas is essentially isothermal. In Figure 4, the computed velocity fields, in both fluids, and the temperature throughout the geometry, are plotted next to each other.

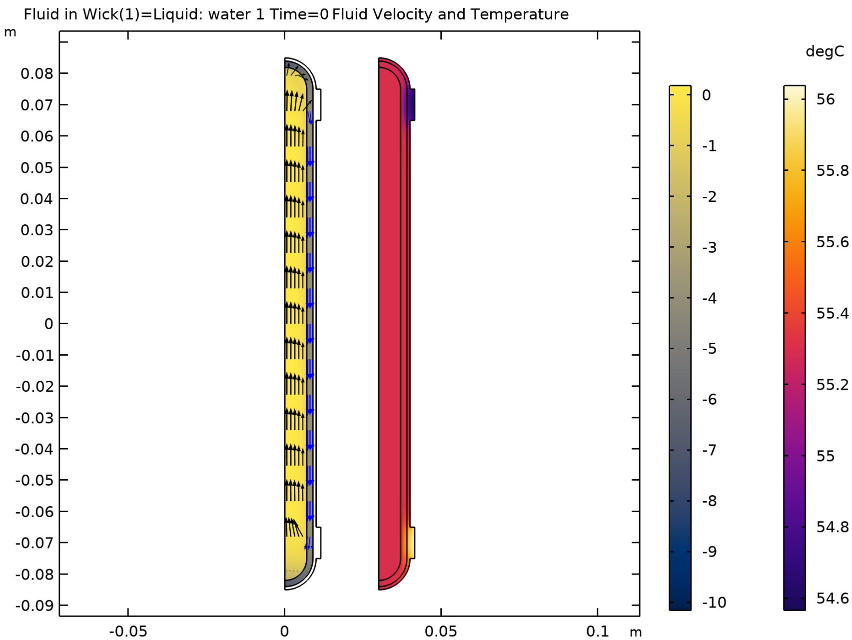

Figure 4: Fluid velocities — $\lg(|\mathbf{u}| / \mathrm{m \cdot s^{-1}})$ *and temperature in the heat pipe running at the design point.*

The heat transfer process can be decomposed into different contributions, by computing line integrals across the cavity, wick, and casing at the middle of the pipe. In Table 1, the relative importance of the different heat transfer mechanisms in the saturated heat pipe.

TABLE 1: RESULTS FROM THE ENERGY BALANCE EVALUATION GROUP.

Process	**Heat transfer rate / W**
Conductive heat transfer in casing	$4 \cdot 10^{-5}$
Conductive heat transfer in wick	$3 \cdot 10^{-5}$
Latent heat transfer in cavity	30

It is seen that, at normal operating conditions, vapor mass transfer (and its associated phase changes) is the completely dominating mechanism by which the heat pipe transfers heat.

References

1. I. Shishido, I. Oishi, and S. Ohtani, "Capillary Limit in Heat Pipes," *J. Chem. Eng. Japan*, vol. 17, no. 2, pp. 179–186, 1986.

2. W. Wagner and H.J. Kretzschmar *IAPWS industrial formulation 1997 for the thermodynamic properties of water and steam. International steam tables: properties of water and steam based on the industrial formulation IAPWS-IF97*, pp. 7–150, 2008.

Application Library path: `Chemical_Reaction_Engineering_Module/Thermodynamics/heat_pipe`

Note: This model is included in the booklet *Introduction to the Liquid & Gas Properties Module*.

Model 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 Axisymmetric**.
2. In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.
3. Right-click and choose **Add Physics**.
4. In the **Select Physics** tree, select **Fluid Flow>Porous Media and Subsurface Flow>Brinkman Equations (br)**.
5. Right-click and choose **Add Physics**.
6. In the **Select Physics** tree, select **Heat Transfer>Porous Media>Heat Transfer in Porous Media (ht)**.
7. Right-click and choose **Add Physics**.
8. Click **Study**.

9 In the **Select Study** tree, select **General Studies>Stationary**.

10 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Start by reading in a set of parameters defining the dimensions and specific properties.

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `heat_pipe_parameters.txt`.

GEOMETRY 1

Define a cylinder with rounded ends, use a sector of a circle, three rectangles and a mirror plane.

Circle 1 (c1)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r_outer`.

4 In the **Sector angle** text field, type `90`.

5 Locate the **Position** section. In the **z** text field, type `length/2`.

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

Layer name	Thickness (m)
Layer 1	w_casing
Layer 2	w_wick

Rectangle 1 (r1)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `r_outer + w_contact`.

4 In the **Height** text field, type `l_heatsource`.

5 Locate the **Position** section. In the **z** text field, type `length/2-l_heatsource`.

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

Layer name	Thickness (m)
`Layer 1`	`w_casing + w_contact`
`Layer 2`	`w_wick`

7 Select the **Layers to the right** check box.

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

Rectangle 2 (r2)

1 Right-click **Rectangle 1 (r1)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `r_outer`.

4 Locate the **Position** section. In the **z** text field, type `length/2-l_heatsource*2`.

5 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`w_casing`

Rectangle 3 (r3)

1 Right-click **Rectangle 2 (r2)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Height** text field, type `length/2-l_heatsource*2`.

4 Locate the **Position** section. In the **z** text field, type `0`.

At this point, the geometry primitives for the upper half of the heat pipe has been added, go ahead and add a mirror plane to obtain a complete heat pipe.

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Locate the **Normal Vector to Line of Reflection** section. In the **r** text field, type `0`.

6 In the **z** text field, type `1`.

Use a quadrilateral mesh for the middle part of the heat pipe and triangular meshes for the ends. To facilitate this mesh transition, define mesh control edges; these can be referenced during meshing, while not contributing to splitting of domains in other contexts.

Mesh Control Edges 1 (mce1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Edges**.

2 On the object **fin**, select Boundaries 9, 13, 25, 29, 38, and 42 only.

Analogously, ignore some other edges completely, not only during meshing.

Ignore Edges 1 (ige1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **mce1**, select Boundaries 5, 7, 11, 13, 17, 19, 23, and 24 only.

Ignore Edges 2 (ige2)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **ige1**, select Boundaries 12, 14, 18, and 19 only.

3 In the **Settings** window for **Ignore Edges**, locate the **Input** section.

4 Clear the **Ignore adjacent vertices** check box.

Ignore Vertices 1 (igv1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Vertices**.

2 On the object **ige2**, select Points 9, 10, 12, and 13 only.

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

Vapor Cavity

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

It is a good practice to introduce explicit selections with descriptive names, these will help when making selections in the physics interfaces later on. Therefore, define selections for the three domains, and a handful of boundaries of interest.

2 In the **Settings** window for **Explicit Selection**, type `Vapor Cavity` in the **Label** text field.

3 On the object **igv1**, select Domains 3 and 4 only.

Porous Copper Wick

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Porous Copper Wick` in the **Label** text field.

3 On the object **igv1**, select Domains 2 and 5 only.

Solid Copper Tube Casing

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Solid Copper Tube Casing` in the **Label** text field.

3 On the object **igv1**, select Domains 1 and 6 only.

Heat Sink

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Heat Sink` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **igv1**, select Boundary 21 only.

Heat Source

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 On the object **igv1**, select Boundary 20 only.

5 In the **Label** text field, type `Heat Source`.

Cross Section of Cavity

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Cross Section of Cavity` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **igv1**, select Boundary 5 only.

Cross Section of Wick

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Cross Section of Wick` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **igv1**, select Boundary 10 only.

Cross Section of Casing

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Cross Section of Casing` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **igv1**, select Boundary 13 only.

Inner Wick Boundary

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Inner Wick Boundary` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **igv1**, select Boundaries 8 and 9 only.

All Wick Boundaries

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `All Wick Boundaries` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Boundary**.

4 On the object **igv1**, select Boundaries 8, 9, 11, and 12 only.

Next, a union selection of the cross sections is created. Those cross sections have no physical importance, and will only be used for analysis in the postprocessing stage. Therefore, hide them during the setup of the model.

All Cross Sections

1 In the **Geometry** toolbar, click **Selections** and choose **Union Selection**.

2 In the **Settings** window for **Union Selection**, type `All Cross Sections` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Cross Section of Cavity**, **Cross Section of Wick**, and **Cross Section of Casing**.

6 Click **OK**.

DEFINITIONS

Hide for Geometry 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **View 1** and choose **Hide for Geometry**.

3 In the **Settings** window for **Hide for Geometry**, locate the **Selection** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **All Cross Sections**.

MESH 1

The following steps will set up a mesh with its element size controlled by a global parameter, and use a different mesh for the middle segment of the heat pipe.

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 4, 7–9, 11, and 12 only.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

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

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type `length/r_outer/mesh_factor`.

5 In the **Element ratio** text field, type `10`.

6 Select the **Symmetric distribution** check box.

7 Click **Build Selected**.

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

1 In the **Model Builder** window, click **Size**.

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.9*min(w_casing, w_wick)*mesh_factor`.

5 In the **Minimum element size** text field, type `0.3*min(w_casing, w_wick)*mesh_factor`.

Boundary Layers 1

1 In the **Mesh** toolbar, click **Boundary Layers**.
2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domains 2–5 and 8–11 only.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.
2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All Wick Boundaries**.
4 Click **Build All**.

MATERIALS

You need to add materials for the metal casing and the wick, as well as for the working fluid (both in gaseous and liquid form). Copper will be added from the built-in materials. For water vapor and liquid water, two thermodynamic systems are created, from which materials are then generated. Note that you need to add copper to global materials for it to be accessible in the porous material node, which will be added to Component 1. To use this global Copper material in Component 1 for the Solid Copper Tube Casing domain selection, a material link will be added.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Built-in>Copper**.
4 Right-click and choose **Add to Global Materials**.
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

GLOBAL DEFINITIONS

For water vapor use an ideal gas model.

1 In the **Physics** toolbar, click **Thermodynamics** and choose **Thermodynamic System**.

SELECT SYSTEM

1 Go to the **Select System** window.
2 Click **Next** in the window toolbar.

SELECT SPECIES

1 Go to the **Select Species** window.

2 In the **Species** list, select **water (7732-18-5, H2O)**.

3 Click **Add Selected**.

4 Click **Next** in the window toolbar.

SELECT THERMODYNAMIC MODEL

1 Go to the **Select Thermodynamic Model** window.

2 From the **Gas phase model** list, choose **Ideal gas**.

3 Click **Finish** in the window toolbar.

GLOBAL DEFINITIONS

Gas System 1 (pp1)

In the **Model Builder** window, under **Global Definitions>Thermodynamics** right-click **Gas System 1 (pp1)** and choose **Generate Material**.

SELECT PHASE

1 Go to the **Select Phase** window.

2 Click **Next** in the window toolbar.

SELECT SPECIES

1 Go to the **Select Species** window.

2 Click **Add All**.

3 Click **Next** in the window toolbar.

SELECT PROPERTIES

1 Go to the **Select Properties** window.

2 Click **Next** in the window toolbar.

DEFINE MATERIAL

1 Go to the **Define Material** window.

2 From the **Component** list, choose **Global**.

3 Click **Finish** in the window toolbar.

GLOBAL DEFINITIONS

1 In the **Physics** toolbar, click **Thermodynamics** and choose **Thermodynamic System**.
For liquid water, use the IAPWS model.

SELECT SYSTEM

1 Go to the **Select System** window.
2 From the **Phase** list, choose **Vapor-liquid**.
3 Click **Next** in the window toolbar.

SELECT SPECIES

1 Go to the **Select Species** window.
2 In the **Species** list, select **water (7732-18-5, H2O)**.
3 Click + **Add Selected**.
4 Click **Next** in the window toolbar.

SELECT THERMODYNAMIC MODEL

1 Go to the **Select Thermodynamic Model** window.
2 Click **Finish** in the window toolbar.

GLOBAL DEFINITIONS

Vapor-Liquid System 1 (pp2)

In the **Model Builder** window, under **Global Definitions>Thermodynamics** right-click **Vapor-Liquid System 1 (pp2)** and choose **Species Property**.

SELECT PROPERTIES

1 Go to the **Select Properties** window.
2 From the **Amount base unit** list, choose **kg**.
3 In the list, select **Heat of vaporization (J/kg)**.
4 Click + **Add Selected**.
5 In the list, select **Ln vapor pressure, Pa**.
6 Click + **Add Selected**.
7 Click **Next** in the window toolbar.

SELECT PHASE

1 Go to the **Select Phase** window.

2 Click **Next** in the window toolbar.

SELECT SPECIES

1 Go to the **Select Species** window.

2 Click **Add All**.

3 Click **Next** in the window toolbar.

SPECIES PROPERTY OVERVIEW

1 Go to the **Species Property Overview** window.

2 Click **Finish** in the window toolbar.

GLOBAL DEFINITIONS

Vapor pressure of water

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Analytic**.

2 In the **Settings** window for **Analytic**, type `Vapor pressure of water` in the **Label** text field.

3 In the **Function name** text field, type `pH2O`.

4 Locate the **Definition** section. In the **Expression** text field, type `exp(LnVaporPressure_water22(T))`.

5 In the **Arguments** text field, type `T`.

6 Locate the **Units** section. In the table, enter the following settings:

Argument	Unit
T	K

7 In the **Function** text field, type `Pa`.

8 Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit	Unit
T	273.15	373.15	K

Vapor-Liquid System 1 (pp2)

Right-click **Vapor-Liquid System 1 (pp2)** and choose **Generate Material**.

SELECT PHASE

1 Go to the **Select Phase** window.

2 From the list, choose **Liquid**.

3 Click **Next** in the window toolbar.

SELECT SPECIES

1 Go to the **Select Species** window.

2 Click **Add All**.

3 Click **Next** in the window toolbar.

SELECT PROPERTIES

1 Go to the **Select Properties** window.

2 Click **Next** in the window toolbar.

DEFINE MATERIAL

1 Go to the **Define Material** window.

2 Click **Finish** in the window toolbar.

GLOBAL DEFINITIONS

Two cases will be investigated: a dry wick, and a saturated wick. By introducing a material switch, later interfaces can refer to this switch, which can either refer to vapor or liquid. Its state will be controlled from the Study nodes.

Fluid in Wick

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Material Switch**.

2 In the **Settings** window for **Material Switch**, type `Fluid in Wick` in the **Label** text field.

Liquid: water 1 (pp2mat1)

In the **Model Builder** window, right-click **Liquid: water 1 (pp2mat1)** and choose **Copy**.

Liquid: water 1 (sw1.pp2mat1)

In the **Model Builder** window, right-click **Fluid in Wick (sw1)** and choose **Paste Material**.

Gas: water 1 (pp1mat1)

In the **Model Builder** window, under **Global Definitions>Materials** right-click **Gas: water 1 (pp1mat1)** and choose **Copy**.

Gas: water 1 (sw1.pp1mat1)

In the **Model Builder** window, right-click **Fluid in Wick (sw1)** and choose **Paste Material**.

MATERIALS

Add a Porous Material for use in the wick, two sub-features are added to the Porous Material node, one for the fluid, and one for the solid.

Porous Material 1 (pmat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Porous Material**.

2 In the **Settings** window for **Porous Material**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **Porous Copper Wick**.

Fluid 1 (pmat1.fluid1)

1 Right-click **Porous Material 1 (pmat1)** and choose **Fluid**.

2 In the **Settings** window for **Fluid**, locate the **Fluid Properties** section.

3 From the **Material** list, choose **Fluid in Wick (sw1)**.

Solid 1 (pmat1.solid1)

1 In the **Model Builder** window, right-click **Porous Material 1 (pmat1)** and choose **Solid**.

2 In the **Settings** window for **Solid**, locate the **Solid Properties** section.

3 In the θ_s text field, type `1-wick_porosity`.

The material for the wick is now defined, continue to add links to water vapor and copper.

Water Vapor

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

2 In the **Settings** window for **Material Link**, type `Water Vapor` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Vapor Cavity**.

4 Locate the **Link Settings** section. From the **Material** list, choose **Gas: water 1 (pp1mat1)**.

Copper Metal

1 Right-click **Materials** and choose **More Materials>Material Link**.

2 In the **Settings** window for **Material Link**, type `Copper Metal` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Solid Copper Tube Casing**.

Now that the geometry, selections and materials are in place, go on to set up the physics.

MULTIPHYSICS

Nonisothermal Flow 1 (nitf1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Nonisothermal Flow**.

Nonisothermal Flow 2 (nitf2)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Nonisothermal Flow**.
2 In the **Settings** window for **Nonisothermal Flow**, locate the **Coupled Interfaces** section.
3 From the **Fluid flow** list, choose **Brinkman Equations (br)**.

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 **Domain Selection** section.
3 From the **Selection** list, choose **Vapor Cavity**.
4 Locate the **Physical Model** section. From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.
5 In the p_{ref} text field, type `p_ref`.

Apply a pressure condition on the cavity boundary and set it equal to the vapor pressure of water. Not suppressing backflow allows vapor to enter at the hot side and exit at the cold side.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2 In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Inner Wick Boundary**.
4 Locate the **Boundary Condition** section. From the list, choose **Pressure**.
5 Locate the **Pressure Conditions** section. In the p_0 text field, type `pH2O(T)-p_ref`.
6 Clear the **Suppress backflow** check box.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.
2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3 In the p text field, type `p_ref`.

BRINKMAN EQUATIONS (BR)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Brinkman Equations (br)**.
2. In the **Settings** window for **Brinkman Equations**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Porous Copper Wick**.
4. Locate the **Physical Model** section. From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.
5. In the p_{ref} text field, type `p_ref`.

Porous Matrix 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Brinkman Equations (br)>Porous Medium 1** click **Porous Matrix 1**.
2. In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.
3. From the κ list, choose **User defined**. In the associated text field, type `wick_permeability`.

The mass flux of water through the cavity–wick interface must be equal in both the Laminar Flow interface, as well as the Brinkman Equations interface. Using the ratios of densities between vapor and liquid, you can apply a velocity boundary condition on the wick.

Inlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Inner Wick Boundary**.
4. Locate the **Velocity** section. Click the **Velocity field** button.
5. Specify the $\mathbf{u}_0$ vector as

`u*spf.rho/br.rho`	r
`w*spf.rho/br.rho`	z

Apply a pressure constraint of 0[Pa] to the point on the r axis at the interface between the wick and the casing:

Pressure Point Constraint 1

1. In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.
2. Select Point 9 only.
3. In the **Settings** window for **Pressure Point Constraint**, locate the **Pressure Constraint** section.

4 In the p_0 text field, type `pH2O(T)-p_ref`.

HEAT TRANSFER IN POROUS MEDIA (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Porous Media (ht)**.

Solid 1

1 In the **Physics** toolbar, click **Domains** and choose **Solid**.

2 In the **Settings** window for **Solid**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Solid Copper Tube Casing**.

Fluid 1

1 In the **Physics** toolbar, click **Domains** and choose **Fluid**.

2 In the **Settings** window for **Fluid**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Vapor Cavity**.

Porous Matrix 1

Use copper as the solid phase in the porous material. Because the material contains properties for the dense bulk material, you need to specify that the material properties are bulk properties and therefore need to be scaled by the porosity.

1 In the **Model Builder** window, under **Component 1 (comp1)> Heat Transfer in Porous Media (ht)>Porous Medium 1** click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.

3 From the **Define** list, choose **Solid phase properties**.

If you had a predefined material which had data for its porous state, say “copper sponge”, you would have kept the “Define” choice under Matrix Properties as “Dry bulk properties”.

Next, specify the boundary condition corresponding to the heat source.

Heat Flux 1

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

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Heat Source**.

4 Locate the **Heat Flux** section. In the q_0 text field, type `phi_in`.

The boundary condition for the heat sink is set up analogously, but with a convective heat-flux condition. That is, the heat flux out of the pipe through the heat sink is proportional to the temperature difference between the heat sink and the external environment. The magnitude of the proportionality constant (the heat-transfer coefficient) depends on external flow conditions such as the presence of an external fan (and its speed), the external surface area, and the geometry of fins (if present).

Heat Flux 2

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

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Heat Sink**.

4 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

5 In the h text field, type `h_conv`.

The heat associated with the phase change of water, removes heat at the hot side (evaporation) and contributes heat at the cold side (condensation), the amount of energy involved is the heat of vaporization. Add a boundary heat source at the cavity/wick interface for this process.

Boundary Heat Source 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Heat Source**.

2 In the **Settings** window for **Boundary Heat Source**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inner Wick Boundary**.

4 Locate the **Boundary Heat Source** section. In the Q_b text field, type `(u*spf.nr+w*spf.nz)*HeatOfVaporization_water21(T)*spf.rho`.

STUDY 1 - DRY WICK

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Study 1 - Dry Wick` in the **Label** text field.

In order to select the state of our material switch "Fluid in Wick" to correspond to the dry case, a material sweep is added to the study, but is only given a single case, the index (1) of our first material in the switch, "Gas: water 1".

Material Sweep

1 In the **Study** toolbar, click **Material Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Switch	Cases	Case numbers
Fluid in Wick (sw1)	User defined	2

Step 1: Stationary

1 In the **Model Builder** window, click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check boxes for **Laminar Flow (spf)** and **Brinkman Equations (br)**.

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

RESULTS

Temperature, 3D (ht) - Dry Wick

Once the solver is done, look at the temperature profile along the heat pipe for this "dry" case.

1 In the **Settings** window for **3D Plot Group**, type `Temperature, 3D (ht) - Dry Wick` in the **Label** text field.

2 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

3 In the **Title** text area, type `Temperature`.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

5 Select the **Show units** check box.

Surface

1 In the **Model Builder** window, expand the **Temperature, 3D (ht) - Dry Wick** node, then click **Surface**.

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

3 From the **Unit** list, choose **degC**.

4 In the **Temperature, 3D (ht) - Dry Wick** toolbar, click **Plot**.

This is Figure 2 from the earlier Results section. Note that for the parameters at hand, the pipe would get quite hot.

Remove one of the plots before moving on.

Isothermal Contours (ht)

In the **Model Builder** window, under **Results** right-click **Isothermal Contours (ht)** and choose **Delete**.

ROOT

Now solve the model for the normal case, with actual liquid in the wick.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Right-click and choose **Add Study**.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2 - SATURATED WICK

In the **Settings** window for **Study**, type `Study 2 - Saturated Wick` in the **Label** text field.

Material Sweep

1 In the **Study** toolbar, click **Material Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Switch	Cases	Case numbers
Fluid in Wick (sw1)	User defined	1

Step 1: Stationary

Solve Study 2 in two steps: first without the Brinkman Equations and then with all interfaces active. Solving in this order will give the Brinkman Equations an initial guess close to the final solution, making the process more robust and efficient.

Step 2: Stationary 1

In the **Model Builder** window, under **Study 2 - Saturated Wick** right-click **Step 1: Stationary** and choose **Duplicate**.

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for **Brinkman Equations (br)**.

3 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Velocity (spf) and Temperature (ht)

By requesting initial values, default plot groups are created which you can now modify.

1 In the **Settings** window for **2D Plot Group**, type `Velocity (spf) and Temperature (ht)` in the **Label** text field.

2 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

3 In the **Title** text area, type `Fluid Velocity and Temperature`.

4 Locate the **Color Legend** section. Select the **Show units** check box.

Surface 1 - Fluid Velocity, lg(|u|)

1 In the **Model Builder** window, expand the **Velocity (spf) and Temperature (ht)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, type `Surface 1 - Fluid Velocity, lg(|u|)` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `log10(ht.uz^2 + ht.ur^2)/2`.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Linear>Cividis** in the tree.

6 Click **OK**.

Arrow Surface 1

In the **Model Builder** window, right-click **Velocity (spf) and Temperature (ht)** and choose **Arrow Surface**.

Arrow Surface 1, Surface 1 - Fluid Velocity, lg(|u|)

1 In the **Model Builder** window, under **Results>Velocity (spf) and Temperature (ht)**, Ctrl-click to select **Surface 1 - Fluid Velocity, lg(|u|)** and **Arrow Surface 1**.

2 Right-click and choose **Duplicate**.

Surface 2 - Temperature

1 In the **Settings** window for **Surface**, type `Surface 2 - Temperature` in the **Label** text field.

2 Locate the **Expression** section. In the **Expression** text field, type `T`.

3 From the **Unit** list, choose **degC**.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Thermal>HeatCamera** in the tree.

6 Click **OK**.

Use a deformation to allow showing two plots side by side.

Deformation 1

1 Right-click **Surface 2 - Temperature** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Expression** section.

3 In the **r-component** text field, type `r_outer`.

4 Locate the **Scale** section.

5 Select the **Scale factor** check box. In the associated text field, type `3`.

Arrow Surface 1 - Vapor Flow

1 In the **Model Builder** window, under **Results>Velocity (spf) and Temperature (ht)** click **Arrow Surface 1**.

2 In the **Settings** window for **Arrow Surface**, type `Arrow Surface 1 - Vapor Flow` in the **Label** text field.

3 Locate the **Arrow Positioning** section. Find the **r grid points** subsection. In the **Points** text field, type `9`.

4 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

5 Select the **Scale factor** check box. In the associated text field, type `0.005`.

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

Arrow Surface 1 - Vapor Flow 1

Right-click **Arrow Surface 1 - Vapor Flow** and choose **Duplicate**.

Arrow Surface 2 - Liquid Flow

1 In the **Settings** window for **Arrow Surface**, type `Arrow Surface 2 - Liquid Flow` in the **Label** text field.

2 Locate the **Expression** section. In the **r-component** text field, type `u2`.

3 In the **z-component** text field, type `w2`.

4 Locate the **Coloring and Style** section.

5 Select the **Scale factor** check box. In the associated text field, type `50`.

6 From the **Color** list, choose **Blue**.

Line 1 - Material Boundaries

1 In the **Model Builder** window, right-click **Velocity (spf) and Temperature (ht)** and choose **Line**.

2 In the **Settings** window for **Line**, type `Line 1 - Material Boundaries` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

Deformation 1

1 Right-click **Line 1 - Material Boundaries** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Expression** section.

3 In the **r-component** text field, type `r_outer`.

4 Locate the **Scale** section.

5 Select the **Scale factor** check box. In the associated text field, type `3`.

STUDY 2 - SATURATED WICK

Solver Configurations

In the **Model Builder** window, expand the **Study 2 - Saturated Wick>Solver Configurations** node.

Solution 4 (sol4)

1 In the **Model Builder** window, expand the **Study 2 - Saturated Wick>Solver Configurations>Solution 4 (sol4)>Stationary Solver 1** node.

2 Right-click **Stationary Solver 1** and choose **Fully Coupled**.

3 In the **Settings** window for **Fully Coupled**, click to expand the **Results While Solving** section.

4 Select the **Plot** check box.

5 From the **Plot group** list, choose **Velocity (spf) and Temperature (ht)**.

6 In the **Model Builder** window, right-click **Stationary Solver 2** and choose **Fully Coupled**.

7 In the **Settings** window for **Fully Coupled**, locate the **Results While Solving** section.

8 Select the **Plot** check box.

9 From the **Plot group** list, choose **Velocity (spf) and Temperature (ht)**.

RESULTS

To aid the analysis of the results, create a series of line integrals of the heat flux across boundaries, in an Evaluation Group. Let the line integrals run across the middle section of the pipe (on the r-axis), and the contact surfaces for heat source and heat sink.

Energy Balance

1 In the **Results** toolbar, click **Evaluation Group**.
2 In the **Settings** window for **Evaluation Group**, type `Energy Balance` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - Saturated Wick/ Parametric Solutions 2 (sol6)**.
4 Locate the **Transformation** section. Select the **Transpose** check box.

Heat Sink

1 Right-click **Energy Balance** and choose **Integration>Line Integration**.
2 In the **Settings** window for **Line Integration**, type `Heat Sink` in the **Label** text field.
3 Locate the **Selection** section. From the **Selection** list, choose **Heat Sink**.
4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Porous Media>Boundary fluxes> ht.ndflux - Normal conductive heat flux - W/m²**.
5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`ht.ndflux`	`W`	`sink: ndflux`

Heat Source

1 Right-click **Heat Sink** and choose **Duplicate**.
2 In the **Settings** window for **Line Integration**, type `Heat Source` in the **Label** text field.
3 Locate the **Selection** section. From the **Selection** list, choose **Heat Source**.
4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`ht.ndflux`	`W`	`source: ndflux`

Casing

1 Right-click **Heat Source** and choose **Duplicate**.
2 In the **Settings** window for **Line Integration**, type `Casing` in the **Label** text field.
3 Locate the **Selection** section. From the **Selection** list, choose **Cross Section of Casing**.

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

Expression	Unit	Description
ht.ndflux	W	casing: ndflux

Wick

1 Right-click **Casing** and choose **Duplicate**.

2 In the **Settings** window for **Line Integration**, type Wick in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Cross Section of Wick**.

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

Expression	Unit	Description
ht.ndflux	W	wick: ndflux

Cavity

1 Right-click **Wick** and choose **Duplicate**.

2 In the **Settings** window for **Line Integration**, type Cavity in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Cross Section of Cavity**.

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

Expression	Unit	Description
ht.ndflux	W	cavity: ndflux
w*spf.rho* HeatOfVaporization_water21(T)	W	cavity: latent heat

To investigate how the phase change along the wick transfers thermal energy, create a line graph plotting this heat flux along the cavity–wick boundary.

Latent heat flux from phase change

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type Latent heat flux from phase change in the **Label** text field.

Line Graph 1

1 Right-click **Latent heat flux from phase change** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2 - Saturated Wick/Parametric Solutions 2 (sol6)**.

4 Locate the **Selection** section. From the **Selection** list, choose **Inner Wick Boundary**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type (u*spf.nr + w*spf.nz)*spf.rho*HeatOfVaporization_water21(T).

6 Select the **Description** check box. In the associated text field, type (<B>u</B>\cdot<B>n</B>)\rho\DELTA H_{vap}.

7 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

8 In the **Title** text area, type Latent heat flux from phase change.

9 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

10 In the **Expression** text field, type z.

STUDY 2 - SATURATED WICK

In the **Home** toolbar, click **Compute**.

RESULTS

Energy Balance

1 In the **Model Builder** window, under **Results** click **Energy Balance**.

2 In the **Energy Balance** toolbar, click **Evaluate**.

Temperature, 3D (ht)

1 In the **Model Builder** window, click **Temperature, 3D (ht)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type Temperature.

5 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

6 Select the **Show units** check box.

Surface

1 In the **Model Builder** window, expand the **Temperature, 3D (ht)** node, then click **Surface**.

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

3 From the **Unit** list, choose **degC**.

Temperature, 3D (ht)

1 In the **Model Builder** window, click **Temperature, 3D (ht)**.

2 In the **Temperature, 3D (ht)** toolbar, click **Plot**.

This is Figure 3 from the results section earlier. It is striking how even the temperature is throughout the heat pipe. Also the maximum temperature is dramatically lower compared to the dry wick case investigated previously.