Model created in COMSOL Multiphysics 6.4

Continuous Mixer

Introduction

Continuous mixing is used in process equipment to mix chemical species in a single pass. Compared to batch mixing, this operation has the advantage that the tank filling and emptying steps are eliminated, meaning that the process can be run without interruptions. On the other hand, the disadvantage of continuous mixing is that time the material spends in the mixer, the residence time, is not uniform. The residence time is to a large extent dependent on the details of the mixer construction and the operation. Parameters that affect the residence time are, for example, the position and speed of the impeller and the feed rate.

Model Definition

In this model a tank with an asymmetrically positioned impeller is studied. The impeller is attached to a shaft protruding from the wall as seen in Figure 1. The tank is also fitted with a tubular inlet and outlet, positioned at the top and bottom of the tank. The tank and impeller geometry are defined using parts from the Mixer Module Parts Library. A liquid is continuously fed into the tank near the top. At the same time liquid exits through an outlet at the bottom of the tank. The impeller rotates at 15 rpm resulting in a turbulent flow of the liquid. The flow field resulting from the impeller agitation is solved for using a frozen rotor analysis.

The mixing process and the residence time distribution are evaluated using particle tracing of massless particles. A benefit of using particles is that a distinct pulse to be injected is easily modeled, for example by distributing all particles on the inlet boundary at the initial time step. In addition, the number of particles to track, and thus the computational requirements, can be controlled. This in contrast to solving for a continuous concentration field, in which case the concentration in all points of the mesh need to be solved for in each time step. This could be a substantial effort since a long simulation time is typically needed when evaluating the mixing in a stirred tank.

In the present model, a pulse containing 50,000 massless particles are injected at the feed inlet and subjected to the flow field from the frozen rotor simulation. The particles are tracked as they flow through the mixer until they exit at the outlet. A particle counter feature is used at the outlet to track the number of particles that exit the tank.

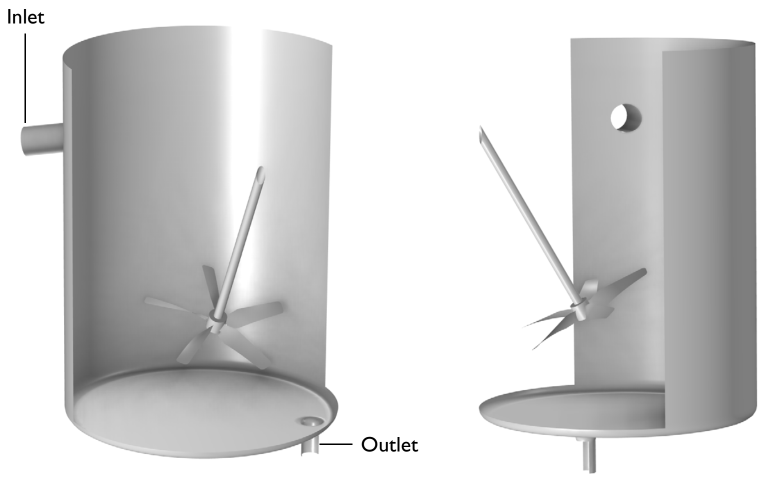

Figure 1: The continuous mixer geometry including a tank fitted with a tubular inlet and outlet. The tank is agitated by an impeller attached to a shaft protruding from the tank wall.

Results and Discussion

Figure 2 shows the computed velocity field. The inlet stream is seen to propagate through the tank and turn at the opposite wall. Fluid is drawn downward in the vicinity of the impeller. Along the outer walls the fluid in general flows upward. This implies that a fluid parcel that does not pass close to the outlet when arriving at the bottom will be transported up again along the tank walls.

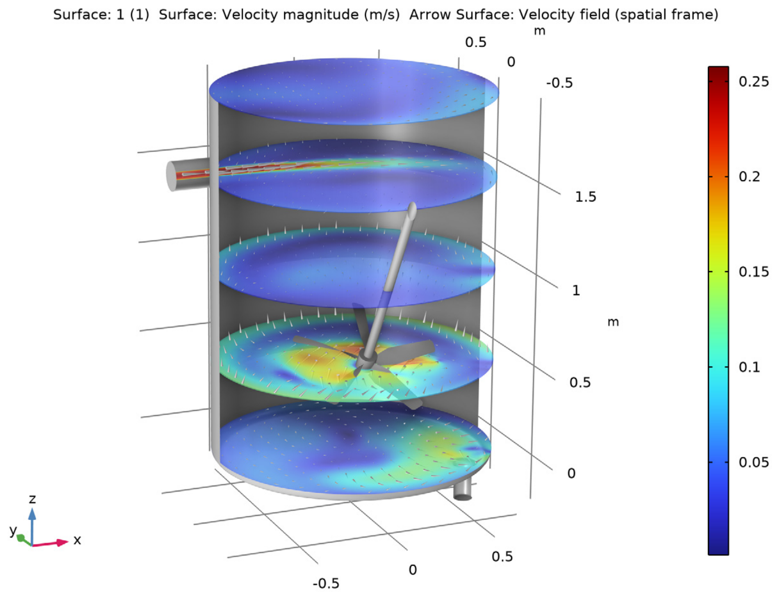

Figure 2: The frozen rotor flow field in the mixer visualized at five positions.

The trajectories of the injected particles after 100 s of simulation time are plotted in Figure 3. Each particle is represented by a cone pointing in direction of the particle velocity. At this time, the particles are distributed throughout the whole vessel, but still mostly in certain bands. Some of the particles are seen to have passed the outlet. These particles are still at the outlet since a freeze condition is applied. The particles are colored by the velocity magnitude. As expected, particles close to the impeller are subjected to higher velocities. However, the highest velocities occur inside the tubular outlet channel.

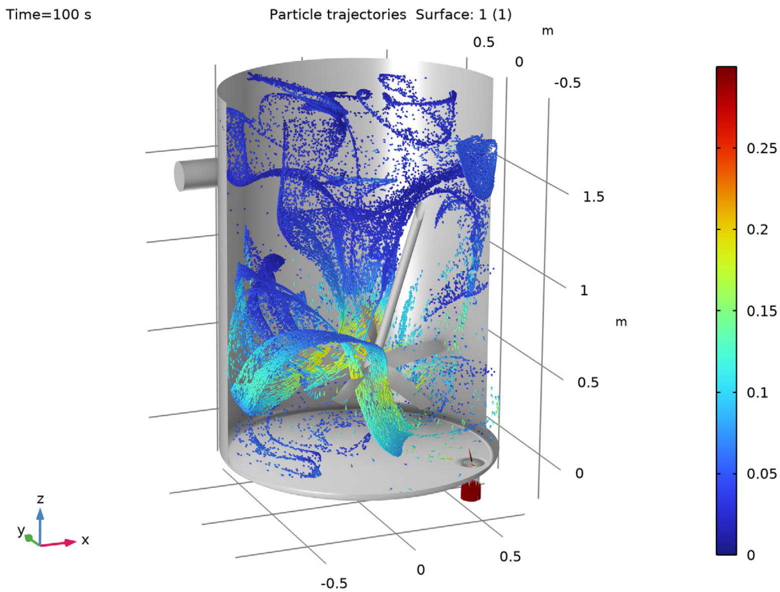

Figure 3: Flow patterns inside the test apparatus with water at 1 m/s.

By injecting a pulse input of a tracer species into a reactor or mixing vessel, and subsequently measuring the concentration $C(t)$ at the outlet, the residence time distribution function $E(t)$ can be defined as (Ref. 1)

$$E(t) = \frac{C(t)}{\int_0^{\infty} C(t)dt} \qquad (1)$$

The residence time distribution function can be said to quantify how much time different fractions of the total amount of species being mixed have spent in the vessel. Using an integral form, the fraction of the tracer species leaving the vessel that have spent between times t_1 and t_2 inside the vessel can be defined as

$$\delta_{t_1-t_2} = \int_{t_1}^{t_2} E(t)dt \qquad (2)$$

In the present model, the tracer species are as discrete particles, meaning that the concentration at the outlet is not known in each time step. To estimate the concentration, the number of particles that exit the tank over a short time is computed. In Figure 4, the residence time distribution, using a sampling time of 10 s, is plotted. No particles exit before 80 s. There are two notable peaks, at 150 s and 300 s, with significant ejection of particles, before the distribution levels out. The fraction remaining in the tank is also plotted. More than 60% of the particles remain in the tank after 500 s, corresponding to 125 rotations of the impeller.

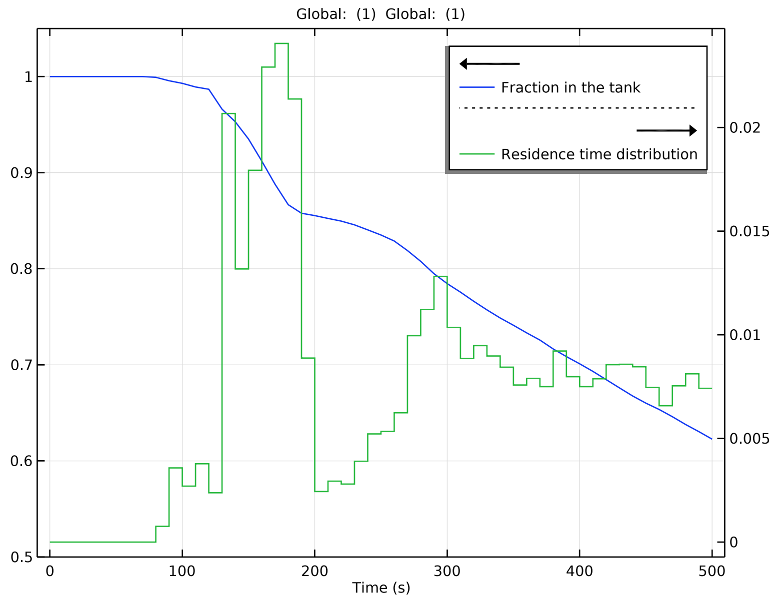

Figure 4: Residence time distribution for the particles injected into the mixer.

Reference

1. H.S. Fogler, *Elements of Chemical Reaction Engineering*, 2nd Ed., Prentice-Hall, 1992

Application Library path: `Mixer_Module/Tutorials/continuous_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** > **Rotating Machinery, Fluid Flow** > **Turbulent Flow** > **Turbulent Flow, k-ε**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Frozen Rotor**.
6 Click **Done**.

Load the model geometry from a geometry sequence.

GEOMETRY 1

1 In the **Geometry** toolbar, click **Insert Sequence**.
2 Browse to the model's Application Libraries folder and double-click the file `continuous_mixer_geom_sequence.mph`.
3 In the **Geometry** toolbar, click **Build All**.

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 `continuous_mixer_parameters.txt`.

Parameters 2

1 In the **Home** toolbar, click **Parameters** and choose **Add** > **Parameters**.
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 `continuous_mixer_mesh_parameters.txt`.

DEFINITIONS

Impeller Blades

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node.

2 Right-click **Definitions** and choose **Selections** > **Explicit**.

3 In the **Settings** window for **Explicit**, type `Impeller Blades` in the **Label** text field.

4 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 22, 23, 25, 26, 35, and 44–48 only.

Blade edges

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Edge**.

4 Select Edges 31–42, 47–52, 54, 60–64, 75, 76, 101–104, 106, 108, 109, 112–122, and 124–139 only.

5 In the **Label** text field, type `Blade edges`.

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.

TURBULENT FLOW, K-ε (SPF)

1 In the **Settings** window for **Turbulent Flow, k-ε**, locate the **Physical Model** section.

2 Select the **Include gravity** checkbox.

3 Select the **Use reduced pressure** checkbox.

Interior Wall 1

1 In the **Model Builder** window, expand the **Turbulent Flow, k-ε (spf)** node.

2 Right-click **Component 1 (comp1)** > **Turbulent Flow, k-ε** (spf) and choose **Interior Wall**.

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

4 From the **Selection** list, choose **Impeller Blades**.

Inlet 1

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

2 Select Boundary 1 only.

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

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

5 Clear the **Apply condition on each disjoint selection separately** checkbox.

6 Locate the **Fully Developed Flow** section. In the U_{av} text field, type `v_in`.

Outlet 1

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

2 Select Boundaries 51, 52, 54, and 55 only.

Wall 2

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

2 Select Boundary 10 only.

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

4 From the **Wall condition** list, choose **Slip**.

MOVING MESH

In the **Model Builder** window, expand the **Moving Mesh** node.

Rotating Domain 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Moving Mesh** > **Rotating Domain 1** node, then click **Rotating Domain 1**.

2 In the **Settings** window for **Rotating Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Manual**.

4 Click **Clear Selection**.

5 Select Domain 4 only.

6 Locate the **Rotation** section. From the **Rotational velocity expression** list, choose **Constant angular velocity**.

7 In the ω text field, type `Omega`.

8 Locate the **Axis** section. Specify the $\mathbf{r}_{\mathrm{ax}}$ vector as

0	X
-0.175	Y
0.5	Z

9 Specify the $\mathbf{u}_{rot}$ vector as

0	X
-sin(impellerTilt[deg])	Y
cos(impellerTilt[deg])	Z

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

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

4 Click the **Add to Component 1** button in the window toolbar.

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

6 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** 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 **Massless**.

3 Locate the **Additional Variables** section. Select the **Store particle release statistics** checkbox.

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

Inlet 1

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

2 Select Boundary 1 only.

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

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

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

Outlet 1

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

2 Select Boundaries 51, 52, 54, and 55 only.

Particle Counter 1

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

2 Select Boundaries 51, 52, 54, and 55 only.

MESH 1

Size 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Size**.

Size

1 In the **Settings** window for **Size**, locate the **Element Size** section.

2 From the **Predefined** list, choose **Coarse**.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `h_dom_max`.

5 In the **Minimum element size** text field, type `h_dom_min`.

6 In the **Maximum element growth rate** text field, type `h_dom_growth`.

7 In the **Curvature factor** text field, type `h_dom_curvef`.

8 In the **Resolution of narrow regions** text field, type `h_dom_narrowres`.

Size 1

1 In the **Model Builder** window, click **Size 1**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Click to select the **Activate Selection** toggle button.

5 Select Boundaries 2–9, 14, 15, 22, 23, 25–38, 40, 42–50, 53, and 56 only.

6 Locate the **Element Size** section. From the **Predefined** list, choose **Fine**.

7 From the **Calibrate for** list, choose **Fluid dynamics**.

8 Click the **Custom** button.

9 Locate the **Element Size Parameters** section.

10 Select the **Maximum element size** checkbox. In the associated text field, type `h_bnd_max`.

11 Select the **Minimum element size** checkbox. In the associated text field, type `h_bnd_min`.

12 Select the **Curvature factor** checkbox. In the associated text field, type `h_bnd_curvef`.

Size 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Impeller Blades**.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.

6 Click the **Custom** button.

7 Locate the **Element Size Parameters** section.

8 Select the **Maximum element size** checkbox. In the associated text field, type `h_bnd_max/2.5`.

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

2 Select Boundaries 1, 51, 52, 54, and 55 only.

Size 1

1 Right-click **Free Triangular 1** and choose **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.

5 Select the **Maximum element growth rate** checkbox. In the associated text field, type `h_inout_growth`.

Free Triangular 1

Right-click **Free Triangular 1** and choose **Size**.

Size 2

1 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
2 From the **Geometric entity level** list, choose **Edge**.
3 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.
4 From the **Predefined** list, choose **Extra fine**.
5 Select Edges 1, 2, 4, 6, 142, 144, 150, and 155 only.
6 Click the **Custom** button.
7 Locate the **Element Size Parameters** section.
8 Select the **Maximum element size** checkbox. In the associated text field, type `h_bnd_max*0.45`.

Swept 1

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 1** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 From the **Distribution type** list, choose **Predefined**.
4 In the **Number of elements** text field, type `n_inlet`.
5 In the **Element ratio** text field, type `n_inlet_hratio`.
6 Select the **Reverse direction** checkbox.

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 3 only.

Distribution 1

1 Right-click **Swept 2** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type `n_inlet*2`.

5 In the **Element ratio** text field, type `n_inlet_hratio`.

Free Triangular 2

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

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

3 From the **Selection** list, choose **Impeller Blades**.

Size 1

1 Right-click **Free Triangular 2** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Edge**.

4 From the **Selection** list, choose **Blade edges**.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `h_blade_max`.

8 Select the **Minimum element size** checkbox. In the associated text field, type `h_blade_min`.

Corner Refinement 1

1 In the **Mesh** toolbar, click **More Attributes** and choose **Corner Refinement**.

2 In the **Settings** window for **Corner Refinement**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Click in the **Graphics** window and then press Ctrl+A to select all domains.

5 Locate the **Boundary Selection** section. Click to select the **Activate Selection** toggle button.

6 Select Boundaries 2–10, 14, 15, 22, 23, 25–38, 40, 42–50, 53, and 56 only.

Free Tetrahedral 1

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

Size 1

1 Right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 4 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element growth rate** checkbox. In the associated text field, type `h_blade_growth`.

Boundary Layers 1

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

2 In the **Settings** window for **Boundary Layers**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Click in the **Graphics** window and then press Ctrl+A to select all domains.

5 Click to expand the **Corner Settings** section. From the **Handling of sharp edges** list, choose **Trimming**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

3 In the **Number of layers** text field, type `5`.

4 In the **Thickness adjustment factor** text field, type `bl_thicknessf`.

5 Select Boundaries 2–9, 14, 15, 22, 23, 25–38, 40, 42–50, 53, and 56 only.

6 Click **Build All**.

STUDY 1

1 In the **Model Builder** window, click **Study 1**.

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

3 Clear the **Generate default plots** checkbox.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** node, then click **Segregated 1**.

4 In the **Settings** window for **Segregated**, locate the **General** section.

5 In the **Target error estimate** text field, type `0.1`.

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

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(0,1,100) range(100+deltaT,deltaT, tEnd)`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Turbulent Flow, k-ε** (spf).

5 Click **Disable in Solvers**.

6 In the tree, select **Component 1 (comp1)** > **Moving Mesh, Controls spatial frame** > **Rotating Domain 1**.

7 Click **Disable**.

8 Click **Disable**.

9 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**.

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

11 From the **Study** list, choose **Study 1, Frozen Rotor**.

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

DEFINITIONS

Hide for Geometry 1

1 In the **Model Builder** window, expand the **Definitions** node.

2 Right-click **View 1** and choose **Hide for Geometry**.

3 In the **Settings** window for **Hide for Geometry**, locate the **Selection** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 On the object **igf1**, select Boundaries 2, 3, 6, 10, 12, 13, 16–21, 24, 36, 39, 41, 49, and 53 only.

RESULTS

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

Cut Plane 1

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** and choose **Cut Plane**.

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

4 Select the **Additional parallel planes** checkbox.

5 From the **Plane** list, choose **xy-planes**.

6 In the **Distances** text field, type `0.5 1.0 1.5 1.95`.

All Walls

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `All Walls` in the **Label** text field.

3 Select Boundaries 2–10, 14, 15, 22, 23, 25–30–32–38, 40, 42–50, 53, and 56 only.

Exterior Walls

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Exterior Walls` in the **Label** text field.

3 Select Boundaries 2–10, 14, 15, 27–34, 36–38, 40, 42, 43, 49, 50, 53, and 56 only.

Interior Walls

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Interior Walls` in the **Label** text field.

3 Select Boundaries 22, 23, 25, 26, 35, and 44–48 only.

Velocity (spf)

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

2 In the **Settings** window for **3D Plot Group**, type `Velocity (spf)` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

1 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 **Gray**.

Surface 2

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

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

3 From the **Dataset** list, choose **Cut Plane 1**.

Transparency 1

1 In the **Model Builder** window, right-click **Surface 2** and choose **Transparency**.

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

3 Find the **Transparency** subsection. Set the **Transparency** value to **0.4**.

4 In the **Velocity (spf)** toolbar, click **Plot**.

5 In the **Graphics** window toolbar, click next to **Scene Light**, then choose **Ambient Occlusion**.

Arrow Surface 1

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

2 In the **Settings** window for **Arrow Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Cut Plane 1**.

4 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `900`.

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

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

Pressure (spf)

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

2 In the **Settings** window for **3D Plot Group**, type `Pressure (spf)` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

1 Right-click **Pressure (spf)** and choose **Surface**.

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

3 From the **Dataset** list, choose **All Walls**.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

Pressure

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

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

3 From the **Dataset** list, choose **Exterior Walls**.

4 Locate the **Expression** section. In the **Expression** text field, type `p`.

5 Locate the **Levels** section. Clear the **Round the levels** checkbox.

6 In the **Label** text field, type `Pressure`.

7 Locate the **Levels** section. In the **Total levels** text field, type `40`.

Upside Pressure

1 Right-click **Pressure (spf)** and choose **Contour**.

2 In the **Settings** window for **Contour**, type `Upside Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Interior Walls**.

4 Locate the **Levels** section. Clear the **Round the levels** checkbox.

5 Locate the **Expression** section. In the **Expression** text field, type `up(p)`.

6 Locate the **Levels** section. In the **Total levels** text field, type `40`.

Deformation 1

1 Right-click **Upside Pressure** and choose **Deformation**.

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

3 In the **x-component** text field, type `nx/sqrt(up(tremetric))`.

4 In the **y-component** text field, type `ny/sqrt(up(tremetric))`.

5 In the **z-component** text field, type `nz/sqrt(up(tremetric))`.

6 Locate the **Scale** section.

7 Select the **Scale factor** checkbox. In the associated text field, type `0.1`.

Downside Pressure

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

2 In the **Settings** window for **Contour**, type `Downside Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Interior Walls**.

4 Locate the **Expression** section. In the **Expression** text field, type `down(p)`.

5 Locate the **Levels** section. In the **Total levels** text field, type `40`.

6 Clear the **Round the levels** checkbox.

Deformation 1

1 Right-click **Downside Pressure** and choose **Deformation**.

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

3 In the **x-component** text field, type `-nx/sqrt(down(tremetric))`.

4 In the **y-component** text field, type `-ny/sqrt(down(tremetric))`.

5 In the **z-component** text field, type `-nz/sqrt(down(tremetric))`.

6 Locate the **Scale** section.

7 Select the **Scale factor** checkbox. In the associated text field, type `0.1`.

Wall Resolution (spf)

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

2 In the **Settings** window for **3D Plot Group**, type `Wall Resolution (spf)` in the **Label** text field.

Wall Resolution

1 Right-click **Wall Resolution (spf)** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Wall Resolution` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Exterior Walls**.

4 Locate the **Expression** section. In the **Expression** text field, type `spf.Delta_wPlus`.

Wall Resolution (spf)

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

Wall Resolution, Interior Walls

1 In the **Wall Resolution (spf)** toolbar, click **More Plots** and choose **Surface Slit**.

2 In the **Settings** window for **Surface Slit**, type `Wall Resolution, Interior Walls` in the **Label** text field.

3 In the **Model Builder** window, click **Wall Resolution, Interior Walls**.

4 In the **Label** text field, type `Wall Resolution, Interior Walls`.

5 Locate the **Data** section. From the **Dataset** list, choose **Interior Walls**.

6 Locate the **Expression on the Upside** section. In the **Expression** text field, type `spf.Delta_wPlus_u`.

7 Locate the **Expression on the Downside** section. In the **Expression** text field, type `spf.Delta_wPlus_d`.

Residence time distribution

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

2 In the **Settings** window for **1D Plot Group**, type `Residence time distribution` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

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

5 In the **Times (s)** text field, type `range(0,deltaT,tEnd)`.

Global 1

1 Right-click **Residence time distribution** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`1-fpt.pcnt1.Nsel/nPart`	1	

Global 2

1 In the **Model Builder** window, right-click **Residence time distribution** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`(fpt.pcnt1.Nsel-if(t>deltaT,at(t-deltaT, fpt.pcnt1.Nsel),0))/nPart`	1	

4 In the **Residence time distribution** toolbar, click **Plot**.

Residence time distribution

1 In the **Model Builder** window, click **Residence time distribution**.

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

3 Select the **Two y-axes** checkbox.

4 In the table, select the **Plot on secondary y-axis** checkbox for **Global 2**.

5 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

6 In the **y minimum** text field, type `0.5`.

7 In the **y maximum** text field, type `1.05`.

8 Locate the **Grid** section. Select the **Manual spacing** checkbox.

9 In the **x spacing** text field, type `100`.

10 In the **y spacing** text field, type `0.1`.

11 In the **Secondary y spacing** text field, type `0.005`.

Global 1

1 In the **Model Builder** window, click **Global 1**.

2 In the **Settings** window for **Global**, click to expand the **Legends** section.

3 From the **Legends** list, choose **Manual**.

4 In the table, enter the following settings:

Legends
`Fraction in the tank`

Global 2

1 In the **Model Builder** window, click **Global 2**.

2 In the **Settings** window for **Global**, click to expand the **Coloring and Style** section.

3 From the **Function type** list, choose **Discrete**.

4 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`Residence time distribution`

6 Click to collapse the **Legends** section. In the **Residence time distribution** toolbar, click **Plot**.

Particle Trajectories (fpt)

1 In the **Model Builder** window, under **Results** click **Particle Trajectories (fpt)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Time (s)** list, choose **100**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

Right-click **Particle Trajectories (fpt)** and choose **Surface**.

Surface 1

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

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 **Gray**.

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 **Point style** subsection. From the **Type** list, choose **Comet tail**.

Color Expression 1

1 In the **Model Builder** window, expand the **Particle Trajectories 1** node, then click **Color Expression 1**.

2 In the **Settings** window for **Color Expression**, click to expand the **Range** section.

3 Select the **Manual color range** checkbox.

4 In the **Maximum** text field, type `0.3`.

5 In the **Minimum** text field, type `0`.

6 In the **Particle Trajectories (fpt)** toolbar, click **Plot**.

Geometry Modeling Instructions

Follow steps below to generate the geometry used in this model.

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **3D**.

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
impellerTilt	30	30	Impeller shaft tilt angle
r_in	0.08[m]	0.08 m	Inlet radius
Vol	3.38[m^3]	3.38 m^3	Tank volume

PART LIBRARIES

1 In the **Geometry** toolbar, click **Part Libraries**.
2 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
3 In the **Part Libraries** window, select **Mixer Module** > **Tanks** > **flat_bottom_tank** in the tree.
4 Click **Add to Geometry**.

GEOMETRY 1

Flat Bottom Tank 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Flat Bottom Tank 1 (pi1)**.
2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
n_ba	0[1]	0	Number of baffles
w_ba	0.15[m]	0.15 m	Baffle width
d_im	1[m]	1 m	Impeller diameter
d_ta	1.5[m]	1.5 m	Tank diameter
h_ta	2.0[m]	2 m	Tank height
rm_b_ta	0.3[m]	0.3 m	Minor radius of the tank bottom
hp_ta	0[m]	0 m	Height position, cylindrical surface
rf_ta	0.05[m]	0.05 m	Fillet radius of lower tank edge

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.
2 Select the object **pi1** only.
3 In the **Settings** window for **Union**, locate the **Union** section.
4 Clear the **Keep interior boundaries** checkbox.
5 Click **Build Selected**.

PART LIBRARIES

1 In the **Home** toolbar, click **Windows** and choose **Part Libraries**.
2 In the **Model Builder** window, click **Geometry 1**.
3 In the **Part Libraries** window, select **Mixer Module** > **Shafts** > **impeller_shaft** in the tree.
4 Click **Add to Geometry**.

GEOMETRY 1

Impeller Shaft 1 (pi2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Impeller Shaft 1 (pi2)**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
hp_im	`0.5[m]`	0.5 m	Position of the lowest part of the impeller hub or impeller shaft along the z-axis
l_is	`2[m]`	2 m	Impeller shaft length

PART LIBRARIES

1 In the **Home** toolbar, click **Windows** and choose **Part Libraries**.

2 In the **Model Builder** window, click **Geometry 1**.

3 In the **Part Libraries** window, select **Mixer Module** > **Impellers, Axial** > **pitched_blade_impeller_constant_pitch** in the tree.

4 Click **Add to Geometry**.

GEOMETRY 1

Pitched Blade Impeller with Constant Pitch 1 (pi3)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Pitched Blade Impeller with Constant Pitch 1 (pi3)**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
n_ib	`5[1]`	5	Number of impeller blades
d_im	`0.8[m]`	0.8 m	Impeller diameter
hp_im	`0.5[m]`	0.5 m	Position of the lowest part of the impeller hub or impeller shaft along the z-axis
pa_cs_im	`1`	1	Add cross-section planes for flow evaluation above and below the impeller: 1 = add planes, 0 = do not add planes.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the objects **pi2**, **pi3(1)**, and **pi3(2)** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 From the **Axis type** list, choose **x-axis**.

5 In the **Angle** text field, type `impellerTilt`.

6 Locate the **Point on Axis of Rotation** section. In the **z** text field, type `0.5`.

7 Click **Build Selected**.

Move 1 (mov1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Move**.

2 Select the objects **rot1(1)**, **rot1(2)**, and **rot1(3)** only.

3 In the **Settings** window for **Move**, locate the **Displacement** section.

4 In the **y** text field, type `-0.175`.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the objects **mov1(3)** and **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 objects **mov1(1)** and **mov1(2)** only.

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 `r_in`.

4 In the **Height** text field, type `r_in*8`.

5 Locate the **Position** section. In the **x** text field, type `-0.9`.

6 In the **y** text field, type `0.3`.

7 In the **z** text field, type `1.5`.

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

Work Plane 1 (wp1)

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**.

4 In the **y-coordinate** text field, type `-0.35`.

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 `0.2`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **xw** text field, type `0.25`.

6 In the **yw** text field, type `-0.25`.

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 `0.25`.

4 In the **Height** text field, type `0.2`.

5 Locate the **Position** section. In the **xw** text field, type `0.25`.

6 In the **yw** text field, type `-0.25`.

Work Plane 1 (wp1) > Fillet 1 (fil1)

1 In the **Work Plane** toolbar, click **Fillet**.

2 On the object **r1**, select Point 3 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `0.03`.

Work Plane 1 (wp1) > Difference 1 (dif1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r2** 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 **fil1** only.

Geometry 1

Work Plane 1 (wp1)

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** node.

Revolve 1 (rev1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Revolve**.

2 In the **Settings** window for **Revolve**, locate the **Revolution Axis** section.

3 Find the **Point on the revolution axis** subsection. In the **xw** text field, type `0.5`.

4 In the **yw** text field, type `-0.25`.

5 Locate the **Revolution Angles** section. Clear the **Keep original faces** checkbox.

6 In the **Geometry** toolbar, click **Build All**.

Ignore Edges 1 (ige1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 In the **Settings** window for **Ignore Edges**, locate the **Input** section.

3 Click the **Paste Selection** button for **Edges to ignore**.

4 In the **Paste Selection** dialog, type `13, 27-29, 60, 85, 88, 90, 96, 97, 99, 101, 105, 109, 115, 116, 120, 121, 147, 148, 171, 180, 188, 189, 197, 198` in the **Selection** text field.

5 Click **OK**.

Ignore Faces 1 (igf1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Faces**.

2 In the **Settings** window for **Ignore Faces**, locate the **Input** section.

3 Click the **Paste Selection** button for **Faces to ignore**.

4 In the **Paste Selection** dialog, type `51` in the **Selection** text field.

5 Click **OK**.

6 In the **Geometry** toolbar, click **Build All**.