Turbulent Mixing in a Stirred Tank

Introduction

This tutorial demonstrates how mixing can be simulated in a stirred vessel by seeding a trace species from a point. The flow is modeled using the Rotating Machinery, Fluid Flow interface, which solves the Navier-Stokes equations on geometries with rotating parts such as impellers. The transport of the trace species is modeled using the Transport of Diluted Species interface. The trace species is added when the mixer is running in steady operation. The mixing time is evaluated by comparing the concentration in a measurement point to the average concentration in the vessel.

Model Definition

MODEL GEOMETRY

Figure 1 shows the model geometry, which is a schematic cross section of a tank with a four-blade impeller. The tank has four baffles attached to the wall to enhance mixing. The mixer blades and the impellers are approximated to be infinitely thin. The seeding of the trace species is done at the marked seeding point. The tank also contains a measurement point, close to the wall, where the concentration can be probed.

The circle between the impeller and the tank wall is the assembly boundary where the mesh is allowed to slide when the impeller rotates.

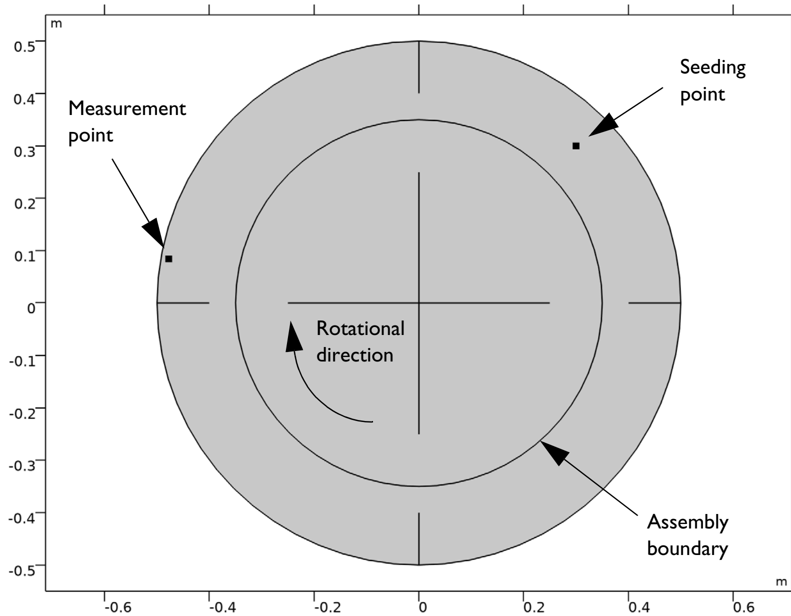

Figure 1: Model geometry.

DOMAIN EQUATION AND BOUNDARY CONDITIONS

The mixer fluid is water, and a rotational speed of 20 rpm is prescribed for the impeller. This rotation is achieved by prescribing the inner domain to be a rotating domain. The boundary between the inner and the outer domain is prescribed to be a continuity boundary that transfers momentum to the fluid in the outer domain.

The Reynolds number based on the impeller radius and the impeller tip speed is approximately 1.9×10^6, which means that the flow is turbulent. The k–ε turbulence model is applied in this example.

There are two methods to reach operating conditions. One is to accelerate the impeller up to full speed and wait for the flow to reach a quasi steady-state. This approach is simple but can be time consuming. A computationally more efficient method is to first simulate the flow using the frozen rotor approach. Frozen rotor means that the impeller, or rotor, is frozen in position. The flow in the rotating domain is assumed to be stationary in terms of a rotating coordinate system. The effect of the rotation is then accounted for by Coriolis and centrifugal forces. This solution couples to the nonrotating parts where the flow is also assumed to be stationary, but in a nonrotating coordinate system. See the *CFD Module User's Guide* for more information about frozen rotor.

The result of a frozen rotor simulation is an approximation to the flow at operating conditions. The result depends on the angular position of the impeller and cannot represent transient effects. It is still a very good starting condition to reach operating conditions. Here, to make sure that the correct operating conditions are reached, a time dependent simulations in ran starting from the frozen rotor solution. The average flow quantities are inspected in order to determine when a quasi-steady state is reached.

A trace species is a species introduced in very small quantities. It is often of a sharp color to be clearly visible even in small amounts. A trace species is not supposed to affect the flow, and hence, the flow can be solved for first and then the trace species transport solved for subsequently. The seeding is performed by injection at the seeding point during one second. Then the mixing in the vessel is simulated for 40 revolutions. The degree of mixing is evaluated by comparing the averaged concentration to the concentration measured near the wall behind one of the baffles.

To reduce the computational effort, the fluid flow field is not solved for during the mixing. The flow field and turbulent diffusion during the last revolution of the flow simulation is reused repeatedly during the mixing stage.

Note that in order to be able to reuse the flow field in a periodic fashion, variables are re-defined in the Equation View of the multiphysics coupling feature (Reacting Flow, Diluted Species).

Results and Discussion

Figure 2 shows the frozen rotor velocity field. As expected, the highest velocity magnitude is found at the tip of the mixer blades. There are also clearly visible recirculation zones both before and behind the baffles.

Figure 3 shows the velocity field at the end of the fluid flow simulation (at t = 30 s, corresponding to 10 revolutions). The rotor position is the same as in Figure 2, and the results in the figures are similar. There are, however, differences. The most notably difference is the recirculation zones before the baffles that are smaller in Figure 3 than in Figure 2. The size and shape of the recirculation zones for the time-dependent simulation also vary with the position of the impeller.

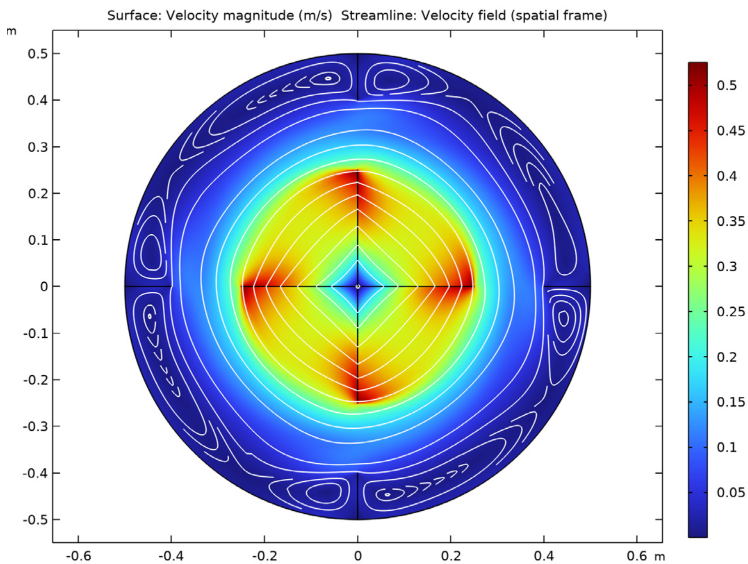

Figure 2: The velocity field obtained from the frozen rotor simulation.

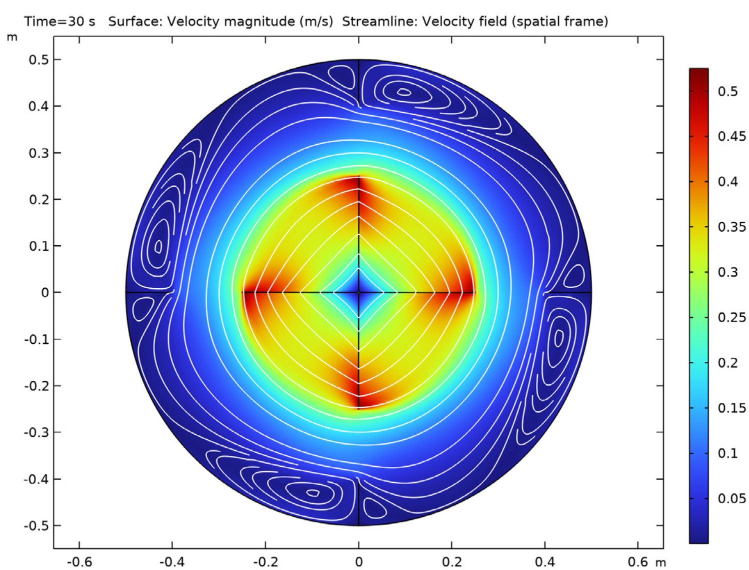

Figure 3: A snapshot of the time-dependent velocity field at t=30 s.

The average velocity and the average turbulent dynamic viscosity are plotted in Figure 4 to visualize the flow field development over time. The variables are normalized using the results from the frozen rotor solution. It can be noted that there is a drift over the first couple of rotations, but the magnitude of the variations are small. The frequency with which the impeller blades are passing the baffles is clearly seen in the velocity variations. The flow is noted to reached a cyclic steady state after around seven revolutions. This implies that the last revolution of the simulation can confidently be used repeatedly for the mixing simulation.

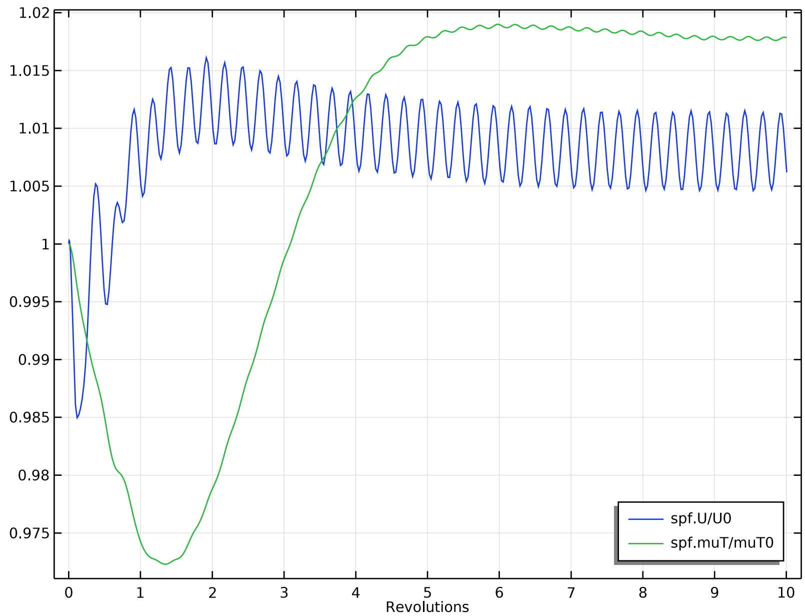

Figure 4: Flow development starting from the frozen rotor solution. The average velocity and average turbulent diffusion normalized by the respective initial value are shown.

Figure 5 shows four snapshots of the mixing process, with the time running from the upper-left to the lower-right picture. Since the velocity is rather slow at the seeding point (see also Figure 3), the initial transport is almost isotropic around the seeding point (t = 2 s). Some trace species is, however, entrained in the faster velocity field in the center of the mixer and becomes thereby spread in the azimuthal direction (t = 8 s). The slowest spreading is to the regions behind the left baffle, close to the measuring point, as well as behind the top baffle. The mixing in also poor in between the impeller blades. This is a well-known phenomenon and is the reason to why chemical substances are commonly added as close to the impeller axis as possible.

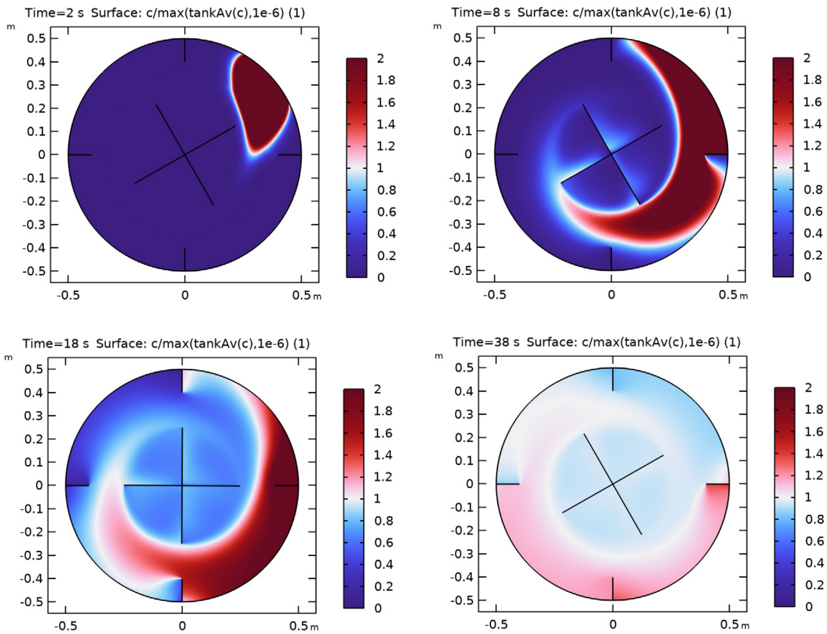

Figure 5: The mixing process. Surface plots of the trace species, normalized by the average concentration, at t = 2, 8, 18 and 38 seconds.

The mixing development can be evaluated in Figure 5 where the concentration at the measurement position and the average concentration are plotted. As was noted in the previous figure it takes a number of rotations of the impeller before the species is spread to the measurement position. After about 14 rotations the concentration behind the baffle surpasses the averaged concentrations, and after around 30 rotations the average concentration has been reached.

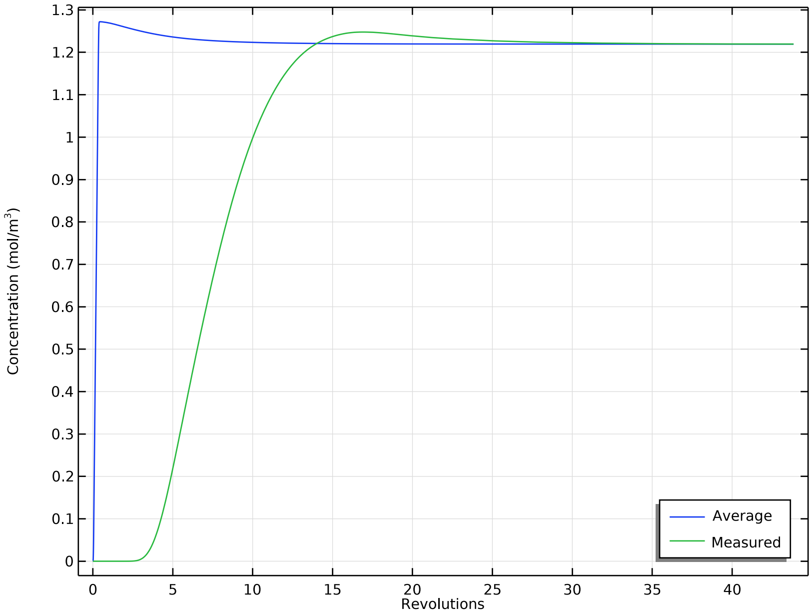

Figure 6: Concentration development during mixing. Comparing the average concentration in the tank to the one in the measurement point.

Application Library path: `CFD_Module/Single-Phase_Flow/turbulent_mixing`

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 **2D**.

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**.

GLOBAL DEFINITIONS

Import some model parameters from a text file.

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 `turbulent_mixing_parameters.txt`.

GEOMETRY 1

Circle 1 (c1)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `tank_R`.

Circle 2 (c2)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.7*tank_R`.

4 Click **Build Selected**.

Difference 1 (dif1)

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

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

3 Select the **Keep objects to subtract** check box.

4 Select the object **c1** only.

5 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

6 Select the object **c2** only.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **x** text field, type `-tank_R`.

6 Locate the **Endpoint** section. In the **x** text field, type `-tank_R+baffle_L`.

Rotate 1 (rot1)

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

2 Select the object **ls1** only.

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

4 Click **Range**.

5 In the **Range** dialog box, type `90` in the **Start** text field.

6 In the **Step** text field, type `90`.

7 In the **Stop** text field, type `270`.

8 Click **Replace**.

9 In the **Settings** window for **Rotate**, locate the **Input** section.

10 Select the **Keep input objects** check box.

Point 1 (pt1)

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

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

3 In the **x** text field, type `inject_x`.

4 In the **y** text field, type `inject_y`.

5 Click **Build Selected**.

Point 2 (pt2)

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

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

3 In the **x** text field, type `-tank_R*0.97`.

4 In the **y** text field, type `0`.

5 Click **Build Selected**.

Rotate 2 (rot2)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.
2 Select the object **pt2** only.
3 In the **Settings** window for **Rotate**, locate the **Rotation** section.
4 In the **Angle** text field, type `-10`.
5 Click **Build Selected**.

Union 1 (uni1)

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

   The final operation will later be set to **Form an assembly**. Union operations are therefore necessary to merge the domains and the lines.
2 Select the objects **dif1**, **ls1**, **pt1**, **rot1(1)**, **rot1(2)**, **rot1(3)**, and **rot2** only.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.
2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.
3 From the **Specify** list, choose **Coordinates**.
4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.
5 Locate the **Starting Point** section. In the **x** text field, type `-0.25`.
6 Locate the **Endpoint** section. In the **x** text field, type `0.25`.

Line Segment 3 (ls3)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.
2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.
3 From the **Specify** list, choose **Coordinates**.
4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.
5 Locate the **Starting Point** section. In the **y** text field, type `-0.25`.
6 Locate the **Endpoint** section. In the **y** text field, type `0.25`.

Union 2 (uni2)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.
2 Select the objects **c2**, **ls2**, and **ls3** only.

Form Union (fin)

The boundary between the rotating and nonrotating domain must be an assembly boundary so that the parts can move relative to each other.

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.
2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.
3 From the **Action** list, choose **Form an assembly**.
4 In the **Geometry** toolbar, click **Build All**.
5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Create a selection for the interior boundaries representing the impeller and baffles.

DEFINITIONS

Impeller and Baffles

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 **Boundary**.
4 Select Boundaries 1–4 and 13–16 only.
5 In the **Label** text field, type `Impeller and Baffles`.

Create an average operator to be used for computing average quantities in the tank.

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.
2 In the **Settings** window for **Average**, type `tankAv` in the **Operator name** text field.
3 Locate the **Source Selection** section. From the **Selection** list, choose **All domains**.

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.

MOVING MESH

Rotating Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Moving Mesh** click **Rotating Domain 1**.
2 Select Domain 2 only.

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

4 In the *f* text field, type `-rpm`.

TURBULENT FLOW, K-ε (SPF)

Interior Wall 1

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

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

3 From the **Selection** list, choose **Impeller and Baffles**.

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

2 Select Point 10 only.

The pressure level must be specified to obtain a unique solution since water is an incompressible liquid.

Hide the internal boundaries between the moving and the stationary part of the geometry. First copy the boundary selection from the continuity feature.

Flow Continuity 1

1 In the **Model Builder** window, click **Flow Continuity 1**.

2 In the **Settings** window for **Flow Continuity**, click to expand the **Boundary Selection** section.

3 Click **Copy Selection**.

DEFINITIONS

Hide for Physics 1

1 In the **Model Builder** window, right-click **View 1** and choose **Hide for Physics**.

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, Press CTRL+V to paste the copied boundary selection.

6 click **OK**.

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 **Fine**.

Size

Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size 1

1 In the **Model Builder** window, right-click **Free Triangular 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 **Point**.

4 Select Point 11 only.

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

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `3e-3`.

8 Click **Build All**.

STUDY 1

Step 1: Frozen Rotor

In the **Home** toolbar, click **Compute**.

RESULTS

Velocity (spf)

Create Figure 2 using the following steps.

Streamline 1

1 Right-click **Velocity (spf)** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 From the **Positioning** list, choose **Uniform density**.

4 In the **Separating distance** text field, type `0.02`.

5 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **White**.

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

Add a **Time Dependent** study.

Pressure (spf), Velocity (spf), Wall Resolution (spf)

1. In the **Model Builder** window, under **Results**, Ctrl-click to select **Velocity (spf)**, **Pressure (spf)**, and **Wall Resolution (spf)**.
2. Right-click and choose **Group**.

Fluid Flow, Frozen Rotor

In the **Settings** window for **Group**, type `Fluid Flow, Frozen Rotor` in the **Label** text field.

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

DEFINITIONS

Variables 1

1. In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
2. In the **Settings** window for **Variables**, locate the **Variables** section.
3. In the table, enter the following settings:

Name	Expression	Unit	Description
U0	withsol('sol1', tankAv(spf.U))	m/s	Average velocity, frozen rotor
muT0	withsol('sol1', tankAv(spf.muT))	Pa·s	Average turbulent dynamic viscosity, frozen rotor

Domain Probe: U

1. In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.
2. In the **Settings** window for **Domain Probe**, type `Domain Probe: U` in the **Label** text field.
3. Locate the **Expression** section. In the **Expression** text field, type `spf.U/U0`.

Domain Probe: muT

1. Right-click **Domain Probe: U** and choose **Duplicate**.

2 In the **Settings** window for **Domain Probe**, type `Domain Probe: muT` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `spf.muT/muT0`.

STUDY 2

In the **Study** toolbar, click **Show Default Plots**.

Step 1: Time Dependent

Store time steps at the end of the simulation when the flow field is expected to have reached a quasi-steady state.

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

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

3 In the **Output times** text field, type `0 range(frev_t0,rev_dt,frev_tEnd)`.

Enable results while solving, and use the probes to monitor when a quasi-steady state is reached.

4 Click to expand the **Results While Solving** section. Select the **Plot** check box.

5 From the **Plot group** list, choose **Velocity (spf) 1**.

6 From the **Update at** list, choose **Time steps taken by solver**.

7 From the **Probes** list, choose **Manual**.

Start from the frozen rotor solution.

8 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

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

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

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

RESULTS

Pressure (spf) 1, Probe Plot Group 7, Velocity (spf) 1, Wall Resolution (spf) 1

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Velocity (spf) 1**, **Pressure (spf) 1**, **Wall Resolution (spf) 1**, and **Probe Plot Group 7**.

2 Right-click and choose **Group**.

Fluid Flow, Time Dependent

In the **Settings** window for **Group**, type `Fluid Flow, Time Dependent` in the **Label** text field.

Probe Plot Group 7

1. In the **Model Builder** window, click **Probe Plot Group 7**.
2. In the **Settings** window for **1D Plot Group**, locate the **Legend** section.
3. From the **Position** list, choose **Lower right**.

Probe Table Graph 1

1. In the **Model Builder** window, expand the **Probe Plot Group 7** node, then click **Probe Table Graph 1**.
2. In the **Settings** window for **Table Graph**, click to expand the **Preprocessing** section.
3. Find the **x-axis column** subsection. From the **Preprocessing** list, choose **Linear**.
4. In the **Scaling** text field, type `rpm`.

Probe Plot: Fluid flow development

1. In the **Model Builder** window, under **Results>Fluid Flow, Time Dependent** click **Probe Plot Group 7**.
2. In the **Settings** window for **1D Plot Group**, type `Probe Plot: Fluid flow development` in the **Label** text field.
3. Locate the **Plot Settings** section. Select the **x-axis label** check box.
4. In the associated text field, type `Revolutions`.

Probe Table Graph 1

1. In the **Model Builder** window, expand the **Probe Plot: Fluid flow development** node, then click **Probe Table Graph 1**.
2. In the **Settings** window for **Table Graph**, click to expand the **Legends** section.
3. From the **Legends** list, choose **Manual**.
4. In the table, enter the following settings:

Legends
`spf.U/U0`
`spf.muT/muT0`

5. In the **Probe Plot: Fluid flow development** toolbar, click **Plot**.

Velocity (spf) 1

Recreate Figure 3 using the following steps.

Streamline 1

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

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 From the **Positioning** list, choose **Uniform density**.

4 In the **Separating distance** text field, type `0.02`.

5 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **White**.

6 In the **Velocity (spf) 1** toolbar, click **Plot**.

DEFINITIONS

Rectangle 1 (rect1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Local>Rectangle**.

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

3 In the **Lower limit** text field, type `0.1`.

4 In the **Upper limit** text field, type `1.1`.

5 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `0.2`.

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 **Chemical Species Transport>Transport of Diluted Species (tds)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Study 1** and **Study 2**.

5 Click **Add to Component 1** in the window toolbar.

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

MULTIPHYSICS

Reacting Flow, Diluted Species 1 (rfd1)

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

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 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Turbulent Flow, k-ε (spf)**.

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

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

TRANSPORT OF DILUTED SPECIES (TDS)

In the **Model Builder** window, under **Component 1 (comp1)** click **Transport of Diluted Species (tds)**.

Thin Impermeable Barrier 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Thin Impermeable Barrier**.

2 In the **Settings** window for **Thin Impermeable Barrier**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Impeller and Baffles**.

Line Mass Source 1

1 In the **Physics** toolbar, click **Points** and choose **Line Mass Source**.

2 Select Point 11 only.

3 In the **Species Source** text field, type `rect1(t[1/s])`.

Define cyclic variables to repeatedly re-use the last rotation computed in Study 2. Here the withsol operator is used to evaluate the variables from the previous solution. During evaluation, the setval operator specifies the time using a modulo operator taking the current time and the single revolution time as input.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Variables 1**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`ucyc`	`withsol('sol2',u, setval(t,frev_tEnd-rev_t+mod(t,rev_t)))`	m/s	`Cyclic fluid velocity, x-component`
`vcyc`	`withsol('sol2',v, setval(t,frev_tEnd-rev_t+mod(t,rev_t)))`	m/s	`Cyclic fluid velocity, y-component`
`nuTcyc`	`withsol('sol2',spf.nuT, setval(t,frev_tEnd-rev_t+mod(t,rev_t)))`	m^2/s	`Cyclic turbulent viscosity`

4 Click the **Show More Options** button in the **Model Builder** toolbar.

5 In the **Show More Options** dialog box, select **Physics>Equation View** in the tree.

6 In the tree, select the check box for the node **Physics>Equation View**.

7 Click **OK**.

MULTIPHYSICS

Reacting Flow, Diluted Species 1 (rfd1)

The multiphysics coupling node supplies the velocity and turbulent diffusion used in the species transport equations. Re-define these in the **Equation View** node to apply the cyclic variables defined above. In this manner the quasi-steady flow solution can be re-used during the entire mixing simulation.

1 In the **Model Builder** window, expand the **Reacting Flow, Diluted Species 1 (rfd1)** node, then click **Equation View**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description	Selection	Details
tds.DiT_c	`nuTcyc/ rfd1.ScT_c`	m^2/s	Turbulent diffusivity	Domains 1ñ2	
rfd1.ux	`ucyc`	m/s	Velocity field, x component	Domains 1ñ2	
rfd1.uy	`vcyc`	m/s	Velocity field, y component	Domains 1ñ2	

DEFINITIONS

Domain Probe : c_av

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Probe**.

2 In the **Settings** window for **Domain Probe**, type `Domain Probe : c_av` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `c`.

4 Click to expand the **Table and Window Settings** section. From the **Output table** list, choose **New table**.

5 Click **Add Table**.

6 From the **Plot window** list, choose **New window**.

7 Click **Add Plot Window**.

Point Probe : c

1 In the **Definitions** toolbar, click **Probes** and choose **Point Probe**.

2 In the **Settings** window for **Point Probe**, type `Point Probe : c` in the **Label** text field.

3 Locate the **Probe Type** section. From the **Type** list, choose **Integral**.

4 Locate the **Source Selection** section. Click **Clear Selection**.

5 Select Point 2 only.

6 Locate the **Expression** section. In the **Expression** text field, type `c`.

7 Click to expand the **Table and Window Settings** section. From the **Output table** list, choose **Table 2**.

8 From the **Plot window** list, choose **Probe Plot 2**.

9 Click **Go to Source**.

RESULTS

Probe Table 2

1 In the **Model Builder** window, under **Results>Tables** click **Table 2**.

2 In the **Settings** window for **Table**, type `Probe Table 2` in the **Label** text field.

STUDY 3

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 3** click **Step 1: Time Dependent**.

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

3 In the **Output times** text field, type `range(0,rev_t/3,rev_t*mix_revs)`.

4 Click to expand the **Results While Solving** section. Select the **Plot** check box.

5 From the **Update at** list, choose **Time steps taken by solver**.

6 From the **Probes** list, choose **Manual**.

7 In the **Probes** list, choose **Domain Probe: U (dom1)** and **Domain Probe: muT (dom2)**.

8 Under **Probes**, click **Delete**.

9 Locate 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 2, Time Dependent**.

12 In the **Study** toolbar, click **Show Default Plots**.

Plot the dataset edges on the spatial frame to make them follow the rotation.

RESULTS

Concentration (tds)

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

2 From the **Frame** list, choose **Spatial (x, y, z)**.

STUDY 3

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 3** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Results While Solving** section.

3 From the **Plot group** list, choose **Concentration (tds)**.

4 From the **Update at** list, choose **Time steps taken by solver**.

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

RESULTS

Concentration (tds), Probe Plot Group 9

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Concentration (tds)** and **Probe Plot Group 9**.

2 Right-click and choose **Group**.

Species Mixing, Time Dependent

In the **Settings** window for **Group**, type Species Mixing, Time Dependent in the **Label** text field.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Probe Plot Group 9** node, then click **Probe Table Graph 1**.

2 In the **Settings** window for **Table Graph**, click to expand the **Preprocessing** section.

3 Find the **x-axis column** subsection. From the **Preprocessing** list, choose **Linear**.

4 In the **Scaling** text field, type rpm.

Probe Plot: Mixing

1 In the **Model Builder** window, under **Results>Species Mixing, Time Dependent** click **Probe Plot Group 9**.

2 In the **Settings** window for **1D Plot Group**, type Probe Plot: Mixing in the **Label** text field.

3 Locate the **Plot Settings** section. Select the **x-axis label** check box.

4 In the associated text field, type Revolutions .

5 Select the **y-axis label** check box.

6 In the associated text field, type Concentration (mol/m³).

7 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

8 In the **Probe Plot: Mixing** toolbar, click **Plot**.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Probe Plot: Mixing** node, then click **Probe Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Legends** section.

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

4 In the table, enter the following settings:

Legends
Average
Measured

5 In the **Probe Plot: Mixing** toolbar, click **Plot**.

Concentration (tds)

The following steps create an animation that contains the plots in Figure 5.

Surface 1

1 In the **Model Builder** window, expand the **Concentration (tds)** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `c/max(tankAv(c),1e-6)`.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

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

6 In the **Maximum** text field, type `2`.

7 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

Streamline 1

In the **Model Builder** window, right-click **Streamline 1** and choose **Disable**.

Animation 1

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

2 In the **Settings** window for **Animation**, locate the **Target** section.

3 From the **Target** list, choose **Player**.

4 Locate the **Scene** section. From the **Subject** list, choose **Concentration (tds)**.

   Set the frames to be displayed for as long as the time between the saved solutions.

5 Locate the **Playing** section. In the **Display each frame for** text field, type `0.15`.

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

7 Click the **Play** button in the **Graphics** toolbar.