

Natural Convection Cooling of a Vacuum Flask

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.

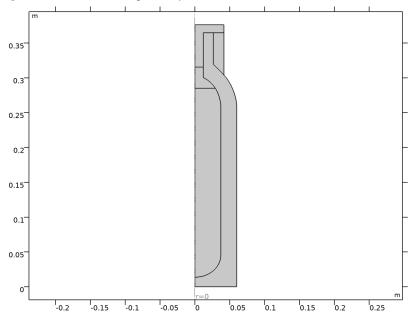


Figure 2 shows the model geometry.

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 I: 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 thermalconduction 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 $Ra_I \le 10^9$ is

$$h = \frac{k \cdot \overline{Nu_L}}{L}$$

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

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

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

with g as the gravity (SI unit: m/s²), κ the thermal diffusivity (SI unit: m²/s), ΔT the Temperature difference (SI unit: K), h the height of the convective object (SI unit: m), α_n the coefficient of thermal expansion (SI unit: 1/K), and v the kinematic viscosity (SI unit: 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 $1\cdot10^9$, 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.

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.

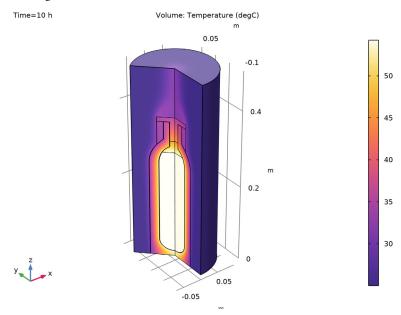


Figure 3: Temperature results for the model including the fluid flow.

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.

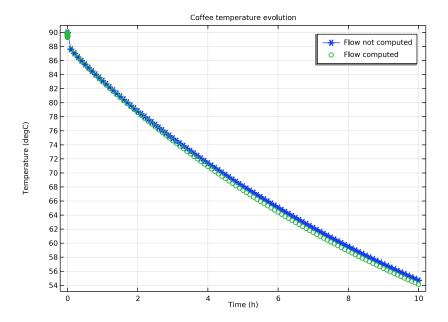


Figure 4: Isothermal domain temperature over time for both approaches.

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.

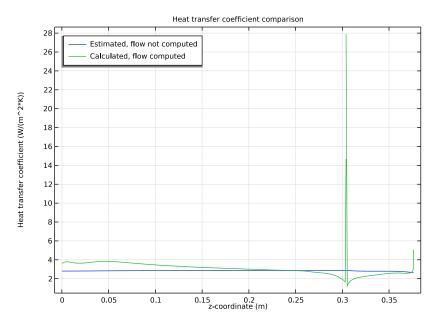


Figure 5: Heat transfer coefficient along the vertical flask walls. Blue line: modeling approach using the heat transfer coefficient library, green line: modeling approach including the fluid flow.

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.

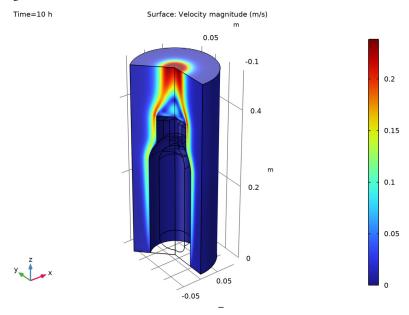


Figure 6: Fluid velocity for air around the flask.

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

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

- I 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 M Done.

GEOMETRY I

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:

- I 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.

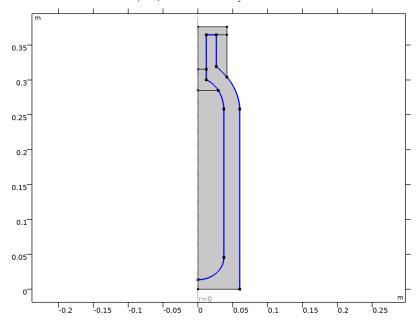
You should now see the geometry shown in Figure 2.

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

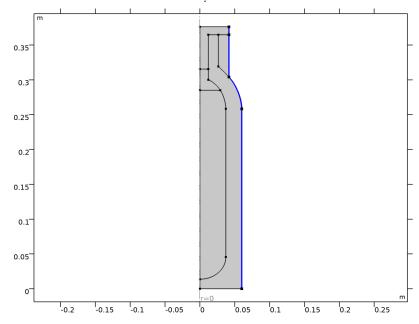
- I In the **Definitions** toolbar, click **\(\big|_{\text{a}} \) Explicit**.
- 2 In the Settings window for Explicit, type Shell in the Label text field.
- 3 Locate the Input Entities section. From the Geometric entity level list, choose Boundary.
- **4** Select Boundaries 9–12, 14, and 17–22 only.



Flask, Vertical Walls

- I In the **Definitions** toolbar, click **\(\bigcap_{\text{a}} \) 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

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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

- I 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>Air.

- 4 Click Add to Component in the window toolbar.
- 5 In the tree, select Built-in>Water, liquid.
- 6 Click Add to Component in the window toolbar.
- 7 In the tree, select Built-in>Nylon.
- **8** Click **Add to Component** in the window toolbar.
- 9 In the tree, select Built-in>Steel AISI 4340.
- **10** Click **Add to Component** in the window toolbar.
- II In the Home 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)

- I In the Model Builder window, click Water, liquid (mat2).
- 2 Select Domain 2 only.

Nylon (mat3)

- I In the Model Builder window, click Nylon (mat3).
- **2** Select Domains 4 and 5 only.

Steel AISI 4340 (mat4)

- I 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

- I 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)	Basic
Density	rho	60	kg/m³	Basic
Heat capacity at constant pressure	Ср	200	J/(kg·K)	Basic

DEFINITIONS (COMPI)

Ambient Properties I (ampr I)

- I 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 $T_{\rm amb}$ text field, type T_amb.

HEAT TRANSFER IN SOLIDS (HT)

Fluid 1

- I In the Model Builder window, under Component I (compl) right-click Heat Transfer in Solids (ht) and choose Fluid.
- **2** Select Domain 3 only.

Initial Values 1

- I In the Model Builder window, click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- 3 From the T list, choose Ambient temperature (amprl).

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

- I 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

- I In the Model Builder window, click Isothermal Domain Interface I.
- 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.

Initial Values 2

- I 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 I

- I 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 I

- I 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 $T_{\rm ext}$ list, choose Ambient temperature (amprl).

Heat Flux 2

- I 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 (amprl).

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Physics-Controlled Mesh section.
- 3 From the Element size list, choose Extra fine.
- 4 Click III Build All.

STUDY I

Step 1: Time Dependent

- I In the Model Builder window, under Study I click Step I: 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 Home toolbar, click **Compute**.

RESULTS

Temperature (ht)

A 2D temperature plot group showing temperature in the domain and in the layered shell is created by default. To display the temperatures in degrees Celsius, you can edit these existing plots:

Domain

- I In the Model Builder window, expand the Temperature (ht) node, then click Domain.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 From the Unit list, choose degC.

Layered Shell

- I In the Model Builder window, click Layered Shell.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 From the Unit list, choose degC.
- 4 In the Temperature (ht) toolbar, click **Plot**.

Add a predefined plot demonstrating the 3D temperature distribution:

ADD PREDEFINED PLOT

- I In the Home toolbar, click Windows and choose Add Predefined Plot.
- 2 Go to the Add Predefined Plot window.
- 3 In the tree, select Study I/Solution I (soll)>Heat Transfer in Solids>Temperature (ht).
- 4 Click Add Plot in the window toolbar.

RESULTS

Temperature 3D (ht)

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

Domain

- I In the Model Builder window, expand the Temperature 3D (ht) node, then click Domain.
- 2 In the Settings window for Volume, locate the Expression section.
- 3 From the Unit list, choose degC.

Layered Shell

- I In the Model Builder window, click Layered Shell.
- 2 In the Settings window for Volume, locate the Expression section.

- 3 From the Unit list, choose degC.
- 4 In the Temperature 3D (ht) toolbar, click Plot.
- **5** Click the **Zoom Extents** button in the **Graphics** toolbar.

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

In the Model Builder window, right-click the root node and choose Add Component> 2D Axisymmetric.

ADD PHYSICS

- I In the Home toolbar, click 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 check box for Study 1.
- 5 Click Add to Component 2 in the window toolbar.
- 6 In the Home toolbar, click and Physics to close the Add Physics window.

ADD STUDY

- I 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** check box for Heat Transfer in Solids (ht).
 - This way, the new study solves for the coupled heat transfer and laminar flow interfaces only.
- 5 Click Add Study 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 I (impl)

- I In the **Home** toolbar, click **Import**.
- 2 In the Settings window for Import, locate the Import 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 I (rI)

- I 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

- I 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

- I In the **Definitions** toolbar, click **\(\bigcap_{\bigcap} \) 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

- I 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>Air.
- 4 Click Add to Component in the window toolbar.
- 5 In the tree, select Built-in>Water, liquid.
- **6** Click **Add to Component** in the window toolbar.
- 7 In the tree, select Built-in>Nylon.
- **8** Click **Add to Component** in the window toolbar.
- 9 In the tree, select Built-in>Steel AISI 4340.
- **10** Click **Add to Component** in the window toolbar.
- II In the Home 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)

- I In the Model Builder window, click Water, liquid (mat7).
- 2 Select Domain 2 only.

Nylon (mat8)

- I In the Model Builder window, click Nylon (mat8).
- 2 Select Domains 4 and 6 only.

Steel AISI 4340 (mat9)

- I 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

- I 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)	Basic
Density	rho	60	kg/m³	Basic
Heat capacity at constant pressure	Ср	200	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)

- I 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** check box.
- 5 Locate the Domain Selection section. Click \(\frac{1}{2} \) Create Selection.
- 6 In the Create Selection dialog box, type Air in the Selection name text field.
- 7 Click OK.

DEFINITIONS (COMP2)

Ambient Properties 2 (ampr2)

- I 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 $T_{\rm amb}$ text field, type T_amb.

HEAT TRANSFER IN SOLIDS AND FLUIDS 2 (HT2)

- I 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.

- I 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

- I In the Model Builder window, click Initial Values I.
- 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 I

- I 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

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

Isothermal Domain 1

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

Isothermal Domain Interface 1

- I In the Model Builder window, click Isothermal Domain Interface I.
- 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.

Initial Values 2

- I 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 Laver 1

- I 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 I

- I 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.

- I 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

- I 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)

- I 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)

- I In the Model Builder window, under Results, Ctrl-click to select Temperature (ht) and Temperature 3D (ht).
- 2 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)

I 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

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

Temperature (ht2)

Change the default temperature plot to use degrees Celsius as quantity unit (as in Figure 3).

Domain

- I In the Model Builder window, expand the Temperature (ht2) node, then click Domain.
- 2 In the Settings window for Surface, locate the Expression section.
- **3** From the **Unit** list, choose **degC**.

Layered Shell

- I In the Model Builder window, click Layered Shell.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 From the Unit list, choose degC.
- 4 Click the **Zoom Extents** button in the **Graphics** toolbar.

Study 2

Add a predefined plot to get the 3D distribution of the temperature field (as in Figure 3).

ADD PREDEFINED PLOT

- I In the Home toolbar, click Windows and choose Add Predefined Plot.
- 2 Go to the Add Predefined Plot window.
- 3 In the tree, select Study 2/Solution 2 (3) (sol2)>Heat Transfer in Solids and Fluids 2> Temperature (ht2).
- 4 Click Add Plot in the window toolbar.

RESULTS

Temberature 3D (ht2)

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

Domain

- I In the Model Builder window, expand the Temperature 3D (ht2) node, then click Domain.
- 2 In the Settings window for Volume, locate the Expression section.

3 From the Unit list, choose degC.

Layered Shell

- I In the Model Builder window, click Layered Shell.
- 2 In the Settings window for Volume, locate the Expression section.
- 3 From the Unit list, choose degC.

Velocity, 3D (sbf)

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

Create a 2D plot of the velocity streamlines.

Velocity, Streamlines

- I In the Home toolbar, click Add Plot Group and choose 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** check box.

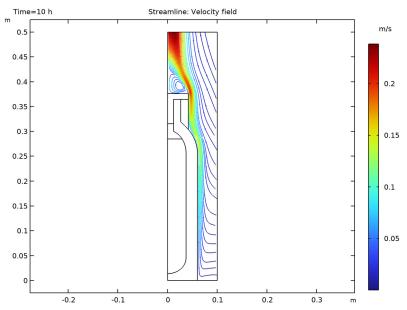
Streamline 1

- I Right-click Velocity, Streamlines and choose 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 I

- I 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

- I In the Home toolbar, click Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for ID 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 check box. In the associated text field, type Temperature (degC).

Global I

- I 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 I (compl)> 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	degC	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

I 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	degC	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 **10** Plot.

Heat Transfer Coefficient

- I In the Home toolbar, click **Add Plot Group** and choose ID Plot Group.
- 2 In the Settings window for ID 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 check box. 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

- I 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** check box.
- 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

- I In the Heat Transfer Coefficient toolbar, click \(\subseteq \) 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** check box.
- 10 From the Legends list, choose Manual.
- II In the table, enter the following settings:

Legends			
Calculated,	flow	computed	

Heat Transfer Coefficient

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

Geometry Modeling Instructions

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

GLOBAL DEFINITIONS

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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 I

Rectangle I (rI)

- I 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 I (boll)

- I 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 I (qb I)

- I 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 I, 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 I, 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)

- I 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 I, 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 I, 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 (bol2)

- I 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 0.83*height 0.83*height 0.79*height.

Quadratic Bézier 3 (ab3)

- I 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 I, 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 I, 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.

Ouadratic Bézier 4 (ab4)

- I 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 I, 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 I, 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 I (Is I)

- I 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)

- I 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 I, 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 I, 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 (Is2)

- I 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 **poll**, 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 Pauld Selected.

Convert to Solid I (csoll)

- I In the Geometry toolbar, click Conversions and choose Convert to Solid.
- 2 Select the objects Is1, Is2, pol1, pol2, qb1, qb2, qb3, qb4, and qb5 only.

Convert to Solid 2 (csol2)

- I 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 (Is3)

- I 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 (Is4)

- I 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 Del Activate Selection toggle button for End vertex.
- **7** On the object **csol2**, select Point 9 only.

Polygon 3 (bol3)

- I 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 0.96*height.

Ignore Edges I (ige I)

- I In the Geometry toolbar, click \to 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**.