Created in COMSOL Multiphysics 6.1

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.

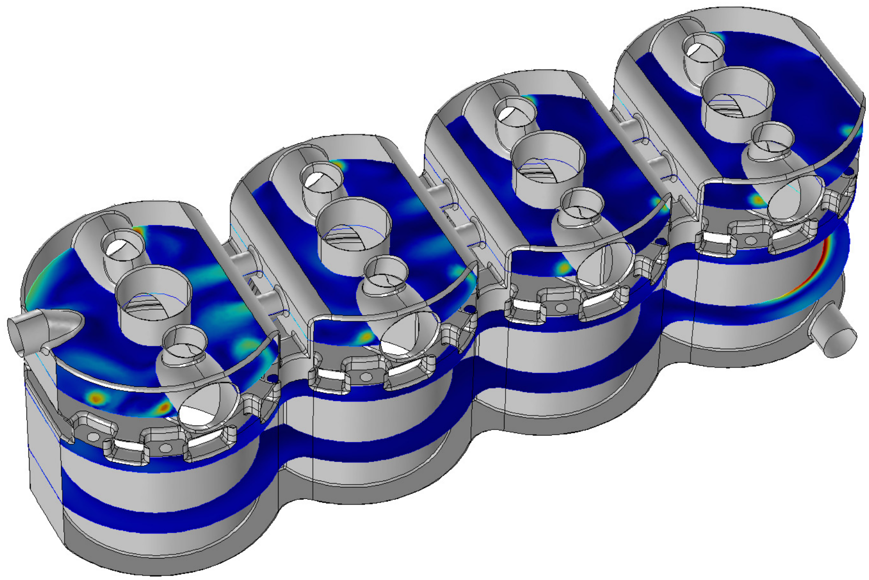


Figure 1: The coolant flow inside the cooling jacket of a four cylinder engine.

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-ε 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. Finally the results are used to compute average mixture properties.

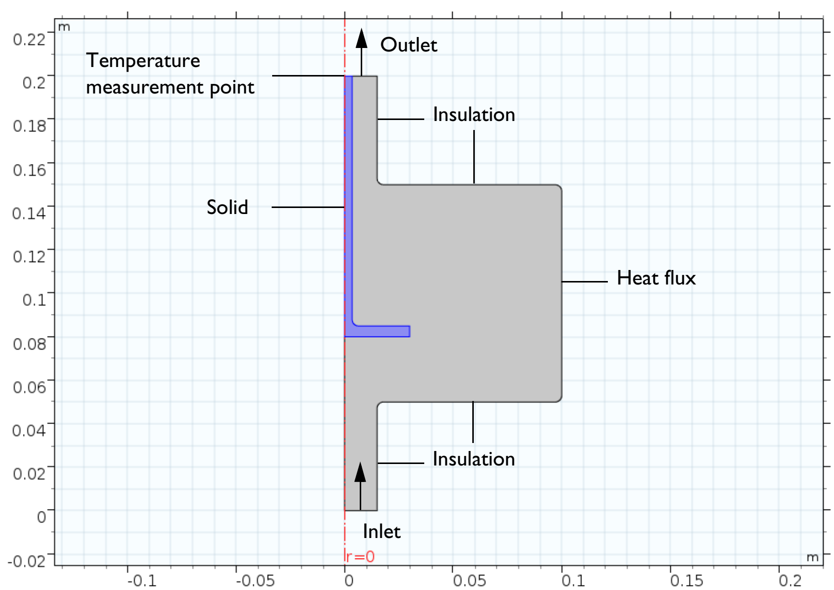


Figure 2: Axially symmetric engine coolant test apparatus.

Results and Discussion

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

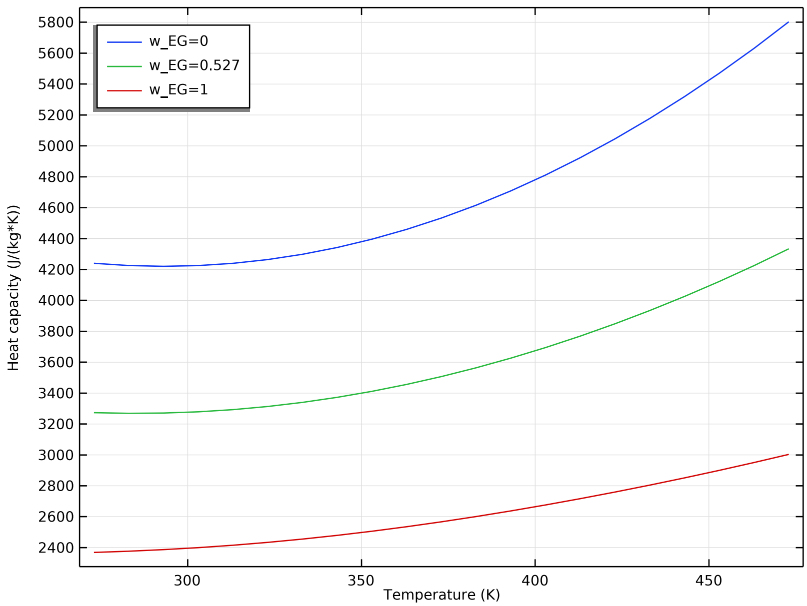


Figure 3: Heat capacity as a function of temperature and composition for ethylene glycol water mixtures.

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.

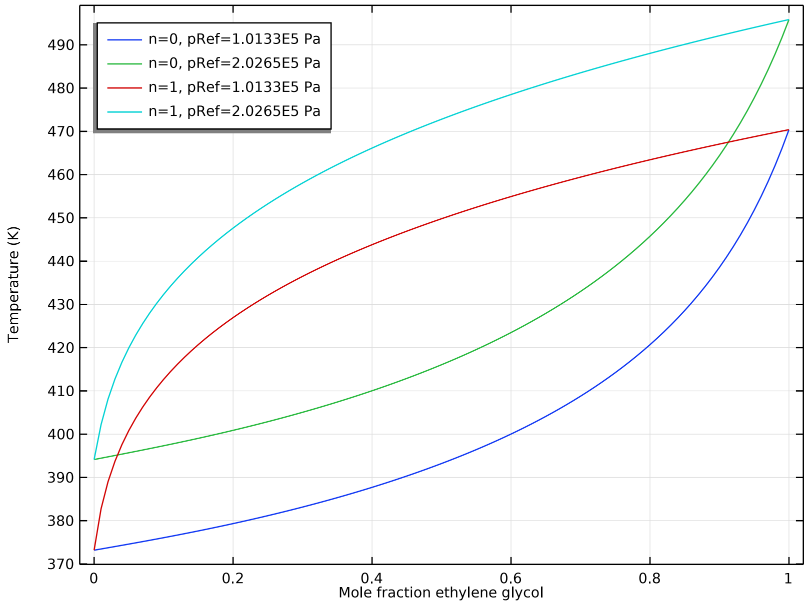


Figure 4: Phase envelope for the equilibrium temperature of ethylene glycol-water mixtures at two pressures.

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.

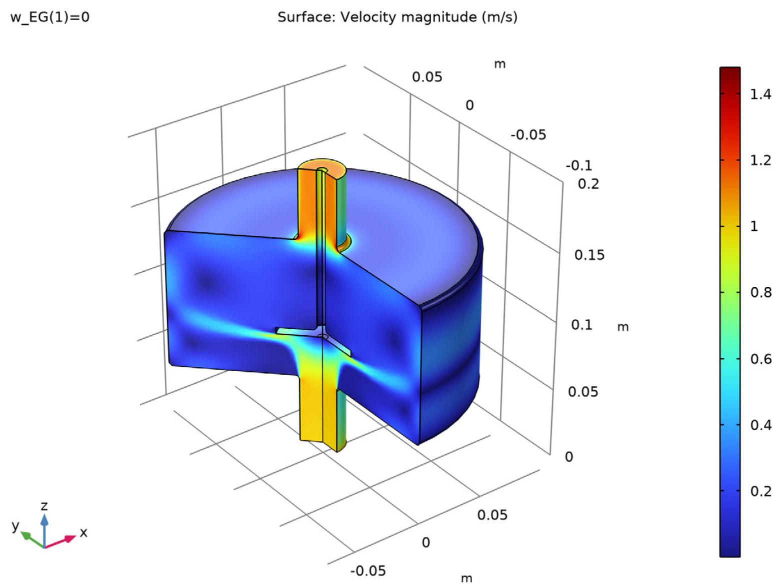


Figure 5: Flow patterns inside the test apparatus with water at 1 m/s.

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.

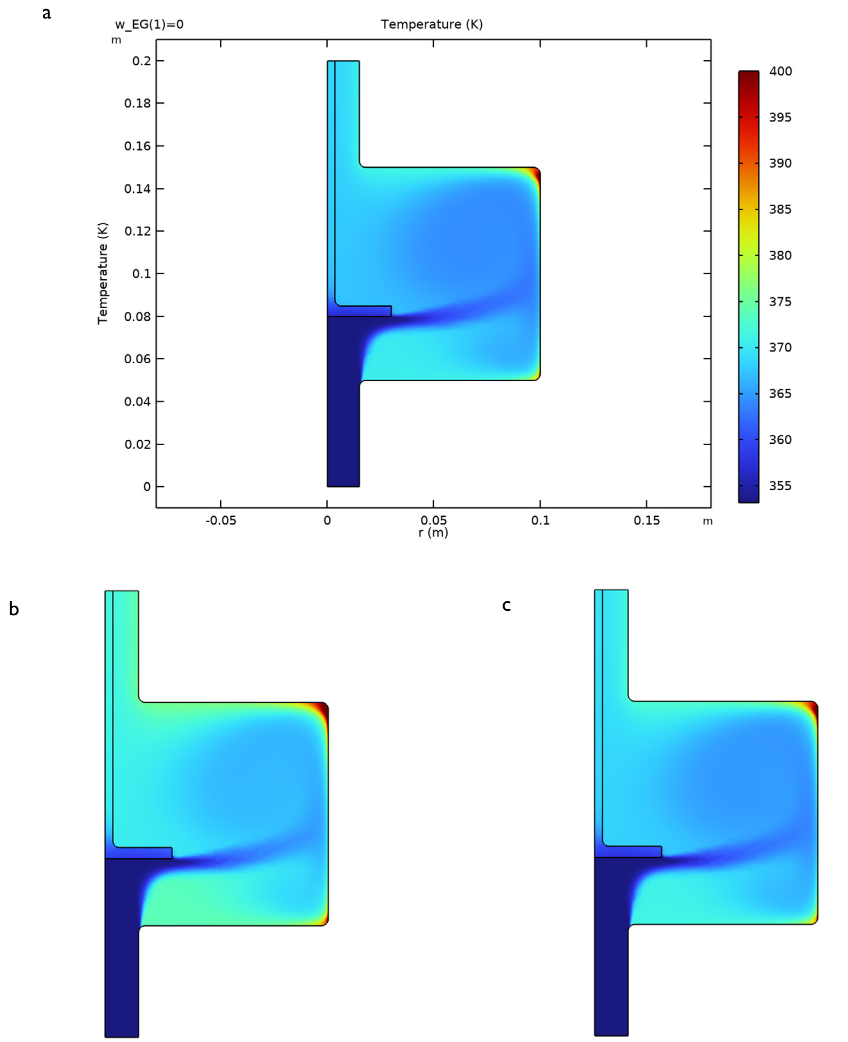


Figure 6: Temperature within the test apparatus for three cases: (a) water at 1 m/s, (b) 50 volume percent ethylene glycol at 1 m/s, and (c) 50 volume percent ethylene glycol at 1.15 m/s.

Table 1 provides a comparison of results for pressure drop, outlet temperature, and outlet density.

TABLE 1: 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[1]	1	608	373	1010

[1] Using constant mixture properties.

Considering the graphical results for the various coolant properties, it might be reasonable to use approximate averages for the relatively small temperature range considered, between 353 and 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.

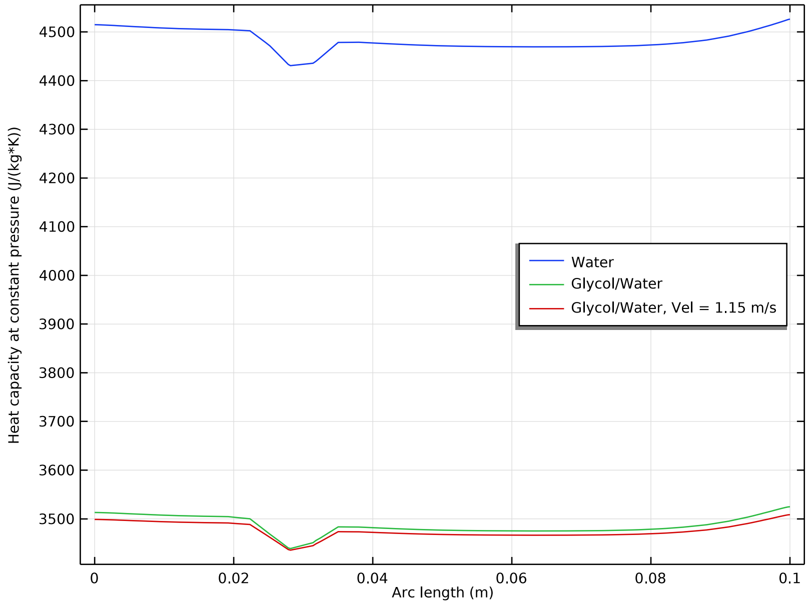
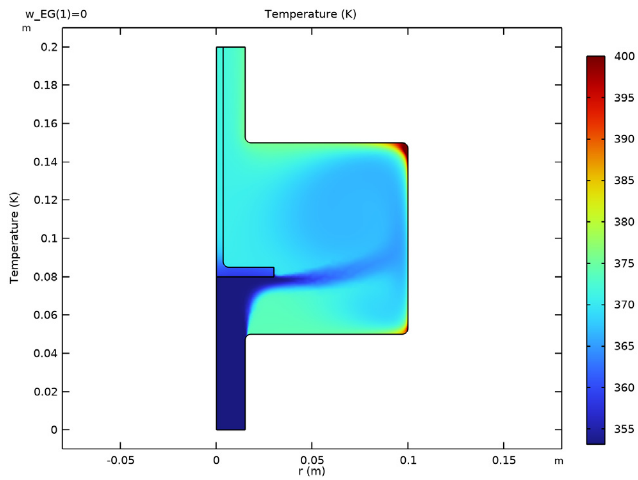


Figure 7: Coolant heat capacity plotted along a vertical cut line at half the radius of the test apparatus chamber.

Using the solution for a mixture with 50 volume percent ethylene glycol in water, the following average values are computed; density = 1010 kg/m^3, viscosity = $9.07 \cdot 10^{-4}$ Pa·s, thermal conductivity = 0.574 W/(m·K), and heat capacity = 3,486 J/(kg·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.

a



b

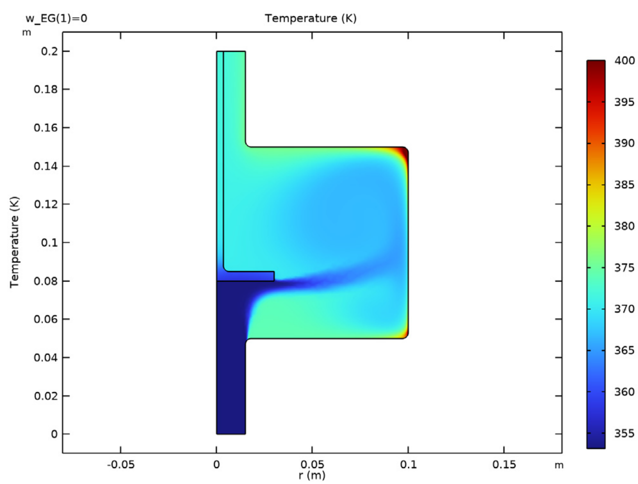


Figure 8: Comparison of temperature within the test apparatus for 50 volume percent ethylene glycol in water at 1 m/s using: (a) temperature dependent properties, (b) approximate average properties.

Reference

1. http://www.engineeringtoolbox.com/ethylene-glycol-d_146.html

Application Library path: `Chemical_Reaction_Engineering_Module/Thermodynamics/engine_coolant_properties`

Note: This model is included in the booklets *Introduction to Thermodynamic Properties*, and *Introduction to the Liquid & Gas Properties Module*.

Modeling Instructions

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

NEW

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

MODEL WIZARD

1. In the **Model Wizard** window, click **2D Axisymmetric**.
2. In the **Select Physics** tree, select **Fluid Flow>Nonisothermal Flow>Turbulent Flow>Turbulent Flow, k-ε**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Stationary**.
6. Click **Done**.

GLOBAL DEFINITIONS

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

SELECT SYSTEM

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

SELECT SPECIES

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

2 In the **Species** list, select **ethylene glycol (107-21-1, C2H6O2)**.

3 Click **Add Selected**.

4 In the **Species** text field, type `water`.

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

6 Click **Add Selected**.

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

SELECT THERMODYNAMIC MODEL

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

2 From the list, choose **UNIFAC VLE**.

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

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

GLOBAL DEFINITIONS

Vapor-Liquid System 1 (pp1)

When modeling a fixed composition mixture, it is convenient to use a material. The reason is that the default behavior in physics interfaces is to use properties from the domain material. Use the **Thermodynamic System** to automatically define 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**.

SELECT PHASE

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

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

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

SELECT SPECIES

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

Notice that both species are already added to the Selected species list.

2 Find the **Material composition** subsection. Click the **Mass fraction** button.

Keep the default composition in the Material composition subsection. The composition will be redefined in the generated Material.

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

SELECT PROPERTIES

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

  Use the default properties. The diffusion coefficients are not needed since a constant composition mixture is studied.

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

DEFINE MATERIAL

1 Go to 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**.

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

GLOBAL DEFINITIONS

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

Parameters 1

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

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

3 Click **Load from File**.

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

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.

GEOMETRY 1

Rectangle 1 (r1)

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

2 Right-click **Geometry 1** and choose **Rectangle**.

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

4 In the **Width** text field, type `r_p`.

5 In the **Height** text field, type `l_p`.

6 Click **Build Selected**.

Rectangle 2 (r2)

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`.

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

5 Locate the **Position** section. In the **z** text field, type `zpos_c`.

6 Click **Build Selected**.

Rectangle 3 (r3)

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**.

Rectangle 4 (r4)

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**.

Union 1 (uni1)

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

2 Select the objects **r3** and **r4** only.

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

4 Clear the **Keep interior boundaries** check box.

5 Click **Build Selected**.

Union 2 (uni2)

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

2 Select the objects **r1** and **r2** only.

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

4 Clear the **Keep interior boundaries** check box.

5 Click **Build Selected**.

Fillet 1 (fil1)

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

2 On the object **uni1**, select Point 5 only.

3 On the object **uni2**, 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**.

STUDY 1: MIXTURE PROPERTIES PARAMETERIZATION

Now, compute and plot the properties of the glycol-water coolant as defined by the **Thermodynamics** functions.

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.

3 Select the **Auxiliary sweep** check box.

4 Click **Add** twice.

5 In the table, enter the following settings:

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

6 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.

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

8 In the 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.

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

RESULTS

Now, inspect the results from the study by creating a plot group and plot the results. Start with the density.

Density

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

2 In the **Settings** window for **1D Plot Group**, type `Density` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Global 1

1 In the **Density** toolbar, click **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Global definitions>Functions> Densitypp1(temperature, pressure, massfraction_ethylene_glycol, massfraction_water) - Density 1**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`Densitypp1(Tc,pRef,w_EG,w_W)`		

4 Click to expand the **Legends** section. 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.

Density

Return to the plot group to improve the plot settings.

1. In the **Model Builder** window, click **Density**.
2. In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.
3. Select the **x-axis label** check box.
4. Select the **y-axis label** check box.
5. In the **x-axis label** text field, type `Temperature (K)`.
6. In the **y-axis label** text field, type Density (kg/m³).
7. In the **Density** toolbar, 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** text field.
3. Locate the **Title** section. From the **Title type** list, choose **None**.

Global 1

1. In the **Viscosity** toolbar, click **Global**.
2. In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Global definitions>Functions> Viscositypp1(temperature, pressure, massfraction_ethylene_glycol, massfraction_water) - Viscosity 1**.
3. Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`Viscositypp1(Tc,pRef,w_EG,w_W)`		

4. Locate the **Legends** section. Find the **Include** subsection. Clear the **Description** check box.

Viscosity

1. In the **Model Builder** window, click **Viscosity**.
2. In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.
3. Select the **x-axis label** check box.
4. Select the **y-axis label** check box.
5. In the **x-axis label** text field, type `Temperature (K)`.

6 In the **y-axis label** text field, type `Viscosity (Pa*s)`.
7 In the **Viscosity** toolbar, click **Plot**.

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** text field.
3 Locate the **Title** section. From the **Title type** list, choose **None**.

Global 1

1 In the **Thermal Conductivity** toolbar, click **Global**.
2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Global definitions>Functions> ThermalConductivitypp1(temperature, pressure, massfraction_ethylene_glycol, massfraction_water) - Thermal conductivity 1**.
3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`ThermalConductivitypp1(Tc,pRef,w_EG,w_W)`		

4 Locate the **Legends** section. Find the **Include** subsection. Clear the **Description** check box.

Thermal Conductivity

1 In the **Model Builder** window, click **Thermal Conductivity**.
2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.
3 Select the **x-axis label** check box. In the associated text field, type `Temperature (K)`.
4 Select the **y-axis label** check box. In the associated text field, type `Thermal conductivity (W/(m*K))`.
5 Locate the **Legend** section. From the **Position** list, choose **Middle right**.
6 In the **Thermal Conductivity** toolbar, click **Plot**.

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** text field.
3 Locate the **Title** section. From the **Title type** list, choose **None**.

Global 1

1 In the **Heat Capacity** toolbar, click **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Global definitions>Functions> HeatCapacityCppp1(temperature, pressure, massfraction_ethylene_glycol, massfraction_water) - Heat capacity (Cp) 1**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`HeatCapacityCppp1(Tc,pRef,w_EG,w_W)`		

4 Locate the **Legends** section. Find the **Include** subsection. Clear the **Description** check box.

Heat Capacity

1 In the **Model Builder** window, click **Heat Capacity**.

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

3 Select the **x-axis label** check box. In the associated text field, type `Temperature (K)`.

4 Select the **y-axis label** check box. In the associated text field, type `Heat capacity (J/(kg*K))`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

6 In the **Heat Capacity** toolbar, click **Plot**.

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

GLOBAL DEFINITIONS

Vapor-Liquid System 1 (pp1)

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

SELECT SPECIES

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

2 Click **Add All**.

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

EQUILIBRIUM SPECIFICATIONS

1 Go to the **Equilibrium Specifications** window.

2 From the **Amount base unit** list, choose **mol**.

3 Find the **Equilibrium conditions** subsection. From the **First condition** list, choose **Pressure**.

4 From the **Second condition** list, choose **Phase fraction**.

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

EQUILIBRIUM FUNCTION OVERVIEW

1 Go to the **Equilibrium Function Overview** window.

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

GLOBAL DEFINITIONS

You can now create an Analytic function to plot the phase envelope. 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. Also, use the Analytic function to change the composition arguments from moles to mole fractions.

Phase envelope

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

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)`.

5 In the **Arguments** text field, type `p, n, w_EG, w_W`.

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

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

Argument	Unit
p	Pa
n	1
w_EG	mol/mol
w_W	mol/mol

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

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

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

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

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

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

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

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

STUDY 2

Step 1: Stationary

Add settings to perform an Auxiliary sweep study, where we vary 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.

4 In the table, enter the following settings:

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**.

6 In the **Model Builder** window, click **Study 2**.

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

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

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

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

11 In the table, clear the **Solve for** check boxes for **Turbulent Flow, k-ε (spf)** and **Heat Transfer in Fluids (ht)**.

12 In the table, clear the **Solve for** check box for **Nonisothermal Flow 1 (nitf1)**.

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

RESULTS

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

1D Plot Group 5

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.
6. Select the **x-axis label** check box. In the associated text field, type `Mole fraction ethylene glycol`.
7. Select the **y-axis label** check box. In the associated text field, type `Temperature (K)`.

Global 1

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

Expression	Unit	Description
`T_x_y(pRef, n, w_EG, w_W)`		

4. Locate the **x-Axis Data** section. From the **Axis source data** list, choose **w_EG**.
5. In the **1D Plot Group 5** toolbar, click **Plot**.

Phase Envelope

1. In the **Model Builder** window, click **1D Plot Group 5**.
2. In the **Settings** window for **1D Plot Group**, locate the **Legend** section.
3. From the **Position** list, choose **Upper left**.
4. In the **Label** text field, type `Phase Envelope`.

MATERIALS

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)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials** node, then click **Liquid: ethylene glycol-water 1 (pp1mat1)**.
2 Select Domain 1 only.
3 In the **Settings** window for **Material**, locate the **Material Contents** section.
4 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.

TURBULENT FLOW, K-ε (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, k-ε (spf)**.
2 Select Domain 1 only.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2 Select Boundary 2 only.
3 In the **Settings** window for **Inlet**, locate the **Velocity** section.
4 In the U_0 text field, type `Vel`.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2 Select Boundary 11 only.

HEAT TRANSFER IN FLUIDS (HT)

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.
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.

Heat Flux 1

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

2 Select Boundary 18 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 From the **Flux type** list, choose **Heat rate**.

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 properties for the solid steel part from **Materials**.

ADD MATERIAL

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

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

3 In the tree, select **Built-in>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.

MATERIALS

Structural steel (mat1)

Select Domain 2 only.

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

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

ADD STUDY

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

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

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

4 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

Step 1: Stationary

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

6 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 as initial conditions. In this case Anderson acceleration can be enabled to reduce the simulation time.

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

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

4 In the **Settings** window for **Segregated**, locate the **General** section.

5 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.

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

Plot group 10 displays the flow field in the test apparatus using a revolved dataset.

RESULTS

Velocity, 3D (spf)

Delete some superfluous plot groups.

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

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Velocity (spf)**, **Pressure (spf)**, **Temperature, 3D (ht)**, and **Isothermal Contours (ht)**.

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

Create a 2D plot group for the temperature.

2D Plot Group 10

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

3 From the **Dataset** list, choose **Study 3: Water/Solution 3 (sol3)**.

Surface 1

1 In the **2D Plot Group 10** 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 **2D Plot Group 10** 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 **2D Plot Group 10** toolbar, click **Plot**.

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

Temperature

1 In the **Model Builder** window, under **Results** click **2D Plot Group 10**.

2 In the **Settings** window for **2D Plot Group**, type `Temperature` in the **Label** text field.

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

4 In the **Title** text area, type `Temperature (K)`.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `r (m)`.

7 Select the **y-axis label** check box. In the associated text field, type `Temperature (K)`.

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 using the same inlet velocity as for the pure water case, and one 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.

3 Find the **Studies** subsection. In the **Select Study** 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 4

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.

2 Select the **Auxiliary sweep** check box.

3 Click **Add** twice.

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`

Step 2: Stationary 1

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**
Vel (Pipe inlet velocity)	`1.15`	`m/s`

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**.

Vel = 1.15 m/s

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 **Solution 5 (sol5)** node.

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

4 In the **Settings** window for **Segregated**, locate the **General** section.

5 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.

6 In the **Model Builder** window, collapse the **Study 4>Solver Configurations>Solution 5 (sol5)>Stationary Solver 1** node.

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

8 In the **Settings** window for **Segregated**, locate the **General** section.

9 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.

10 In the **Model Builder** window, click **Study 4**.

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

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

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

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

15 In the **Settings** window for **Solution Store**, type `Vel = 1.0 m/s` in the **Label** text field.

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

17 In the **Settings** window for **Solution**, type `Vel = 1.15 m/s` in the **Label** text field.

Plot the temperature for the glycol/water mixture to reproduce the plots in Figure 6.

RESULTS

Temperature

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

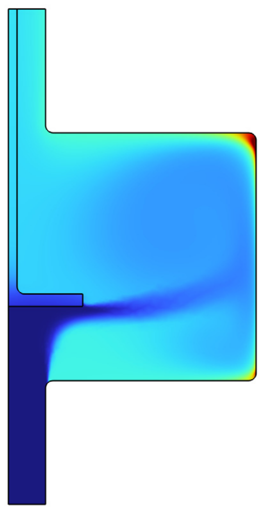
2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 4: Glycol and Water /Vel = 1.0 m/s (sol6)**.

4 In the **Temperature** toolbar, click **Plot**.

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

6 In the **Temperature** toolbar, click **Plot**.



Create cut line datasets to evaluate the heat capacity throughout the chamber section of the apparatus. Create one line for each study case.

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)**.

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)**.

1D Plot Group 11

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.

3 From the **Dataset** list, choose **Cut Line 2D 1**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `ht.Cp`.

5 In the **1D Plot Group 11** toolbar, click **Plot**.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Water`

9 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.

3 From the **Dataset** list, choose **Cut Line 2D 2**.

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

Legends
`Glycol/Water`

5 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.

3 From the **Dataset** list, choose **Cut Line 2D 3**.

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

Legends
Glycol/Water, Vel = 1.15 m/s

5 In the **1D Plot Group 11** toolbar, click **Plot**.

Heat Capacity, Chamber Cut Line

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 In the **Label** text field, type `Heat Capacity, Chamber Cut Line`.

Compute the average mixture property values.

Surface Average 1

1 In the **Results** toolbar, click 8.85 e-12 **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.

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

Expression	Unit	Description
ht.rho	kg/m^3	Density
ht.Cp	J/(kg*K)	Heat capacity at constant pressure
ht.krr	W/(m*K)	Thermal conductivity, rr component
spf.mu	Pa*s	Dynamic viscosity

6 Click **Evaluate**.

Store the average property values as parameters.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
rhoC	1010[kg/m^3]	1010 kg/m³	Average constant density
CpC	3486[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	9.07e-4[Pa*s]	9.07E-4 Pa·s	Average constant viscosity

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

HEAT TRANSFER IN FLUIDS (HT)

Fluid 1

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 ρ list, choose **User defined**. In the associated text field, type `rhoC`.

6 From the C_p list, choose **User defined**. In the associated text field, type `CpC`.

TURBULENT FLOW, K-ε (SPF)

Fluid Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Turbulent Flow, k-ε (spf)** click **Fluid Properties 1**.

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

3 From the μ list, choose **User defined**. In the associated text field, type `muC`.

ADD STUDY

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

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

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

4 Click **Add Study** in the window toolbar.

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

STUDY 5

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 **Model Builder** window, click **Study 5**.

6 In the **Settings** window for **Study**, type `Study 5: Glycol and Water, Constant Properties` in the **Label** text field.

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

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

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

RESULTS

Temperature

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 5: Glycol and Water, Constant Properties / Solution 7 (sol7)**.

4 In the **Temperature** toolbar, click **Plot**.

Compute the outlet temperature, the average pressure drop, and the average outlet density.

Point Evaluation 1

1 In the **Results** toolbar, click 8.85 e-12 **Point Evaluation**.

2 Select Point 6 only.

3 In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
T	K	

5 Locate the **Data** section. From the **Dataset** list, choose **Study 3: Water/Solution 3 (sol3)**.

6 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 **New Table**.

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

10 Click next to **Evaluate**, then choose **New Table**.

11 From the **Dataset** list, choose **Study 5: Glycol and Water, Constant Properties / Solution 7 (sol7)**.

12 Click next to **Evaluate**, then choose **New Table**.

Line Average 2

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Average>Line Average**.

2 Select Boundary 2 only.

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 next to **Evaluate**, then choose **Table 2 - Point Evaluation 1**.

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**.

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**.

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**.

Line Average 3

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Average>Line Average**.

2 Select Boundary 11 only.

3 In the **Settings** window for **Line Average**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`ht.rho`	`kg/m^3`	

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**.

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**.

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**.

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**.