Model created in COMSOL Multiphysics 6.4

Annular Ultraviolet Reactor with Particle Tracing

Introduction

In this example, the sterilization of water in a simple ultraviolet (UV) water purification reactor is modeled using a combination of optical ray tracing, computational fluid dynamics (CFD), and Lagrangian particle tracking methods.

The model consists of three studies. In the first study, the fluence rate in the annular region surrounding a UV lamp is solved for, using the Geometrical Optics physics interface. The fluence rate represents the amount of radiation that a small spherical detector would absorb per unit time, divided by the cross sectional area of such a detector. It has the same units as irradiance (SI unit: W/m^2), but is a volume quantity and can be defined at any arbitrary point in space, not just on surfaces.

In the second study, the velocity profile of water flowing through the reactor is solved for, using the Turbulent Flow, k-ε interface. The reactor in this example is a U-shape with inlet and outlet tubes running perpendicular to the orientation of the lamp.

In the third study, particles are traced through the reactor using the Particle Tracing for Fluid Flow interface. The particle velocity is based on the velocity of the water solved for in the second study. As the particles move through the reactor, the accumulated dose along their trajectories is computed, based on the fluence rate solved for in the first study.

The results show that all particles arriving at the outlet have been exposed to a certain minimum dose. A small number of particles, mostly those passing very close to the lamp, are subject to a substantially higher dose. This model could be used as a starting point to develop more energy-efficient UV water purification reactors by seeking designs with a more uniform accumulated dose.

This model requires the Ray Optics Module, CFD Module, and Particle Tracing Module.

Model Definition

The model geometry consists of the annular region between a cylindrical UV lamp and the cylindrical reactor that surrounds it, along with inlet and outlet tubes. All of the domains contain flowing water at room temperature.

ULTRAVIOLET LAMP MODEL

The surface of the lamp is treated as a diffuse (Lambertian) emitter using the **Release from Boundary** node. Rays are released at the lamp surface with uniform spatial density, and the initial direction of each ray is sampled according to the cosine law. The total power of the lamp is specified; in this example the power distribution over the lamp surface area is assumed to be uniform, although it is also possible to define a weighting factor.

As rays propagate through the water, away from the lamp surface, the power of each ray is attenuated based on the internal transmittance of the water, which in this example is taken to be 70% per centimeter of propagation. For distilled water, the internal transmittance of germicidal UV radiation is approximately 98%, so the value of 70% used here could represent that the water is less pure.

As rays propagate through the water, the volumetric fluence rate is computed using the dedicated **Fluence Rate Calculation** node. To obtain an accurate distribution of the fluence rate, it is important to release a sufficiently large number of rays and to use a sufficiently fine mesh in the domain. In this example, 100,000 rays were released in order to balance accuracy with solution time and file size considerations, but in some publications the number of rays could be orders of magnitude greater (Ref. 1).

Rather than releasing rays diffusely at the lamp surface, a possible extension of this model would be to release rays from positions inside the lamp. At the lamp surface, then, the light could be refracted into the water domain according to Snell's law and the Fresnel equations. Another possible extension is to use the **Mirror** boundary condition on the exterior surfaces of the reactor, rather than an absorbing **Wall** boundary condition.

FLUID FLOW

The Turbulent Flow, k-ε physics interface was used to solve for the fluid velocity and pressure in the reactor. At the **Inlet** boundary, the built-in **Fully developed flow** option was used to define the inflow velocity profile.

The default mesh is controlled by the Turbulent Flow, k-ε interface in this model, which causes boundary layers to grow on surfaces with the **Wall** boundary condition. Note that a **Coarse** mesh was used to reduce solution time in this example model, so the results should be verified using a finer mesh. The mesh also controls the resolution of the fluence rate computed by the Geometrical Optics interface, which uses constant shape functions.

PARTICLE TRACING

The Particle Tracing for Fluid Flow interface was used to track particles as they flow past the lamp. The fraction of deactivated or killed bacteria was predicted using a simple exponential decay model with a fixed inactivation constant k (SI unit: m^2/J),

$$N(t) = N_0 \exp\left(-k\int_0^t E_0 \mathrm{d}t'\right)$$

where N is the number or concentration of active bacteria at any point along a particle trajectory and N_0 is the initial number or concentration. The variable E_0 appearing in the time integral is the fluence rate obtained from the **Ray Tracing** study.

To integrate the fluence rate along each particle trajectory, an **Auxiliary Dependent Variable** was defined with time derivative equal to E_0. At any time, the value of E_0 for any particle is found by evaluating the volumetric fluence rate at the particle's instantaneous position.

Simulations of particles in a fluid are prone to numerical stiffness when the particles are small and the fluid velocity is large. This is because accelerations of small particles in response to drag by the surrounding fluid occur over the time scale τ_p given by

$$\tau_p = \frac{\rho_p d_p^2}{18\mu}$$

where

- ρ_p (SI unit: kg/m^3) is the density of the particle,
- d_p (SI unit: m) is the particle diameter, and
- μ (SI unit: Pa s) is the dynamic viscosity of the surrounding fluid.

For a micron-sized particle in water, this time scale is on the order of 10^{-7} seconds. A full inertial treatment of the particle motion would require time steps close to this order of magnitude, whereas the maximum time for particles to cross through the reactor is on the order of 10 seconds. It would therefore be possible to include the effects of particle inertia but the model would be significantly more time-consuming. For a similar reason, the effects of turbulent dispersion have been neglected in this example; each model particle follows a streamline of the mean fluid velocity. In an inertial particle tracing simulation with very small time steps, it would also be possible to get a more realistic solution by enabling turbulent dispersion in the settings for the **Drag Force**.

Results and Discussion

A **Slice** plot of the fluence rate is shown in Figure 1. The fluence rate is greatest in the region adjacent the lamp, and significantly smaller at the outermost extents of the reactor volume. Thus, particles passing through the center of the reactor are subjected to more intense UV radiation compared to particles moving along the outer edges.

The fluid velocity is shown in Figure 2 and the particle trajectories in Figure 3. The **Color Expression** in the trajectory plot shows the fraction of bacteria yet to be inactivated, starting at 1 and with 0 representing complete inactivation.

In Figure 4 a **Histogram** plot shows the accumulated dose of particles that reach the outlet by the final time. All particles receive a dose of at least 10 mJ/cm^2, but for some particles the accumulated dose is substantially higher.

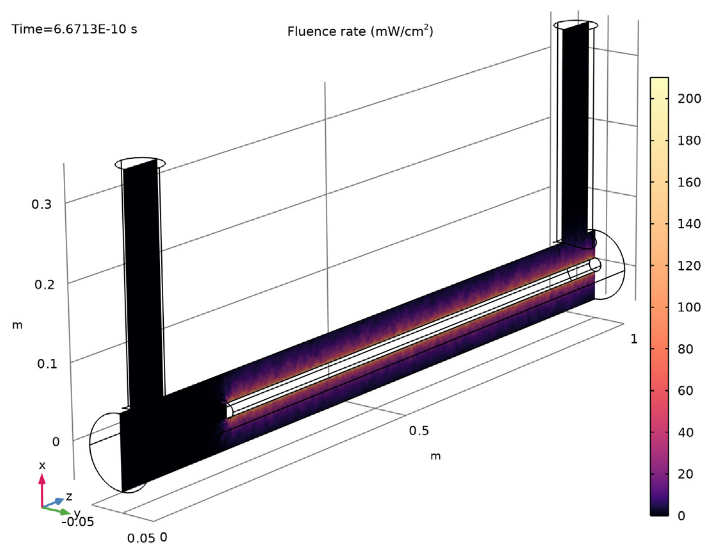

Figure 1: Fluence rate distribution in a cross section of the UV reactor.

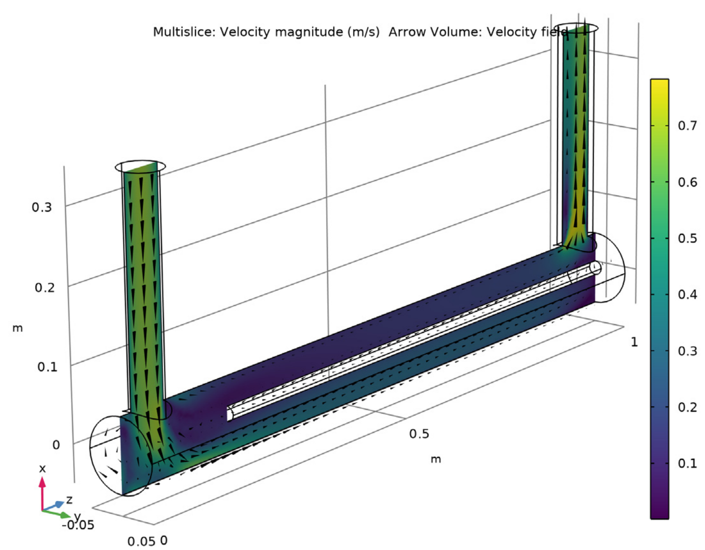

Figure 2: Slice plot of the fluid velocity norm, with arrows indicating the velocity direction.

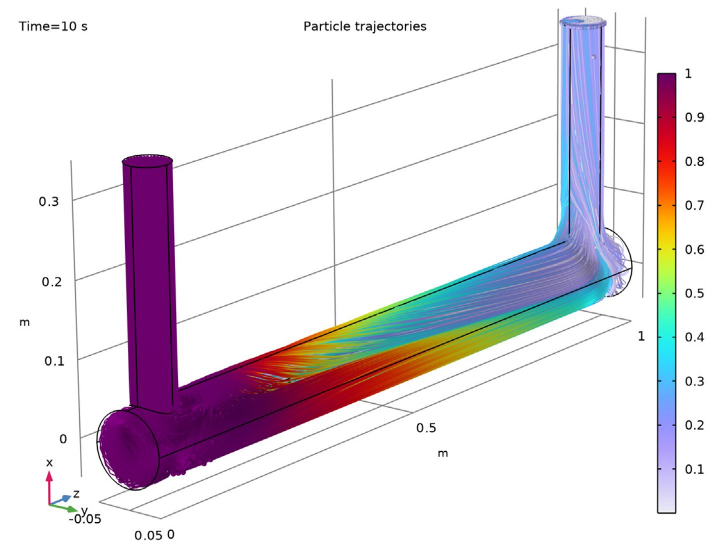

Figure 3: Particle trajectories in the UV reactor. The color expression shows the fraction of surviving bacteria.

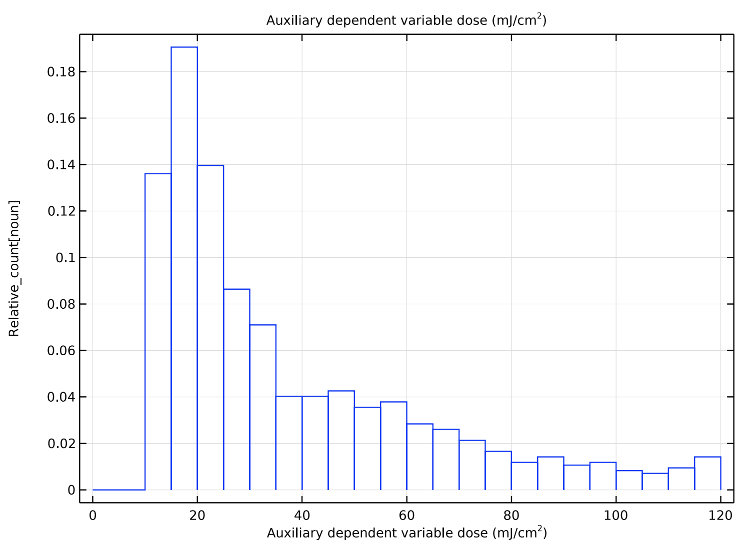

Figure 4: Histogram of the accumulated dose along particle trajectories. The plot was filtered to exclude particles that did not reach the outlet by the final time.

References

1. Y.M. Ahmed, M. Jongewaard, M. Li, and E.R. Blatchley III, "Ray Tracing for Fluence Rate Simulations in Ultraviolet Photoreactors," *Environ. Sci. Technol.*, vol. 52, no. 8, pp. 4738–4745, 2018.

2. D.A. Sozzi and F. Taghipour, "UV Reactor Performance Modeling by Eulerian and Lagrangian Methods," *Environ. Sci. Technol.*, vol. 40, no. 5, pp. 1609–1615, 2006.

Application Library path: `Ray_Optics_Module/Ultraviolet_Sterilization/annular_ultraviolet_reactor_particle`

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 **Optics** > **Ray Optics** > **Geometrical Optics (gop)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Fluid Flow** > **Single-Phase Flow** > **Turbulent Flow** > **Turbulent Flow, k-ε** (spf).

5 Click **Add**.

6 In the **Select Physics** tree, select **Fluid Flow** > **Particle Tracing** > **Particle Tracing for Fluid Flow (fpt)**.

7 Click **Add**.

8 Click **Study**.

9 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Geometrical Optics** > **Ray Tracing**.

10 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
r_lamp	1[cm]	0.01 m	Lamp radius
r_reac	5[cm]	0.05 m	Reactor radius
L_reac	100[cm]	1 m	Reactor length
L_lamp	80[cm]	0.8 m	Lamp length
d_lamp	L_reac-L_lamp	0.2 m	Lamp displacement
mid_lamp	d_lamp+L_lamp/2	0.6 m	Lamp midplane location
P	40[W]	40 W	Total source power
r_inl	3[cm]	0.03 m	Inlet radius
L_inl	30[cm]	0.3 m	Inlet length
z_inl	5[cm]	0.05 m	Inlet z-coordinate
k_inact	0.1[cm^2/mJ]	0.01 s^2/kg	Inactivation rate constant

GEOMETRY 1

Reactor

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

2 In the **Settings** window for **Cylinder**, type `Reactor` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_reac`.

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

Inlet Pipe

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

2 In the **Settings** window for **Cylinder**, type `Inlet Pipe` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_inl`.

4 In the **Height** text field, type `L_inl+r_reac`.

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

6 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

Outlet Pipe

1 In the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for **Cylinder**, type `Outlet Pipe` in the **Label** text field.
3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_inl`.
4 In the **Height** text field, type `L_inl+r_reac`.
5 Locate the **Position** section. In the **z** text field, type `L_reac-z_inl`.
6 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.
2 Click in the **Graphics** window and then press Ctrl+A to select all objects.
3 In the **Settings** window for **Union**, locate the **Union** section.
4 Clear the **Keep interior boundaries** checkbox.

Lamp

1 In the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for **Cylinder**, type `Lamp` in the **Label** text field.
3 Locate the **Size and Shape** section. In the **Radius** text field, type `r_lamp`.
4 In the **Height** text field, type `L_lamp`.
5 Locate the **Position** section. In the **z** text field, type `d_lamp`.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
2 Select the object **uni1** 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 object **cyl4** only.
6 In the **Geometry** toolbar, click **Build All**.

GEOMETRICAL OPTICS (GOP)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometrical Optics (gop)**.
2 In the **Settings** window for **Geometrical Optics**, locate the **Ray Release and Propagation** section.
3 In the **Maximum number of secondary rays** text field, type `0`.
4 Select the **Only store accumulated variables in solution** checkbox.

5 Locate the **Intensity Computation** section. From the **Intensity computation** list, choose **Compute power**.

Ray Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometrical Optics (gop)** click **Ray Properties 1**.

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

3 In the λ_0 text field, type `254[nm]`. A low-pressure mercury vapor lamp would emit most of its UV radiation at this wavelength (Ref. 1).

Medium Properties 1

1 In the **Model Builder** window, click **Medium Properties 1**.

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

3 From the n list, choose **User defined**. In the associated text field, type `1.38`. This is approximately the refractive index of water at 254 nm (Ref. 1).

4 From the **Optical attenuation model** list, choose **Internal transmittance, 10 mm sample thickness**.

5 From the $\tau_{i,10}$ list, choose **User defined**. In the associated text field, type `0.7`. Different values of the internal transmittance of water can be used here, depending on the clarity of the water. For pure water, the internal transmittance of germicidal UV radiation is about 0.98 per centimeter (Ref. 1). The value of 0.7 shown here indicates that the water is less clear.

Release from Boundary 1

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

2 Select Boundaries 5, 6, 9, and 10 only. These are the curved surfaces of the inner cylinder. Use the middle mouse wheel to select an interior boundary in the **Graphics** window. You can also make it easier to view the interior boundaries by enabling wireframe rendering.

3 In the **Settings** window for **Release from Boundary**, locate the **Initial Position** section.

4 From the **Initial position** list, choose **Density**.

5 In the N text field, type `100000`.

6 Locate the **Ray Direction Vector** section. From the **Ray direction vector** list, choose **Lambertian**.

7 Select the **Specify tangential and normal vector components** checkbox.

8 In the N_w text field, type `1`.

9 Specify the **r** vector as

0	t1
0	t2
1	n

10 From the **Sampling from distribution** list, choose **Random**.

These settings will cause rays to be released from the lamp surface with uniform spatial density, with a distribution of initial directions following the cosine law with respect to the surface normal at each release position. To ensure that the surface normal points in the outward direction rather than the inward direction, look for the arrows in the **Graphics** window.

11 Locate the **Total Source Power** section. In the P_{src} text field, type `P`.

Fluence Rate Calculation 1

1 In the **Physics** toolbar, click **Domains** and choose **Fluence Rate Calculation**.

2 Select Domain 1 only.

Wall 1

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

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

3 From the **Selection** list, choose **All boundaries**.

TURBULENT FLOW, K-ε (SPF)

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

Inlet 1

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

2 Select Boundary 20 only.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the list, choose **Fully developed flow**.

5 Locate the **Fully Developed Flow** section. Click the **Flow rate** button.

6 In the V_0 text field, type `25[gal/min]`.

Outlet 1

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

2 Select Boundary 21 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 > Water, liquid**.

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.

PARTICLE TRACING FOR FLUID FLOW (FPT)

1 In the **Settings** window for **Particle Tracing for Fluid Flow**, locate the **Particle Release and Propagation** section.

2 From the **Formulation** list, choose **Newtonian, ignore inertial terms**.

Drag Force 1

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

2 Select Domain 1 only.

3 In the **Settings** window for **Drag Force**, locate the **Drag Force** section.

4 From the **u** list, choose **Velocity field (spf)**.

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 From the ρ_p list, choose **User defined**. The default value of the particle density will be used.

Inlet 1

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

2 Select Boundary 20 only.

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

4 From the **Initial position** list, choose **Density**.

5 In the N text field, type `1000`.

6 In the ρ text field, type `spf.U`.

Outlet 1

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

2 Select Boundary 21 only.

Particle Counter 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Particle Counter**.

2 Select Boundary 21 only.

Absorbed Dose

1 In the **Physics** toolbar, click **Global** and choose **Auxiliary Dependent Variable**.

2 In the **Settings** window for **Auxiliary Dependent Variable**, type `Absorbed Dose` in the **Label** text field.

3 Locate the **Auxiliary Dependent Variable** section. In the **Field variable name** text field, type `dose`.

4 In the R text field, type `gop.frc1.E0`.

5 Locate the **Units** section. Click **Custom Unit**.

6 In the **Dependent variable quantity** table, enter the following settings:

Dependent variable quantity	Unit
Custom unit	J/m^2

DEFINITIONS

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
rs	exp(-k_inact*dose)		Survival rate

MESH 1

In this example a **Free Tetrahedral** mesh is used. To get higher resolution of the radial dependence of the fluence rate, a structured mesh could be created, although this may require the addition of mesh control surfaces to the geometry sequence.

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

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

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

4 Click **Build All**.

STUDY 1

Step 1: Ray Tracing

1 In the **Model Builder** window, under **Study 1** click **Step 1: Ray Tracing**.

2 In the **Settings** window for **Ray Tracing**, locate the **Study Settings** section.

3 From the **Time-step specification** list, choose **Specify maximum path length**.

4 In the **Lengths** text field, type `0 0.2`.

5 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Turbulent Flow, k-ε (spf)** and **Particle Tracing for Fluid Flow (fpt)**.

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

RESULTS

Fluence Rate Slice Plot

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Fluence Rate Slice Plot` in the **Label** text field.

Slice 1

1 In the **Fluence Rate Slice Plot** toolbar, click **Slice**.

2 In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Geometrical Optics > Heating and losses > gop.frc1.E0 - Fluence rate - W/m²**.

3 Locate the **Expression** section. In the **Unit** field, type `mW/cm^2`.

4 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

5 In the **Planes** text field, type `1`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **Magma**.

7 From the **Color table transformation** list, choose **Nonlinear**.

8 Set the **Color calibration parameter** value to **-1.5**.

9 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

10 From the **Resolution** list, choose **No refinement**.

11 In the **Fluence Rate Slice Plot** toolbar, click **Plot**. Compare the resulting plot to Figure 1.

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** checkboxes for **Geometrical Optics (gop)** and **Particle Tracing for Fluid Flow (fpt)**.
4. Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Stationary**.
5. Click the **Add Study** button in the window toolbar.
6. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

1. In the **Settings** window for **Stationary**, click to expand the **Values of Dependent Variables** section.
2. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
3. From the **Method** list, choose **Solution**.
4. From the **Study** list, choose **Study 1, Ray Tracing**.
5. From the **Time (s)** list, choose **Last**.
6. In the **Study** toolbar, click **Compute**.

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 **Multiplane 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 **Viridis**.

Velocity (spf)

In the **Model Builder** window, click **Velocity (spf)**.

Arrow Volume 1

1. In the **Velocity (spf)** toolbar, click **Arrow Volume**.
2. In the **Settings** window for **Arrow Volume**, locate the **Arrow Positioning** section.
3. Find the **x grid points** subsection. In the **Points** text field, type `15`.

4 Find the **y grid points** subsection. In the **Points** text field, type 3.

5 Find the **z grid points** subsection. In the **Points** text field, type 50.

6 Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.

7 From the **Color** list, choose **Black**.

8 In the **Velocity (spf)** toolbar, click **Plot**. Compare the resulting plot to Figure 2.

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** checkboxes for **Geometrical Optics (gop)** and **Turbulent Flow, k-ε (spf)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Time Dependent**.

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

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

STUDY 3

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(0,0.1,10)`.

3 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**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Study 2, Stationary**.

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

RESULTS

Particle Trajectories (fpt)

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

Color Expression 1

1 In the **Model Builder** window, expand the **Results** > **Particle Trajectories (fpt)** > **Particle Trajectories 1** node, then click **Color Expression 1**.

2 In the **Settings** window for **Color Expression**, locate the **Expression** section.

3 In the **Expression** text field, type `rs`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

Particle Trajectories 1

1 In the **Model Builder** window, click **Particle Trajectories 1**.

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 **Line**.

4 In the **Particle Trajectories (fpt)** toolbar, click **Plot**. Compare the resulting plot to Figure 3.

1D Plot Group 6

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

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

3 From the **Dataset** list, choose **Particle 1**.

4 From the **Time selection** list, choose **Last**.

Histogram 1

1 In the **1D Plot Group 6** toolbar, click **Histogram**.

2 In the **Settings** window for **Histogram**, locate the **Expression** section.

3 In the **Expression** text field, type `dose`.

4 In the **Unit** field, type `mJ/cm^2`.

5 Locate the **Bins** section. From the **Entry method** list, choose **Limits**.

6 In the **Limits** text field, type `range(0,5,120)`.

7 Locate the **Output** section. From the **Function** list, choose **Discrete**.

8 From the **Normalization** list, choose **Sum of values**.

Filter 1

1 In the **1D Plot Group 6** toolbar, click **Filter**.

2 In the **Settings** window for **Filter**, click **Replace Expression** in the upper-right corner of the **Point Selection** section. From the menu, choose **Component 1 (comp1)** > **Particle Tracing for Fluid Flow** > **Particle Counter 1** > **fpt.pcnt1.rL - Logical expression for particle inclusion - 1**.

3 In the **1D Plot Group 6** toolbar, click **Plot**. Compare the resulting plot to Figure 4.