Model created in COMSOL Multiphysics 6.4

Natural Convection Cooling of a Vacuum Flask

Introduction

This example solves a pure conduction problem and a free-convection problem in which a vacuum flask holding hot coffee dissipates thermal energy. The main interest is to calculate the flask's cooling power; that is, how much heat it loses per unit time.

Figure 1: Schematic picture of the flask.

The coffee has an initial temperature of 90°C and cools down over time. The observation period is 10 h. This tutorial compares two different approaches to model natural convection cooling:

- Using heat transfer coefficients to describe the thermal dissipation
- Modeling the convective flow of air outside the flask to describe the thermal dissipation

The first approach describes the outside heat flux using a heat transfer coefficient function from the Heat Transfer Coefficients library included with the Heat Transfer Module. This results in a rather simple model that predicts the stationary cooling well and produces accurate results for temperature distribution and cooling power.

The second approach solves for both the total energy balance and the flow equations of the outside cooling air. This application produces detailed results for the flow field around the flask as well as for the temperature distribution and cooling power. However, it is more complex and requires more computational resources than the first version.

Model Definition

Figure 2 shows the model geometry.

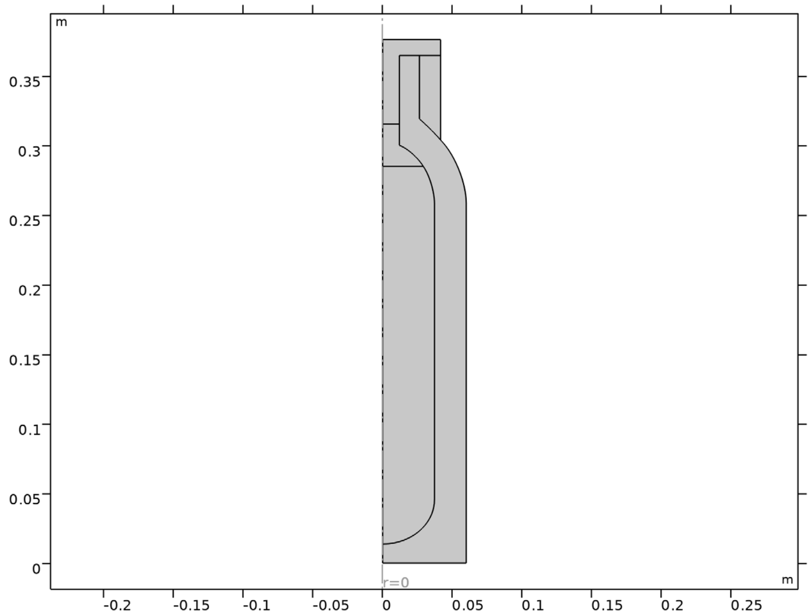


Figure 2: The 2D-axisymmetric representation of the vacuum flask.

CONTROL VOLUME

For the first approach, the model does not include a control volume around the flask to represent the domain of the surrounding air. Instead, it uses a heat transfer coefficient correlation for vertical and horizontal plates.

For the second approach, the model uses a control volume around the flask to represent the domain of the surrounding air. Choosing an appropriate control volume for natural convection models is difficult. Your choice strongly influences the model, the mesh, the convergence, and especially the flow behavior. The real-world air domain surrounding the flask is the entire room or atmosphere in which the flask is placed. Making the rectangle as large as the external room would result in a very large model requiring a supercomputer to solve. At the other extreme, if you make the control volume too small, the solution is affected by the imposed artificial boundary conditions, and there can also be a truncation of flow eddies, making convergence difficult.

An appropriate truncation should resolve the flow field around the flask but avoid modeling a large surrounding. One way to approach this task is to start with a small control volume, set up and solve the model, then expand the control volume, solve the model again, and see if the results change. This example uses a sufficiently large control volume by truncating the air domain at $r = 0.1$ m and $z = 0.5$ m. The boundary condition at the boundaries that are open to large volumes can handle both entering and leaving fluid. The entering fluid has the temperature of the surroundings whereas the leaving fluid has an unknown temperature that results from the cooling effects of the flow field.

MATERIAL PROPERTIES

Next consider the materials that make up the flask model. The flask contains coffee that has almost exactly the same material properties as water. The screw stopper and insulation ring are made of nylon. The flask bottle consists of stainless steel, and the filling material between the inner and outer walls is a plastic foam. The material library includes all materials used in this model except the foam, which you specify manually. Table 1 provides a list of standard foam's thermal properties.

TABLE 1: FOAM MATERIAL PROPERTIES.

PROPERTY	VALUE
Conductivity	0.03 W/(m·K)
Density	60 kg/m^3
Heat Capacity	200 J/(kg·K)

HEAT TRANSFER PHYSICS

This example assumes the hot liquid (coffee) to have a uniform temperature distribution that changes only with time. This is a reasonable approximation since the observation period is long and effects of spatially varying temperatures in the liquid are small. The Heat Transfer Module provides the Isothermal Domain feature. It solves only an additional ordinary differential equation of the form:

$$mC_p\frac{dT}{dt} = Q$$

where m is the total mass of the domain, which can be prescribed or is calculated automatically from the material properties. The source term Q is calculated from the adjacent entities.

The hot liquid does not fill completely the flask, and air is present in the remaining interior domain. Natural convection can be neglected in this domain, and the Fluid feature with zero velocity is therefore applied.

The walls of the flask, that are made of steel are modeled as thin conductive layers, so that their thickness does not need to be resolved by the mesh.

A heat transfer coefficient is used to model the exchange between the liquid and the walls and air domain above the liquid. Because of the insulating properties of other parts of the vacuum flask, this interface is not limiting the overall heat transfer. Hence the magnitude of the heat transfer coefficient used at this interface does not really matter, provided it is large enough.

Approach 1 — Loading a Heat Transfer Coefficient Function

This application uses a simplified approach and solves the time-dependent thermal-conduction equation making use of a heat transfer coefficient, h, to describe the natural convection cooling on the outside surfaces of the flask. This approach is very powerful in many situations, especially if the main interest is not the flow behavior but rather its cooling power. By using the appropriate h correlations, you can generally arrive at accurate results at a very low computational cost. In addition, many correlations are valid for a wide range of flow regimes, from laminar to turbulent flow. This makes it possible to approach the problem directly without predicting whether the flow is laminar or turbulent.

BOUNDARY CONDITIONS

Vertical boundaries along the axis of symmetry have a symmetry condition (zero gradients, set by COMSOL Multiphysics automatically); the bottom is modeled as perfectly insulated (zero flux). The flask surfaces are exposed to air and are cooled by convection. The use of a thin layer feature models the thickness of the steel shell.

The only remaining energy-balance boundary condition is for the flask surface. In the first approach a convective heat transfer coefficient together with the ambient temperature, 25°C, describes the heat flux.

CONVECTIVE HEAT TRANSFER COEFFICIENT

The outer surfaces dissipate heat via natural convection. This loss is characterized by the convective heat transfer coefficient, h, which in practice you often determine with empirical handbook correlations. Because these correlations depend on the surface temperature, T_{surface}, engineers must estimate T_{surface} and then iterate between h and T_{surface} to obtain a converged value for h. Most of these correlations require tedious computations and property interpolations that make this iterative process quite unpleasant and labor intensive.

A typical handbook correlation (see Ref. 1) for h for the case of natural convection in air on a vertical heated wall $\mathrm{Ra}_L \leq 10^9$ is

$$h = \frac{k \cdot \overline{\mathrm{Nu}_L}}{L}$$

$$\overline{\mathrm{Nu}_L} = 0.68 + \frac{0.670\mathrm{Ra}_L^{1/4}}{\left[1 + \left(\frac{0.492}{\mathrm{Pr}}\right)^{9/16}\right]^{4/9}}$$

where Ra_L and Pr are the Rayleigh and Prandtl dimensionless numbers. A similar relation involving Nusselt numbers holds for inclined and horizontal planes (see *Convective Heat Transfer Correlations* in the *Heat Transfer Module User's Guide* for details).

COMSOL Multiphysics handles these types of nonlinearities internally and adds much convenience to such computations, so there is no need to iterate.

The Heat Transfer Module provides heat transfer coefficient functions that you can access easily in the Convective Heat Flux feature.

Approach 2 — Modeling the External Flow

Another approach for simulating the cooled flask is to produce a model that computes the convective velocity field around the flask in detail. Before proceeding with a simulation of this kind, it is a good idea to try to estimate the Rayleigh number because that number influences the choice between assuming laminar flow and applying a turbulence model.

The Rayleigh number describes the ratio between buoyancy and viscous forces in free convection problems. It is defined as

$$\mathrm{Ra} = \frac{g\alpha_p \Delta T h^3}{\kappa\nu}$$

with g as the gravity (SI unit: $\mathrm{m/s^2}$), κ the thermal diffusivity (SI unit: $\mathrm{m^2/s}$), ΔT the Temperature difference (SI unit: K), h the height of the convective object (SI unit: m), α_p the coefficient of thermal expansion (SI unit: 1/K), and ν the kinematic viscosity (SI unit: $\mathrm{m^2/s}$).

The model's length scale is the length of the heated fluid's flow path, in this example 0.5 m. Notice that this value increases if the modeled flow domain is extended in the direction of the flow. ΔT is about 15 K (assuming that the flask surface temperature is 15°C above the ambient temperature). Together with the material properties of air at atmospheric pressure and T about 25°C the result is below $1 \cdot 10^9$, which indicates that the flow is still laminar rather than turbulent. Thus, it makes sense to model the flow using a physics interface for laminar flow.

BUOYANCY-DRIVEN FLOW

To model nonisothermal buoyancy-driven flow, the following example implements a buoyancy force in the fluid. It provides a generalized Navier–Stokes formulation that takes varying density into account as well as the energy equation.

The buoyancy forces are included using the Gravity feature described in the *Gravity* section in the *CFD Module User's Guide*.

BOUNDARY CONDITIONS

When solving flow problems numerically, your engineering intuition is crucial in setting good boundary conditions. In this problem, the warm flask drives vertical air currents along its walls, and they eventually join in a thermal plume above the top of the flask. Air is pulled from the surroundings toward the flask where it eventually feeds into the vertical flow.

The open boundary condition is a boundary condition for the heat and flow equation and can handle incoming flow with ambient temperature and leaving flow with a-priori unknown temperature.

You would expect this flow to be quite weak and therefore do not anticipate any significant changes in dynamic pressure.

Flow Boundary Conditions

- On the top and right boundaries, the normal stress is zero as an open boundary with ambient temperature of 25°C.
- On the upper-left boundary, the flow domain coincides with the axis of symmetry where the Axial Symmetry condition is applied automatically.
- All other boundaries (the flask surface and the bottom horizontal line) are walls with a No slip condition.

Thermal Boundary Conditions

- The top and right boundaries are the exit and entrance of the flow domain respectively where convection dominates; accordingly, use an open boundary condition.
- Again, the top left is described by axial symmetry, which is set by default.
- Assume that the bottom is perfectly insulated.

All other boundaries (the flask surface) have continuity in temperature and flux by default.

Results and Discussion

Figure 3 shows the temperature distribution in both the flask and the surrounding air. However, the temperature results in the solid parts are close to identical for the case of modeling with a heat transfer coefficient.

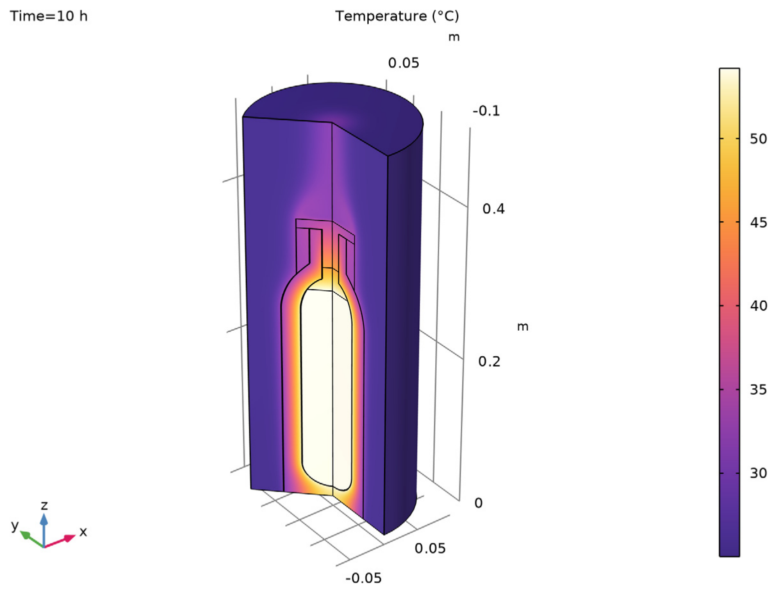


Figure 3: Temperature results for the model including the fluid flow.

One objective of the model is to predict the coffee temperature over time. The plot below shows the results of both approaches and one can see that both results produce almost exactly the same curve.

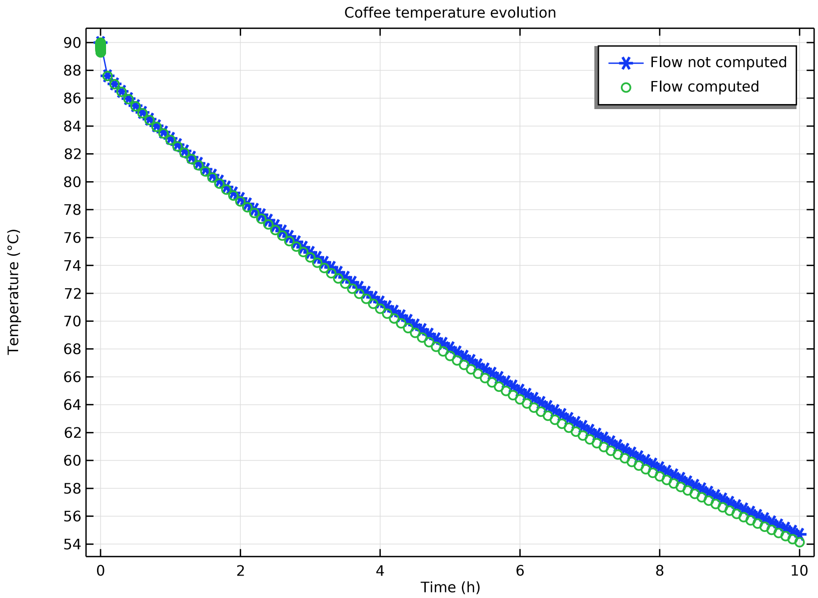


Figure 4: Isothermal domain temperature over time for both approaches.

A second question concerns how the cooling power is distributed on the flask surface. The heat transfer coefficient represents this property. Figure 5 shows a comparison of the predicted distribution of h along the height of the flask between the two models.

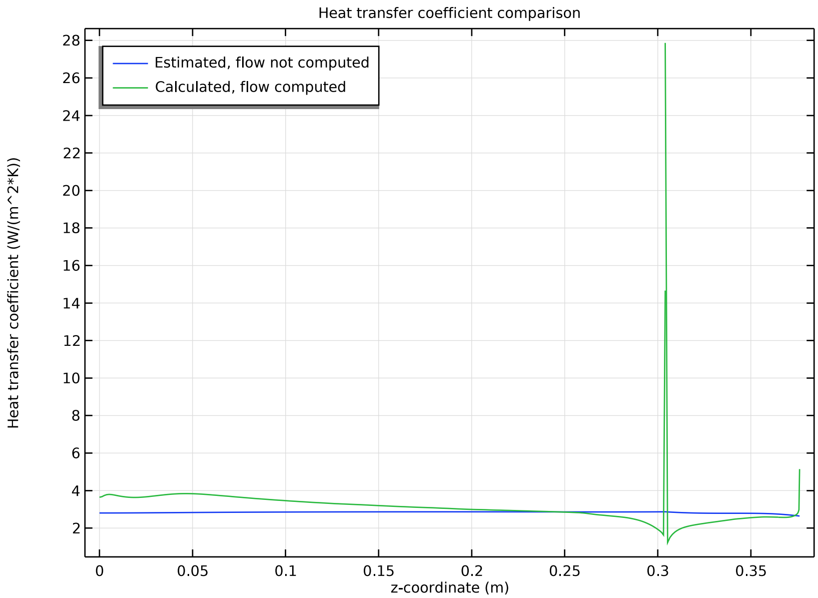


Figure 5: Heat transfer coefficient along the vertical flask walls. Blue line: modeling approach using the heat transfer coefficient library, green line: modeling approach including the fluid flow.

Figure 6 depicts the flow of air around the flask calculated from the flow model. This fluid flow model does a better job at describing local cooling power.

One interesting result is the vortex formed above the lid. It reduces the cooling in this region.

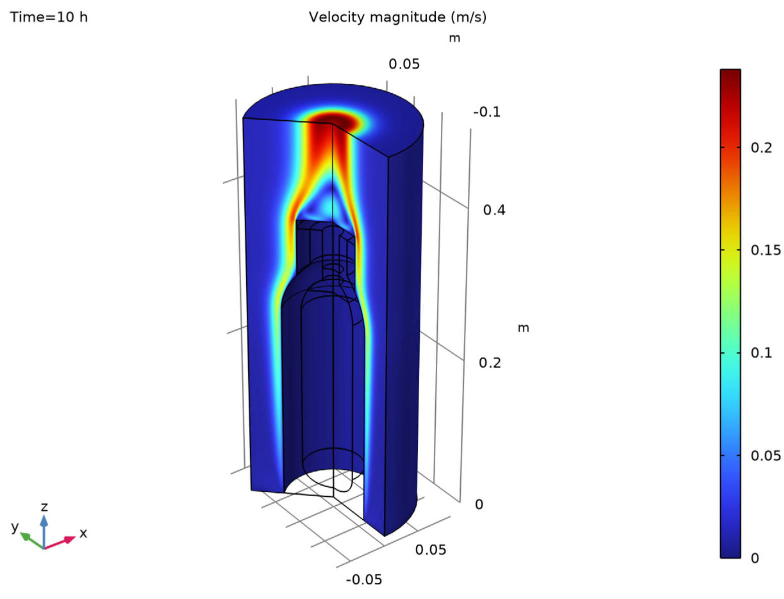


Figure 6: Fluid velocity for air around the flask.

CONCLUSIONS

By using a convective heat flux condition you can easily obtain simulation results. The predicted heat transfer coefficient is in the same range as the results from the model that includes the correlations, and the total cooling power is almost identical.

However, the predefined heat transfer coefficients do not predict the local effects of airflow surrounding the flask. For this purpose, a flow model is more accurate. This means that you can use this type of model to create and calibrate functions for heat transfer coefficients for your geometries. Once calibrated, the functions allow you to use the first approach when solving large-scale and time-dependent models.

Application Library path: `Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/vacuum_flask`

Reference

1. F. Incropera, D. Dewitt, T. Bergman, and A. Lavine, *Fundamentals of Heat and Mass Transfer*, 6th ed., John Wiley & Sons, 2007.

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 **Heat Transfer > Heat Transfer in Solids (ht)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

The geometry sequence for the model is available in a file. If you want to create it from scratch yourself, you can follow the instructions in the Geometry Modeling Instructions section. Otherwise, insert the geometry sequence as follows:

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `vacuum_flask_geom_sequence.mph`.

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

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

You should now see the geometry shown in Figure 2.

In the following section you define selections which will be needed during the model setup, for example the boundaries that represent the steel shell of the flask and the boundaries that are convectively cooled by the surrounding air.

DEFINITIONS

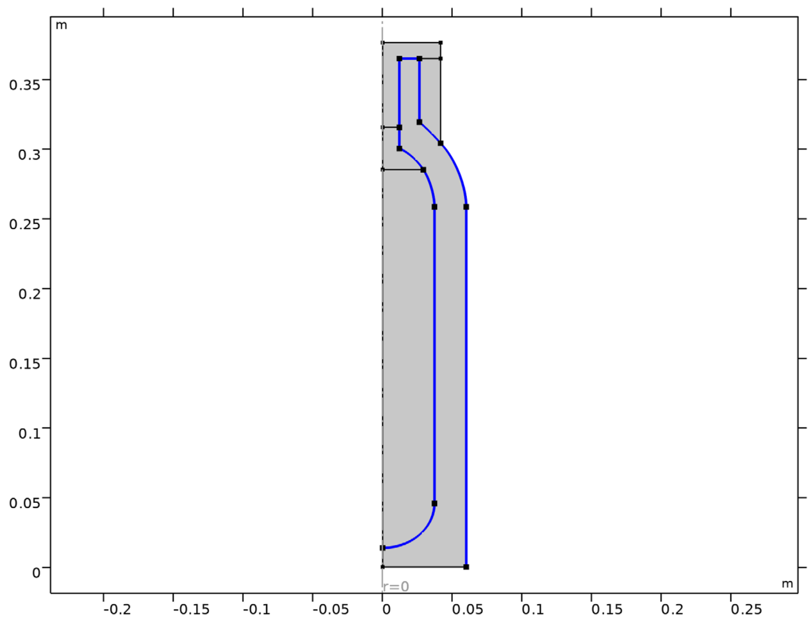
Shell

1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Boundaries 9–12, 14, and 17–22 only.



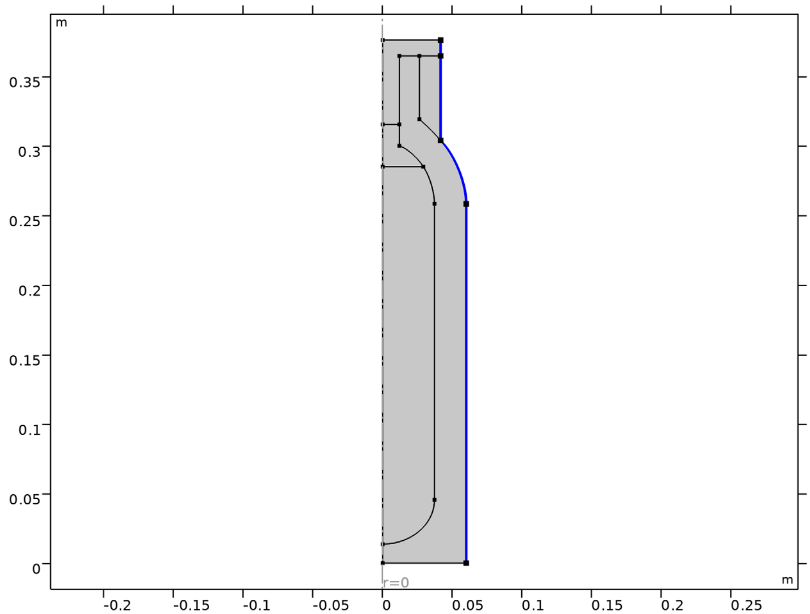
Flask, Vertical Walls

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Flask, Vertical Walls` in the **Label** text field.

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

4 Select Boundaries 15–17 and 22 only.



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
T_amb	25[degC]	298.15 K	Temperature of surrounding air
T_coffee	90[degC]	363.15 K	Coffee temperature
d_shell	0.5[mm]	5E-4 m	Steel-shell thickness
p_amb	1[atm]	1.0133E5 Pa	Ambient pressure

ADD MATERIAL

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

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

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

4 Click the **Add to Component** button in the window toolbar.

5 In the tree, select **Built-in** > **Water, liquid**.

6 Click the **Add to Component** button in the window toolbar.

7 In the tree, select **Built-in** > **Nylon**.

8 Click the **Add to Component** button in the window toolbar.

9 In the tree, select **Built-in** > **Steel AISI 4340**.

10 Click the **Add to Component** button in the window toolbar.

11 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat1)

Leave the default geometric entity selection; subsequent materials that you add will override air as the material for the domains where it does not apply.

The properties of coffee are almost the same as for water.

Water, liquid (mat2)

1 In the **Model Builder** window, click **Water, liquid (mat2)**.

2 Select Domain 2 only.

Nylon (mat3)

1 In the **Model Builder** window, click **Nylon (mat3)**.

2 Select Domains 4 and 5 only.

Steel AISI 4340 (mat4)

1 In the **Model Builder** window, click **Steel AISI 4340 (mat4)**.

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

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

4 From the **Selection** list, choose **Shell**.

5 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Geometric Properties** > **Shell** > **Thickness (lth)**.

6 Click **Add to Material**.

7 Click to collapse the **Material Properties** section. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thickness	lth	`d_shell`	m	Shell

Foam

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

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

3 Select Domain 1 only.

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

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`0.03[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`60[kg/m^3]`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`200[J/(kg*K)]`	J/(kg·K)	Basic

DEFINITIONS (COMP1)

Ambient Properties 1 (ampr1)

1 In the **Physics** toolbar, click **Shared Properties** and choose **Ambient Properties**.

Define the ambient temperature used in boundary conditions and initial values.

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

3 In the T_{amb} text field, type `T_amb`.

HEAT TRANSFER IN SOLIDS (HT)

Fluid 1

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

2 Select Domain 3 only.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

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

3 From the T list, choose **Ambient temperature (ampr1)**.

Assuming that the coffee temperature is uniform and depends on the time only, the domain can be defined as isothermal domain with the initial coffee temperature.

Isothermal Domain 1

1. In the **Physics** toolbar, click **Domains** and choose **Isothermal Domain**.
2. Select Domain 2 only.

   For the **Isothermal Domain Interface** boundary condition use a heat flux condition that describes a good heat transmission from the coffee to the thin conductive shell boundary.

Isothermal Domain Interface 1

1. In the **Model Builder** window, click **Isothermal Domain Interface 1**.
2. In the **Settings** window for **Isothermal Domain Interface**, locate the **Isothermal Domain Interface** section.
3. From the **Interface type** list, choose **Convective heat flux**.
4. In the h text field, type `100[W/(m^2*K)]`.

Initial Values 2

1. In the **Physics** toolbar, click **Domains** and choose **Initial Values**.
2. Select Domain 2 only.
3. In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
4. In the T text field, type `T_coffee`.

   The steel walls of the flask are represented by a special boundary condition for highly conductive layers:

Thin Layer 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.
2. In the **Settings** window for **Thin Layer**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Shell**.
4. Locate the **Layer Model** section. From the **Layer type** list, choose **Thermally thin approximation**.

   To allow for cooling to the surrounding, add heat flux conditions that use appropriate heat transfer coefficients from a library.

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 **Flask, Vertical Walls**.

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

5 From the **Heat transfer coefficient** list, choose **External natural convection**.

6 In the L text field, type `height`.

7 From the T_{ext} list, choose **Ambient temperature (ampr1)**.

Heat Flux 2

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

2 Select Boundary 8 only.

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

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

5 From the **Heat transfer coefficient** list, choose **External natural convection**.

6 From the list, choose **Horizontal plate, upside**.

7 In the L text field, type `radius`.

8 From the T_{ext} list, choose **Ambient temperature (ampr1)**.

MESH 1

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

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

3 From the **Element size** list, choose **Extra fine**.

4 Click **Build All**.

STUDY 1

Step 1: Time Dependent

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

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

3 From the **Time unit** list, choose **h**.

4 In the **Output times** text field, type `range(0,0.1,10)`.

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

RESULTS

Change the unit of the temperature results to degrees Celsius.

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **General** > **Temperature (K)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Temperature	K	°C

8 Click **Apply**.

Temperature (ht)

A 2D temperature plot group showing temperature in the domain and in the layered shell is created by default.

Add a plot from the **Result Templates** demonstrating the 3D temperature distribution:

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Heat Transfer in Solids** > **Temperature (ht)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Temperature 3D (ht)

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

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

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

Approach 2 — Modeling the External Flow

The second modeling approach is done within the same MPH file. This way the results from both approaches can be compared directly. Add a second model as follows:

ADD COMPONENT

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

2 Right-click the root node and choose **Add Component** > **2D Axisymmetric**.

ADD PHYSICS

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

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

3 In the tree, select **Heat Transfer** > **Conjugate Heat Transfer** > **Laminar Flow**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Study 1**.

5 Click the **Add to Component 2** button in the window toolbar.

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

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** > **Time Dependent**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Heat Transfer in Solids (ht)**.

   This way, the new study solves for the coupled heat transfer and laminar flow interfaces only.

5 Click the **Add Study** button in the window toolbar.

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

GEOMETRY 2

The flask's geometry is already present in **Component 1**. Import the geometry sequence from above as follows:

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

2 In the **Settings** window for **Import**, locate the **Source** section.

3 From the **Source** list, choose **Geometry sequence**.

4 From the **Geometry** list, choose **Geometry 1**.

5 Click **Import**.

In this approach, you model the fluid flow explicitly, so you need to add a flow domain to the model.

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `0.1[m]`.

4 In the **Height** text field, type `0.5[m]`.

5 Click **Build All Objects**.

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

Define the same selections as before, which you can use during the model setup and for comparing the results of this approach to the first one.

DEFINITIONS (COMP2)

Shell

1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Boundaries 11–14, 16, 19, and 22–26 only.

Flask, Vertical Walls

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Flask, Vertical Walls` in the **Label** text field.

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

4 Select Boundaries 17–19 and 26 only.

ADD MATERIAL

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

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

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

4 Click the **Add to Component** button in the window toolbar.

5 In the tree, select **Built-in** > **Water, liquid**.

6 Click the **Add to Component** button in the window toolbar.

7 In the tree, select **Built-in** > **Nylon**.

8 Click the **Add to Component** button in the window toolbar.

9 In the tree, select **Built-in** > **Steel AISI 4340**.

10 Click the **Add to Component** button in the window toolbar.

11 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat6)

Again, leave the default geometric entity selection; subsequent materials that you add will override air as the material for the domains where it does not apply.

The properties of coffee are almost the same as for water.

Water, liquid (mat7)

1 In the **Model Builder** window, click **Water, liquid (mat7)**.

2 Select Domain 2 only.

Nylon (mat8)

1 In the **Model Builder** window, click **Nylon (mat8)**.

2 Select Domains 4 and 6 only.

Steel AISI 4340 (mat9)

1 In the **Model Builder** window, click **Steel AISI 4340 (mat9)**.

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

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

4 From the **Selection** list, choose **Shell**.

5 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Geometric Properties** > **Shell** > **Thickness (lth)**.

6 Click **Add to Material**.

7 Click to collapse the **Material Properties** section. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thickness	lth	`d_shell`	m	Shell

Foam

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

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

3 Select Domain 1 only.

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

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`0.03[W/(m*K)]`	W/(m·K)	Basic
Density	rho	`60[kg/m^3]`	kg/m³	Basic
Heat capacity at constant pressure	Cp	`200[J/(kg*K)]`	J/(kg·K)	Basic

After setting up the physics interfaces, the warning for missing material, marked with red crosses in the materials node, will disappear.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Laminar Flow (spf)**.

2 Select Domain 5 only.

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

4 Select the **Include gravity** checkbox.

5 Locate the **Domain Selection** section. Click **Create Selection**.

6 In the **Create Selection** dialog, type `Air` in the **Selection name** text field.

7 Click **OK**.

DEFINITIONS (COMP2)

Ambient Properties 2 (ampr2)

1 In the **Physics** toolbar, click **Shared Properties** and choose **Ambient Properties**.

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

3 In the T_{amb} text field, type `T_amb`.

HEAT TRANSFER IN SOLIDS AND FLUIDS 2 (HT2)

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Heat Transfer in Solids and Fluids 2 (ht2)**.

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

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

Fluid 1

The interface provides nodes for the solid and fluid domain by default and the remaining step is to assign the surrounding air domain to the **Fluid Properties** node. Because the density depends on the temperature and the pressure, choose the calculated pressure as input for this material property.

1. In the **Model Builder** window, under **Component 2 (comp2)** > **Heat Transfer in Solids and Fluids 2 (ht2)** click **Fluid 1**.
2. In the **Settings** window for **Fluid**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Air**.

   To get a good initial guess for the solver, set the initial value for the temperature to the ambient temperature.

Initial Values 1

1. In the **Model Builder** window, click **Initial Values 1**.
2. In the **Settings** window for **Initial Values**, locate the **Initial Values** section.
3. From the $T2$ list, choose **Ambient temperature (ampr2)**.

LAMINAR FLOW (SPF)

In the **Model Builder** window, under **Component 2 (comp2)** click **Laminar Flow (spf)**.

Open Boundary 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.
2. Select Boundaries 10 and 21 only.

HEAT TRANSFER IN SOLIDS AND FLUIDS 2 (HT2)

In the **Model Builder** window, under **Component 2 (comp2)** click **Heat Transfer in Solids and Fluids 2 (ht2)**.

Fluid 2

1. In the **Physics** toolbar, click **Domains** and choose **Fluid**.
2. Select Domain 3 only.

Isothermal Domain 1

1. In the **Physics** toolbar, click **Domains** and choose **Isothermal Domain**.
2. Select Domain 2 only.

Isothermal Domain Interface 1

1. In the **Model Builder** window, click **Isothermal Domain Interface 1**.

2 In the **Settings** window for **Isothermal Domain Interface**, locate the **Isothermal Domain Interface** section.

3 From the **Interface type** list, choose **Convective heat flux**.

4 In the h text field, type `100[W/(m^2*K)]`.

Initial Values 2

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

2 Select Domain 2 only.

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

4 In the $T2$ text field, type `T_coffee`.

Thin Layer 1

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

2 In the **Settings** window for **Thin Layer**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Shell**.

4 Locate the **Layer Model** section. From the **Layer type** list, choose **Thermally thin approximation**.

Open Boundary 1

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

2 Select Boundaries 10 and 21 only.

3 In the **Settings** window for **Open Boundary**, locate the **Upstream Properties** section.

4 In the T_{ustr} text field, type `T_amb`.

MESH 2

Use a finer mesh to get a good resolution of the flow field.

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Mesh 2**.

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

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

4 Click **Build All**.

Large differences of the initial values `T_coffee` and `T_amb` can cause numerical instabilities if time steps become too large. Forcing the solver to use small time steps in the beginning until the strong gradients are blurred out helps to overcome this problem. To do so, add two time dependent solver steps in one study. Set up the time intervals as follows:

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

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

3 From the **Time unit** list, choose **h**.

4 In the **Output times** text field, type `range(0,0.1,10)[s] range(0.1,0.1,10)`.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Steps taken by solver** list, choose **Intermediate**.

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

For clarity, assort the plot groups by study.

RESULTS

Temperature (ht), Temperature 3D (ht)

Right-click and choose **Group**.

Study 1

In the **Settings** window for **Group**, type `Study 1` in the **Label** text field.

Pressure (spf), Temperature (ht2), Temperature and Fluid Flow (nitf1), Velocity (spf), Velocity, 3D (spf)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Temperature (ht2)**, **Velocity (spf)**, **Pressure (spf)**, **Velocity, 3D (spf)**, and **Temperature and Fluid Flow (nitf1)**.

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

Study 2

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

Add a plot from the **Result Templates** to get the 3D distribution of the temperature field (as in Figure 3).

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 2/Solution 2 (3) (sol2)** > **Heat Transfer in Solids and Fluids 2** > **Temperature (ht2)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Temperature 3D (ht2)

In the **Settings** window for **3D Plot Group**, type `Temperature 3D (ht2)` in the **Label** text field.

Velocity, 3D (spf)

The velocity field is automatically shown in its dedicated plot group (see Figure 6).

Study 2

Create a 2D plot of the velocity streamlines.

Velocity, Streamlines

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 From the **Time (h)** list, choose **10**.

5 Locate the **Color Legend** section. Select the **Show units** checkbox.

Streamline 1

1 In the **Velocity, Streamlines** toolbar, click **Streamline**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2 (comp2)** > **Laminar Flow** > **Velocity and pressure** > **u,w - Velocity field**.

3 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Magnitude controlled**.

4 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

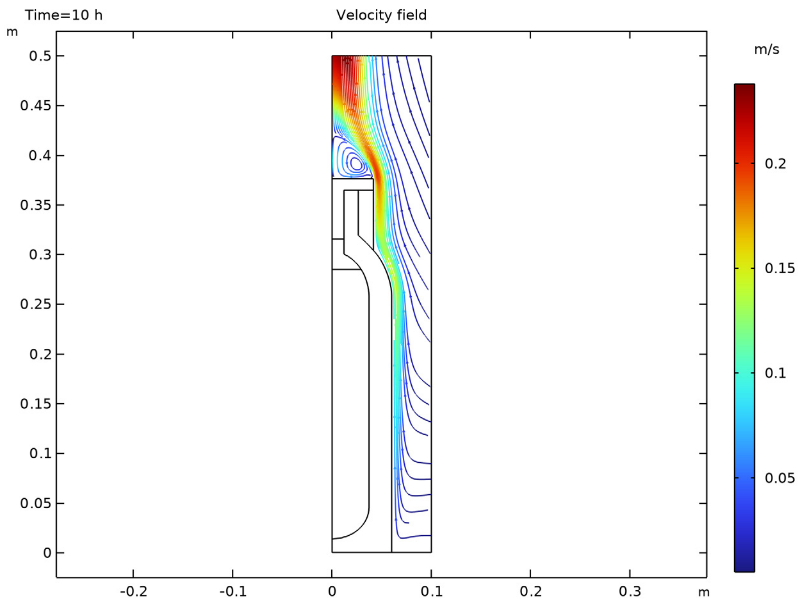
Color Expression 1

1 In the **Velocity, Streamlines** toolbar, click **Color Expression**.

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2 (comp2)** > **Laminar Flow** > **Velocity and pressure** > **spf.U - Velocity magnitude - m/s**.

3 In the **Velocity, Streamlines** toolbar, click **Plot**.

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



To compare both approaches, evaluate the coffee temperature over time and use a 1D Plot to visualize the results. The solutions are stored under the **Datasets** node and for each plot you can choose which dataset should be used.

Coffee Temperature vs. Time

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

2 In the **Settings** window for **1D Plot Group**, type `Coffee Temperature vs. Time` 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 `Coffee temperature evolution`.

5 Locate the **Plot Settings** section.

6 Select the **y-axis label** checkbox. In the associated text field, type `Temperature (°C)`.

Global 1

1 In the **Coffee Temperature vs. Time** 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 **Component 1 (comp1)** > **Heat Transfer in Solids** > **Temperature** > **ht.id1.T - Isothermal domain temperature - K**.

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

Expression	Unit	Description
`ht.id1.T`	`°C`	`Isothermal domain temperature - flow not computed`

4 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

5 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
`Flow not computed`

Coffee Temperature vs. Time

In the **Model Builder** window, click **Coffee Temperature vs. Time**.

Global 2

1 In the **Coffee Temperature vs. Time** toolbar, click **Global**.

To compare the heat transfer coefficients for the two modeling approaches (Figure 4), do the following:

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 2 (comp2)** > **Heat Transfer in Solids and Fluids 2** > **Temperature** > **ht2.id1.T - Isothermal domain temperature - K**.

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

Expression	Unit	Description
`ht2.id1.T`	`°C`	`Isothermal domain temperature - flow computed`

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

8 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
Flow computed

10 In the **Coffee Temperature vs. Time** toolbar, click **Plot**.

Heat Transfer Coefficient

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

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

3 Locate the **Data** section. From the **Time selection** list, choose **Last**.

4 Locate the **Title** section. From the **Title type** list, choose **Manual**.

5 In the **Title** text area, type `Heat transfer coefficient comparison`.

6 Locate the **Plot Settings** section.

7 Select the **y-axis label** checkbox. In the associated text field, type `Heat transfer coefficient (W/(m^2*K))`.

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

Line Graph 1

1 Right-click **Heat Transfer Coefficient** and choose **Line Graph**.

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

3 From the **Selection** list, choose **Flask, Vertical Walls**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `ht.hf1.h`.

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

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

7 Click to expand the **Legends** section. Select the **Show legends** checkbox.

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

9 In the table, enter the following settings:

Legends
Estimated, flow not computed

Heat Transfer Coefficient

In the **Model Builder** window, click **Heat Transfer Coefficient**.

Line Graph 2

1 In the **Heat Transfer Coefficient** toolbar, click **Line Graph**.
2 In the **Settings** window for **Line Graph**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2/Solution 2 (3) (sol2)**.
4 From the **Time selection** list, choose **Last**.
5 Locate the **Selection** section. From the **Selection** list, choose **Flask, Vertical Walls**.
6 Locate the **y-Axis Data** section. In the **Expression** text field, type `abs(ht2.ntflux)/(T2-T_amb)`.
7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
8 In the **Expression** text field, type `z`.
9 Locate the **Legends** section. Select the **Show legends** checkbox.
10 From the **Legends** list, choose **Manual**.
11 In the table, enter the following settings:

Legends
Calculated, flow computed

Heat Transfer Coefficient

1 In the **Model Builder** window, click **Heat Transfer Coefficient**.
2 In the **Heat Transfer Coefficient** toolbar, click **Plot**.

Geometry Modeling Instructions

If you want to create the geometry yourself, follow these steps.

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
height	380[mm]	0.38 m	Flask height
radius	40[mm]	0.04 m	Bottleneck radius

GEOMETRY 1

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `1.04*radius`.

4 In the **Height** text field, type `0.96*height`.

5 Locate the **Position** section. In the **r** text field, type `0*radius`.

6 In the **z** text field, type `0*height`.

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.

5 In the **r** text field, type `0 1.5*radius 1.5*radius 1.5*radius`.

6 In the **z** text field, type `0 0 0 0.68*height`.

Quadratic Bézier 1 (qb1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Quadratic Bézier**.

2 In the **Settings** window for **Quadratic Bézier**, locate the **Control Points** section.

3 In row **1**, set **r** to `1.5*radius`.

4 In row **2**, set **r** to `1.5*radius`.

5 In row **3**, set **r** to `1.04*radius`.

6 In row **1**, set **z** to `0.68*height`.

7 In row **2**, set **z** to `0.751*height`.

8 In row **3**, set **z** to `0.80*height`.

Quadratic Bézier 2 (qb2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Quadratic Bézier**.

2 In the **Settings** window for **Quadratic Bézier**, locate the **Control Points** section.

3 In row **1**, set **r** to `1.04*radius`.

4 In row **2**, set **r** to `0.88*radius`.

5 In row **3**, set **r** to `0.66*radius`.

6 In row **1**, set **z** to `0.80*height`.

7 In row **2**, set **z** to `0.82*height`.

8 In row **3**, set **z** to `0.84*height`.

Polygon 2 (pol2)

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

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.

5 In the **r** text field, type `0.66*radius 0.66*radius 0.66*radius 0.3*radius 0.3*radius 0.3*radius 0.3*radius 0.3*radius`.

6 In the **z** text field, type `0.84*height 0.96*height 0.96*height 0.96*height 0.96*height 0.83*height 0.83*height 0.79*height`.

Quadratic Bézier 3 (qb3)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Quadratic Bézier**.

2 In the **Settings** window for **Quadratic Bézier**, locate the **Control Points** section.

3 In row **1**, set **r** to `0.3*radius`.

4 In row **2**, set **r** to `0.56*radius`.

5 In row **3**, set **r** to `0.73*radius`.

6 In row **1**, set **z** to `0.79*height`.

7 In row **2**, set **z** to `0.78*height`.

8 In row **3**, set **z** to `0.75*height`.

Quadratic Bézier 4 (qb4)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Quadratic Bézier**.

2 In the **Settings** window for **Quadratic Bézier**, locate the **Control Points** section.

3 In row **1**, set **r** to `0.73*radius`.

4 In row **2**, set **r** to `0.93*radius`.

5 In row **3**, set **r** to `0.93*radius`.

6 In row **1**, set **z** to `0.75*height`.

7 In row **2**, set **z** to `0.72*height`.

8 In row **3**, set **z** to `0.68*height`.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **r** text field, type `0.93*radius`.

6 Locate the **Endpoint** section. In the **r** text field, type `0.93*radius`.

7 Locate the **Starting Point** section. In the **z** text field, type `0.68*height`.

8 Locate the **Endpoint** section. In the **z** text field, type `0.12*height`.

Quadratic Bézier 5 (qb5)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Quadratic Bézier**.

2 In the **Settings** window for **Quadratic Bézier**, locate the **Control Points** section.

3 In row **1**, set **r** to `0.93*radius`.

4 In row **2**, set **r** to `0.93*radius`.

5 In row **3**, set **r** to `0*radius`.

6 In row **1**, set **z** to `0.12*height`.

7 In row **2**, set **z** to `0.036*height`.

8 In row **3**, set **z** to `0.036*height`.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 Click to select the **Activate Selection** toggle button for **Start vertex**.

4 On the object **pol1**, select Point 1 only.

5 Locate the **Endpoint** section. Click to select the **Activate Selection** toggle button for **End vertex**.

6 On the object **qb5**, select Point 2 only.

7 Click **Build Selected**.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Select the objects **ls1**, **ls2**, **pol1**, **pol2**, **qb1**, **qb2**, **qb3**, **qb4**, and **qb5** only.

Convert to Solid 2 (csol2)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

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

Line Segment 3 (ls3)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **r** text field, type `0*radius`.

6 In the **z** text field, type `0.83*height`.

7 Locate the **Endpoint** section. In the **r** text field, type `0.3*radius`.

8 In the **z** text field, type `0.83*height`.

Line Segment 4 (ls4)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **r** text field, type `0*radius`.

5 In the **z** text field, type `0.75*height`.

6 Locate the **Endpoint** section. Click to select the **Activate Selection** toggle button for **End vertex**.

7 On the object **csol2**, select Point 9 only.

Polygon 3 (pol3)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 From the **Data source** list, choose **Vectors**.

4 In the **r** text field, type `0*radius 1.04*radius 1.04*radius 0*radius 0*radius`.

5 In the **z** text field, type `0.96*height 0.96*height 0.99*height 0.99*height 0.96*height`.

Ignore Edges 1 (ige1)

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

2 On the object **fin**, select Boundaries 9 and 17 only.

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