

Smoke from an Incense Stick — Visualizing the Laminar to Turbulent Transition in Natural Convection

This model is licensed under the COMSOL Software License Agreement 6.1. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This example considers natural convection of air above a smoldering incense stick. In reality, this kind of flow often shows a transition from laminar to turbulent, which is nicely visualized by the smoke produced by the slow burning of the incense. The aim of this model is to simulate this transition and to illustrate it by reproducing the shape of the smoke plume. To be able to capture the transition to turbulence, the model uses the Nonisothermal Flow, LES RBVM interface. For the visualization of the flow, the Particle Tracing for Fluid Flow interface is used.

Note: This model requires both the CFD Module and the Particle Tracing Module.

Model Definition

The flow domain is a cylinder with a diameter of 40 cm and a height of 185 cm. At the bottom of the cylinder, a rectangular holder (20-by-5-by-1 cm) and an incense stick (diameter 0.5 cm and length 20 cm) are located. The model geometry includes a number of cylindrical domains that allow for a better control of the mesh. See Figure 1 for a graphic representation of the geometry.

In the literature, peak temperatures in smoldering materials of around 500–700°C are reported (see Ref. 1). Here it is assumed that the tip of the incense stick (2 cm long) has an average temperature of 520°C. In an open environment, the combustion process in and the air flow around the incense stick will be influenced by variations in the incense composition, nearby moving objects, nearby heat sources, and so on. To get a lifelike smoke plume from the simulation, (some of) these disturbances need to be included in the model. In the present case, a more realistic plume shape is created by assuming a nonconstant burning rate, which is modeled by random fluctuations of the tip temperature:

$$T_{\rm tip} = 793.15 \text{K} + 50 \text{K} \times \text{rand}(\text{floor}(t)) \tag{1}$$

Here, the random function has a uniform distribution with mean 0 and range 2. The simulated time interval is 10 seconds.

The Nonisothermal Flow, LES RBVM interface is set up to treat the flow as incompressible and to use the Boussinesq approximation to compute the buoyancy forces for the natural convection. The density and viscosity of the flowing air are taken from the Air material from COMSOL Multiphysics' material library. Initially, the air is at rest and the temperature is equal to the ambient temperature of 20°C.

The smoke particles are assumed to be small and light enough to almost immediately adopt the velocity of the surrounding air. This means that the particle inertia can be neglected, allowing for a massless formulation of the particle dynamics, which in turn generally allows for larger time steps in the time-dependent solver. The particles are released each 0.01 second in a 11-by-11 grid on a patch of 1 cm² that is located approximately 13 mm above the tip of the incense stick.



Figure 1: Wireframe rendering of the complete geometry including the domains for controlling the mesh. The incense stick and holder are indicated in red.

Results and Discussion

In Figure 2, slice plots in the *xz*-plane of the velocity magnitude are shown for t = 4 s, t = 6 s, t = 8 s, and t = 10 s, respectively. The velocity magnitude is highest just above the

tip of the incense stick, with a maximum value of approximately 0.7 m/s. From these plots it is clearly seen that the air flow is initially laminar as it rises from the incense stick, and that at a distance of 40 to 50 cm above the stick it develops into a turbulent flow. As time progresses, the turbulent part of the flow grows and spreads upward and sideways.



Figure 2: Slice plots in the center xz-plane of the velocity magnitude for t=4 s (top left), t=6 s (top right), t=8 s (bottom left), and t=10 s (bottom right).

In Figure 3, the particle positions are plotted at different moments in time. The plot shows, as expected, the same pattern as the velocity plots: first, right above the hot tip, the particles travel upward in more or less straight lines, following the laminar part of the air flow. Then, at a certain height above the tip of the incense stick, the flow becomes unstable and transitions to a less organized pattern, spreading the smoke plume horizontally. The height at which the flow becomes unstable is seen to be quite constant in time.



Figure 3: The shape of the smoke plume visualized by the particle positions after (from left to right) t=2 s, t=4 s, t=6 s, t=8 s, and t=10 s, respectively.

Reference

1. G. Rein, "Smouldering Combustion Phenomena in Science and Technology," *Int. Rev. Chem. Eng.*, vol. 1, pp. 3–18, 2009.

Application Library path: CFD_Module/Nonisothermal_Flow/incense_stick

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 间 3D.
- 2 In the Select Physics tree, select Fluid Flow>Nonisothermal Flow>Large Eddy Simulation> LES RBVM.
- 3 Click Add.
- 4 In the Select Physics tree, select Fluid Flow>Particle Tracing> Particle Tracing for Fluid Flow (fpt).
- 5 Click Add.
- 6 Click 🔿 Study.
- 7 In the Select Study tree, select General Studies>Time Dependent.
- 8 Click **M** Done.

DEFINITIONS

Random I (rn1)

- I In the Model Builder window, expand the Component I (compl)>Definitions node.
- 2 Right-click Definitions and choose Functions>Random.
- 3 In the Settings window for Random, locate the Parameters section.
- 4 In the Range text field, type 2.
- 5 Select the Use random seed check box.
- 6 In the Random seed text field, type 11301.

Analytic I (an I)

- I In the Home toolbar, click f(x) Functions and choose Local>Analytic.
- 2 In the Settings window for Analytic, locate the Definition section.
- 3 In the Expression text field, type rn1(floor(x)).

GEOMETRY I

The computational domain is essentially a cylinder. However, to be able to control the mesh size in various parts of the domain, the geometry will be built as a number of concentric cylinders, varying in location and height. This is achieved by revolving a plane geometry, consisting of rectangles, around the *z*-axis.

Work Plane I (wp1)

- I In the Geometry toolbar, click 😓 Work Plane.
- 2 In the Settings window for Work Plane, locate the Plane Definition section.

3 From the Plane list, choose xz-plane.

Work Plane I (wpI)>Plane Geometry

In the Model Builder window, click Plane Geometry.

Work Plane I (wp1)>Rectangle I (r1)

- I In the Work Plane toolbar, click Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type **0.4**.
- 4 In the **Height** text field, type 1.85.
- 5 Click to expand the Layers section. Select the Layers to the right check box.
- 6 Clear the Layers on bottom check box.
- 7 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.12
Layer 2	0.08

Work Plane 1 (wp1)>Rectangle 2 (r2)

I In the Work Plane toolbar, click Rectangle.

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

- **3** In the **Width** text field, type **0.4**.
- 4 In the **Height** text field, type 1.35.

Work Plane I (wp1)>Rectangle 3 (r3)

- I In the Work Plane toolbar, click 📃 Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type **0.2**.
- 4 In the **Height** text field, type 0.53.
- **5** Locate the **Position** section. In the **yw** text field, type **0.12**.
- 6 Locate the Layers section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.18

Work Plane 1 (wp1)>Rectangle 4 (r4)

I In the Work Plane toolbar, click 📃 Rectangle.

- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type 0.2.
- 4 In the **Height** text field, type 1.73.
- 5 Locate the **Position** section. In the **yw** text field, type 0.12.
- 6 Locate the Layers section. Select the Layers to the right check box.
- 7 Clear the Layers on bottom check box.
- 8 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.08
Layer 2	0.085

Revolve 1 (rev1)

In the Model Builder window, right-click Geometry I and choose Revolve.

Cylinder I (cyl1)

- I In the **Geometry** toolbar, click **(D)** Cylinder.
- 2 In the Settings window for Cylinder, locate the Size and Shape section.
- 3 In the Radius text field, type 0.25e-2.
- 4 In the **Height** text field, type 0.2.
- 5 Locate the Position section. In the x text field, type -0.005.
- 6 In the z text field, type 0.145.
- 7 Locate the Axis section. From the Axis type list, choose Cartesian.
- **8** In the **x** text field, type **1**.
- **9** In the **z** text field, type -1.
- 10 Click to expand the Layers section. Select the Layers on bottom check box.
- II Clear the Layers on side check box.

12 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.02

Block I (blkI)

- I In the **Geometry** toolbar, click 🗍 **Block**.
- 2 In the Settings window for Block, locate the Size and Shape section.

- **3** In the **Width** text field, type 0.2.
- 4 In the **Depth** text field, type 0.05.
- 5 In the **Height** text field, type 0.01.
- 6 Locate the Position section. In the x text field, type -0.05.
- **7** In the **y** text field, type -0.025.

Difference I (dif1)

- I In the Geometry toolbar, click Pooleans and Partitions and choose Difference.
- 2 Select the object revI only.
- 3 In the Settings window for Difference, locate the Difference section.
- **4** Find the **Objects to subtract** subsection. Click to select the **Calculate Selection** toggle button.
- 5 Select the objects **blk1** and **cyl1** 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 Home toolbar, click 🙀 Add Material to close the Add Material window.

LES RBVM (SPF)

- I In the Model Builder window, under Component I (compl) click LES RBVM (spf).
- 2 In the Settings window for LES RBVM, locate the Physical Model section.
- **3** Select the **Include gravity** check box.
- 4 Select the Use reduced pressure check box.

Outlet I

- I In the Physics toolbar, click 🔚 Boundaries and choose Outlet.
- **2** Select Boundaries 9, 10, 19, 20, 41, 42, 59, 60, 81, 82, 92, 97, 108, 117, 126, 137, 147, 156, 164, and 169 only.
- 3 In the Settings window for Outlet, locate the Boundary Selection section.
- 4 Click 🐚 Create Selection.
- 5 In the Create Selection dialog box, type Top in the Selection name text field.
- 6 Click OK.

Open Boundary I

- I In the Physics toolbar, click 📄 Boundaries and choose Open Boundary.
- **2** Select Boundaries 1, 2, 5, 6, 88, 90, 170, and 171 only.
- 3 In the Settings window for Open Boundary, locate the Boundary Selection section.
- 4 Click here are a create Selection.
- 5 In the Create Selection dialog box, type Sides in the Selection name text field.
- 6 Click OK.

HEAT TRANSFER IN FLUIDS (HT)

In the Model Builder window, under Component I (compl) click Heat Transfer in Fluids (ht).

Temperature 1

- I In the Physics toolbar, click 🔚 Boundaries and choose Temperature.
- **2** Select Boundaries 83–87 only.
- 3 In the Settings window for Temperature, locate the Boundary Selection section.
- 4 Click here **Create Selection**.
- 5 In the Create Selection dialog box, type Tip in the Selection name text field.
- 6 Click OK.
- 7 In the Settings window for Temperature, locate the Temperature section.
- 8 In the T_0 text field, type 293.15[K]+500[K]+50[K]*an1(t).

Open Boundary I

- I In the Physics toolbar, click 📄 Boundaries and choose Open Boundary.
- 2 In the Settings window for Open Boundary, locate the Boundary Selection section.
- **3** From the **Selection** list, choose **Sides**.

Outflow I

- I In the Physics toolbar, click 📄 Boundaries and choose Outflow.
- 2 In the Settings window for Outflow, locate the Boundary Selection section.
- **3** From the **Selection** list, choose **Top**.

PARTICLE TRACING FOR FLUID FLOW (FPT)

- I In the Model Builder window, under Component I (comp1) click Particle Tracing for Fluid Flow (fpt).
- 2 In the Settings window for Particle Tracing for Fluid Flow, locate the Particle Release and Propagation section.

3 From the Formulation list, choose Massless.

Wall I

- I In the Model Builder window, under Component I (compl)> Particle Tracing for Fluid Flow (fpt) click Wall I.
- 2 In the Settings window for Wall, locate the Wall Condition section.
- 3 From the Wall condition list, choose Disappear.

Particle Properties 1

- I In the Model Builder window, click Particle Properties I.
- 2 In the Settings window for Particle Properties, locate the Particle Properties section.
- **3** Specify the **v** vector as

u x v y w z

Release from Grid 1

- I In the Physics toolbar, click 🗱 Global and choose Release from Grid.
- 2 In the Settings window for Release from Grid, locate the Release Times section.
- 3 In the **Release times** text field, type range(0,0.01,10).
- **4** Locate the **Initial Coordinates** section. In the $q_{x,0}$ text field, type range (-0.005,0.001, 0.005).
- **5** In the $q_{y,0}$ text field, type range(-0.005,0.001,0.005).
- 6 In the $q_{z,0}$ text field, type 0.16.

MULTIPHYSICS

Nonisothermal Flow 1 (nitf1)

- I In the Model Builder window, under Component I (compl)>Multiphysics click Nonisothermal Flow I (nitfl).
- 2 In the Settings window for Nonisothermal Flow, locate the Material Properties section.
- **3** Select the **Boussinesq approximation** check box.

MESH I

Set up the mesh so that it is finest in an area above the tip of the incense stick, extending upward. This is the area where the flow is expected to develop from laminar to turbulent

and where an accurate air velocity is needed. In the remaining parts of the geometry, a much courser mesh can be used for increased efficiency.

Free Tetrahedral I

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

Size

- I In the Model Builder window, click Size.
- 2 In the Settings window for Size, locate the Element Size section.
- **3** Click the **Custom** button.
- **4** Locate the **Element Size Parameters** section. In the **Minimum element size** text field, type 0.002.

Free Tetrahedral I

- I In the Model Builder window, click Free Tetrahedral I.
- 2 In the Settings window for Free Tetrahedral, locate the Domain Selection section.
- **3** From the Geometric entity level list, choose Domain.
- **4** Select Domains 4, 8, 9, and 11–17 only.

Size 1

- I Right-click Free Tetrahedral I and choose Size.
- 2 In the Settings window for Size, locate the Element Size section.
- **3** From the **Calibrate for** list, choose **Fluid dynamics**.
- 4 From the Predefined list, choose Extra fine.

Mapped I

- I In the Mesh toolbar, click \bigwedge Boundary and choose Mapped.
- **2** Select Boundaries 9, 10, 92, and 169 only.

Distribution I

- I Right-click Mapped I and choose Distribution.
- 2 Select Edge 9 only.
- 3 In the Settings window for Distribution, locate the Distribution section.
- 4 In the Number of elements text field, type 1.

Swept I

- I In the Mesh toolbar, click A Swept.
- 2 In the Settings window for Swept, locate the Domain Selection section.

- 3 From the Geometric entity level list, choose Domain.
- **4** Select Domain 2 only.

Swept 2

- I In the Mesh toolbar, click 🎪 Swept.
- 2 In the Settings window for Swept, locate the Domain Selection section.
- **3** From the Geometric entity level list, choose Domain.
- **4** Select Domain 1 only.

Distribution I

- I Right-click Swept 2 and choose Distribution.
- 2 In the Settings window for Distribution, locate the Distribution section.
- 3 In the Number of elements text field, type 20.

Free Tetrahedral 2

In the Mesh toolbar, click \land Free Tetrahedral.

Boundary Layers 1

In the Mesh toolbar, click **Boundary Layers**.

Boundary Layer Properties

- I In the Model Builder window, click Boundary Layer Properties.
- 2 Select Boundaries 83–87 and 172–179 only.
- 3 In the Settings window for Boundary Layer Properties, locate the Layers section.
- 4 In the Number of layers text field, type 2.
- **5** In the **Thickness adjustment factor** text field, type **5**.
- 6 Click 📗 Build All.

LES requires an adequate resolution of the convective time scale. This puts a constraint on the time steps taken by the solver: $\Delta t < h/(2U)$, where Δt is the time step, h the mesh size in the streamline direction, and U the velocity magnitude. The following instructions are meant to give an indication of h in the important part of the domain, that is, in the area where the air velocities are expected to be the highest and where the transition from laminar to turbulent flow is taking place.

RESULTS

Mesh I

I In the Model Builder window, expand the Results node.

2 Right-click Results>Datasets and choose Mesh.

Volume Minimum 1

- I In the Results toolbar, click ^{8,85}_{e-12} More Derived Values and choose Minimum> Volume Minimum.
- **2** Select Domains 8, 11, 12, 15, and 16 only.
- 3 In the Settings window for Volume Minimum, locate the Expressions section.
- **4** In the table, enter the following settings:

Expression	Unit	Description
h	m	Element size

5 Click **= Evaluate**.

The evaluated minimum mesh size should be approximately equal to 0.0065 m. In addition, the maximum air velocities are expected to be around 0.6 to 0.7 m/s. Therefore, a reasonable time-step size for the LES is 0.005 s. The simulation is split over two studies. The first study computes the flow and temperature fields, and the second computes the particle trajectories. Since the particle tracing will be expected to take time steps of the same order of magnitude, the solver in the first study for the flow is set to store the velocity field every 0.005 s, and the steps taken by the solver is set to strict to ensure that the maximum time step is also 0.005 s. Storing the solution for this many time steps takes a large amount of disk space. Since the particle tracing only needs the air velocity as input, the required disk space can be reduced by storing only the air velocity, thus not storing the pressure and temperature fields. The following instructions set up the solver in the first study according to these considerations.

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 In the **Output times** text field, type range(0,0.005,10).
- 4 Locate the Physics and Variables Selection section. In the table, clear the Solve for check box for Particle Tracing for Fluid Flow (fpt).

Solution 1 (soll)

- I In the Study toolbar, click **The Show Default Solver**.
- 2 In the Model Builder window, expand the Solution I (soll) node.

- 3 In the Model Builder window, expand the Study I>Solver Configurations> Solution I (soll)>Dependent Variables I node, then click Pressure (compl.p).
- 4 In the Settings window for Field, locate the General section.
- **5** Clear the **Store in output** check box.
- 6 In the Model Builder window, under Study I>Solver Configurations>Solution I (soll)> Dependent Variables I click Particle position (compl.qfpt).
- 7 In the Settings window for Field, locate the General section.
- 8 Clear the Store in output check box.
- 9 In the Model Builder window, under Study I>Solver Configurations>Solution I (soll)> Dependent Variables I click Temperature (comp I.T).
- 10 In the Settings window for Field, locate the General section.
- II Clear the **Store in output** check box.
- 12 In the Model Builder window, under Study I>Solver Configurations>Solution I (soll) click Time-Dependent Solver I.
- **13** In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- 14 From the Steps taken by solver list, choose Strict.
- IS Select the Initial step check box. In the associated text field, type 0.0001.

To further reduce the required disk space, instruct the solver not to store the reaction forces and time derivatives.

- 16 Click to expand the Output section. Clear the Store reaction forces check box.
- **I7** Clear the **Store time derivatives** check box.
- **18** In the **Study** toolbar, click **= Compute**.

The following instructions add a second study for the particle tracing. This study uses the stored results of the first study. Only the particle positions are stored, and in order to make it possible to produce an animation of the smoke plume with 25 frames per second, they are stored every 0.04 s.

ADD STUDY

- I In the Study toolbar, click 2 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 Click Add Study in the window toolbar.

5 In the Study toolbar, click 2 Add Study to close the Add Study window.

STUDY 2

Step 1: Time Dependent

- I In the Settings window for Time Dependent, locate the Study Settings section.
- 2 In the **Output times** text field, type range(0,0.04,10).
- **3** Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check boxes for **LES RBVM (spf)** and **Heat Transfer in Fluids (ht)**.
- 4 In the table, clear the Solve for check box for Nonisothermal Flow I (nitfl).
- 5 Click to expand the Values of Dependent Variables section. Find the Values of variables not solved for subsection. From the Settings list, choose User controlled.
- 6 From the Method list, choose Solution.
- 7 From the Study list, choose Study I, Time Dependent.
- 8 From the Time (s) list, choose Automatic (all solutions).

Solution 2 (sol2)

- I In the Study toolbar, click **The Show Default Solver**.
- 2 In the Model Builder window, expand the Solution 2 (sol2) node.
- 3 In the Model Builder window, expand the Study 2>Solver Configurations> Solution 2 (sol2)>Dependent Variables I node, then click Pressure (compl.p).
- 4 In the Settings window for Field, locate the General section.
- 5 Clear the Store in output check box.
- 6 In the Model Builder window, under Study 2>Solver Configurations>Solution 2 (sol2)> Dependent Variables I click Temperature (comp I.T).
- 7 In the Settings window for Field, locate the General section.
- 8 Clear the Store in output check box.
- 9 In the Model Builder window, under Study 2>Solver Configurations>Solution 2 (sol2) click Time-Dependent Solver I.
- 10 In the Settings window for Time-Dependent Solver, locate the Output section.
- II Clear the Store reaction forces check box.
- **12** Clear the **Store time derivatives** check box.
- **I3** In the **Study** toolbar, click **= Compute**.

Now that the particle positions have been computed, you can optionally discard the results of the first study in order to reduce disk usage; the stored air velocity fields computed in the first study take up several GBs of disk space. Note that the velocity field is still stored in the solution of the second study since the **Store in output** check box was not cleared for this field. However, the second study only stores the solution every 0.04 s instead of every 0.005 s. The next instruction will remove the solution of the first study from the model.

STUDY I

- I Click 📐 Clear Solutions.
- 2 In the Model Builder window, click Study I.
- 3 Click Yes to confirm.

The following instructions reproduce the figures shown in the Results and Discussion section.

RESULTS

Velocity (spf)

- I In the Model Builder window, under Results click Velocity (spf).
- 2 In the Settings window for 3D Plot Group, locate the Data section.
- 3 From the Dataset list, choose Study 2/Solution 2 (sol2).

Slice

- I In the Model Builder window, expand the Velocity (spf) node, then click Slice.
- 2 In the Settings window for Slice, locate the Plane Data section.
- 3 From the Plane list, choose zx-planes.
- **4** In the **Planes** text field, type 1.
- 5 Locate the Coloring and Style section. Click Change Color Table.
- 6 In the Color Table dialog box, select Aurora>AuroraAustralisDark in the tree.
- 7 Click OK.

Surface 1

- I In the Model Builder window, right-click Velocity (spf) and choose Surface.
- 2 In the Settings window for Surface, locate the Coloring and Style section.
- 3 From the Coloring list, choose Uniform.
- 4 From the Color list, choose Custom.

- **5** On Windows, click the colored bar underneath, or if you are running the cross-platform desktop the **Color** button.
- 6 Click Define custom colors.
- 7 Set the RGB values to 196, 106, and 72, respectively.
- 8 Click Add to custom colors.
- 9 Click Show color palette only or OK on the cross-platform desktop.

Selection I

- I Right-click Surface I and choose Selection.
- **2** Select Boundaries 61–64, 189, and 190 only.
- 3 In the Settings window for Selection, locate the Selection section.
- 4 Click here a Create Selection.
- 5 In the Create Selection dialog box, type holder in the Selection name text field.
- 6 Click OK.

Surface 2

- I In the Model Builder window, right-click Velocity (spf) and choose Surface.
- 2 In the Settings window for Surface, locate the Coloring and Style section.
- 3 From the Coloring list, choose Uniform.
- 4 From the Color list, choose Gray.

Selection I

- I Right-click Surface 2 and choose Selection.
- 2 Select Boundaries 83–87 and 172–179 only.
- 3 In the Settings window for Selection, locate the Selection section.
- 4 Click here a Create Selection.
- 5 In the Create Selection dialog box, type incense stick in the Selection name text field.
- 6 Click OK.
- 7 Click the Z Go to XZ View button in the Graphics toolbar.

Velocity (spf)

- I In the Model Builder window, under Results click Velocity (spf).
- 2 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- **3** Clear the **Plot dataset edges** check box.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

- 5 Locate the Data section. From the Time (s) list, choose 4.
- 6 In the Velocity (spf) toolbar, click 💿 Plot.

This reproduces the first slice plot in Figure 2. Choose different times to reproduce the other plots. The following instructions reproduce Figure 3.

3D Plot Group 7

- I In the Home toolbar, click 🚛 Add Plot Group and choose 3D Plot Group.
- 2 In the Settings window for 3D Plot Group, locate the Data section.
- 3 From the Dataset list, choose Particle I.
- 4 From the Time (s) list, choose 2.
- 5 Locate the Plot Settings section. Clear the Plot dataset edges check box.
- 6 Locate the Title section. From the Title type list, choose None.
- 7 Click to expand the **Plot Array** section. Select the **Enable** check box.
- 8 In the **Relative padding** text field, type 0.

Particle Trajectories 1

- I In the **3D Plot Group 7** toolbar, click **More Plots** and choose **Particle Trajectories**.
- 2 In the Settings window for Particle Trajectories, locate the Coloring and Style section.
- **3** Find the Line style subsection. From the Type list, choose None.
- 4 Find the **Point style** subsection. From the **Type** list, choose **Point**.
- 5 Select the Radius scale factor check box. In the associated text field, type 2.
- 6 From the Color list, choose Gray.

Transparency I

- I Right-click Particle Trajectories I and choose Transparency.
- 2 In the Settings window for Transparency, locate the Transparency section.
- **3** Set the **Transparency** value to **0.8**.
- 4 In the Graphics window toolbar, click ▼ next to
 Scene Light, then choose
 Ambient Occlusion.
- **5** Click the **Show Grid** button in the **Graphics** toolbar.
- 6 Click the 🖈 Show Axis Orientation button in the Graphics toolbar.

Particle Trajectories 2

I In the Model Builder window, under Results>3D Plot Group 7 right-click Particle Trajectories I and choose Duplicate.

- 2 In the Settings window for Particle Trajectories, locate the Data section.
- 3 From the Dataset list, choose Particle I.
- 4 From the Time (s) list, choose 4.

Repeat duplicating the **Surface** nodes and adjust the times to plot the particle positions at t = 2 s, t = 4 s, t = 6 s, t = 8 s, and t = 10 s, as in Figure 3.

Particle Trajectories (fpt)

The last steps of the instructions create the plot that is used as the model thumbnail.

In the Model Builder window, expand the Results>Particle Trajectories (fpt) node.

Particle Trajectories (fpt)

- I In the Model Builder window, expand the Results>Particle Trajectories (fpt)> Particle Trajectories I node, then click Results>Particle Trajectories (fpt).
- 2 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- **3** Clear the **Plot dataset edges** check box.

Color Expression 1

In the Model Builder window, under Results>Particle Trajectories (fpt)> Particle Trajectories I right-click Color Expression I and choose Disable.

Particle Trajectories I

- I In the Model Builder window, click Particle Trajectories I.
- 2 In the Settings window for Particle Trajectories, locate the Coloring and Style section.
- **3** Find the **Point style** subsection. From the **Color** list, choose **Gray**.

Transparency I

- I Right-click Particle Trajectories I and choose Transparency.
- 2 In the Settings window for Transparency, locate the Transparency section.
- 3 Set the Transparency value to 0.8.

Surface 1

- I In the Model Builder window, right-click Particle Trajectories (fpt) and choose Surface.
- 2 In the Settings window for Surface, click to expand the Title section.
- **3** From the **Title type** list, choose **None**.
- 4 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
- **5** From the **Color** list, choose **Custom**.

- 6 On Windows, click the colored bar underneath, or if you are running the crossplatform desktop — the **Color** button.
- 7 Click Define custom colors.
- 8 Set the RGB values to 196, 106, and 72, respectively.
- 9 Click Add to custom colors.
- **IO** Click **Show color palette only** or **OK** on the cross-platform desktop.

Selection I

- I Right-click Surface I and choose Selection.
- 2 In the Settings window for Selection, locate the Selection section.
- **3** From the **Selection** list, choose **holder**.

Surface 2

- I In the Model Builder window, right-click Particle Trajectories (fpt) and choose Surface.
- 2 In the Settings window for Surface, locate the Title section.
- **3** From the **Title type** list, choose **None**.
- 4 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
- 5 From the Color list, choose Gray.

Selection 1

- I Right-click Surface 2 and choose Selection.
- 2 In the Settings window for Selection, locate the Selection section.
- **3** From the **Selection** list, choose **incense stick**.
- 4 In the Particle Trajectories (fpt) toolbar, click 🗿 Plot.

To use this plot to create an animation, proceed as follows:

Animation I

I In the Particle Trajectories (fpt) toolbar, click Animation and choose File.

Then, in order to ensure that all stored solutions are used and that the animation plays at the correct speed, do the following before exporting the animation in the desired format:

- 2 In the Settings window for Animation, locate the Output section.
- 3 In the Frames per second text field, type 25.
- 4 Locate the Frames section. From the Frame selection list, choose All.