



Model created in COMSOL Multiphysics 6.4

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

## 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:

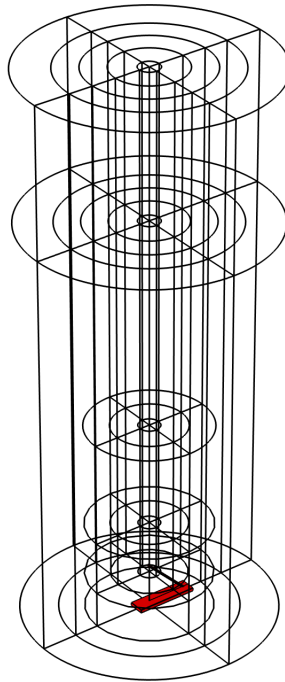
$$T_{\text{tip}} = 793,15\text{K} + 50\text{K} \times \text{rand}(\text{floor}(t)) \quad (1)$$

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 cm<sup>2</sup> 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.

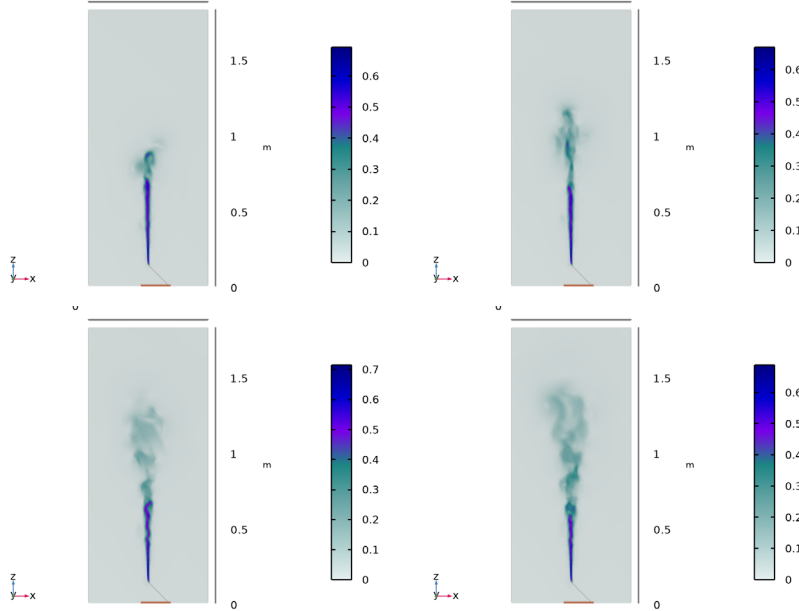
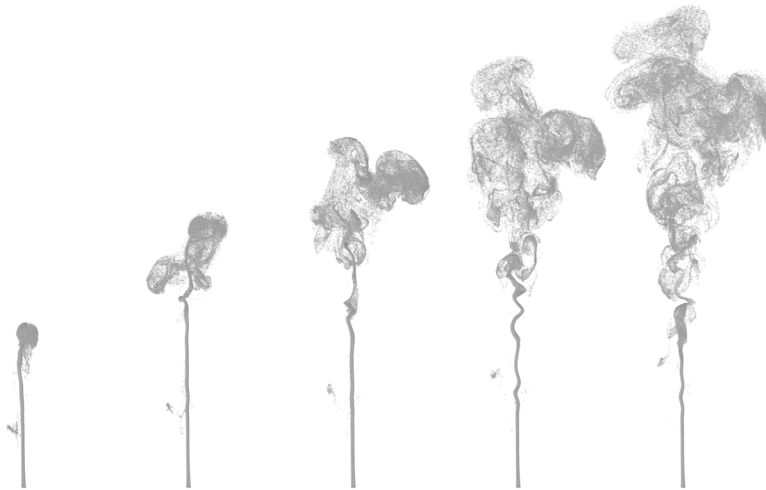


Figure 2: Slice plots in the center  $xz$ -plane of the velocity magnitude for  $t = 4$  s (top left),  $t = 6$  s (top right),  $t = 8$  s (bottom left), and  $t = 10$  s (bottom right).

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*

---

I. 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


- 1 In the **Model Wizard** window, click  **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  **Done**.

## DEFINITIONS

### *Random 1 (rn1)*

- 1 In the **Model Builder** window, expand the **Component 1 (comp1) > 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** checkbox.
- 6 In the **Random seed** text field, type 11301.

### *Analytic 1 (an1)*

- 1 In the **Definitions** toolbar, click  **Analytic**.
- 2 In the **Settings** window for **Analytic**, locate the **Definition** section.
- 3 In the **Expression** text field, type  $\text{rn1}(\text{floor}(x))$ .

## GEOMETRY 1

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 1 (wpl)*


- 1 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 1 (wp1) > Plane Geometry*

In the **Model Builder** window, click **Plane Geometry**.

*Work Plane 1 (wp1) > Rectangle 1 (r1)*

1 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** checkbox.

6 Clear the **Layers on bottom** checkbox.

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)*


1 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 1 (wp1) > Rectangle 3 (r3)*

1 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)*

1 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** checkbox.
- 7 Clear the **Layers on bottom** checkbox.
- 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 1** and choose **Revolve**.

*Cylinder 1 (cyl1)*

- 1 In the **Geometry** toolbar, click  **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** checkbox.
- 11 Clear the **Layers on side** checkbox.
- 12 In the table, enter the following settings:



Layer name	Thickness (m)
Layer 1	0.02

*Block 1 (blk1)*



- 1 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 1 (dif1)*

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 Select the object **rev1** only.
- 3 In the **Settings** window for **Difference**, locate the **Difference** section.
- 4 Click to select the  **Activate Selection** toggle button for **Objects to subtract**.
- 5 Select the objects **blk1** and **cyll** 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 **Materials** toolbar, click  **Add Material** to close the **Add Material** window.

#### **LES RBVM (SPF)**


- 1 In the **Settings** window for **LES RBVM**, locate the **Physical Model** section.
- 2 Select the **Include gravity** checkbox.
- 3 Select the **Use reduced pressure** checkbox.

#### *Outlet 1*

- 1 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, type Top in the **Selection name** text field.
- 6 Click **OK**.



#### *Open Boundary 1*

- 1 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  **Create Selection**.
- 5 In the **Create Selection** dialog, type Sides in the **Selection name** text field.
- 6 Click **OK**.

## HEAT TRANSFER IN FLUIDS (HT)


### *Temperature 1*

- 1 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  **Create Selection**.
- 5 In the **Create Selection** dialog, 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  $293.15[\text{K}] + 500[\text{K}] + 50[\text{K}] * \text{an1}(t)$ .

### *Open Boundary 1*

- 1 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 1*

- 1 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)

- 1 In the **Model Builder** window, under **Component 1 (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 1


- 1 In the **Model Builder** window, under **Component 1 (comp1) > Particle Tracing for Fluid Flow (fpt)** click **Wall 1**.
- 2 In the **Settings** window for **Wall**, locate the **Wall Condition** section.
- 3 From the **Wall condition** list, choose **Disappear**.

### Particle Properties 1

- 1 In the **Model Builder** window, click **Particle Properties 1**.
- 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

- 1 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)

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Multiphysics** click **Nonisothermal Flow 1 (nitf1)**.
- 2 In the **Settings** window for **Nonisothermal Flow**, locate the **Material Properties** section.
- 3 Select the **Boussinesq approximation** checkbox.

## MESH 1

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 coarser mesh can be used for increased efficiency.

### *Free Tetrahedral I*

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

#### *Size*

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


### *Free Tetrahedral I*

- 1 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 I*

- 1 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*

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Mapped**.
- 2 Select Boundaries 9, 10, 92, and 169 only.


#### *Distribution I*

- 1 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*

- 1 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 2 only.

### *Swept 2*

- 1 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 1*

- 1 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  **Free Tetrahedral**.

### *Boundary Layers 1*

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

### *Boundary Layer Properties*

- 1 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  $\Delta t$  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 1*

- 1 In the **Model Builder** window, expand the **Results** node.
- 2 Right-click **Results > Datasets** and choose **Mesh**.

### Volume Minimum I

- 1 In the **Results** toolbar, click  $8.85 \times 10^{-12}$  **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



- 1 In the **Model Builder** window, under **Study I** click **Step 1: 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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Particle Tracing for Fluid Flow (fpt)**.
- 5 In the **Model Builder** window, click **Study I**.
- 6 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 7 Clear the **Generate default plots** checkbox, as the results will be discarded for this study.

### *Solution 1 (sol1)*

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Dependent Variables 1**.
- 3 In the **Settings** window for **Dependent Variables**, locate the **General** section.
- 4 From the **Defined by study step** list, choose **User defined**.
- 5 In the **Model Builder** window, expand the **Study 1 > Solver Configurations > Solution 1 (sol1) > Dependent Variables 1** node, then click **Pressure (comp1.p)**.
- 6 In the **Settings** window for **Field**, locate the **General** section.
- 7 From the **Store in output** list, choose **None**.
- 8 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1) > Dependent Variables 1** click **Particle Position (comp1.qfpt)**.
- 9 In the **Settings** window for **Field**, locate the **General** section.
- 10 From the **Store in output** list, choose **None**.
- 11 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1) > Dependent Variables 1** click **Temperature (comp1.T)**.
- 12 In the **Settings** window for **Field**, locate the **General** section.
- 13 From the **Store in output** list, choose **None**.
- 14 In the **Model Builder** window, under **Study 1 > Solver Configurations > Solution 1 (sol1)** click **Time-Dependent Solver 1**.
- 15 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- 16 From the **Steps taken by solver** list, choose **Strict**.
- 17 Select the **Initial step** checkbox. 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.
- 18 Click to expand the **Time Stepping** section. From the **Maximum step constraint** list, choose **Automatic**.  
Use automatic time-stepping to speed up the simulation.
- 19 Click to expand the **Output** section. Clear the **Store reaction forces** checkbox.
- 20 Clear the **Store the first time derivative** checkbox.
- 21 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

- 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 Click the **Add Study** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.


## STUDY 2

### *Step 1: Time Dependent*

- 1 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 **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **LES RBVM (spf)** and **Heat Transfer in Fluids (ht)**.
- 4 In the **Solve for** column of the table, under **Component 1 (comp1)** > **Multiphysics**, clear the checkbox for **Nonisothermal Flow 1 (nitf1)**.
- 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 1, Time Dependent**.
- 8 From the **Time (s)** list, choose **Automatic (all solutions)**.
- 9 In the **Model Builder** window, click **Study 2**.
- 10 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 11 Clear the **Generate default plots** checkbox, because the desired plots can be added later using the **Result Templates**.


### *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 **Dependent Variables 1**.
- 3 In the **Settings** window for **Dependent Variables**, locate the **General** section.
- 4 From the **Defined by study step** list, choose **User defined**.
- 5 In the **Model Builder** window, expand the **Study 2 > Solver Configurations > Solution 2 (sol2) > Dependent Variables 1** node, then click **Pressure (comp1.p)**.
- 6 In the **Settings** window for **Field**, locate the **General** section.
- 7 From the **Store in output** list, choose **None**.
- 8 In the **Model Builder** window, under **Study 2 > Solver Configurations > Solution 2 (sol2) > Dependent Variables 1** click **Temperature (comp1.T)**.
- 9 In the **Settings** window for **Field**, locate the **General** section.
- 10 From the **Store in output** list, choose **None**.
- 11 In the **Model Builder** window, under **Study 2 > Solver Configurations > Solution 2 (sol2)** click **Time-Dependent Solver 1**.
- 12 In the **Settings** window for **Time-Dependent Solver**, locate the **Output** section.
- 13 Clear the **Store reaction forces** checkbox.
- 14 Clear the **Store the first time derivative** checkbox.
- 15 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** checkbox 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 1

- 1 Click  **Clear Solutions**.
- 2 In the **Model Builder** window, click **Study 1**.
- 3 Click **Yes** to confirm.

The following instructions reproduce the figures shown in the [Results and Discussion](#) section.

## 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 (sol2) > LES RBVM > Velocity (spf)**.
- 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



### *Multislice 1*


- 1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Multislice 1**.
- 2 In the **Settings** window for **Multislice**, locate the **Multipane Data** section.
- 3 Find the **x-planes** subsection. In the **Planes** text field, type 0.
- 4 Find the **z-planes** subsection. In the **Planes** text field, type 0.
- 5 Locate the **Coloring and Style** section. From the **Color table** list, choose **AuroraAustralisDark**.

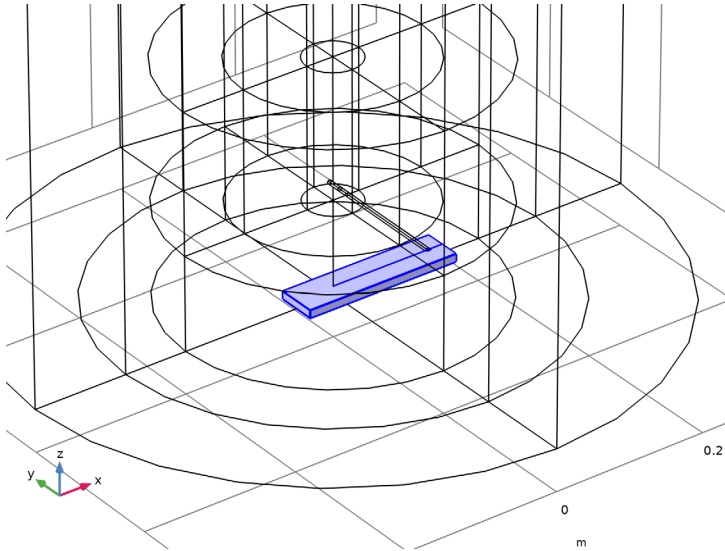
### *Surface 1*

- 1 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type 1.
- 4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 5 From the **Color** list, choose **Custom**.
- 6 On Windows, click the colored bar underneath, or — if you are running the cross-platform 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**.
- 10 Click **Show color palette only** or **OK** on the cross-platform desktop.

### *Selection 1*

- 1 Right-click **Surface 1** and choose **Selection**.
- 2 Click the  **Wireframe Rendering** button in the **Graphics** toolbar.
- 3 In the **Settings** window for **Selection**, locate the **Selection** section.
- 4 Click  **Paste Selection**.
- 5 In the **Paste Selection** dialog, type 61-64, 189, 190 in the **Selection** text field.
- 6 Click **OK**.



- 7 Click the  **Zoom to Selection** button in the **Graphics** toolbar and zoom out a little bit to get a better view.



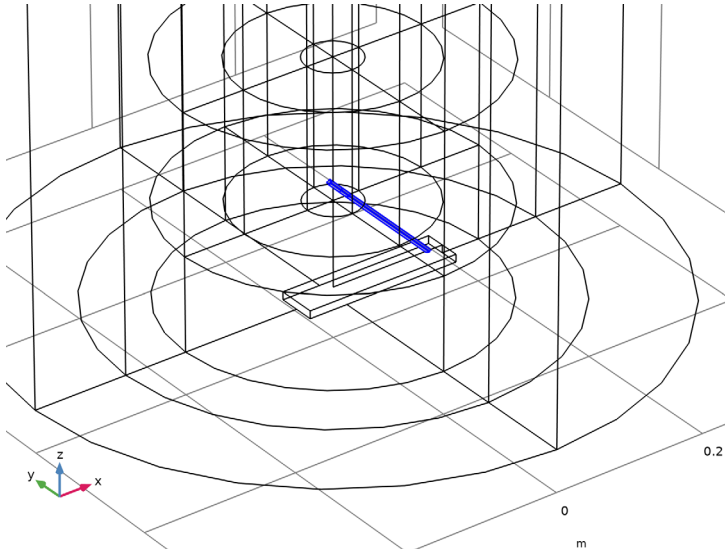
### *Surface 2*

- 1 In the **Model Builder** window, under **Results > Velocity (spf)** right-click **Surface 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
- 3 From the **Color** list, choose **Gray**.




### *Selection 1*

- 1 In the **Model Builder** window, expand the **Surface 2** node, then click **Selection 1**.
- 2 In the **Settings** window for **Selection**, locate the **Selection** section.
- 3 Click  **Clear Selection**.
- 4 Click  **Paste Selection**.
- 5 In the **Paste Selection** dialog, type 83 84 85 86 87 172 173 174 175 176 177 178 179 in the **Selection** text field.

6 Click **OK**.




*Velocity (spf)*

- 1 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** checkbox.
- 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**.
- 7 Click the  **Go to XZ View** button in the **Graphics** toolbar.
- 8 Click the  **Zoom Extents** button in the **Graphics** toolbar.

This reproduces the first slice plot in [Figure 2](#). Choose the times  $t = 6$  s,  $t = 8$  s, and  $t = 10$  s to reproduce the other plots in [Figure 2](#).

## 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 (sol2) > Particle Tracing for Fluid Flow > Particle Trajectories (fpt)**.
- 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

### *Particle Trajectories over Time*

- 1 In the **Settings** window for **3D Plot Group**, type **Particle Trajectories over Time** in the **Label** text field.
- 2 Locate the **Data** section. From the **Time (s)** list, choose **2**.
- 3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.
- 4 Locate the **Title** section. From the **Title type** list, choose **None**.
- 5 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.
- 6 In the **Relative padding** text field, type **0**.
- 7 In the **Model Builder** window, expand the **Particle Trajectories over Time** node.





### *Color Expression 1*

- 1 In the **Model Builder** window, expand the **Results > Particle Trajectories over Time > Particle Trajectories 1** node.
- 2 Right-click **Color Expression 1** and choose **Delete**.

### *Particle Trajectories 1*

- 1 In the **Settings** window for **Particle Trajectories**, locate the **Coloring and Style** section.
- 2 Find the **Point style** subsection.
- 3 Select the **Radius scale factor** checkbox. In the associated text field, type **2**.
- 4 From the **Color** list, choose **Gray**.

### *Transparency 1*

- 1 Right-click **Particle Trajectories 1** and choose **Transparency**.
- 2 In the **Settings** window for **Transparency**, locate the **Transparency** section.
- 3 Find the **Transparency** subsection. 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*

- 1 Right-click **Particle Trajectories 1** 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](#).

#### *Velocity (spf)*

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

#### *Particle Trajectories*

**1** In the **Model Builder** window, right-click **Velocity (spf)** and choose **Duplicate**.

**2** In the **Settings** window for **3D Plot Group**, type **Particle Trajectories** in the **Label** text field.

#### *Multislice I*

**1** In the **Model Builder** window, expand the **Particle Trajectories** node.

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

#### *Particle Trajectories*

In the **Model Builder** window, under **Results** click **Particle Trajectories**.

#### *Particle Trajectories I*

**1** In the **Particle Trajectories** toolbar, click  **More Plots** and choose **Particle Trajectories**.

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

**3** From the **Dataset** list, choose **Particle I**.

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

**5** Find the **Point style** subsection. From the **Type** list, choose **Point**.

**6** From the **Color** list, choose **Gray**.

#### *Transparency I*

**1** Right-click **Particle Trajectories I** and choose **Transparency**.


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

**3** Find the **Transparency** subsection. Set the **Transparency** value to **0.8**.

**4** Click the  **Go to Default View** button in the **Graphics** toolbar.

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

#### *Animation I*

**1** In the **Particle Trajectories** toolbar, click  **Animation** and choose **File**.

**2** Enter a filename or click **Browse** to navigate the file system.

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:

**3** In the **Settings** window for **Animation**, locate the **Output** section.

**4** In the **Frames per second** text field, type 25.

**5** Locate the **Frames** section. From the **Frame selection** list, choose **All**.

**6** Click **Export** in the settings toolbar.