

INTRODUCTION TO  
Liquid & Gas Properties

# Introduction to the Liquid & Gas Properties Module

© 1998–2022 COMSOL

Protected by patents listed on [www.comsol.com/patents](http://www.comsol.com/patents), or see Help>About COMSOL Multiphysics on the File menu in the COMSOL Desktop for a less detailed lists of U.S. Patents that may apply. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement ([www.comsol.com/sla](http://www.comsol.com/sla)) and may be used or copied only under the terms of the license agreement.

COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, COMSOL Compiler, COMSOL Server, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see [www.comsol.com/trademarks](http://www.comsol.com/trademarks).

Version: COMSOL 6.1

## Contact Information

Visit the Contact COMSOL page at [www.comsol.com/contact](http://www.comsol.com/contact) to submit general inquiries or search for an address and phone number. You can also visit the Worldwide Sales Offices page at [www.comsol.com/contact/offices](http://www.comsol.com/contact/offices) for address and contact information.

If you need to contact Support, an online request form is located on the COMSOL Access page at [www.comsol.com/support/case](http://www.comsol.com/support/case). Other useful links include:

- Support Center: [www.comsol.com/support](http://www.comsol.com/support)
- Product Download: [www.comsol.com/product-download](http://www.comsol.com/product-download)
- Product Updates: [www.comsol.com/support/updates](http://www.comsol.com/support/updates)
- COMSOL Blog: [www.comsol.com/blogs](http://www.comsol.com/blogs)
- Discussion Forum: [www.comsol.com/forum](http://www.comsol.com/forum)
- Events: [www.comsol.com/events](http://www.comsol.com/events)
- COMSOL Video Gallery: [www.comsol.com/videos](http://www.comsol.com/videos)
- Support Knowledge Base: [www.comsol.com/support/knowledgebase](http://www.comsol.com/support/knowledgebase)

Part number: CM025202

# Contents

---

The Thermodynamic Properties Database.....	5
Engine Coolant Properties .....	6
Introduction.....	6
Model Definition.....	6
Results .....	8
Reference.....	15
Modeling Instructions: Engine Coolant Properties .....	16
.....	52
Heat Pipe with Accurate Liquid and Gas Properties.....	53
Model Definition.....	53
Results and Discussion.....	56
References.....	59
Modeling Instructions: Heat Pipe with Accurate Liquid and Gas Properties .....	59



## The Thermodynamic Properties Database

---

This introduction booklet demonstrates the use of the built-in thermodynamic properties database in the Liquid & Gas Properties Module. The purpose of this database is to calculate thermodynamic and transport properties for pure solutions and mixtures of chemical compounds. Properties such as enthalpy of formation, heat capacity, thermal conductivity, density, and diffusivity can be computed using a range of models. These properties can be calculated for fluids consisting of a single gas phase or a single liquid phase and for liquid-liquid, vapor-liquid, and vapor-liquid-liquid systems. For multiphase systems, the equilibrium composition can also be calculated, for example, to calculate the phase envelope for a liquid mixture at equilibrium with its vapor phase (flash calculations).

- The first example in this booklet, namely [Engine Coolant Properties](#), shows the use of the thermodynamic properties database in the investigation of a liquid coolant for internal combustion engines. A mixture of ethylene glycol and water is studied, and the built-in thermodynamics functionality is used to show how the boiling point, density, viscosity, thermal conductivity, and heat capacity all depend on the composition of the coolant mixture, and how changes in these properties affect the cooling process
- The second example considers a tubular [Heat Pipe with Accurate Liquid and Gas Properties](#). A copper cylinder, with an inner cavity and a porous water saturated copper wick, allows evaporated water to flow from the hot end to the cold, depositing its latent heat. The effect of loss of working fluid is analyzed, and the relative importance of transfer of vapor at normal operating conditions is evaluated.

The thermodynamic properties database in the Liquid & Gas Properties Module can be combined with any module that deals with transport in fluids — for example, the CFD Module, Mixer Module, Heat Transfer Module, Pipe Flow Module, and Subsurface Flow Module.

# Engine Coolant Properties

---

## *Introduction*

The engine block of a car includes a cooling jacket to remove excess heat from combustion. The cooling jacket consists of open spaces in the cylinder block and the cylinder head. When the engine is running, a coolant fluid is pumped through the jacket to keep the engine from overheating. Optimizing the heat removal is important to minimize coolant boiling, prevent engine failure, and, more recently, improve overall efficiency through waste heat recovery. This example demonstrates how the Thermodynamics feature can be used to evaluate the performance of different engine coolants.

Although pure water works well as a coolant, to prevent freezing at low temperatures, a mixture of ethylene glycol and water is normally used to lower the freezing point. The Thermodynamics feature is used here to show how the boiling point, density, viscosity, thermal conductivity, and heat capacity also depend on the composition of the coolant mixture and how changes in these properties affect the cooling process.

## *Model Definition*

[Figure 1](#) shows the flow pattern inside the cooling jacket of a representative four cylinder engine. Solving a fully coupled nonisothermal turbulent flow problem with temperature-, pressure-, and composition-dependent coolant properties in this complex geometry typically requires a significant number of computer hours. One approach to obtain a reliable approximate solution in a shorter time is to use the functionality available in the Thermodynamics feature to investigate the coolant property behavior and determine where simplifying assumptions can be made. The consequences of these assumptions can be investigated efficiently in a

simplified geometry in order to provide confidence in their use in more complex geometries.

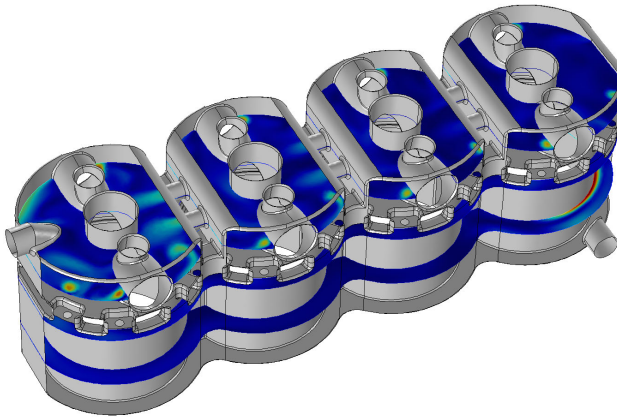


Figure 1: The coolant flow inside the coolant jacket of a four-cylinder engine.

Here, a simplified 2D axially symmetric geometry, shown in [Figure 2](#), is considered as an engine coolant test apparatus. Coolant is introduced at a specified flow rate in the bottom of the device, the coolant hits a solid steel part and is then deflected into a larger flow domain. A heat flux is applied on the outer boundary of the larger section. The resulting temperature is measured at steady state in the solid structure near the coolant outflow at the top.

To solve for the fluid flow and heat transfer in the test apparatus, the current model uses the Single-Phase Flow and the Heat Transfer in Fluids interfaces. The interfaces are coupled using a Nonisothermal Flow multiphysics feature, and the  $k-\epsilon$  model is used to model the fluid flow turbulence.

The properties of the coolant fluid are defined using the Thermodynamics feature. This is done by first defining and adding a Thermodynamic System node to the Thermodynamics feature. Included in the Thermodynamic System are the relevant chemical species, in this case ethylene glycol and water. The Thermodynamic system node in turn can be used to compute property functions for thermodynamic properties and transport properties, both for the pure species and for the resulting mixture. In this case, functions for the density, the viscosity, the thermal conductivity and the heat capacity of the coolant mixture are created.

The analysis of the coolant properties is performed in three steps. First, the mixture properties are evaluated by plotting the functions created by the Thermodynamic System. Then the phase envelope of the coolant vapor-liquid system is visualized by plotting the equilibrium temperatures (for boiling and condensation) as a function of the composition. The required equilibrium

functions are defined by adding an Equilibrium Calculation feature to the Thermodynamic System. Using the equilibrium functions the phase envelope for two different pressures are compared.

The fluid flow and heat transfer of the coolant mixture inside the test apparatus are then solved for. Results for pure water, and a 50 volume percent mixture of ethylene glycol in water are compared. For these chemicals, a 50 volume percent mixture corresponds to 52.7 mass percent of ethylene glycol. Finally the results are used to compute average mixture properties.

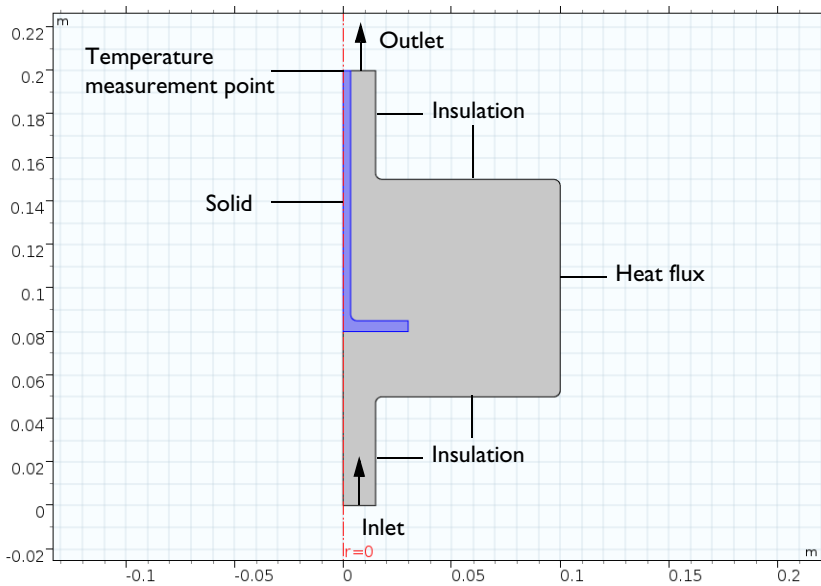


Figure 2: Axially symmetric engine coolant test apparatus.

## Results

Figure 3 shows the temperature and composition dependence of the heat capacity. Similar graphs are generated for density, viscosity, and thermal conductivity. Studying these graphs reveals that the addition of ethylene glycol increases the density and viscosity, but decreases the thermal conductivity and heat capacity when compared with pure water. It should be expected that a 50 volume percent



mixture will yield a higher pressure drop and require a higher flow rate to achieve the same cooling effect as that of pure water.

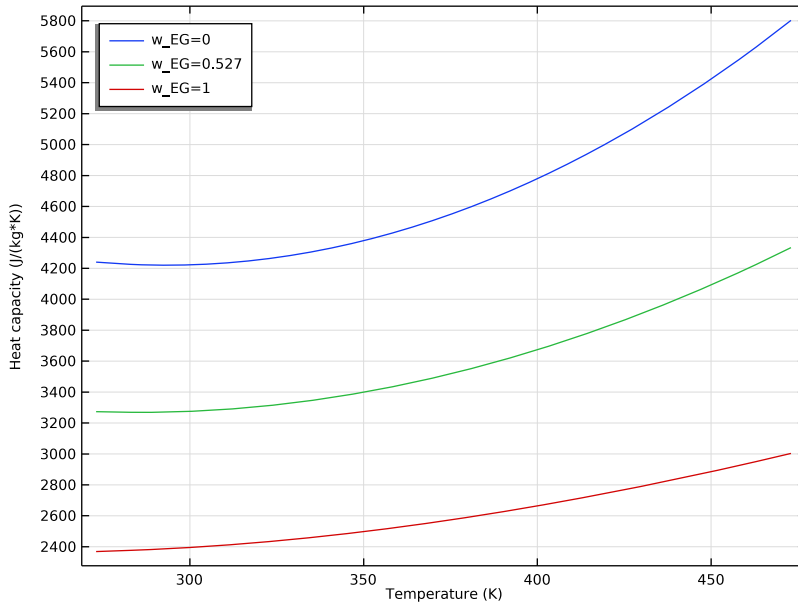


Figure 3: Heat capacity as a function of temperature and composition for ethylene glycol water mixtures.

Figure 4 shows the phase envelope for ethylene glycol-water mixtures produced using the Equilibrium Calculation feature of the Thermodynamic System. A car coolant system typically operates at about 2 atm pressure. Here we can see that a

50 volume percent (24.4 mole percent) mixture should boil at a temperature slightly higher than 400 K at this pressure.

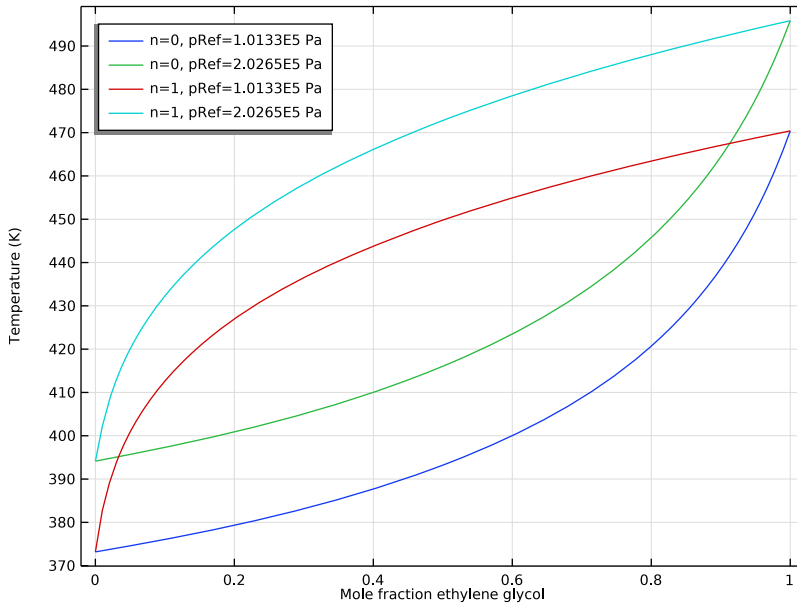


Figure 4: Phase envelope for the equilibrium temperature of ethylene glycol-water mixtures at two pressures.

Figure 5 shows the flow pattern inside the test apparatus with water entering at 1 m/s. The coolant flow of 42 l/min and a heat input of 50 kW used here in the

test apparatus are on the same order of magnitude as in a conventional car cooling system.

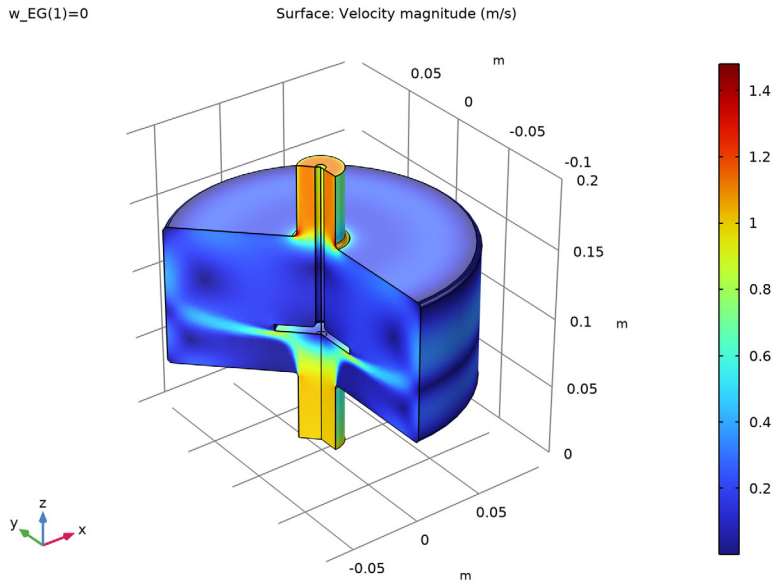


Figure 5: Flow patterns inside the test apparatus with water at 1 m/s.

As expected, Figure 6 shows that an ethylene glycol-water mixture will provide less cooling than pure water at a fixed flow rate. About 15 percent more coolant flow is required to produce the same cooling as when using pure water. It can also be

seen that some boiling of the coolant (at  $T > 400$  K) is expected in the recirculation zones in the outer corners of the apparatus.

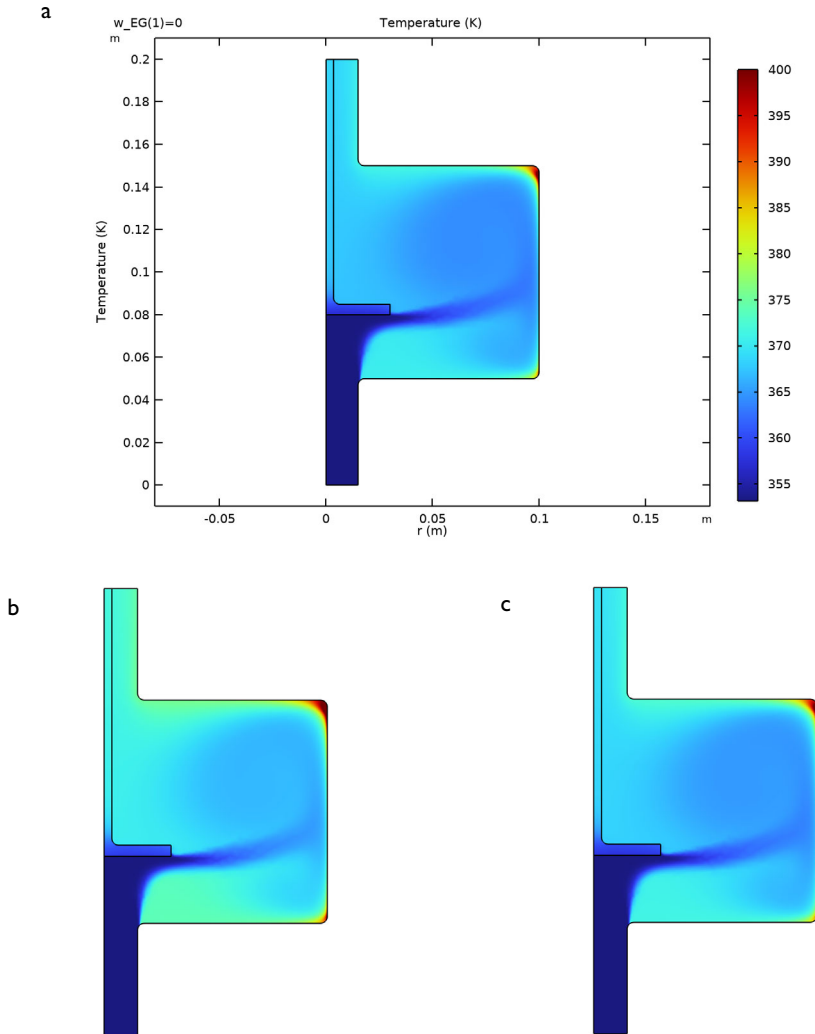


Figure 6: Temperature within the test apparatus for three cases: (a) water at 1 m/s, (b) 50 volume percent ethylene glycol at 1 m/s, and (c) 50 volume percent ethylene glycol at 1.15 m/s.

The table below provides a comparison of results for pressure drop, outlet temperature, and outlet density.

### SIMULATION RESULTS

WEIGHT FRACTION, ETHYLENE GLYCOL	VELOCITY (M/S)	PRESSURE DROP (PA)	OUTLET TEMPERATURE (K)	OUTLET DENSITY (KG/M)
0	1	554	370	961
0.527	1	626	373	1007
0.527	1.15	822	371	1009
0.527 <sup>1</sup>	1	608	373	1010

<sup>1</sup> Using constant mixture properties.

Considering the graphical results for the various coolant properties, it might be best to use approximate averages for the relatively small temperature range of 353–400 K. In [Figure 7](#) the resulting heat capacity for the pure water and the two ethylene glycol-water mixture cases is plotted. As seen before, the heat capacity differs significantly when comparing pure water and the mixture. But, the individual variation for each coolant however is seen to be small, about 2% for this mixture property and location. Analyzing the density in the same manner, the variation can be seen to be in the same order of magnitude.

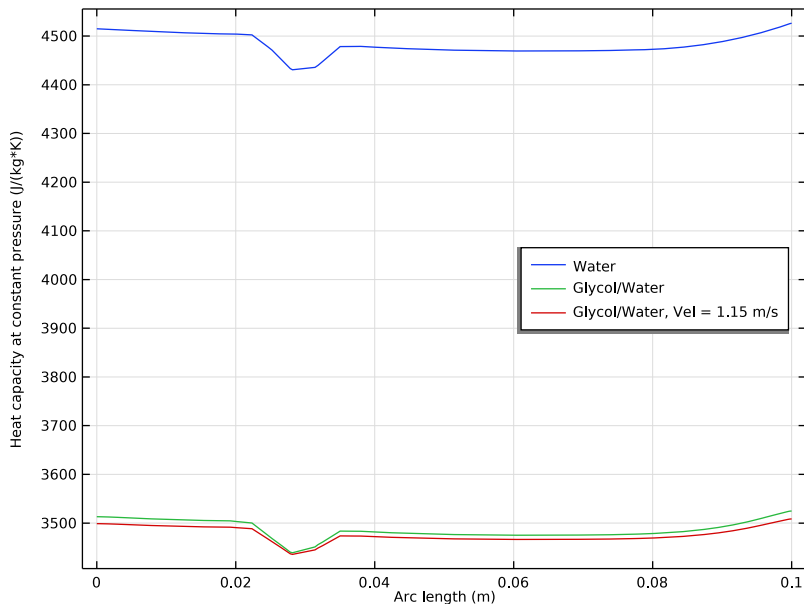


Figure 7: Coolant heat capacity plotted along a vertical cut line at half the radius of the test apparatus chamber.

Using the solution for a mixture with 50 volume percent ethylene glycol in water, the following average values are computed: density =  $1010 \text{ kg/m}^3$ , viscosity =  $9.07 \cdot 10^{-4} \text{ Pa}\cdot\text{s}$ , thermal conductivity =  $0.574 \text{ W}/(\text{m}\cdot\text{K})$ , and heat capacity =  $3486 \text{ J}/(\text{kg}\cdot\text{K})$ . [Figure 8](#) shows a comparison of the temperature results obtained using these approximations with those using the fully coupled temperature-dependent properties in our test device. The similarity between these results is sufficient to justify the use of the approximate average values in a cooling jacket model with a realistic geometry. Solving the flow and heat transfer equations requires considerably less computational effort for the constant average property value case.

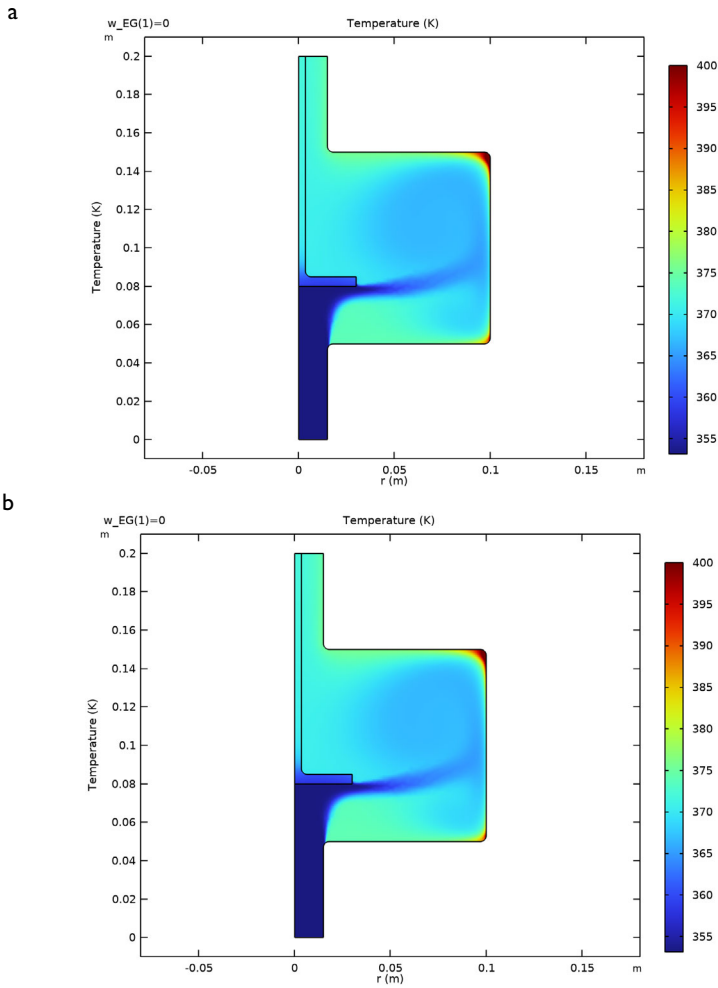


Figure 8: Comparison of temperature within our test apparatus for 50 volume percent ethylene glycol in water at 1 m/s using: (a) temperature dependent properties, (b) approximate average properties.

## Reference

1. [http://www.engineeringtoolbox.com/ethylene-glycol-d\\_146.html](http://www.engineeringtoolbox.com/ethylene-glycol-d_146.html)

## Modeling Instructions: Engine Coolant Properties

The following step-by-step instructions guide you through the process of setting up a Thermodynamic System, build a geometry, and simulate the engine coolant physics using the Single Phase Flow, and Heat Transfer in Fluids interfaces. The Thermodynamic System is created from the COMSOL thermodynamic database, which provides all necessary thermodynamic and transport properties.



### Model Wizard

---

**Note:** These instructions are for the user interface on Windows but apply, with minor differences, also to Linux and macOS.



- 1 To start the software, double-click the COMSOL icon on the desktop. When the software opens, choose to use the Model Wizard to create a new COMSOL model, or choose Blank Model to create one manually. For this tutorial, click the Model Wizard button.




If COMSOL is already open, you can start the Model Wizard by choosing

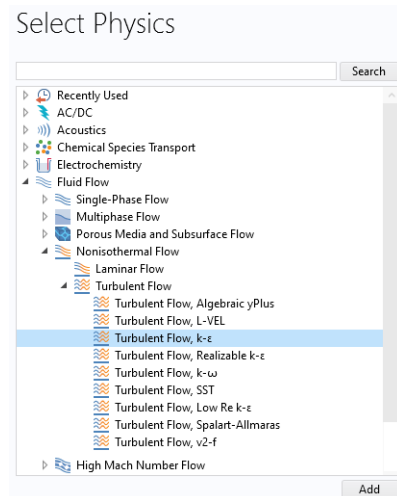
 New from the File menu and then click  Model Wizard.

The Model Wizard guides you through the first steps of setting up a model. The next window lets you choose the dimension of the modeling space.

- 2 In the Select Space Dimension window, click  2D Axisymmetric.

In the Select Physics tree, expand the nodes Fluid Flow>Nonisothermal Flow>Turbulent Flow and then double-click  Turbulent Flow, k-ε to add it to the Added physics interfaces list. You can also right-click Turbulent Flow, k-ε and choose  Add Physics.



- 3 Click  Study. In the Select Study tree, under General Studies, select  Stationary.
- 4 Click  Done.



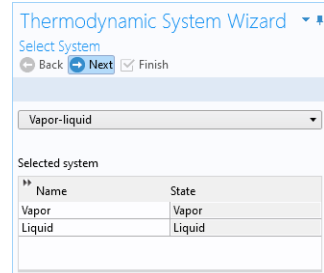


## Thermodynamics


Start by adding a Thermodynamic System, which will be used to compute the transport and thermodynamic properties of the coolant.

- 1 In the Model Builder window, right-click  Global Definitions and choose Thermodynamics >  Thermodynamic System. This opens the Thermodynamic System Wizard. In the first step, select the phases in the system. You can choose among the following systems:

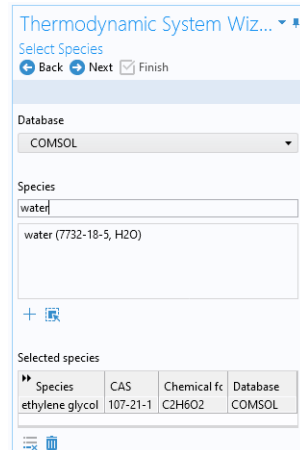
- Gas
- Liquid
- Vapor-liquid
- Vapor-liquid-liquid
- Liquid-liquid




- 2 In the Select System window, choose Vapor - liquid from the list. In this case a two-phase system is needed to study the phase envelope of the coolant mixture. When solving for the heat transfer in the test apparatus, the mixture will be assumed to remain in the liquid phase.


- 3 Click  Next in the window toolbar to proceed to the next step of the wizard.


- 4 In the Select Species window, locate the Species list and type ethylene glycol in the Species filter field. While typing, the available species are dynamically updated in the list below.





- 5 Select ethylene glycol (107-21-1, C2H6O2) in the list of available species and click  Add Selected to add it to the Selected species list.

- 6 Perform the same steps (step 4 and 5) to add water, namely:

- Type water in the Species filter field.
- Select water (7732-18-5, H2O) in the list of available species.
- Click  Add selected.

- 7 Click  Next in the window toolbar to proceed to the last step of the Thermodynamic System Wizard. Here you select the models used to compute thermodynamic and transport properties.

- 8 In the Select Thermodynamic Model window, choose UNIFAC VLE from the list. Then check the Advanced options check box and set Liquid volume to Ideal mixing. Complete the wizard by clicking  Finish in the window toolbar.

This adds a Thermodynamic System node under  Thermodynamics in the Model Builder window. The default label (name) of the system is Vapor-Liquid System 1, indicating the phases in the system.






For more information on how to choose the thermodynamic model see [Selecting the Right Thermodynamic Model](#) in the *Liquid & Gas Module User's Guide*.




## Generate Material

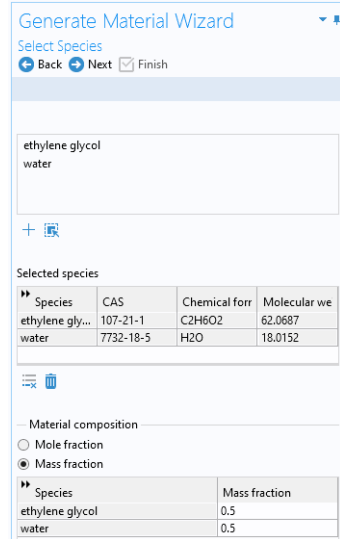
---

When modeling a fixed composition mixture, it is convenient to use a material because the default behavior in physics interfaces is to use properties from the domain material. Ready-to-use materials can be selected from the available Material libraries, or generated from Thermodynamics. In this model, first generate a material node for the coolant, using the Thermodynamic System (Vapor-Liquid System 1). Later on we will use the Material library to create a Material for the Solid Parts of the geometry. Both these materials are then assigned to the relevant selections in the geometry. This allows the physics interfaces in these selections to access the material properties

Use Thermodynamic System (Vapor-Liquid System 1) to create a Material node for the properties of the mixture.




- 1 In the Model Builder window, under  Global Definitions >  Thermodynamics right-click  Vapor-Liquid System 1 (pp1) and choose  Generate Material. This opens the Generate Material Wizard.
- 2 In to the Select Phase window, choose `Liquid` from the list.  Click Next in the window toolbar.

- 3 In the Select Species window, notice how both species are already added to the Selected species list.
- 4 In the Material composition subsection, click the Mass fraction button and keep the default composition. The composition will be redefined later in the generated Material.
- 5 Click  Next in the window toolbar.
- 6 In the Select Properties window, use the default properties.
- 7 Click  Next in the window toolbar.
- 8 In the Define Material window, add the material to Component 1. Also keep the Function type set to Thermodynamics. This means that material properties will directly use functions defined by the Vapor-Liquid System. It is also possible to create interpolation functions and use these in the material properties.
- 9 Click  Finish in the window toolbar to create the material.



## Global Definitions

Load the needed parameters from a file. Alternatively, you could have created the parameters in the Parameter window directly.

- 1 In the Model Builder window, under  Global Definitions click  Parameters 1.
- 2 In the Settings window for Parameters, locate the Parameters section and click  Load from File.
- 3 Browse to the model's Application Libraries folder and double-click the file engine\_coolant\_properties\_parameters.txt.

The parameters from the txt-file are added to the table.

Settings Parameters

Label: Parameters 1


Name	Expression	Value	Description
w_EG	0.527	0.527	Mass fraction, ethylene g...
w_W	1-w_EG	0.473	Mass fraction, water
Vel	1.0[m/s]	1 m/s	Pipe inlet velocity
r_p	1.5[cm]	0.015 m	Pipe radius
L_p	20[cm]	0.2 m	Pipe length
r_c	10[cm]	0.1 m	Chamber radius
L_c	10[cm]	0.1 m	Chamber length
zpos_c	Lp/4	0.05 m	Chamber position along...
r_s1	3[cm]	0.03 m	Solid part 1, radius
L_s1	0.5[cm]	0.005 m	Solid part 1, length
r_s2	0.35[cm]	0.0035 m	Solid part 2, radius
L_s2	L_p-zpos_s	0.12 m	Solid part 2, length
zpos_s	zpos_c+0.3*L_c	0.08 m	Solid parts position along...
Tc	353.15[K]	353.15 K	Coolant temperature
pRef	2[atm]	2.0265E5 Pa	Coolant pressure
n	0	0	Phase fraction
P0	50[kW]	50000 W	Combustion heat flux

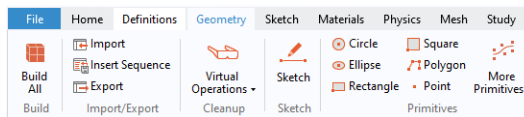
## Geometry 1


---

Now that we have defined our thermodynamic system and used that to generate a material, it is time to build the geometry for the axially symmetric engine coolant test apparatus (see [Figure 2](#)).



### PIPE

- 1 In the Model Builder window, expand the Component 1 (comp1)>Geometry 1 node, right-click Geometry 1 and choose Rectangle. As an option, you can also click  Rectangle in the Geometry ribbon tab.





- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type  $r_p$  (parameter name for pipe radius).
- 4 In the Height text field, type  $l_p$  (parameter name for pipe length). Both  $r_p$  and  $l_p$  are defined in the Parameters node, see section [Global Definitions](#).
- 5 Click  Build Selected.



### CHAMBER

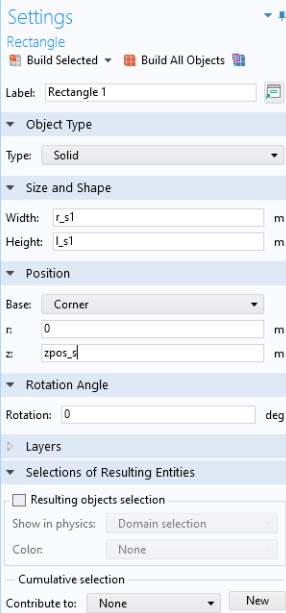
- 1 In the Geometry toolbar, click  Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type  $r_c$  (parameter name for chamber radius).
- 4 In the Height text field, type  $l_c$  (parameter name for chamber length).
- 5 Locate the Position section. In the z text field, type  $zpos_c$  (chamber position along the pipe).
- 6 Click  Build Selected.

## SOLID PART 1

- 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\_s1.
- 4 In the Height text field, type l\_s1.
- 5 Locate the Position section. In the z text field, type zpos\_s.
- 6 Click  Build Selected.

## SOLID PART 2

- 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\_s2.
- 4 In the Height text field, type l\_s2.
- 5 Locate the Position section. In the z text field, type zpos\_s.
- 6 Click  Build Selected.



Settings  
Rectangle

Build Selected Build All Objects

Label: Rectangle 1

Object Type  
Type: Solid

Size and Shape  
Width: r\_s1 m  
Height: l\_s1 m

Position  
Base: Corner  
r: 0 m  
z: zpos\_s m

Rotation Angle  
Rotation: 0 deg

Layers

Selections of Resulting Entities

Resulting objects selection

Show in physics: Domain selection

Color: None

Cumulative selection

Contribute to: None New

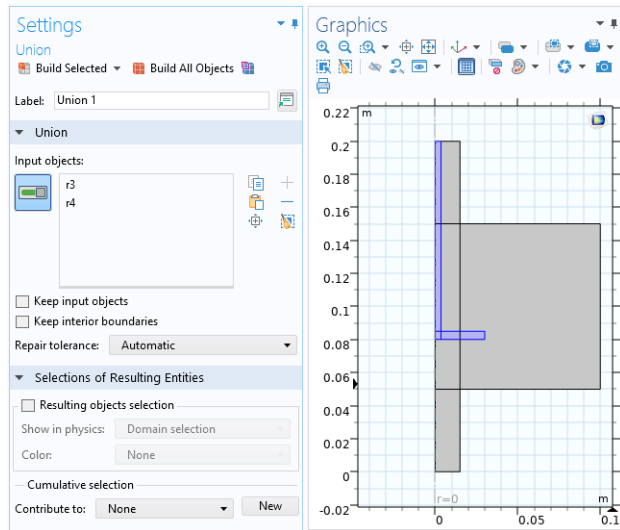
## UNITE THE SOLID PARTS

- 1 In the Geometry toolbar, click  Booleans and Partitions and choose  Union.

2 In the Settings window for Union, locate the Union section and select the objects r3 and r4 only. Do so by clicking the Solid part 1 and Solid part 2 in the Graphics window.

3 Clear the Keep interior boundaries check box.

4 Click  Build Selected.



Now perform the corresponding actions to unite the pipe with the chamber.

#### UNITE THE PIPE AND THE CHAMBER

1 In the Geometry toolbar, click  Booleans and Partitions and choose  Union.



2 Select the objects r1 (pipe) and r2 (chamber) in the Graphics window.

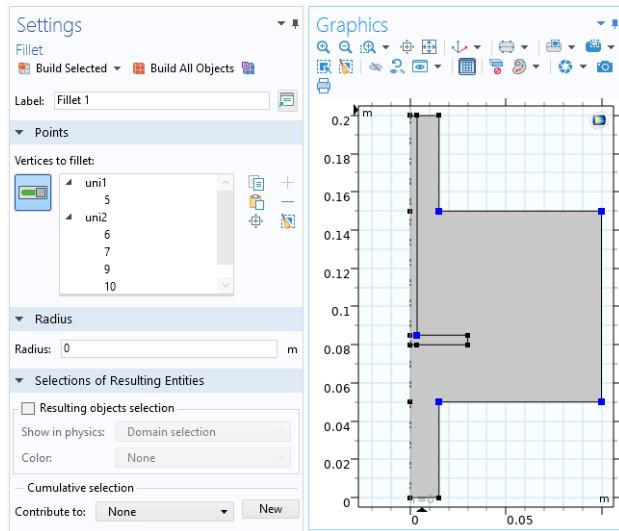
3 In the Settings window for Union, locate the Union section and clear the Keep interior boundaries check box.

4 Click  Build Selected.

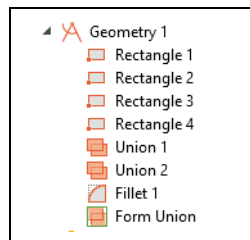
Continue by adding round corners to the geometry.

## FILLET

- 1 In the Geometry toolbar, click  Fillet.
- 2 On the object uni1 (Solid parts), select Point 5 only. Select the point by clicking in the Graphics window and checking the point number in the Points section, Vertices to fillet, found in the Fillet Settings window.
- 3 On the object uni2 (pipe and chamber), select Points 6, 7, 9, and 10 only.
- 4 In the Settings window for Fillet, locate the Radius section.
- 5 In the Radius text field, type 0.3 [cm].
- 6 Click  Build All Objects.




The Geometry node in the Model Builder window should now look like this:



## Study 1: Mixture Properties Parameterization



---

Now, having built the geometry for our system, compute and plot the properties of the glycol-water coolant as defined by the Thermodynamic System.

- 1 In the Model Builder window, click Study 1 .
- 2 In the Settings window for Study, type Study 1: Mixture properties parameterization in the Label text field.
- 3 Locate the Study Settings section. Clear the Generate default plots check box.

### STEP 1: STATIONARY

Start this investigation by performing an Auxiliary sweep study, where you study the effect of varying mass fraction of ethylene, and varying coolant temperature.

- 1 In the Model Builder window, under Study 1: Mixture properties parameterization click  Step 1: Stationary.
- 2 In the Settings window for Stationary, click to expand the Study Extensions section and select the Auxiliary sweep check box.
- 3 Click  Add twice (button located below the table). This adds two rows to the table.
- 4 In the table, edit the two rows by selecting the below parameter names from the drop down menu, and enter parameter values so that the table looks like this:

PARAMETER NAME	PARAMETER VALUE LIST	PARAMETER UNIT
w_EG (Mass fraction, ethylene glycol)	0 0.527 1	
Tc (Coolant temperature)	range(273,10,473)	K




- From the Sweep type list, choose All combinations.

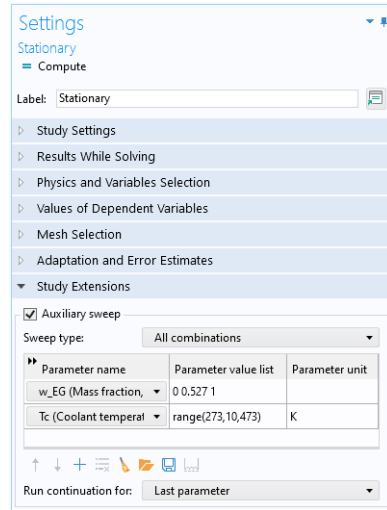
During this first Auxiliary sweep study there is no need to solve for turbulent flow and heat transfer. Change the settings to omit solving for these interfaces.

- Locate the Physics and Variables Selection section. In the Physics interface table, clear the Solve for check boxes for Turbulent Flow, k- $\epsilon$  (spf) and Heat Transfer in Fluids (ht).

- In the Multiphysics couplings table, clear the Solve for check box for Nonisothermal Flow 1 (nitf1).

The parametric solver has now been set up to compute function values for pure water, a 50 volume percent mixture of ethylene glycol and water, and pure ethylene glycol. Furthermore, the solver will compute function values for a temperature range from 273 K to 473 K.





- In the Home toolbar (or in the Settings window for Stationary), click  Compute.



## Results

Now, inspect the mixture properties resulting from the study. Do so by creating a plot group and plot the results. Start with the density.

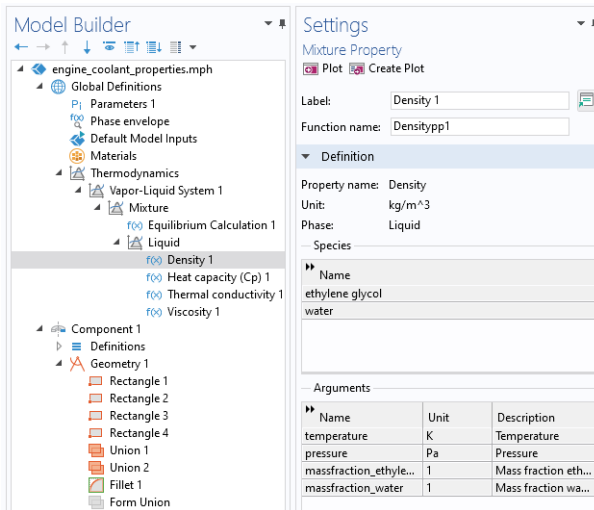
### DENSITY

- In the Home toolbar, click  Add Plot Group and choose  1D Plot Group.
- In the Settings window for 1D Plot Group, type Density in the Label field.
- Click to expand the Title section. From the Title type list, choose None.
- In the Density toolbar, click  Global. This adds a Global plot to the Density plot group.
- In the Settings window for Global, locate the y-Axis Data section. Click Replace Expression  in the upper-right corner and, in the window that opens, either search for Densitypp1, or choose Densitypp1 (...) -Density 1

under Model>Global definitions>Functions. Double-click the expression to add it to the table.

- 6 In the table, enter the following arguments for the function:  
Tc, pRef, w\_EG, w\_W. Make sure to add the arguments in this order. This is the order defined in the Mixture node.
- 7 Locate the Legends section and find the Include subsection. Clear the Description check box.

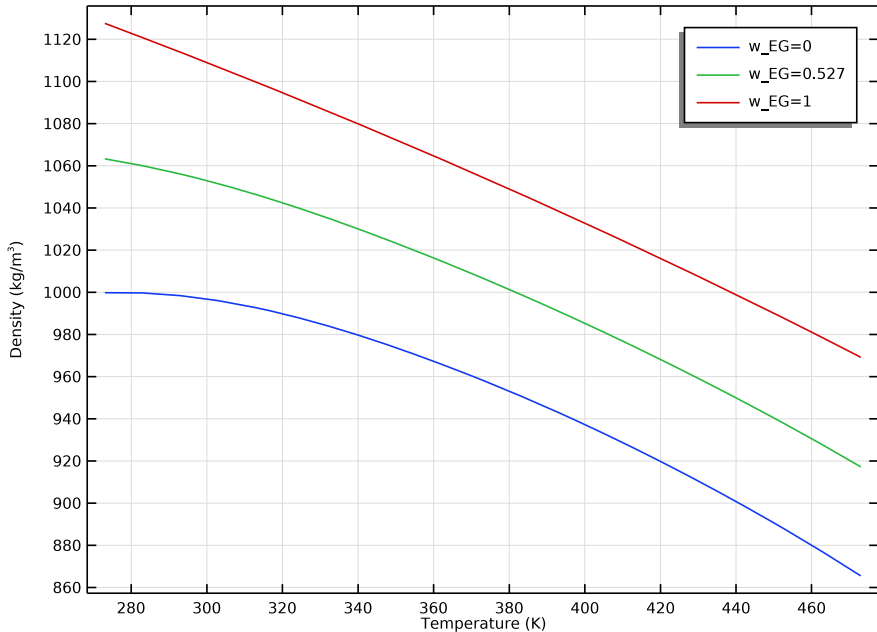
**Note:** The function named Densitypp1, can be found in the Model Builder window, under Global Definitions>Thermodynamics. Click the mixture density function, labeled Density 1, and you will find the Function name in the Settings window. The function was created by the Thermodynamic System and can be used in any physics interface.



Now, return to the plot group to improve the plot settings.





- 1 In the Model Builder window, click Density.
- 2 In the Settings window, locate the Plot Settings section.
- 3 Select the x-axis label check box and type Temperature (K).
- 4 Select the y-axis label check box and type Density (kg/m<sup>3</sup>).
- 5 Click Plot.

The resulting plot shows the coolant density for the three compositions.



Perform the same steps to complete the plots for viscosity, thermal conductivity, and heat capacity.

## VISCOSITY



- 1 In the Home toolbar, click  Add Plot Group and choose  1D Plot Group.
- 2 In the Settings window for 1D Plot Group, type *Viscosity* in the Label field.
- 3 In Title section, from the Title type list, choose None.
- 4 In the Viscosity toolbar, click  Global. This adds a Global plot to the plot group.
- 5 In the Settings window for Global, locate the y-Axis Data section. Click Replace Expression  in the upper-right corner and in the window that opens, search for *Viscositypp1*, or choose *Viscositypp1(...)-Viscosity 1* under Model>Global definitions>Functions. Double-click the expression to add it to the table.

6 In the table, enter the following arguments for the function:

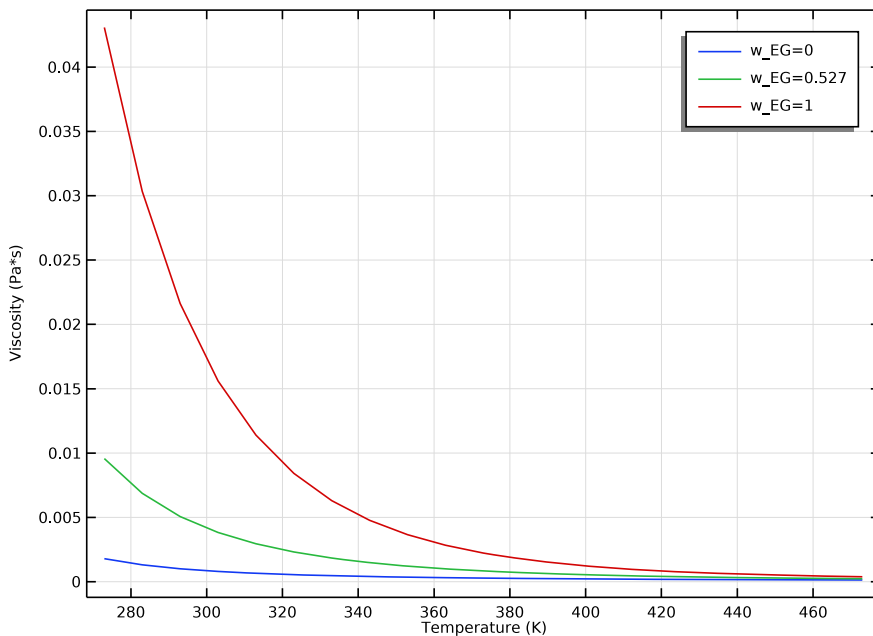
EXPRESSION	UNIT	DESCRIPTION
Viscositypp1(Tc,pRef,w_EG,w_W)		

7 Locate the Legends section and find the Include subsection. Clear the Description check box.





Now, return to the plot group to fine tune the plot settings.

- 1 In the Model Builder window, click  Viscosity.
- 2 In the Settings window, locate the Plot Settings section.
- 3 Select the x-axis label check box and type Temperature (K).
- 4 Select the y-axis label check box and type Viscosity (Pa·s).
- 5 Click  Plot.

The resulting plot shows the coolant viscosity for the three compositions.





## THERMAL CONDUCTIVITY

- 1 In the Home toolbar, click  Add Plot Group and choose  1D Plot Group.
- 2 In the Settings window for 1D Plot Group, type Thermal Conductivity in the Label field.
- 3 In the Title section, from the Title type list, choose None.
- 4 In the Thermal Conductivity toolbar, click  Global.
- 5 In the Settings window for Global, locate the y-Axis Data section. Click Replace Expression  in the upper-right corner and, in the window that opens, either search for ThermalConductivitypp1, or choose ThermalConductivitypp1(...)-Thermal conductivity 1 under Model>Global definitions>Functions. Double-click the expression to add it to the table.
- 6 In the table, enter the following arguments for the function:

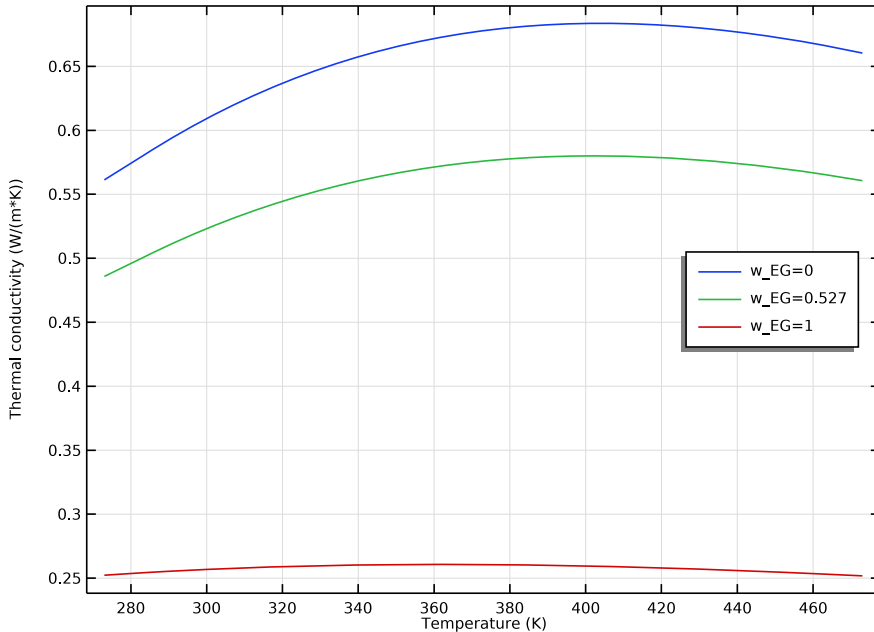
EXPRESSION	UNIT	DESCRIPTION
ThermalConductivitypp1(Tc,pRef,w_EG,w_W)		

- 7 Locate the Legends section and find the Include subsection. Clear the Description check box.





Now, return to the plot group to perfect the plot settings.

- 1 In the Model Builder window, click  Thermal Conductivity.
- 2 In the Settings window, locate the Plot Settings section.
- 3 Select the x-axis label check box and type Temperature (K).
- 4 Select the y-axis label check box and type Thermal conductivity (W/(m\cdot K)).
- 5 Locate the Legend section. From the Position list, choose Middle right.
- 6 Click  Plot.

The resulting plot shows the coolant thermal conductivity for the three compositions.





## HEAT CAPACITY

- 1 In the Home toolbar, click  Add Plot Group and choose  1D Plot Group.
- 2 In the Settings window for 1D Plot Group, type Heat Capacity in the Label field.
- 3 In the Title section, from the Title type list, choose None.
- 4 In the Heat Capacity toolbar, click  Global.
- 5 In the Settings window for Global, locate the y-Axis Data section. Click Replace Expression  in the upper-right corner and, in the window that opens, either search for HeatCapacity, or choose HeatCapacityCp1 (...) - Heat capacity 1 under Model>Global definitions>Functions. Double-click the expression to add it to the table.
- 6 In the table, enter the following arguments for the function:

EXPRESSION	UNIT	DESCRIPTION
HeatCapacityCp1(Tc,pRef,w_EG,w_W)		

- 7 Locate the Legends section and find the Include subsection. Clear the Description check box.

Now, return to the plot group to better the plot settings.





- 1 In the Model Builder window, click  Heat Capacity.
- 2 In the Settings window, locate the Plot Settings section.
- 3 Select the x-axis label check box and type Temperature (K).
- 4 Select the y-axis label check box and type Heat capacity (J/(kg\cdot K)).
- 5 Locate the Legend section. From the Position list, choose Upper left.
- 6 Click  Plot.



The resulting plot is seen in [Figure 3](#).

## Equilibrium Calculation

---

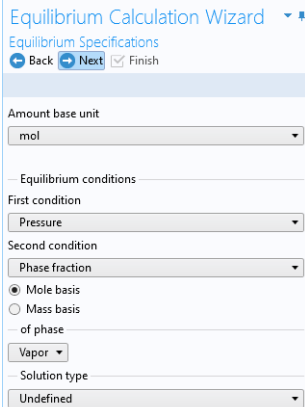
Now, use the thermodynamic system to define an equilibrium function. This will be used to visualize the phase envelope of the coolant mixture.

- 1 In the Model Builder window, under  Global Definitions >  Thermodynamics right-click  Vapor-Liquid System 1 (pp1) and choose  Equilibrium Calculation.

- 1 Go to the Select Species window.
- 2 Click  Add All.
- 3 Click  Next in the window toolbar.
- 4 Go to the Equilibrium Specifications window.
- 5 From the Amount base unit list, choose mol.

Specify the equilibrium conditions. It is possible to create different equilibrium diagrams such as T-x, h-x, P-x, x-y, and so on using two equilibrium conditions. These available conditions are: Temperature, Pressure, Phase fraction, Enthalpy, Entropy, Energy, Density and Volume.

- 6 Find the Equilibrium conditions subsection. From the First condition list, choose Pressure.
- 7 From the Second condition list, choose Phase fraction.



Equilibrium Calculation Wizard ▾

Equilibrium Specifications

Back Next Finish

Amount base unit

mol ▾

Equilibrium conditions

First condition

Pressure ▾

Second condition

Phase fraction ▾

Mole basis  
 Mass basis  
of phase

Vapor ▾

Solution type

Undefined ▾

8 Click Next in the window toolbar.

9 Go to the Equilibrium Function Overview window. The functions to be created and their arguments are now displayed. Note especially `Flash1_1_Temperature`. This equilibrium function will be used to plot the phase envelope

10 Click Finish in the window toolbar. This creates an Equilibrium Calculation node in the Model Builder window.

You can now create an Analytic function to plot the phase envelope.

Name	Unit
Flash1_1_PhaseExist_Vapor	1
Flash1_1_PhaseExist_Liquid	1
Flash1_1_Temperature	K
Flash1_1_PhaseAmount_Vapor	mol
Flash1_1_PhaseAmount_Liquid	mol
Flash1_1_PhaseComposition_Vapor_ethylene_glycol	mol/mol
Flash1_1_PhaseComposition_Vapor_water	mol/mol
Flash1_1_PhaseComposition_Liquid_ethylene_glycol	mol/mol
Flash1_1_PhaseComposition_Liquid_water	mol/mol

Name	Unit	Description
pressure	Pa	Pressure
phasefraction	mol/...	Vapor Mole fract...
ethylene_glycol	mol	Amount ethylen...
water	mol	Amount water

## Analytic Function

Create an Analytic function from the equilibrium function just defined. Analytic functions are convenient since they do not require the actual argument names in an expression when writing the function. Use the Analytic function to change the composition arguments from moles to mole fractions.

1 In the Home toolbar, click Functions and choose Analytic in the Global section.

2 In the Settings window for Analytic, type Phase envelope in the Label text field.

3 In the Function name text field, type `T_x_y`.

4 Locate the Definition section.

- In the Expression text field, type `Flash1_1_Temperature(p,n,w_EG,w_W)`. You may press `Ctrl + space` after having written a few characters to get suggestions.
- In the Arguments text field, type `p, n, w_EG, w_W`.

Settings  
Analytic  
 Plot  Create Plot

Label:

Function name:

Definition

Expression:

Arguments:

Derivatives:

Periodic Extension

Units

Arguments:

Function:



Remember to use the same order of the arguments as defined in the Equilibrium Calculation node.

**5** Locate the Units section.





- In the Function text field, type K.
- In the Arguments text field, type Pa, 1, mol/mol, mol/mol.

**Note:** The Expression `Flash1_1_Temperature` and what arguments to use can be found in the Model Builder window, under Global Definitions>Thermodynamics>Vapor-Liquid System 1 (pp1)>Mixture. Under Mixture, click Equilibrium Calculation. In the Settings window, locate the Functions subsection, as well as the Arguments subsection, both in the Definition section.

## Add Study

---


Add a Study to compute the phase envelope for the mixture, using the defined analytic function.

- 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 and select  Stationary under General in the Select Study tree.
- 4** Click  Add Study. This adds a Study node in the Model Builder window.
- 5** In the Home toolbar, click  Add Study to close the Add Study window.

## Study 2: Phase Envelope Parameterization

---

Add settings to perform an Auxiliary sweep study, where we vary the mass fraction of ethylene glycol, phase fraction, and coolant pressure.

- 1** In the Settings window for Stationary, locate the Study Extensions section.
- 2** Select the Auxiliary sweep check box.
- 3** Click  Add three times. This will add three rows to the table.

4 In the table, enter the following settings (use the associated drop down list):

PARAMETER NAME	PARAMETER VALUE LIST	PARAMETER UNIT
w_EG (Mass fraction, ethylene glycol)	range(0,0.01,1)	
n (Phase fraction)	0 1	
pRef (Coolant pressure)	1[atm] 2[atm]	Pa


5 From the Sweep type list, choose All combinations.

The parametric solver has now been set up to compute function values for the entire range of compositions for the binary mixture, only vapor and only liquid, as well as two pressure levels.

6 Locate the Physics and Variables Selection section. In the Physics interface table, clear the Solve for check boxes for Turbulent Flow, k- $\epsilon$  (spf) and Heat Transfer in Fluids (ht).

7 In the Multiphysics couplings table, clear the Solve for check box for Nonisothermal Flow 1 (nitf1).

Go back to the Study node.

1 In the Model Builder window, click  Study 2.

2 In the Settings window for Study, type Study 2: Phase envelope parameterization in the Label text field.

3 Locate the Study Settings section. Clear the Generate default plots check box.

4 In the Home toolbar, click  Compute.

The function for the equilibrium values has now been computed for the specified parameters. You can plot the phase envelope, as a function of the mole fraction of ethylene glycol, for the two pressures.

## Results

---

Plot the phase envelope, for the two pressures used, as a function of the mole fraction of ethylene glycol.


### PHASE ENVELOPE

1 In the Home toolbar, click  Add Plot Group and choose  1D Plot Group.


2 In the Settings window for 1D Plot Group, locate the Data section.

- 3 From the Dataset list, choose Study 2: Phase envelope parameterization/Solution 2 (sol2).
- 4 Locate the Title section. From the Title type list, choose None.
- 5 Locate the Plot Settings section. Select the x-axis label check box.
- 6 In the associated text field, type Mole fraction ethylene glycol.
- 7 Select the y-axis label check box.
- 8 In the associated text field, type Temperature (K).

## GLOBAL I

- 1 In the 1D Plot Group 5 toolbar, click  Global.
- 2 In the Label field, enter Phase envelope
- 3 In the Settings window for Global, locate the y-Axis Data section.
- 4 In the table, enter the following settings:

EXPRESSION	UNIT	DESCRIPTION
T_x_y(pRef, n, w_EG, w_W)		

- 5 Locate the x-Axis Data section. From the Axis source data list, choose w\_EG.
- 6 In the Phase Envelope toolbar, click  Plot. This generates [Figure 4](#). The figure shows that the phase envelope is moved to higher temperatures when the pressure increases.



To position the legend and rename the plot group use the following steps.

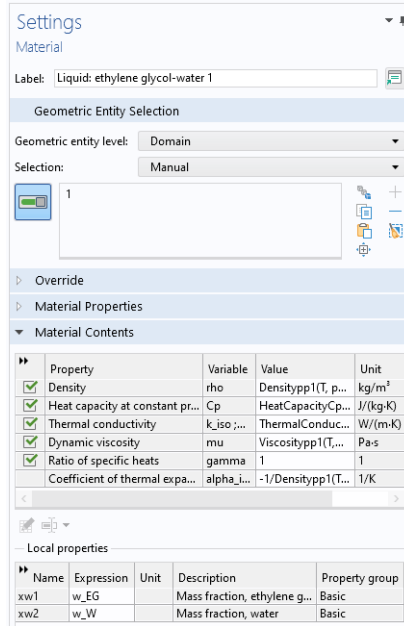
- 1 In the Model Builder window, click Phase Envelope.
- 2 In the Settings window for Phase Envelope, locate the Legend section.
- 3 From the Position list, choose Upper left.
- 4 Click OK.

## Material for Fluid Domain

Apply the material to the fluid domain. Also, use the defined parameters to specify the mixture composition. This makes it easy to vary the composition.

### LIQUID: ETHYLENE GLYCOL-WATER 1 (pp1mat1)



- In the Model Builder window, expand the Component 1 (comp1) >  Materials node, then click the created material  Liquid: ethylene glycol-water 1 (pp1mat1).
- Select Domain 1 only. Select 2 in the Geometric Entity Selection section, click the minus sign.
- Locate the Material Contents section.
- Find the Local properties subsection. In the table, enter the following settings:



NAME	EXPRESSION	UNIT	DESCRIPTION	PROPERTY GROUP
xw1	w_EG		Mass fraction, ethylene glycol	Basic
xw2	w_W		Mass fraction, water	Basic



Now apply boundary conditions for turbulent flow and heat transfer.

## Turbulent Flow, $k-\epsilon$ (spf)



- In the Model Builder window, under  Component 1 (comp1) click  Turbulent Flow,  $k-\epsilon$  (spf).
- Select Domain 1 only. Do so by removing 2 from the Domain Selection section, this can also be achieved by clicking in the Graphics window. Fluid flow will now be solved for Domain 1 only.

Add the inlet and outlet conditions for the fluid flow.

## INLET 1



- 1 In the Physics toolbar, click  Boundaries and choose  Inlet.
- 2 Select Boundary 2, the bottommost boundary.
- 3 In the Settings window for Inlet, locate the Velocity section.
- 4 In the  $U_0$  text field, type  $Ve1$ , the inlet velocity parameter.

## OUTLET 1



- 1 In the Physics toolbar, click  Boundaries and choose  Outlet.
- 2 Select Boundary 11, the boundary at the top of the fluid flow domain.

## Heat Transfer in Fluids (ht)


---

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



## INFLOW 1

- 1 In the Physics toolbar, click  Boundaries and choose  Inflow.
- 2 Select Boundary 2 only, the bottommost boundary.
- 3 In the Settings window for Inflow, locate the Upstream Properties section.
- 4 In the  $T_{ustr}$  text field, type  $Tc$ .
- 5 Select the Specify upstream absolute pressure check box.
- 6 In the  $p_{ustr}$  text field, type  $pRef$ .

## OUTFLOW 1

- 1 In the Physics toolbar, click  Boundaries and choose Outflow.
- 2 Select Boundary 11 only, the boundary at the top of the fluid flow domain.

## HEAT FLUX 1

- 1 In the Physics toolbar, click  Boundaries and choose  Heat Flux.
- 2 Select Boundary 18, the right most boundary adjacent to the fluid flow domain.
- 3 In the Settings window for Heat Flux, locate the Heat Flux section.
- 4 Select Heat rate as the Flux type.
- 5 In the  $P_0$  text field, type  $P0$ .



## SOLID 1

- 1 In the Physics toolbar, click  Domains and choose Solid.
- 2 Select Domain 2 only.

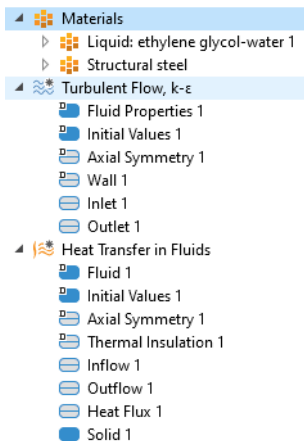
## Add Material

---

Add material properties for the solid steel part from Materials. Do so by using a built-in 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>Structural steel.
- 4 Click Add to Component in the window toolbar.
- 5 In the Home toolbar, click  Add Material to close the Add Material window.
- 6 In the Settings window, select Domain 2 only. Do so by clicking on the Solid part 2 in the Graphics window, and confirming the Selection in the Geometric Entity Selection section.

The Materials node; the Turbulent Flow, k- $\epsilon$  node; and the Heat Transfer in Fluids node should now have the following child nodes in the Model Builder window:






## Add Study

---


Now, add a Stationary study to solve for the fluid flow and heat transfer in the test apparatus, using pure water as the coolant.

You will use two stationary study steps. The first step solves for the fluid flow only. This study step serves as initial conditions for the second step, which in turn solves for both fluid flow and heat transfer.

- 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 Find the Physics interfaces in study subsection. In the table, clear the Solve check box for Heat Transfer in Fluids (ht).
- 5 Click  Add Study in the window toolbar.

## Study 3: Water




---

- 1 In the Settings window for Stationary, locate the Study Extensions section.
- 2 Select the Auxiliary sweep check box.
- 3 Click  Add.
- 4 In the table, enter the following settings:

PARAMETER NAME	PARAMETER VALUE LIST	PARAMETER UNIT
w_EG (Mass fraction, ethylene glycol)	0	

- 5 Click Study 3 in the Model Builder window, and in the Settings window for Study, type **Study 3: Water** in the Label text field.


### STATIONARY 2

- 1 In the Study toolbar, click  Study Steps and choose Stationary >  Stationary.
- 2 In the Settings window for Stationary, click to expand the Study Extensions section.
- 3 Select the Auxiliary sweep check box.
- 4 Click  Add.






5 In the table, enter the following settings:

PARAMETER NAME	PARAMETER VALUE LIST	PARAMETER UNIT
w_EG (Mass fraction, ethylene glycol)	0	

### SOLUTION 3 (SOL3)


1 In the Study toolbar, click  Show Default Solver.

Study step 2 uses the solution for the flow field (in Step 1) as initial conditions. In this case Anderson acceleration can be enabled to reduce the simulation time.

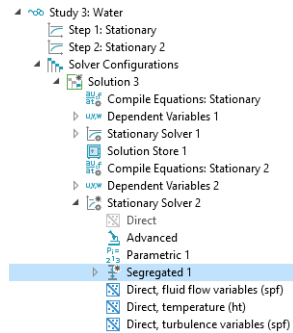
2 In the Model Builder window, expand the  Study 3: Water >  Solver Configurations >  Solution 3 (sol3) >  Stationary Solver 2 node, then click  Segregated 1.

3 In the Settings window for Segregated, locate the General section.

4 From the Stabilization and acceleration list, choose Anderson acceleration.


5 In the Study toolbar, click  Compute.

When the solver finishes the plot group Velocity, 3D (spf) displays the flow field in the apparatus using a revolved dataset.



## Results

Begin by deleting some superfluous plot groups.

1 In the Model Builder window, under  Results, Ctrl-click to select

- Velocity (spf)
- Pressure (spf)
- Wall Resolution (spf)
- Temperature, 3D (ht)
- Isothermal Contours (ht)



2 Right-click and choose Delete.

3 Click Yes.





Create a 2D plot group for the temperature.



## 2D PLOT GROUP I0

- 1 In the Home toolbar, click  Add Plot Group and choose  2D Plot Group.
- 2 Type **Temperature** in the Label text field.
- 3 In the Settings window for 2D Plot Group, locate the Data section.
- 4 From the Dataset list, choose **Study 3: Water/Solution 3 (sol3)**.
- 5 Click to expand the Title section. From the Title type list, choose **Manual**.
- 6 In the Title text area, type **Temperature (K)**.

## SURFACE I





- 1 In the Temperature toolbar, click  Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 In the Expression text field, type **T**.
- 4 In the Temperature toolbar, click  Plot.
- 5 Click to expand the Range section. Select the **Manual color range** check box.
- 6 In the Maximum text field, type **400**.
- 7 In the Temperature toolbar, click  Plot.
- 8 Click the  Zoom Extents button in the Graphics toolbar.

This completes the setup of [Figure 6a](#).

## Add Study

---

Add a new Study to solve for the fluid flow and heat transfer in the test apparatus when using a coolant mixture composed of equal volumes of ethylene glycol and water. Use two study steps: One step using the same inlet velocity as for the pure water case and another study step where the flow rate of the ethylene/water mixture is increased by 15%.



- 1 In the Home toolbar, click  Add Study to open the Add Study window.
- 2 Go to the Add Study window and find the Studies subsection. In the Studies tree, select  General Studies >  Stationary.
- 3 Click Add Study in the window toolbar. If you wish to close the Add Study window, go to the Home toolbar and click  Add Study.

## Study 4: Glycol and Water

---

Enter the settings for the added study. As mentioned, we will use two study steps, both stationary. The output from these studies will result in [Figure 6b](#) and [c](#).

### STEP 1: STATIONARY

- 1 In the Settings window for  Step 1: Stationary, locate the Study Extensions section.
- 2 Select the Auxiliary sweep check box.
- 3 Click  Add twice to add two rows to the table.
- 4 In the table, enter the following settings:

PARAMETER NAME	PARAMETER VALUE LIST	PARAMETER UNIT
w_EG (Mass fraction, ethylene glycol)	0.527	
Vel (Pipe inlet velocity)	1	m/s

Add the second step by duplicating the first one.


### STEP 2: STATIONARY

- 1 Right-click Study 4>Step 1: Stationary and choose Duplicate.
- 2 In the Settings window for Stationary, locate the Study Extensions section.
- 3 In the table, enter the following settings:


PARAMETER NAME	PARAMETER VALUE LIST	PARAMETER UNIT
w_EG (Mass fraction, ethylene glycol)	0.527	
Vel (Pipe inlet velocity)	1.15	m/s

Now enter the needed settings for the two steps.






### STEP 1: STATIONARY

- 1 In the Model Builder window, click  Step 1: Stationary.
- 2 In the Settings window for Stationary, click to expand the Values of Dependent Variables section.
- 3 Find the Initial values of variables solved for subsection. From the Settings list, choose User controlled.
- 4 From the Method list, choose Solution.
- 5 From the Study list, choose Study 3: Water, Stationary 2.

## STEP 2: STATIONARY

1 In the Study toolbar, click  Show Default Solver.

Apply Anderson acceleration also for the cases using ethylene glycol. Note that the previous solution, using water as coolant, was used for the initial conditions.






2 In the Model Builder window, expand the  Study 4 >  Solver Configurations >  Solution 5 (sol5) >  Stationary Solver 1 node, then click  Segregated 1.

3 In the Settings window for Segregated, locate the General section.

4 From the Stabilization and acceleration list, choose Anderson acceleration.


5 In the Model Builder window, collapse the Stationary Solver 1 node.

Now add the same settings to the solver for step 2.

6 In the Model Builder window, expand the  Study 4 >  Solver Configurations >  Solution 5 (sol5) >  Stationary Solver 2 node, then click  Segregated 1.


7 In the Settings window for Segregated, locate the General section.





8 From the Stabilization and acceleration list, choose Anderson acceleration.

9 In the Model Builder window, click  Study 4.


10 In the Settings window for Study, type **Study 4: Glycol and Water** in the Label text field.

11 Locate the Study Settings section. Clear the Generate default plots check box.

12 In the Study toolbar, click  Compute.

13 In the Model Builder window, under  Study 4: Glycol and Water >  Solver Configurations >  Solution 5 (sol5) click  Solution Store 2 (sol6).

14 In the Settings window for Solution Store, type  $V_{e1} = 1.0 \text{ m/s}$  in the Label text field.

15 In the Model Builder window, click  Solution 5 (sol5).





16 In the Settings window for Solution, type  $V_{e1} = 1.15 \text{ m/s}$  in the Label text field.

## Results

---


Let's look at the results from the two study steps just added. Plot the temperature for the glycol/water mixture to reproduce the plots in [Figure 6](#).

## TEMPERATURE PLOT

- 1 In the Model Builder window, under  Results click  Temperature.
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Dataset list, choose Study 4: Glycol and Water /Vel = 1.0 m/s (sol6).
- 4 In the Temperature toolbar, click  Plot. This generates [Figure 6b](#). Note that the temperature in the apparatus in general is higher than for the case of pure water, mainly due to the lower heat capacity of the glycol/water mixture.
- 5 From the Dataset list, choose Study 4: Glycol and Water /Vel = 1.15 m/s (sol5).
- 6 In the Temperature toolbar, click  Plot. This generates [Figure 6c](#). Note now that the increased flow rate reduces the temperature to levels close to the water case.

Next, evaluate the resulting mixture properties within the apparatus. Create three cut line datasets to evaluate the heat capacity throughout the chamber section of the apparatus.

### *Cut Line 2D 1*

- 1 In the Results toolbar, click  Cut Line 2D.
- 2 In the Settings window for Cut Line 2D, locate the Line Data section.
- 3 In row Point 1, set r to  $r_c * 0.5$ .
- 4 In row Point 2, set r to  $r_c * 0.5$ .
- 5 In row Point 2, set z to 1.
- 6 Locate the Data section. From the Dataset list, choose Study 3: Water/Solution 3 (sol3).

Create two more datasets, one for each solution.

### *Cut Line 2D 2*

- 1 Right-click Cut Line 2D 1 and choose Duplicate.
- 2 In the Settings window for Cut Line 2D, locate the Data section.
- 3 From the Dataset list, choose Study 4: Glycol and Water /Vel = 1.0 m/s (sol6).


### *Cut Line 2D 3*

- 1 Right-click Cut Line 2D 2 and choose Duplicate.
- 2 In the Settings window for Cut Line 2D, locate the Data section.


- 3 From the Dataset list, choose Study 4: Glycol and Water /Vel = 1.15 m/s (sol5).

Now use the three datasets to create [Figure 7](#).

## HEAT CAPACITY, CHAMBER CUT LINE PLOT

In the Results toolbar, click  1D Plot Group.

### *Line Graph 1*

- 1 In the 1D Plot Group 11 toolbar, click  Line Graph.
- 2 In the Settings window for Line Graph, locate the Data section. From the Dataset list, choose Cut Line 2D 1.
- 3 Locate the y-Axis Data section. In the Expression text field, type `ht.Cp`.
- 4 Click to expand the Legends section. Select the Show legends check box.
- 5 From the Legends list, choose Manual.
- 6 In the table, enter the following settings:

LEGENDS
Water

- 7 In the 1D Plot Group 11 toolbar, click  Plot.

### *Line Graph 2*

- 1 Right-click Line Graph 1 and choose Duplicate.
- 2 In the Settings window for Line Graph, locate the Data section. From the Dataset list, choose Cut Line 2D 2.
- 3 Locate the Legends section. In the table, enter the following settings:

LEGENDS
Glycol/Water


- 4 In the 1D Plot Group 11 toolbar, click  Plot.

### *Line Graph 3*


- 1 Right-click Line Graph 2 and choose Duplicate.
- 2 In the Settings window for Line Graph, locate the Data section. From the Dataset list, choose Cut Line 2D 3.

3 Locate the Legends section. In the table, enter the following settings:

LEGENDS
Glycol/Water, Vel = 1.15 m/s

4 In the 1D Plot Group 11 toolbar, click  Plot, to visualize the heat capacity along the vertical cut line for the three cases solved (see [Figure 7](#)).

Rename the plot group and improve the plot settings.



- 1 In the Model Builder window, click  1D Plot Group 11.
- 2 In the Settings window for 1D Plot Group, locate the Legend section.
- 3 From the Position list, choose Middle right.
- 4 Locate the Title section. From the Title type list, choose None.
- 5 Right-click 1D Plot Group 11 and choose Rename.
- 6 In the Rename 1D Plot Group dialog box, type Heat Capacity, Chamber Cut Line in the New label text field.
- 7 Click OK.

## Evaluate

---


Compute the average mixture property values. These will be used to compute the flow and heat transfer in the test apparatus, visualized in [Figure 8](#).




### SURFACE AVERAGE 1

- 1 In the Results toolbar, click  More Derived Values and choose Average >  Surface Average.
- 2 In the Settings window for Surface Average, locate the Data section.
- 3 From the Dataset list, choose Study 4: Glycol and Water /Vel = 1.0 m/s (sol6).
- 4 Select Domain 1 only. Do this by pressing Domain 1 in the Graphics window, and verifying the selection in the Selection section in the Surface Average Settings window.
- 5 Locate the Expressions section. In the table, enter the following settings:



EXPRESSION	UNIT	DESCRIPTION
ht.rho	kg/m <sup>3</sup>	Density
ht.Cp	J/(kg*K)	Heat capacity at constant pressure

EXPRESSION	UNIT	DESCRIPTION
ht.krr	W/(m*K)	Thermal conductivity, r component
spf.mu	Pa*s	Dynamic viscosity

6 Click  Evaluate in the toolbar of the Settings window.

The computed averages are found in Table 1 , added under the Tables  node under  Results in the Model Builder. Now store the average property values as parameters.

## PARAMETERS I




- 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, add the following rows:

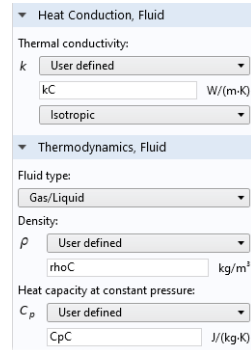
NAME	EXPRESSION	VALUE	DESCRIPTION
rhoC	1010[kg/m^3]	1010 kg/m <sup>3</sup>	Average constant density
CpC	3487[J/kg/K]	3486 J/(kg·K)	Average constant heat capacity
kC	0.574[W/m/K]	0.574 W/(m·K)	Average constant conductivity
muC	9.04e-4[Pa*s]	9.07E-4 Pa·s	Average constant viscosity

Finally, compute the flow and heat transfer in the test apparatus using the averaged mixture properties. First apply the mixture parameters.

## Heat Transfer in Fluids (ht)

Finally compute the flow and heat transfer in the test apparatus using the average values for the mixture properties.




- 1 In the Model Builder window, under  Component 1 (comp1) >  Heat Transfer in Fluids (ht) click  Fluid 1.
- 2 In the Settings window for Fluid, locate the Heat Conduction, Fluid section.
- 3 From the  $k$  list, choose User defined. In the associated text field, type  $kC$ .
- 4 Locate the Thermodynamics, Fluid section. From the Fluid type list, choose Gas/Liquid.
- 5 From the  $\rho$  list, choose User defined. In the associated text field, type  $\rho h o C$ .
- 6 From the  $C_p$  list, choose User defined. In the associated text field, type  $C p C$ .



## Turbulent Flow, k- $\epsilon$ (spf)






---

### FLUID PROPERTIES I

- 1 In the Model Builder window, under  Component 1 (comp1) >  Turbulent Flow, k- $\epsilon$  (spf) click  Fluid Properties 1.
- 2 In the Settings window for Fluid Properties, locate the Fluid Properties section.
- 3 From the  $\mu$  list, choose User defined. In the associated text field, type  $\mu h o C$ .

## 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 Studies tree, select  General Studies >  Stationary.
- 4 Click  Add Study in the window toolbar.
- 5 In the Home toolbar, click  Add Study to close the Add Study window.




## Study 5: Glycol and Water, Constant Properties

---

Add the settings for the final Study in this model. Use the solution from Study 3 as initial values.

### STEP 1: STATIONARY




- 1 In the Settings window for Stationary, locate the Values of Dependent Variables section.
- 2 Find the Initial values of variables solved for subsection. From the Settings list, choose User controlled.
- 3 From the Method list, choose Solution.
- 4 From the Study list, choose Study 3: Water, Stationary 2.
- 5 In the Study Extensions section, enable Auxiliary sweep, add Vel and enter 1.0 as a single entry in Parameter value list.
- 6 In the Model Builder window, click Study 5.
- 7 In the Settings window for Study, type **Study 5: Glycol and Water, Constant Properties** in the Label text field.
- 8 Locate the Study Settings section. Clear the Generate default plots check box.
- 9 In the Home toolbar, click  Compute.

## Results

---

Plot the temperature for the case with average values for the mixture properties.

### TEMPERATURE

- 1 In the Model Builder window click  Results >  Temperature.
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Dataset list, choose Study 5: Glycol and Water, Constant Properties /Solution 7 (sol7).
- 4 In the Temperature toolbar, click  Plot.


You can now compute the outlet temperature, the average pressure drop, and the average outlet density for the four simulations performed:

- Water
- Glycol and Water, Vel = 1.0 m/s








- Glycol and Water, Vel = 1.15 m/s
- Glycol and Water, Constant Properties

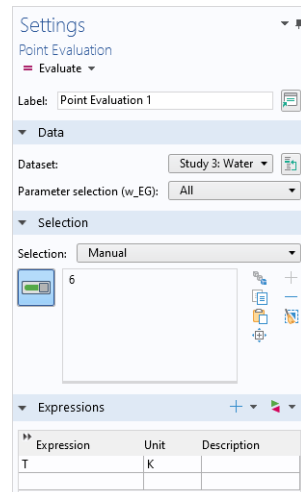
First add a Point Evaluation node to study the outlet temperature.


## POINT EVALUATION: OUTLET TEMPERATURE

- 1 In the Results toolbar, click  Point Evaluation.
- 2 Select Point 6, the uppermost point in the solid domain on the symmetry line. Do so by clicking in the Graphics window and verifying the selection in the Selection section in the Settings window for Point Evaluation.
- 3 Locate the Expressions section and enter the following settings in the table:



EXPRESSION	UNIT	DESCRIPTION
T	K	

- 4 Locate the Data section. From the Dataset list, choose Study 3: Water/Solution 3 (sol3).
- 5 Click  Evaluate at the top of the Settings window.
- 6 From the Dataset list, choose Study 4: Glycol and Water /Vel = 1.0 m/s (sol6).
- 7 Click  next to  Evaluate, then choose New Table to add the evaluation to a new table.
- 8 From the Dataset list, choose Study 4: Glycol and Water /Vel = 1.15 m/s (sol5).
- 9 Click  next to  Evaluate, then choose New Table.
- 10 From the Dataset list, choose Study 5: Glycol and Water, Constant Properties /Solution 7 (sol7).
- 11 Click  next to  Evaluate, then choose New Table.













In this manner, four tables (Table 2–5), containing the outlet temperature, have been added under Tables , one for each of the coolant simulations performed. Next, add a Line Average node to compute the average pressure at the inlet. Since a zero reference pressure is applied at the outlet, the inlet pressure represents the pressure drop over the system.

## LINE AVERAGE: PRESSURE DROP



- 1 In the Results toolbar, click  More Derived Values and choose Average>  Line Average.
- 2 Select Boundary 2 at the bottom of the geometry, corresponding to the inlet.
- 3 In the Settings window for Line Average, locate the Expressions section.
- 4 In the table, enter the following settings:


EXPRESSION	UNIT	DESCRIPTION
p	Pa	

- 5 Locate the Data section. From the Dataset list, choose Study 3: Water/Solution 3 (sol3).
- 6 Click  Evaluate (Table 2 - Point Evaluation 1).  
In the same manner, perform the same steps to derive the pressure drop for the remaining flow solutions. Add the pressure drop to the table containing the outlet temperature for the same case.
- 7 From the Dataset list, choose Study 4: Glycol and Water /Vel = 1.0 m/s (sol6).
- 8 Click  next to  Evaluate, then choose Table 3 - Point Evaluation 1. Click  Evaluate.
- 9 From the Dataset list, choose Study 4: Glycol and Water /Vel = 1.15 m/s (sol5).
- 10 Click  next to  Evaluate, then choose Table 4 - Point Evaluation 1. Click  Evaluate.
- 11 From the Dataset list, choose Study 5: Glycol and Water, Constant Properties /Solution 7 (sol7).
- 12 Click  next to  Evaluate, then choose Table 5 - Point Evaluation 1. Click  Evaluate.









Finally, add another Line Average node to compute the average density at the outlet.

## LINE AVERAGE: OUTLET DENSITY

- 1 In the Results toolbar, click  More Derived Values and choose Average>  Line Average.
- 2 Select Boundary 11, the outlet boundary.
- 3 In the Settings window for Line Average, locate the Expressions section.

4 In the table, enter the following settings (the density defined by the Heat Transfer in Fluids (ht)  interface):

EXPRESSION	UNIT	DESCRIPTION
ht.rho	kg/m <sup>3</sup>	

- 5 Locate the Data section. From the Dataset list, choose Study 3: Water/Solution 3 (sol3).
- 6 Click ▼ next to  Evaluate, then choose Table 2 - Point Evaluation 1. Click  Evaluate.
- 7 From the Dataset list, choose Study 4: Glycol and Water /Vel = 1.0 m/s (sol6).
- 8 Click ▼ next to  Evaluate, then choose Table 3 - Point Evaluation 1. Click  Evaluate.
- 9 From the Dataset list, choose Study 4: Glycol and Water /Vel = 1.15 m/s (sol5).
- 10 Click ▼ next to  Evaluate, then choose Table 4 - Point Evaluation 1. Click  Evaluate.
- 11 From the Dataset list, choose Study 5: Glycol and Water, Constant Properties /Solution 7 (sol7).
- 12 Click ▼ next to  Evaluate, then choose Table 5 - Point Evaluation 1. Click  Evaluate.

## Heat Pipe with Accurate Liquid and Gas Properties

---

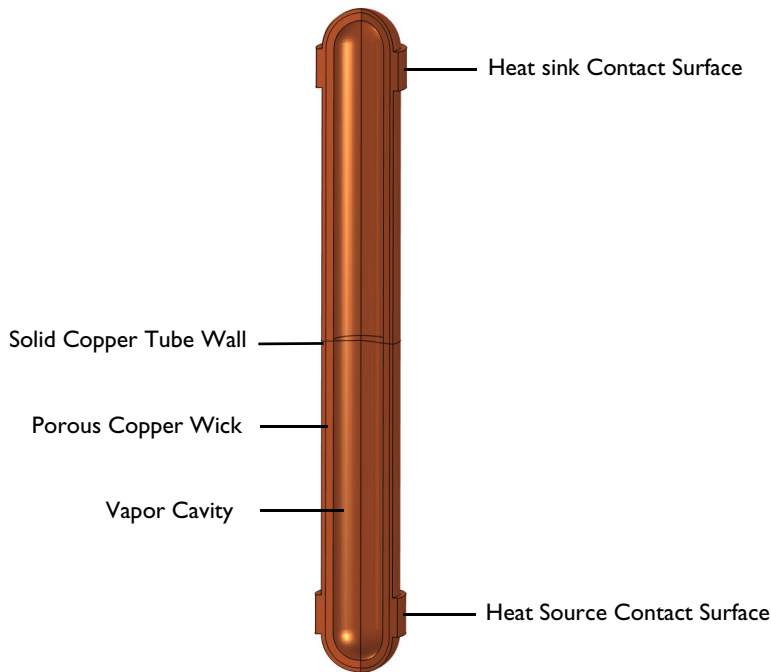
Heat pipes are designed to transfer heat efficiently through vaporization, mass transfer, and condensation of a working fluid. They are found in a wide variety of applications where thermal control is of importance, with cooling of electronics being a prominent example.

Inside a heat pipe, the temperature difference between the hot and cold side along 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. It also shows how thermodynamic properties of water can be obtained from the database in the Liquid & Gas Properties Module. 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 exist in variety of different shapes, with tubular ones probably being the most common kind. 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 which can easily be cooled

by a fan. The geometry used including the different parts is visualized in the figure below.



*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,  $\Delta p_c$ , is usually the limiting factor (Ref. 2):

$$\Delta p_c = \frac{2\sigma}{r_c} \quad (1)$$

Here  $\sigma$  is the surface tension and  $r_c$  is the capillary radius. At the capillary limit, this pressure equals the 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:

$$\Delta p_c = \Delta p_v + \Delta p_g + \Delta p_l \quad (2)$$

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

$$\Delta p_l = \frac{\mu_l L_{\text{eff}} \dot{V}}{KA_w} \quad (3)$$

where  $\mu_l$  is the dynamic viscosity of the liquid,  $L_{\text{eff}}$  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:

$$\dot{V} = \frac{\dot{Q}}{\Delta H_{\text{vap}} \rho} \quad (4)$$

where,  $\dot{Q}$  is the heat transfer rate,  $\rho$  is the density of the liquid,  $\Delta H_{\text{vap}}$  is the latent heat of vaporization (with dimensionality of energy per mass). Inserting Equation 2-4 into Equation 1, and neglecting  $\Delta p_v$  and  $\Delta p_g$  yields:

$$\dot{Q} = 2 \frac{KA_w \Delta H_{\text{vap}} \rho \sigma}{\mu_l L_{\text{eff}} r_c} \quad (5)$$

Evaluating Equation 5 with  $K = 1 \cdot 10^{-9} \text{ m}^2$ ,  $A_w = 1 \cdot 10^{-4} \text{ m}^2$ ,  $\Delta H_{\text{vap}} = 2.5 \cdot 10^6 \text{ J/kg}$ ,  $\rho = 1 \cdot 10^3 \text{ kg/m}^3$ ,  $\sigma = 7 \cdot 10^{-2} \text{ N/m}$ ,  $\mu = 1 \cdot 10^{-3} \text{ Pa}\cdot\text{s}$ ,  $L = 0.15 \text{ m}$ , and  $r = 3.1 \cdot 10^{-5} \text{ m}$ , we obtain 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. As a 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 flow fluid 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.

$$p = p_{\text{H}_2\text{O},\text{sat}}(T) \quad (6)$$

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

$$\mathbf{u}_l = \frac{\mathbf{u}_v \rho_v}{\rho_l} \quad (7)$$

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. 3) 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 9 below.

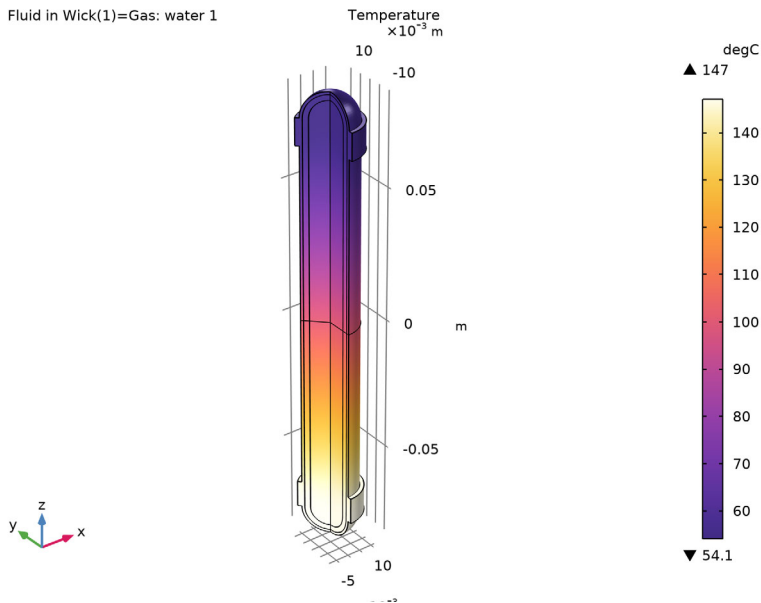


Figure 9: Temperature of heat pipe with dry wick.



We 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 10](#), looks very different.

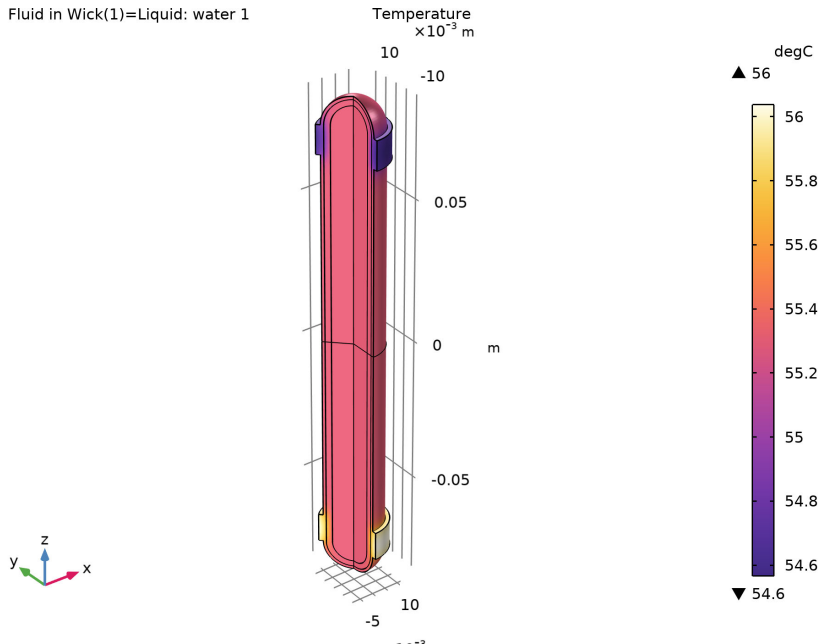


Figure 10: Temperature of heat pipe with saturated wick.

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 11, the computed velocity fields, in both fluids, and the temperature throughout the geometry, are plotted next to each other.

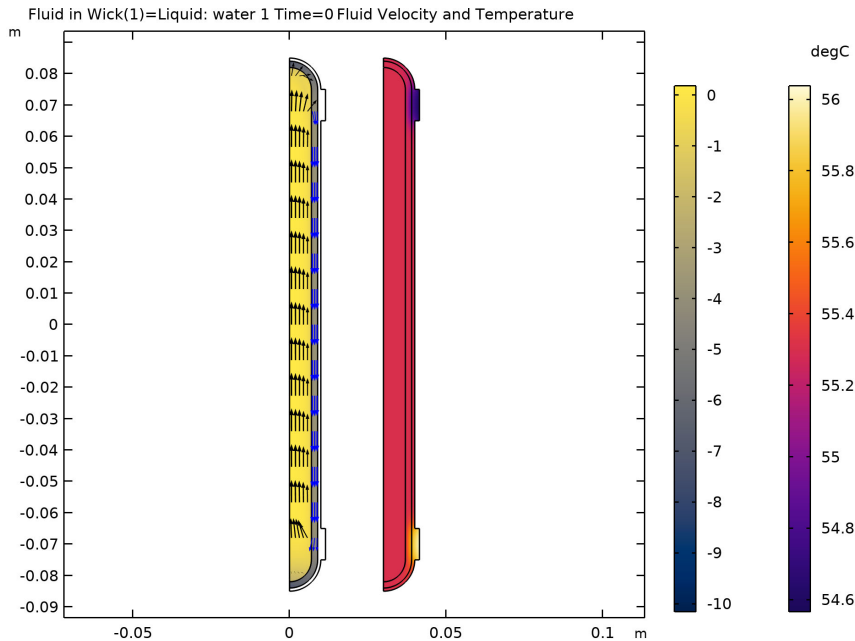


Figure 11: Fluid velocities— $\lg(|\mathbf{u}| / \text{m}\cdot\text{s}^{-1})$  and temperature in the heat pipe running at the design point.

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 the table below, the relative importance of the different heat transfer mechanisms in the saturated heat pipe.

#### 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

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

3. 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.

## Modeling Instructions: Heat Pipe with Accurate Liquid and Gas Properties



By following the steps described below you will be able to set up a model for a Heat Pipe, analyze it at operating conditions, and see the importance of liquid water in the porous wick.

## Model Wizard




---

**Note:** These instructions are for the user interface on Windows but apply, with minor differences, also to Linux and macOS.

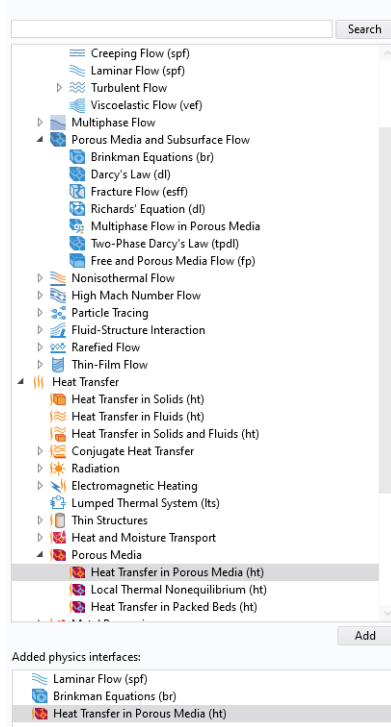
**I** To start the software, double-click the COMSOL icon on the desktop. When the software opens, choose to use the Model Wizard to create a new COMSOL Multiphysics model, or choose Blank Model to create one manually. For this tutorial, click the Model Wizard button.

If COMSOL Multiphysics is already open, you can start the Model Wizard by choosing  New from the File menu and then click  Model Wizard.

The Model Wizard guides you through the first steps of setting up a model. The next window lets you choose the dimension of the modeling space.

- 2 In the Model Wizard window, click  2D Axisymmetric.
- 3 In the Select Physics tree, select Fluid Flow>Single-Phase Flow>Laminar Flow (spf).
- 4 Click Add.
- 5 In the Select Physics tree, select Fluid Flow>Porous Media and Subsurface Flow>Brinkman Equations (br).
- 6 Click Add.
- 7 In the Select Physics tree, select Heat Transfer>Porous Media>Heat Transfer in Porous Media (ht).
- 8 Click Add.
- 9 Click  Study.
- 10 In the Select Study tree, select General Studies>Stationary.
- 11 Click  Done.


## Select Physics



## Global Definitions

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

### PARAMETERS


- 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

---


Build up the geometry using a sector of a circle, three rectangles and a mirror plane.

### CIRCLE 1

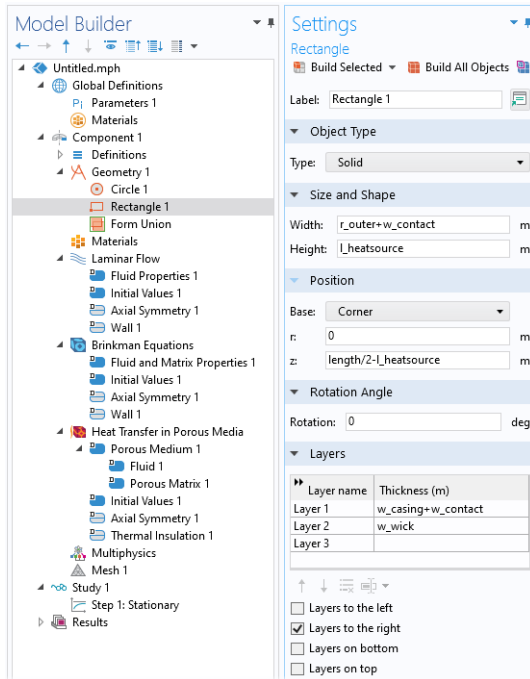
- 1 In the Geometry toolbar, click  Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type `r_outer`.
- 4 In the Sector angle text field, type `90`.
- 5 Locate the Position section. In the `z` text field, type `length/2`.
- 6 Click to expand the Layers section. In the table, enter the following settings:

LAYER NAME	THICKNESS (M)
Layer 1	<code>w_casing</code>
Layer 2	<code>w_wick</code>

## RECTANGLE 1

- 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

- 1 In the Model Builder window, right-click Rectangle 1 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	$w_{casing}$
Layer 2	$w_{wick}$

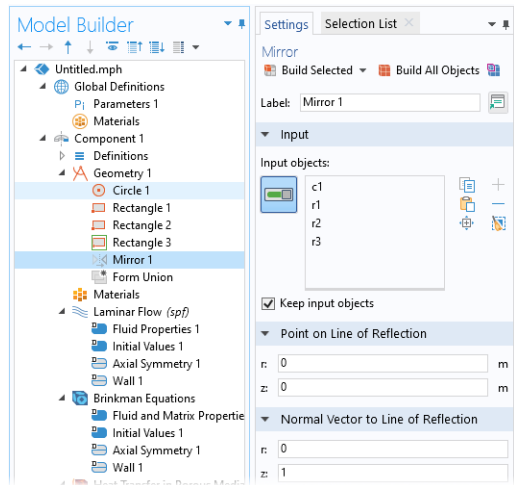
## RECTANGLE 3

- 1 Right-click Rectangle 2 and choose Duplicate.
- 2 Locate the Size and Shape section. In the Height text field, type  $\text{length}/2 - l_{\text{heatsource}} * 2$
- 3 In the Settings window for Rectangle, locate the Position section.
- 4 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

- 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.
- 7 In the Geometry toolbar, click  Build All Objects.
- 8 In the Graphics window, click  Zoom Extents.

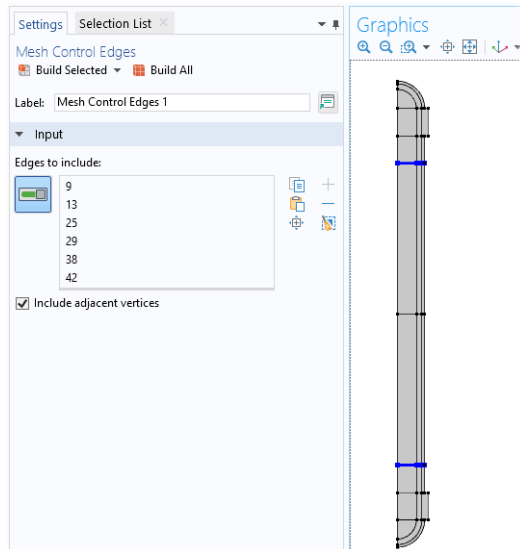


## MESH CONTROL EDGES I


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

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

- 2 In the Graphics window, select Boundaries 9, 13, 25, 29, 38, and 42 by clicking in accordance with the figure on the right. Verify that the selected edges match in the Settings window. In case of trouble, ensure sure that Activate Selection slider button is active (green).

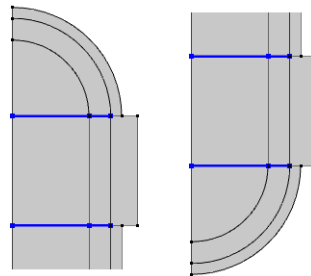


## IGNORE EDGES I

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


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

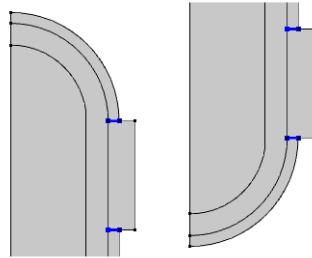
- 2 With the Input selection active in Settings window for Ignore Edges, select Boundaries 5, 7, 11, 13, 17, 19, 23, and 24 in the Graphics window.







## IGNORE EDGES 2

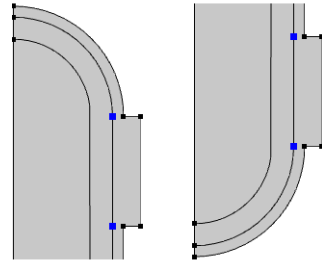
- 1 In the Geometry toolbar, click  Virtual Operations and choose Ignore Edges.
- 2 Select Boundaries 12, 14, 18, and 19.
- 3 In the Settings window for Ignore Edges, locate the Input section.
- 4 Clear the Ignore adjacent vertices check box.




## IGNORE VERTICES I

- 1 In the Geometry toolbar, click  Virtual Operations and choose Ignore Vertices.
- 2 Select Points 9, 10, 12, and 13.
- 3 In the Geometry toolbar, click  Build All.


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.



## VAPOR CAVITY

- 1 In the Geometry toolbar, click  Selections and choose Explicit Selection.
- 2 Select Domains 3 and 4.
- 3 In the Settings window for Explicit Selection, type Vapor Cavity in the Label text field.

## POROUS COPPER WICK


- 1 In the Geometry toolbar, click  Selections and choose Explicit Selection.
- 2 Select Domains 2 and 5.
- 3 In the Settings window for Explicit Selection, type Porous Copper Wick in the Label text field.

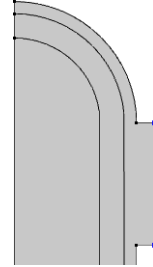
## SOLID COPPER TUBE CASING

- 1 In the Geometry toolbar, click  Selections and choose Explicit Selection.
- 2 Select Domains 1 and 6.


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

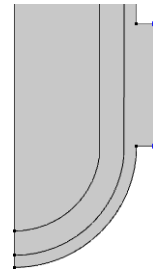
### HEAT SINK

- 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 Select Boundary 21.




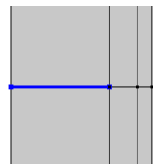
### HEAT SOURCE

- 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 Select Boundary 20.
- 5 In the Label text field, type **Heat Source**.




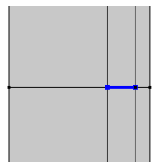
### CROSS SECTION OF CAVITY

- 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 Select Boundary 5.




### CROSS SECTION OF WICK

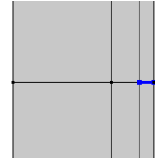
- 1 In the Geometry toolbar, click  Selections and choose Explicit Selection.
- 2 In the Settings window for Explicit Selection, type **Cross Section of Wick** in the Label text field.
- 3 Locate the Entities to Select section. From the Geometric entity level list, choose Boundary.




- 4 Select Boundary 10.

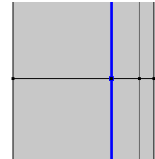
### CROSS SECTION OF CASING

- 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 Select Boundary 13.




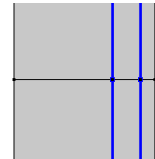
### INNER WICK BOUNDARY

- 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 Select Boundaries 8 and 9.




### 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 Select Boundaries 8, 9, 11 and 12.



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. Choose (holding the Ctrl key allows you to make multiple choices):
  - Cross Section of Cavity.
  - Cross Section of Wick.
  - Cross Section of Casing.
- 5 Click OK.

## 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 I


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  $0.9 * \min(w_{\text{casing}}, w_{\text{wick}}) * \text{mesh\_factor}$ .
- 5 In the Minimum element size text field, type  $0.3 * \min(w_{\text{casing}}, w_{\text{wick}}) * \text{mesh\_factor}$ .

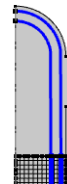


### BOUNDARY LAYERS I

- 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 Selection drop box, choose “All Wick Boundaries”.
- 3 Click  Build All.



## Materials



---

We 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 you will then generate materials. Note that we 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  Windows and choose Add Material from Library.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-in>Copper.
- 4 Click  Add to Global Materials.

## Global Definitions

---


For water vapor we will use an ideal gas model, for liquid water we will use the IAPWS model.

### THERMODYNAMIC SYSTEM FOR WATER VAPOR

- 1 In the Model Builder window, right-click Global Definitions and choose Thermodynamics>Thermodynamic System.
- 2 Click Next in the window toolbar.
- 3 In the Species list, select water (7732-18-5, H<sub>2</sub>O).
- 4 Click  Add Selected.
- 5 Click Next in the window toolbar.
- 6 From the Gas phase model list, choose Ideal gas.
- 7 Click Finish in the window toolbar.
- 8 In the Model Builder window, under Global Definitions>Thermodynamics right-click Gas System 1 (pp1) and choose Generate Material.
- 9 Click Next in the window toolbar three times (until you come to the Define Material window).
- 10 From the Component list, choose Global, and then click Finish.



### THERMODYNAMIC SYSTEM FOR LIQUID WATER

- 1 In the Model Builder window, under Global Definitions right-click Thermodynamics and choose Thermodynamic System.

- 2 From the drop down list, choose Vapor-liquid.
- 3 Click Next in the window toolbar.
- 4 In the Species list, select water (7732-18-5, H<sub>2</sub>O).
- 5 Click  Add Selected.
- 6 Click Next in the window toolbar, and then Finish.


## PROPERTIES OF WATER

Some of the boundary conditions need physical properties of water. These can be generated from the Thermodynamic system.

- 1 In the Model Builder window, under Global Definitions>Thermodynamics right-click Vapor-Liquid System 1 (pp2) and choose Species Property.
- 2 From the Amount base unit list, choose kg.
- 3 From the list, click  Add Selected, for each of:
  - Heat of vaporization (J/kg)
  - Ln vapor pressure, Pa
- 4 Click Next in the window toolbar.
- 5 Choose Liquid, and click Next.
- 6 Click  Add All, and click Next.
- 7 Click Finish in the window toolbar.

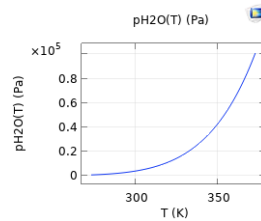
## VAPOR PRESSURE OF WATER

For convenience, a helper function is set up, allowing access to the vapor pressure of water from the generated function (which returns the natural logarithm of the vapor pressure measured in Pascal).

- 1 In the Home toolbar, click  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 pH<sub>2</sub>O.
- 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 Arguments text field, type K.
- 7 In the Function text field, type Pa.

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

ARGUMENT	LOWER LIMIT	UPPER LIMIT
T	273.15	373.15



- Optionally, click Plot to inspect the vapor pressure curve.

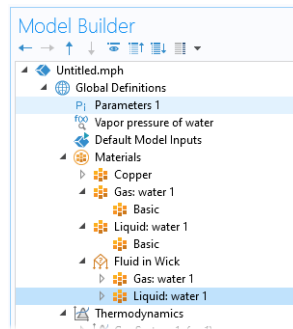
## LIQUID WATER MATERIAL

- Right-click Vapor-Liquid System 1 (pp2) and choose Generate Material.
- From the list, choose Liquid, and click Next in the window toolbar, three times.
- Click Finish.

## FLUID IN WICK

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

- In the Model Builder window right-click Global Definitions>Materials and choose Material Switch.
- In the Settings window for Material Switch, type Fluid in wick in the Label text field.
- In the Model Builder window, right-click Gas: water 1 (pp1mat1) and choose Copy.
- In the Model Builder window, right-click Fluid in wick (sw1) and choose Paste Material.
- In the Model Builder window, right-click Liquid: water 1 (pp2mat1) and choose Copy.
- In the Model Builder window, right-click Fluid in wick (sw1) and choose Paste Material.



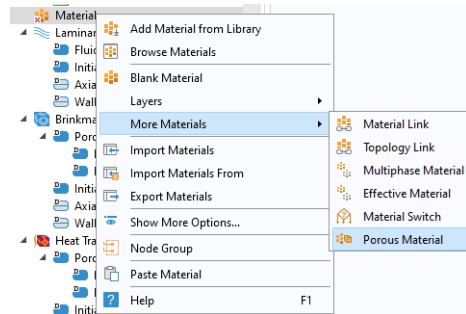
## Materials

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

- 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

- 1 Right-click Porous Material 1 (poromat1) and choose Fluid.
- 2 In the Settings window for Fluid, locate the Fluid Properties section.
- 3 From the Material list, choose Fluid in wick.

## SOLID 1

- 1 In the Model Builder window, right-click Porous Material 1 (poromat1) and choose Solid.
- 2 In the Settings window for Solid, locate the Solid Properties section.
- 3 In the  $\theta_s$  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, 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 In the Settings window for Material Link, locate the Geometric Entity Selection section.
- 4 From the Selection list, choose Vapor Cavity.
- 5 Locate the Link Settings section. From the Material list, choose Gas: water 1 (pp1mat1).

## COPPER METAL


- 1 Right-click Materials (under Component 1) and choose More Materials>Material Link.
- 2 In the Settings window for Material Link, type Copper Metal in the Label text field.
- 3 In the Settings window for Material Link, locate the Geometric Entity Selection section.
- 4 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 AND 2

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


## Laminar Flow

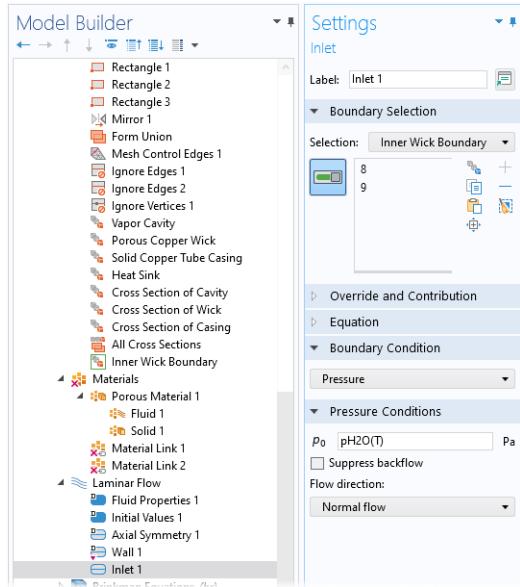
---

- 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  $p_{ref}$ .

### INLET 1

A pressure condition is applied on the cavity boundary, it is set equal to the vapor pressure of water. By not suppressing backflow vapor is allowed to enter at the hot side and exit at the cold side.

- 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  $p_{H2O}(T) - p_{ref}$ .
- 6 Clear the Suppress backflow check box.



## INITIAL VALUES I

- 1 In the Model Builder window under Component 1>Laminar Flow, 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

---


- 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 I

- 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  $\kappa$  list, choose User defined. In the associated text field, type `wick_permeability`.
- 4 From the Fluid material list, choose Fluid in Wick (sw1).
- 5 Locate the Porous Matrix Properties section. From the  $\kappa$  list, choose User defined. In the associated text field, type `wick_permeability`.


## INLET I

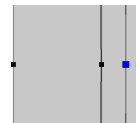
The mass flux of water through the cavity–wick interface must be equal in both the Laminar Flow interface, as well as the Brinkman Equations interface. Using the ratios of densities between vapor and liquid, set a velocity boundary condition on the wick.

- 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

<code>u*spf.rho/br.rho</code>	<code>r</code>
<code>w*spf.rho/br.rho</code>	<code>z</code>

## PRESSURE POINT CONSTRAINT I

- 1 In the Physics toolbar, click  Points and choose Pressure Point Constraint.  
Apply a pressure constraint of 0[Pa] to the point on the  $r$  axis at the interface between the wick and the casing.
- 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 0[Pa].




## Heat Transfer in Porous Media


---

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 Porous Medium 1, click Porous Matrix 1.
- 2 Locate the Matrix Properties section. From the Define list, choose Solid phase 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`.

### HEAT FLUX 2


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 this proportionality constant (the

heat transfer coefficient) depends on external flow conditions such as the presence of an external fan, the external surface area, and the geometry of fins (if present).

- 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. Choose Convective heat flux from the choices of Flux type.
- 5 In the  $h$  text field, type  $h_{\text{conv}}$ .

## BOUNDARY HEAT SOURCE I

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.

- 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  $(u*\text{spf.nr}+w*\text{spf.nz})*\text{HeatOfVaporization\_water21}(T)*\text{spf.rho}$ .



## Study I

---

- 1 In the Model Builder window, click Study I.
- 2 In the Settings window for Study, type Study 1 - Dry Wick in the Label text field.

## MATERIAL SWEEP


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 I”.

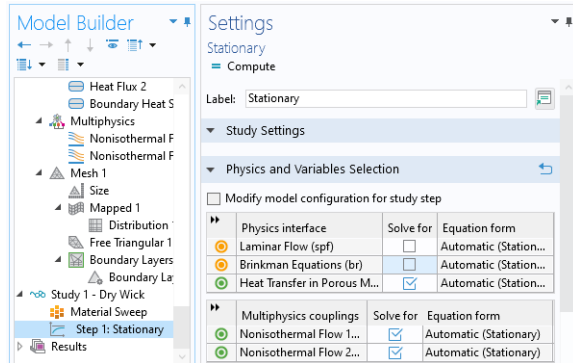
- 1 In the Study toolbar, click  Material Sweep.
- 2 In the Settings window for Material Sweep, locate the Study Settings section.
- 3 Click  Add.

4 In the table, enter the following settings:

SWITCH	CASES	CASE NUMBERS
Fluid in wick (sw1)	User defined	1

### STEP 1: STATIONARY

- 1 In the Model Builder window, under Study 1 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.



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


## Results

Edit the plot groups to create [Figure 9](#).

### TEMPERATURE, 3D (HT) - DRY WICK

- 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 9](#) from the earlier Results section. We 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, right-click Isothermal Contours (ht) and choose Delete.

## Add Study

---

Let us now solve the model for the normal case, with actual liquid in the wick.



- 1 In the Home toolbar, click  Windows and choose Add Study.
- 2 In the Add Study window, select General Studies>Stationary in the Select Study tree.
- 3 Click  Add Study.

## Study 2 - Saturated Wick

---

- 1 In the Model Builder window, click Study 2.
- 2 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	2

## STEP 2: STATIONARY I

Study 2 is solved in two steps, first without 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.

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 the first step in Study 2: “Step 1: 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.

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

## Results


---

### VELOCITY (SPF) AND TEMPERATURE (HT)

- 1 In the Settings window for 2D Plot Group Velocity (spf), add 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 I - 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  $\log_{10}(\text{ht.uz}^2 + \text{ht.ur}^2) / 2$ .
- 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.

## SURFACE 2 - TEMPERATURE

- 1 Under Velocity (spf) and Temperature, right-click Surface 1 - Fluid velocity,  $\lg(|u|)$ , and choose Duplicate.
- 2 In the Settings window for Surface, type Surface 2 - Temperature in the Label text field.
- 3 In the Settings window for Surface, locate the Expression section.
- 4 In the Expression text field, type T.
- 5 Locate the Expression section. From the Unit list, choose degC.
- 6 Locate the Coloring and Style section. From the Color table list, choose HeatCamera.

## DEFORMATION I

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

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

## ARROW SURFACE I

- 1 In the Model Builder window, right-click Velocity (spf) and Temperature (ht) and choose Arrow Surface.
- 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.
- 6 In the associated text field, type 0.005.

7 From the Color list, choose Black.

## ARROW SURFACE 1 - VAPOR FLOW I

Right-click Arrow Surface 1 - Vapor flow and choose Duplicate.

## ARROW SURFACE 2 - LIQUID FLOW

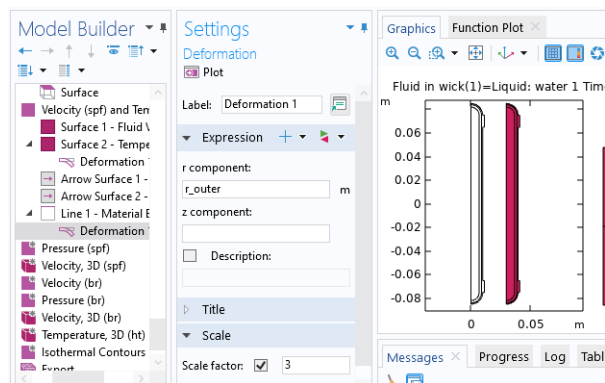
- 1 In the Settings window for Arrow Surface, type Arrow Surface 2 - Liquid flow in the Label text field.
- 2 Locate the Expression section. In the r component text field, type  $u_2$ .
- 3 In the z component text field, type  $w_2$ .
- 4 Locate the Coloring and Style section. Enter the value 50 as the Scale factor.
- 5 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 I

- 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. Select the Scale factor check box.



- 5 In the associated text field, type 3, and click  Plot.

## Study 2 - Saturated Wick

---

### SOLVER CONFIGURATIONS


- 1 In the Model Builder window, expand the Study 2 - Saturated Wick>Solver Configurations>Solution 4 (sol4) 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, select “Velocity (spf) and Temperature (ht)” as the Plot group.
- 5 In the Model Builder window, right-click Stationary Solver 2 and choose Fully Coupled.
- 6 In the Settings window for Fully Coupled, locate the Results While Solving section.
- 7 Select the Plot check box, select “Velocity (spf) and Temperature (ht)” as the Plot group.

### 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 Label text field, type Heat Sink.

- 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/m<sup>2</sup>.
- 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

## LATENT HEAT FLUX FROM PHASE CHANGE

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.

- 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 I

- 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 Locate the y-Axis Data section. Select the Description check box.
- 7 In the associated text field, type  $(\langle B \rangle u \langle /B \rangle \cdot \langle B \rangle n \langle /B \rangle) \rho \Delta H_{\text{vap}}$ .

- 8 Click to expand the Title section. From the Title type list, choose Manual.
- 9 In the Title text area, type `Latent heat flux from phase change`.
- 10 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- 11 In the Expression text field, type `z`.

## Study 2 - Saturated Wick

---


In the Home toolbar, click  Compute.

## Results

---

We can analyze the results, first by numbers, by looking at line integrals defined earlier.

### ENERGY BALANCE

- 1 In the Model Builder window, under Results click Energy Balance.
- 2 In the Energy Balance toolbar, click  Evaluate.

Note how the heat fluxes across the heat sink and heat source match with opposite signs, and how close the magnitude of “cavity: latent heat” is to this number.

Next apply some finishing touches to the 3D plot of the temperature.

### TEMPERATURE, 3D (HT)

- 1 In the Model Builder window, click Temperature, 3D (ht).
- 2 In the Settings window for 3D Plot Group, in the Title section, change the Title type list choice to 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) node, then click Surface.
- 2 In the Settings window for Surface, locate the Expression section.

3 From the Unit list, choose degC.

4 Click  Plot.

This is [Figure 10](#) from the results section earlier. Notice how even the temperature is throughout the heat pipe. Also the maximum temperature is significantly lower compared to the dry wick case investigated previously.