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. It evaluates the mixing performance by calculating the concentration’s standard deviation.

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 6 mm; the length is \( 14R \), and the length of each blade is \( 3R \). The inlet flow is laminar and fully developed with an average velocity of 1 cm/s. At the outlet, the model specifies a constant reference pressure of 0 Pa. The equations for the momentum transport are the stationary Navier-Stokes equations in 3D:
\[ \rho (\mathbf{u} \cdot \nabla) \mathbf{u} = \nabla \cdot \left[ -p \mathbf{I} + \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) \right] \]
\[ \nabla \cdot \mathbf{u} = 0 \]

Here \( \eta \) denotes the dynamic viscosity (kg/(m·s)), \( \mathbf{u} \) is the velocity (m/s), \( \rho \) represents the fluid density (kg/m\(^3\)), and \( p \) denotes the pressure (Pa). The fluid’s properties are not affected by the change in concentration of the dissolved species.

The model studies the mixing performance by assuming a discontinuous concentration profile at the mixer’s inlet. The inlet concentration is defined as

\[
e_{\text{inlet}} = \begin{cases} 
c_0 & x < 0 \\
0 & x \geq 0
\end{cases}
\]

with the line \( x = 0 \) separating the two inlet sides. Diffusion and convection contribute to the mass flux, and the resulting mass transport equation is:

\[
\nabla \cdot (-D \nabla c + c \mathbf{u}) = 0
\]

Here \( D \) denotes the diffusion coefficient (m\(^2\)/s), and \( c \) is the concentration (mol/m\(^3\)).

At the outlet, the mass transport is mainly driven by convection. That is, the transport by diffusion is neglected in the normal direction of the pipe’s cross section. Because the convective term leads to instabilities in the solution, you need a fine mesh to obtain a stable solution for the concentration field.

The low Reynolds numbers, in the mixer implies that the Navier-Stokes equations do not require a particularly dense mesh. You can therefore first solve the Navier-Stokes equations on a coarse mesh and then map the solution onto a finer mesh. In the last solution step you use this mapped velocity field in the convective mass-transport term.

**Results**

Figure 2 shows a slice plot of the concentration in the mixer. The slice at the bottom shows the lighter and darker halves of the fluid with and without the dissolved species, respectively. As the fluid flows upward through the system, the two solutions are mixed and an almost constant concentration is obtained 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 show the concentration at $z = -2$ mm to $z = 30$ mm in steps of 4 mm.

References


Model Library path: Chemical_Reaction_Engineering_Module/Mixing/laminar_static_mixer
**Modeling Instructions**

**MODEL WIZARD**
1. Go to the Model Wizard window.
2. Click Next.
4. Click Add Selected.
5. In the Add physics tree, select Chemical Species Transport>Transport of Diluted Species (chds).
6. Click Add Selected.
7. Click Next.
8. In the Studies tree, select Preset Studies for Selected Physics>Stationary.
9. Click Finish.

**GLOBAL DEFINITIONS**

**Parameters**
1. In the Model Builder window, right-click Global Definitions and choose Parameters.
2. Go to the Settings window for Parameters.
3. Locate the Parameters section. In the Parameters table, enter the following settings:

<table>
<thead>
<tr>
<th>NAME</th>
<th>EXPRESSION</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>ra</td>
<td>3[mm]</td>
<td>Tube radius</td>
</tr>
<tr>
<td>u_mean</td>
<td>10[mm/s]</td>
<td>Mean inlet velocity</td>
</tr>
<tr>
<td>c0</td>
<td>5[mol/m^3]</td>
<td>Inlet concentration</td>
</tr>
<tr>
<td>D</td>
<td>5e-8[m^2/s]</td>
<td>Diffusion coefficient</td>
</tr>
</tbody>
</table>

**Step 1**
1. In the Model Builder window, right-click Global Definitions and choose Functions>Step.
2. Go to the Settings window for Step.
3. Locate the Parameters section. In the To edit field, type 5.
4. Click to expand the Smoothing section.
5. Select the Size of transition zone check box.
6. In the associated edit field, type 3e-4.
GEOMETRY 1
1. In the Model Builder window, click Model 1>Geometry 1.
2. Go to the Settings window for Geometry.
3. Locate the Units section. From the Length unit list, select mm.
4. Right-click Model 1>Geometry 1 and choose Work Plane.

Rectangle 1
1. In the Model Builder window, right-click Geometry and choose Rectangle.
2. Go to the Settings window for Rectangle.
3. Locate the Size section. In the Width edit field, type 2.4*3.
4. In the Height edit field, type 3/8.
5. Locate the Position section. From the Base list, select Center.
6. Click the Build Selected button.

Extrude 1
1. In the Model Builder window, right-click Work Plane 1 and choose Extrude.
2. Go to the Settings window for Extrude.
3. Locate the Distances from Work Plane section. In the associated table, enter the following settings:
   
<table>
<thead>
<tr>
<th>DISTANCES (MM)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.5</td>
</tr>
</tbody>
</table>
   
4. Locate the Twist Angles section. In the associated table, enter the following settings:
   
<table>
<thead>
<tr>
<th>TWIST ANGLES (DEG)</th>
</tr>
</thead>
<tbody>
<tr>
<td>30</td>
</tr>
</tbody>
</table>
   
5. Click the Build Selected button.

Copy 1
1. In the Model Builder window, right-click Geometry 1 and choose Transforms>Copy.
2. Select the object ext1 only.
3. Go to the Settings window for Copy.
4. Locate the Displacement section. In the z edit field, type 1.5.
5. Click the Build Selected button.
Rotate 1
1 In the **Model Builder** window, right-click **Geometry 1** and choose **Transforms>Rotate**.
2 Select the object *copy1* only.
3 Go to the **Settings** window for Rotate.
4 Locate the **Rotation Angle** section. In the **Rotation** edit field, type -30.
5 Click the **Build Selected** button.

Copy 2
1 In the **Model Builder** window, right-click **Geometry 1** and choose **Transforms>Copy**.
2 Select the objects *ext1* and *rot1* only.
3 Go to the **Settings** window for Copy.
4 Locate the **Displacement** section. In the **z** edit field, type 3.
5 Click the **Build Selected** button.

Rotate 2
1 In the **Model Builder** window, right-click **Geometry 1** and choose **Transforms>Rotate**.
2 Select the objects *copy2(1)* and *copy2(2)* only.
3 Go to the **Settings** window for Rotate.
4 Locate the **Rotation Angle** section. In the **Rotation** edit field, type -60.
5 Click the **Build Selected** button.

Copy 3
1 In the **Model Builder** window, right-click **Geometry 1** and choose **Transforms>Copy**.
2 Select the objects *ext1* and *rot1* only.
3 Go to the **Settings** window for Copy.
4 Locate the **Displacement** section. In the **z** edit field, type 6.
5 Click the **Build Selected** button.

Rotate 3
1 In the **Model Builder** window, right-click **Geometry 1** and choose **Transforms>Rotate**.
2 Go to the **Settings** window for Rotate.
3 Locate the **Rotation Angle** section. In the **Rotation** edit field, type -120.
4 Select the objects *copy3(1)* and *copy3(2)* only.
5 Click the **Build Selected** button.
Copy 4
1  In the Model Builder window, right-click Geometry 1 and choose Transforms>Copy.
2  Select the objects ext1, rot1, rot2(1), rot2(2), rot3(1), and rot3(2) only.
3  Go to the Settings window for Copy.
4  Locate the Displacement section. In the z edit field, type 9.
5  Click the Build Selected button.

Scale 1
1  In the Model Builder window, right-click Geometry 1 and choose Transforms>Scale.
2  Select the objects copy4(4), copy4(5), copy4(2), copy4(3), copy4(6), and copy4(1) only.
3  Go to the Settings window for Scale.
4  Locate the Scale Factor section. From the Scaling list, select Anisotropic.
5  In the x edit field, type -1.
6  Click the Build Selected button.

Rotate 4
1  In the Model Builder window, right-click Geometry 1 and choose Transforms>Rotate.
2  Select the objects sca1(2), sca1(3), sca1(1), sca1(6), sca1(4), and sca1(5) only.
3  Go to the Settings window for Rotate.
4  Locate the Rotation Angle section. In the Rotation edit field, type 90.
5  Click the Build Selected button.

Copy 5
1  In the Model Builder window, right-click Geometry 1 and choose Transforms>Copy.
2  Select the objects ext1, rot1, rot2(1), rot2(2), rot3(1), and rot3(2) only.
3  Go to the Settings window for Copy.
4  Locate the Displacement section. In the z edit field, type 18.
5  Click the Build Selected button.

Cylinder 1
1  In the Model Builder window, right-click Geometry 1 and choose Cylinder.
2  Go to the Settings window for Cylinder.
3  Locate the Size and Shape section. In the Radius edit field, type 3.
4  In the Height edit field, type 42.
5  Locate the Position section. In the z edit field, type -6.
6 Locate the Rotation Angle section. In the Rotation edit field, type 15.

7 Click the Build Selected button.

**Union 1**

1 In the Model Builder window, right-click Geometry 1 and choose Boolean Operations>Union.

2 Select the objects copy5(6), rot4(1), rot4(4), rot4(5), rot4(2), rot4(3), rot4(6), ext1, rot1, copy5(3), rot2(1), copy5(2), copy5(5), copy5(4), rot2(2), rot3(1), copy5(1), and rot3(2) only.

3 Click the Build Selected button.

**Difference 1**

1 In the Model Builder window, right-click Geometry 1 and choose Boolean Operations>Difference.

2 Go to the Settings window for Difference.

3 Locate the Difference section. Under Objects to add, click Activate Selection.

4 Select the object cyl1 only to add it to the Objects to add list in the previous instruction.

5 Under Objects to subtract, click Activate Selection.

6 Select the object uni1 only.

7 Click the Build Selected button.

8 Click the Zoom Extents button on the Graphics toolbar.

**MATERIALS**

1 In the Model Builder window, right-click Model 1>Materials and choose Open Material Browser.

2 Go to the Material Browser window.

3 Locate the Materials section. In the Materials tree, select Built-In>Water, liquid.

4 Right-click and choose Add Material to Model from the menu.

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

*Initial Values 1*
1. In the **Model Builder** window, expand the **Model 1>Laminar Flow** node, then click **Initial Values 1**.
2. Go to the **Settings** window for Initial Values.
3. Locate the **Initial Values** section. Specify the **u** vector as

<table>
<thead>
<tr>
<th>x</th>
<th>u_mean</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>y</td>
</tr>
<tr>
<td>0</td>
<td>z</td>
</tr>
</tbody>
</table>

**Inlet 1**
1. In the **Model Builder** window, right-click **Laminar Flow** and choose **Inlet**.
2. Select Boundary 20 only.
3. Go to the **Settings** window for Inlet.
4. Locate the **Velocity** section. In the \( U_0 \) edit field, type \( 2 \times (1 - (x^2+y^2)/ra^2) \times u_{\text{mean}} \).

This gives a parabolic inlet velocity profile appropriate for fully developed laminar flow with mean velocity \( u_{\text{mean}} \).

**Outlet 1**
1. In the **Model Builder** window, right-click **Laminar Flow** and choose **Outlet**.
2. Select Boundary 23 only.

**TRANSPORT OF DILUTED SPECIES**

*Convection and Diffusion 1*
1. In the **Model Builder** window, expand the **Model 1>Transport of Diluted Species** node, then click **Convection and Diffusion 1**.
2. Go to the **Settings** window for Convection and Diffusion.
3. Locate the **Diffusion** section. In the \( D_i \) edit field, type \( D \).
4. Locate the **Model Inputs** section. From the **u** list, select **Velocity field (spf/fp1)**.

**Inflow 1**
1. In the **Model Builder** window, right-click **Transport of Diluted Species** and choose **Inflow**.
2. Go to the **Settings** window for Inflow.
3  Locate the Concentration section. In the $c_0,c$ edit field, type $\text{step1}(x[1/\text{mm}])$.
4  Select Boundary 20 only.

Outflow 1
1  In the Model Builder window, right-click Transport of Diluted Species and choose Outflow.
2  Select Boundary 23 only.

Mesh 1
In the Model Builder window, right-click Model 1>Mesh 1 and choose Free Tetrahedral.
Size
1  In the Model Builder window, click Size.
2  Go to the Settings window for Size.
3  Locate the Element Size section. From the Predefined list, select Extra fine.
4  Click the Build All button.

Mesh 2
1  In the Model Builder window, right-click Model 1 and choose Mesh.
2  Right-click Model 1>Meshes>Mesh 2 and choose Free Tetrahedral.
Size
1  In the Model Builder window, click Size.
2  Go to the Settings window for Size.
3  Locate the Element Size section. Click the Custom button.
4  Locate the Element Size Parameters section. In the Maximum element size edit field, type 0.7.
5  In the Minimum element size edit field, type 0.35.
6  Click the Build All button.

Study 1
Step 1: Stationary
1  In the Model Builder window, right-click Study 1 and choose Study Steps>Stationary.
2  Go to the Settings window for Stationary.
3 Locate the **Physics Selection** section. In the associated table, enter the following settings:

<table>
<thead>
<tr>
<th>PHYSICS INTERFACE</th>
<th>USE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Transport of diluted species (chds)</td>
<td>✗</td>
</tr>
</tbody>
</table>

**Step 2: Stationary 2**

1 In the **Model Builder** window, click **Study 1>Step 2: Stationary 2**.
2 Go to the **Settings** window for Stationary.
3 Locate the **Physics Selection** section. In the associated table, enter the following settings:

<table>
<thead>
<tr>
<th>PHYSICS INTERFACE</th>
<th>USE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Laminar flow (spf)</td>
<td>✗</td>
</tr>
</tbody>
</table>

4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

**RESULTS**

To reproduce plot in **Figure 3** that visualizes the velocity field, follow these steps.

**Concentration (chds)**

1 In the **Model Builder** window, expand the **Results>Concentration (chds)** node, then click **Slice 1**.
2 Go to the **Settings** window for Slice.
3 Locate the **Plane Data** section. From the **Plane** list, select **xy-planes**.
4 In the **Planes** edit field, type **8**.
5 Click the **Plot** button.

**Velocity (spf)**

1 In the **Model Builder** window, expand the **Results>Velocity (spf)** node, then click **Slice 1**.
2 Go to the **Settings** window for Slice.
3 Locate the **Plane Data** section. From the **Plane** list, select **xy-planes**.
4 In the **Planes** edit field, type **8**.
5 Locate the **Expression** section. From the **Unit** list, select **mm/s**.
6 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Streamline**.
7 Go to the **Settings** window for Streamline.
Solved with COMSOL Multiphysics 4.2

8 Locate the Streamline Positioning section. From the Positioning list, select Magnitude controlled.

9 In the Min distance edit field, type 0.025.

10 In the Max distance edit field, type 0.1.

11 Locate the Coloring and Style section. From the Line type list, select Tube.

12 In the Tube radius expression edit field, type 0.05.

13 From the Color list, select Yellow.

14 Select the Radius scale factor check box.

15 In the associated edit field, type 2.

16 Click the Plot button.

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

Data Sets
1 In the Model Builder window, right-click Results>Data Sets and choose Cut Plane.

2 Go to the Settings window for Cut Plane.

3 Locate the Plane Data section. From the Plane list, select xy-planes.

4 In the z-coordinate edit field, type -2.

2D Plot Group 5
1 In the Model Builder window, right-click Results and choose 2D Plot Group.

2 Go to the Settings window for 2D Plot Group.

3 Locate the Data section. From the Data set list, select Cut Plane 3D 1.

4 Right-click 2D Plot Group 5 and choose Surface.

5 Go to the Settings window for Surface.

6 In the upper-right corner of the Expression section, click Replace Expression.

7 From the menu, choose Concentration (c).

8 Locate the Coloring and Style section. Clear the Color legend check box.

9 Click the Plot button.

10 Click the Zoom Extents button on the Graphics toolbar.

Data Sets
1 In the Model Builder window, click Results>Data Sets>Cut Plane 1.

2 Go to the Settings window for Cut Plane.
3. Locate the **Plane Data** section. In the *z-coordinate* edit field, type 2.

4. Right-click **2D Plot Group 5**.
   Repeat these steps for *z-coordinate* 6, 10, 14, 18, 22, 26, and 30 to reproduce the remaining plots in **Figure 4**.