



# Laminar Static Mixer

## Introduction

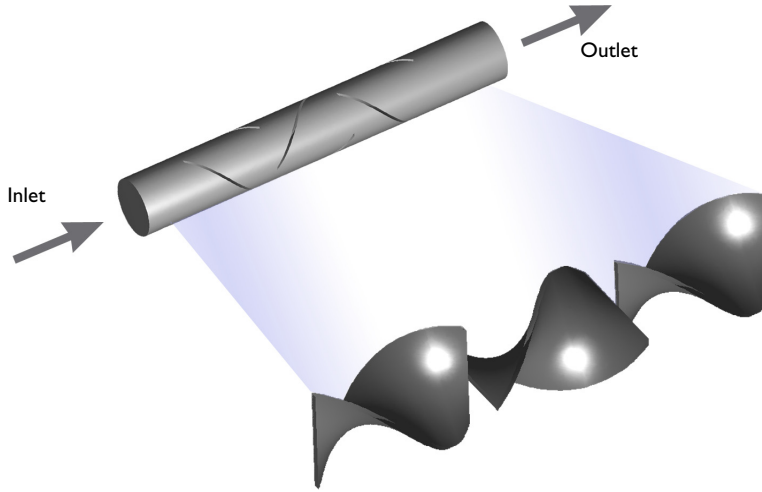
---

In static mixers, also called motionless or in-line mixers, a fluid is pumped through a pipe containing stationary blades. This mixing technique is particularly well suited for laminar flow mixing because it generates only small pressure losses in this flow regime. This example studies the flow in a twisted-blade static mixer.

## Model Definition

---

This model studies the mixing of one species dissolved in water at room temperature. The geometry consists of a tube with three twisted blades of alternating rotations ([Figure 1](#)).



*Figure 1: Depiction of a laminar static mixer containing three blades with alternating rotations.*

The tube's radius,  $R$ , is 3 mm; the length is  $14R$ , and the length of each blade is  $3R$ . The inlet flow is laminar with an average velocity of 10 mm/s. At the outlet, the model specifies a constant reference pressure of 0 Pa. The Laminar Flow interface is used in 3D and solves the Navier-Stokes equations:

$$\begin{aligned}\rho(\mathbf{u} \cdot \nabla)\mathbf{u} &= \nabla \cdot [-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)] \\ \rho(\nabla \cdot \mathbf{u}) &= 0\end{aligned}\tag{1}$$

Here  $\mu$  denotes the dynamic viscosity (SI unit: kg/(m·s)),  $\mathbf{u}$  is the velocity (SI unit: m/s),  $\rho$  represents the fluid density (SI unit: kg/m<sup>3</sup>), and  $p$  denotes the pressure (SI unit: Pa).

These fluid flow properties are not affected by any change in concentration of the dissolved species and are imported from the Material Library.

The species transport, which is defined in the Transport of Diluted Species interface, consists of convection and diffusion. At the outlet an Outflow boundary node is used to prescribe vanishing diffusion in the normal direction. At the inlet, a step change of the concentration is applied using an Inlet node. The inlet concentration is defined as:

$$c_{\text{inlet}} = \begin{cases} 0 & x < -\frac{\delta}{2} \\ c_0 & x \geq \frac{\delta}{2} \end{cases} \quad (2)$$

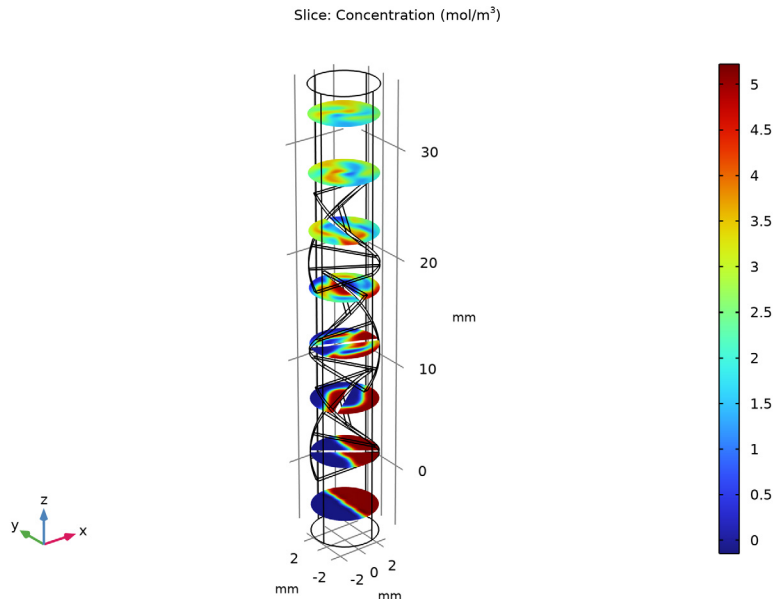
The concentration changes smoothly across a small transition zone  $\delta = 0.3$  mm. Due to the sharp concentration gradient and the fact that the convection is included, a fine mesh is required in order to avoid oscillations in the concentration field. The Reynolds numbers in the mixer, based on the average velocity and the diameter of the pipe, is about 60. This indicates that the flow is laminar and the fluid flow (Equation 1) does not require a particularly dense mesh near the walls. The Peclet number for the mass transport on the other hand is significantly higher.

$$\text{Pe} = \frac{u d_p}{D} = 1200$$

This means that the concentration gradient will be thinner than the shear layers in the flow. Consequently a higher resolution is needed for the mass transport compared to that for fluid flow. Since the concentration does not influence the fluid flow, you can therefore first solve the Navier-Stokes equations on a coarse mesh, and then map the solution onto a finer mesh and solve for the mass transport. In this model the resolution is further increased by using second order elements for the species concentration.

## Results

Figure 2 shows a slice plot of the concentration in the mixer. The slice at the bottom shows the blue and red halves of the fluid with and without the dissolved species, respectively. As the fluid flows upward through the system, the two solutions are mixed producing a more homogeneous concentration profile at the outlet.



*Figure 2: Slice plot of the concentration at different distances from the inlet.*

Figure 3 shows the flow field responsible for the mixing. The streamlines clearly reveal the twisting motion in the fluid that is induced by the mixer blades.

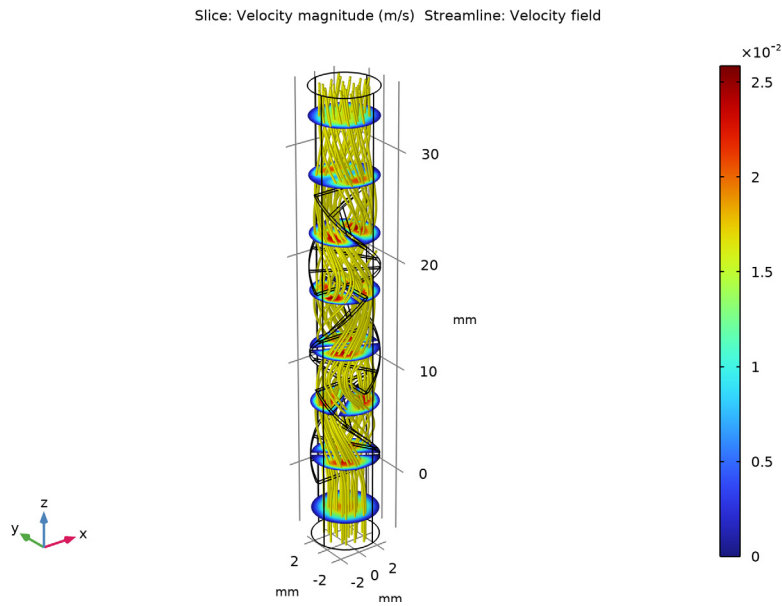


Figure 3: Slice plots of the velocity magnitude field inside the mixer. The streamlines show the flow direction.

You can also visualize the mixing through a series of cross-section plots. Figure 4 contains such a series of plots showing the concentration in the mixer's cross section along the

direction of the flow. The results show that most of the mixing takes place where the blades change rotational direction (the three middle figures).

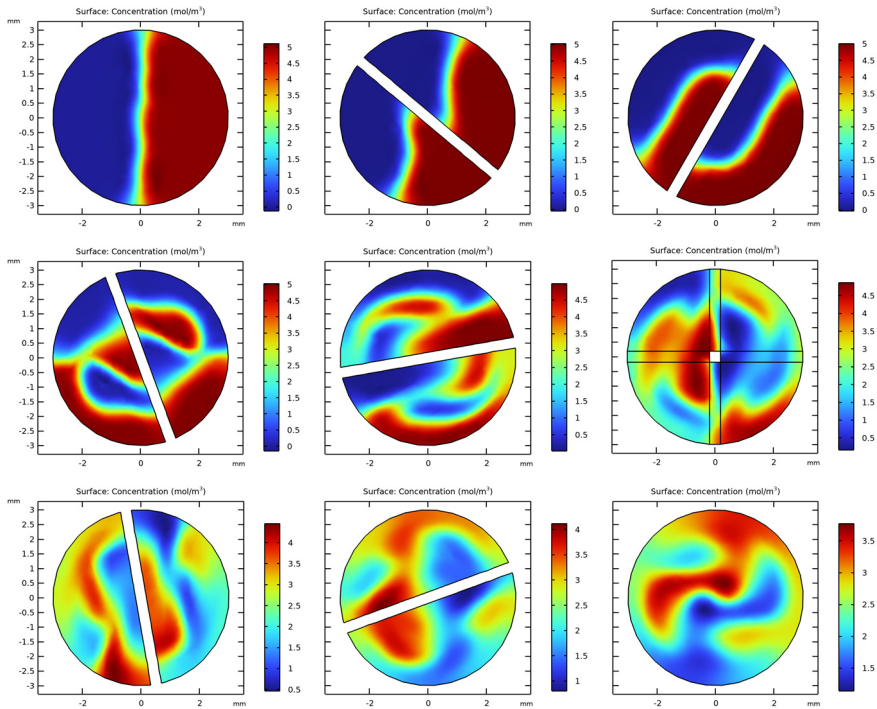


Figure 4: Cross-sectional plots of the concentration at different distances from the inlet. The nine plots shows the concentration at  $z = -2$  mm to  $z = 30$  mm in steps of 4 mm.

## References

1. R. Perry and D. Green, *Perry's Chemical Engineering Handbook*, 7th ed., McGraw Hill, 1997.
2. J.M. Coulson and J.F. Richardson, *Chemical Engineering*, vol. 1, 4th ed., Pergamon Press, 1990.


**Application Library path:** Chemical\_Reaction\_Engineering\_Module/  
Mixing\_and\_Separation/laminar\_static\_mixer

## 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>Single-Phase Flow>Laminar Flow (spf)**.
- 3 Click **Add**.
- 4 In the **Select Physics** tree, select **Chemical Species Transport>Transport of Diluted Species (tds)**.
- 5 Click **Add**.
- 6 Click  **Study**.
- 7 In the **Select Study** tree, select **General Studies>Stationary**.
- 8 Click  **Done**.

Import the model parameters from a text file.


### 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 Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `laminar_static_mixer_parameters.txt`.



A **Step** function is required to create a step change in the concentration at the inlet of the mixer.

#### Step 1 (step1)

- 1 In the **Home** toolbar, click  **Functions** and choose **Global>Step**.
- 2 In the **Settings** window for **Step**, locate the **Parameters** section.
- 3 In the **To** text field, type 5.
- 4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type 3e-4.

Create the geometry. To simplify this step, insert a prepared geometry sequence. In the **Geometry** toolbar, click **Insert Sequence**. Browse to the model's Application Libraries folder and double-click the file `laminar_static_mixer.mph`. Then click **Build All** In the **Geometry** toolbar.

#### ADD MATERIAL

- 1 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>Water, liquid**.
- 4 Click **Add to Component** in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Material** to close the **Add Material** window.

#### MATERIALS

*Water, liquid (mat1)*

The first material you add applies to all domains by default, so you do not need to change any settings.

In the **Laminar Flow** interface, set **Inlet** and **outlet** boundary conditions together with the initial values.

#### LAMINAR FLOW (SPF)

*Inlet 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** 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. In the  $U_{av}$  text field, type `u_av`.

This gives a parabolic inlet velocity profile, appropriate for fully developed laminar flow, with mean velocity `u_av`.

*Outlet 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Outlet**.
- 2 Select Boundary 23 only.

Apply quadratic basis functions for the concentration to further increase the resolution.



## TRANSPORT OF DILUTED SPECIES (TDS)


- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species (tds)**.
- 2 In the **Settings** window for **Transport of Diluted Species**, click to expand the **Discretization** section.
- 3 From the **Concentration** list, choose **Quadratic**.

In the **Transport of Diluted Species** interface, apply inlet and outlet boundary conditions together with initial values. The velocity from the **Laminar Flow** interface is automatically applied for the convective transport by the **Reacting Flow, Diluted Species** Multiphysics feature.


### *Transport Properties 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)**> **Transport of Diluted Species (tds)** click **Transport Properties 1**.
- 2 In the **Settings** window for **Transport Properties**, locate the **Diffusion** section.
- 3 In the  $D_c$  text field, type D.

### *Inflow 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Inflow**.  
At the inlet use the **Step** function to introduce a sharp but continuous step in the concentration.
- 2 In the **Settings** window for **Inflow**, locate the **Concentration** section.
- 3 In the  $c_{0,c}$  text field, type `step1(x[1/mm])`.
- 4 Select Boundary 20 only.

### *Outflow 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Outflow**.
- 2 Select Boundary 23 only.  
The **Outflow** feature assumes that the flow through the outlet is governed by the convection.


## MULTIPHYSICS

### *Reacting Flow, Diluted Species 1 (rfd1)*

In the **Physics** toolbar, click  **Multiphysics Couplings** and choose **Domain>Reacting Flow, Diluted Species**.

Two meshes are required: A coarse one for the **Laminar Flow** interface and a finer one for the **Transport of Diluted Species** interface.

## MESH 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
- 2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.
- 3 From the **Element size** list, choose **Extra coarse**.
- 4 Click  **Build All**.


## MESH 2

In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

### *Free Tetrahedral 1*

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

### *Size*

- 1 In the **Settings** window for **Size**, locate the **Element Size** section.
- 2 Click the **Custom** button.
- 3 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 0.65.
- 4 In the **Minimum element size** text field, type 0.35.
- 5 Click  **Build All**.

The study needs to be solved in two **Stationary** steps. First, the **Laminar Flow** is computed with **Mesh 1**. Second, the **Transport of Diluted Species** is calculated using **Mesh 2**.

## STUDY 1

### *Step 2: Stationary 2*


In the **Study** toolbar, click  **Study Steps** and choose **Stationary>Stationary**.

### *Laminar Flow*

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, type Laminar Flow in the **Label** text field.
- 3 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Transport of Diluted Species (tds)**.

### *Transport of Diluted Species*


- 1 In the **Model Builder** window, under **Study 1** click **Step 2: Stationary 2**.

- 2 In the **Settings** window for **Stationary**, type Transport of Diluted Species in the **Label** text field.
- 3 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Laminar Flow (spf)**.
- 4 In the **Study** toolbar, click  **Compute**.



## RESULTS

To reproduce [Figure 2](#), which uses a slice plot to visualize the concentration within the mixer, follow these steps.

### *Concentration, Slice*

- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type Concentration, Slice in the **Label** text field.

### *Slice 1*

- 1 Right-click **Concentration, Slice** and choose **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)>Transport of Diluted Species>Species c>c - Concentration - mol/m<sup>3</sup>**.
- 3 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.
- 4 In the **Planes** text field, type 8.
- 5 Click the  **Zoom Extents** button in the **Graphics** toolbar.  
Since quadratic elements was applied for the concentration, the resolution of the plot can be increased.
- 6 Click to expand the **Quality** section. From the **Resolution** list, choose **Finer**.
- 7 In the **Concentration, Slice** toolbar, click  **Plot**.

### *Velocity (spf)*

To create [Figure 3](#) that displays the velocity field, follow these steps.

### *Streamline 1*


- 1 In the **Model Builder** window, expand the **Results>Velocity (spf)** node.
- 2 Right-click **Velocity (spf)** and choose **Streamline**.

### *Slice*

- 1 In the **Settings** window for **Slice**, locate the **Plane Data** section.
- 2 From the **Plane** list, choose **xy-planes**.


3 In the **Planes** text field, type 8.

#### *Streamline I*


- 1 In the **Model Builder** window, click **Streamline I**.
- 2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
- 3 From the **Positioning** list, choose **Magnitude controlled**.
- 4 In the **Minimum distance** text field, type 0.025.
- 5 In the **Maximum distance** text field, type 0.1.
- 6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.
- 7 In the **Tube radius expression** text field, type 0.05.
- 8 Find the **Point style** subsection. From the **Color** list, choose **Yellow**.
- 9 Find the **Line style** subsection.
- 10 Select the **Radius scale factor** check box. In the associated text field, type 2.
- 11 In the **Velocity (spf)** toolbar, click  **Plot**.

Finally, reproduce the series of cross-sectional concentration plots for different z-coordinates shown in [Figure 4](#) with the following steps.

#### *Cut Plane I*

- 1 In the **Results** toolbar, click  **Cut Plane**.
- 2 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.
- 3 From the **Plane** list, choose **xy-planes**.
- 4 In the **z-coordinate** text field, type -2.

#### *Cross-sectional concentration plot*

- 1 In the **Results** toolbar, click  **2D Plot Group**.
- 2 In the **Settings** window for **2D Plot Group**, type Cross-sectional concentration plot in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Cut Plane I**.

#### *Surface I*

- 1 Right-click **Cross-sectional concentration plot** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Transport of Diluted Species>Species c>c - Concentration - mol/m<sup>3</sup>**.  
Increase the resolution also for this plot.

3 Click to expand the **Quality** section. From the **Resolution** list, choose **Finer**.

4 In the **Cross-sectional concentration plot** toolbar, click  **Plot**.

#### *Cut Plane 1*

1 In the **Model Builder** window, under **Results>Datasets** click **Cut Plane 1**.

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

3 In the **z-coordinate** text field, type 2.

#### *Cross-sectional concentration plot*

Repeat these steps for z-coordinate 6, 10, 14, 18, 22, 26, and 30 to reproduce the remaining plots in [Figure 4](#).

#### *Cut Plane 1*

1 In the **Model Builder** window, under **Results>Datasets** click **Cut Plane 1**.

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

3 In the **z-coordinate** text field, type 6.

#### *Cut Plane 1*

1 In the **Model Builder** window, under **Results>Datasets** click **Cut Plane 1**.

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

3 In the **z-coordinate** text field, type 10.

#### *Cut Plane 1*

1 In the **Model Builder** window, under **Results>Datasets** click **Cut Plane 1**.

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

3 In the **z-coordinate** text field, type 14.

#### *Cut Plane 1*

1 In the **Model Builder** window, under **Results>Datasets** click **Cut Plane 1**.

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

3 In the **z-coordinate** text field, type 18.

#### *Cut Plane 1*

1 In the **Model Builder** window, under **Results>Datasets** click **Cut Plane 1**.

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

3 In the **z-coordinate** text field, type 22.

#### *Cut Plane 1*

1 In the **Model Builder** window, under **Results>Datasets** click **Cut Plane 1**.

- 2 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.
- 3 In the **z-coordinate** text field, type 26.

#### *Cut Plane 1*

- 1 In the **Model Builder** window, under **Results>Datasets** click **Cut Plane 1**.
- 2 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.
- 3 In the **z-coordinate** text field, type 30.