Model created in COMSOL Multiphysics 6.4

Nonisothermal Flow in a 2D Mixer

Introduction

This model demonstrates how to obtain the temperature distribution in a simplified tabletop lab mixer using the Rotating Machinery, Nonisothermal Flow interface in the Mixer Module. The key instructive element is to demonstrate the Frozen Rotor method, which substantially speeds up the computation time for a mixing study.

General Description

The model geometry is shown in Figure 1. It represents a cross-section view of a table-top lab mixer. Small mixers are not always baffled. Instead, one or several rods are inserted through the mixer lid and function as baffles. The rods typically contain equipment to measure, for example, temperature or pH level. In this model, the mixer is baffled by a simplified immersion heater, which has a constant surface temperature of 60°C.

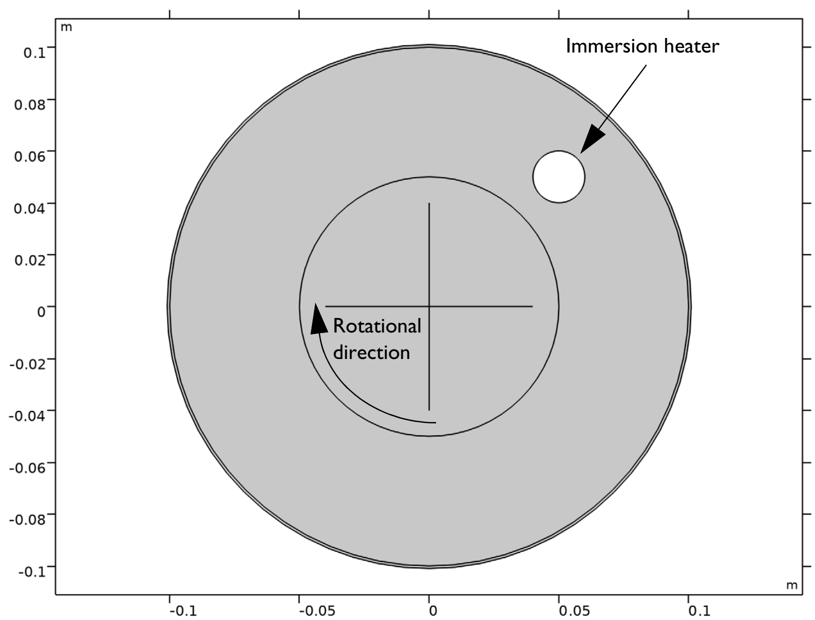

Figure 1: Model geometry.

The mixer tank is filled with water which is agitated by the impeller rotating clockwise (as indicated in Figure 1) with 20 rotations per minute (rpm). The thickness of the impeller blades is significantly smaller than the diameter of the tank, and the impeller blades are therefore modeled as infinitely thin.

The tank is made of steel and is subjected to cooling by natural convection that takes place on the outside of the mixer. The surrounding conditions are standard conditions (temperature equal to 20°C and pressure equal to 1 atmosphere) and the reactor is 0.2 m high. These conditions are needed as input for the natural convection correlations used to calculate the heat transfer coefficient from the tank wall to the surrounding.

Model Setup

The Reynolds number for a mixer is commonly calculated as

$$\mathrm{Re} = \frac{ND_{\mathrm{a}}^2}{\nu} \tag{1}$$

where N is the number of rotations per second, D_{a} the impeller diameter and ν the kinematic viscosity. A high Reynolds number means that the flow has a tendency to become turbulent. Evaluating Equation 1 using ν at 60°C gives Re = 6944.This Reynolds number would warrant the use of a turbulence model, if the model was three-dimensional. But the restriction to two dimensions means that a laminar flow model works at higher Reynolds number, and this model does not employ any turbulence model. A possible extension of the model is to apply a turbulence model and recalculate the result to investigate the effect of possible turbulent structures in the flow.

The objective of this model is to obtain the temperature distribution at operating conditions. One way to get there would be to start from zero velocity and a homogeneous temperature distribution and simulate the startup of the mixer. This approach is simple, but requires a relatively long computation time.

A computationally more efficient method is to first simulate the flow using the frozen rotor approach. The Frozen rotor approach is a modeling concept that treats the rotor as fixed, or frozen in space. 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. The flow in the nonrotating parts is also assumed to be stationary, but in a nonrotating coordinate system. See the section *Frozen Rotor* in the *CFD Module User's Guide* for more information. 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. However, it is still a very good starting condition to reach operating conditions.

The frozen rotor result is used as input in a time-dependent simulation and the progress toward the operating conditions is monitored using probe plots.

Results and Discussion

Figure 2 shows the velocity distribution obtained from the frozen-rotor simulation. As expected, the highest velocity magnitude is found at the tip of the mixer blades. There are three recirculation zones: One downstream of the immersion heater, one along the top wall and one along the bottom wall.

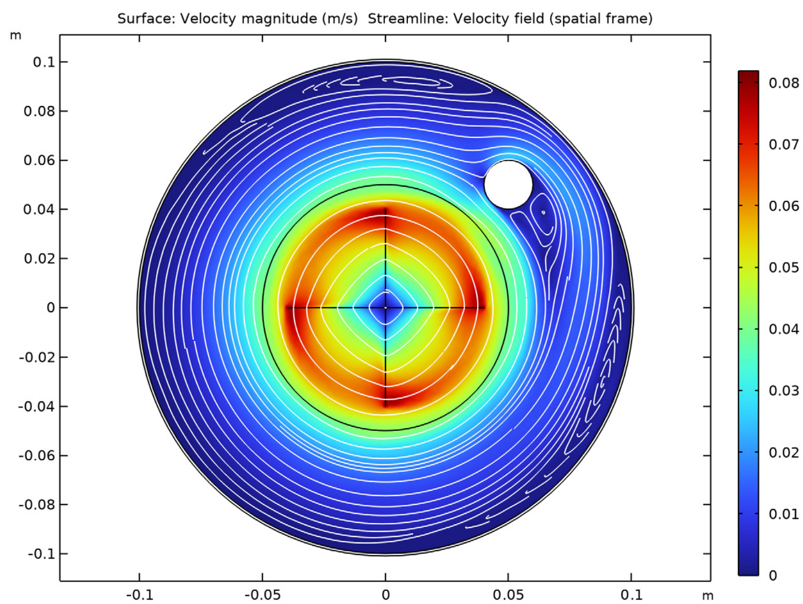

Figure 2: Velocity field obtained from the frozen rotor simulation.

Figure 3 shows the temperature distribution obtained from the frozen-rotor simulation. Streamlines are also included to visualize the flow field. The temperature is relatively homogeneous throughout the mixer. There are some cold spots in connection to the recirculation zones. This is expected since the fluid there has a longer residence time close to the solid wall, and therefore has less contact with the heated fluid closer to the center of the mixer.

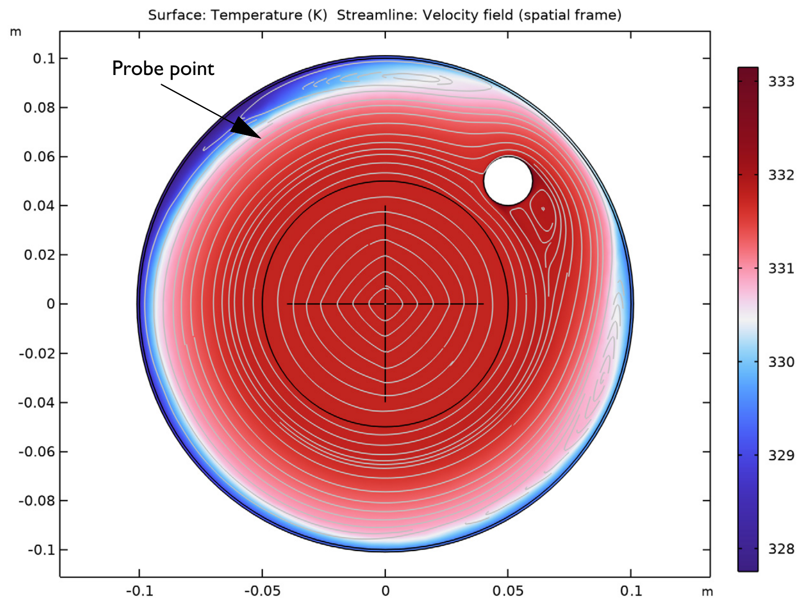

Figure 3: Temperature distribution obtained from the frozen rotor simulation.

To monitor the progress toward steady state, the velocity magnitude and temperature are probed at $(x,y) = (-0.05,0.065)$. The location is indicated in Figure 3. As can be seen, it is located just outside the recirculation zone along the top wall.

The probe plots produced during the time dependent simulation are shown in Figure 4. The velocity probe plot shows that the flow pattern, after an initial transient, oscillates around the frozen-rotor result with an amplitude of about 10%. The deviations in temperature are much smaller.

The velocity probe plot exhibits quasiperiodic structure. Discrete Fourier analysis reproduces a peak at a frequency 1.33 Hz which corresponds to the passing of the blades. Several other frequencies can be identified, the most pronounced peaks are at 0.125 Hz, 0.25 Hz, 0.42 Hz and 1.07 Hz. Evaluating the Strouhal frequency for the immersion heater (for typical value of the Strouhal number $\mathrm{St} = 0.2$) gives a value of about 1.48 Hz. Hence, oscillations in the mixer, which include the intermittent boundary-layer separation, can not be completely explained as being driven by the frequency of the rotating blades and the main frequency of the Kármán vortex street behind the heater. Probably, some more intricate mechanisms are involved. The temperature variations are within the tolerance set in the default solver sequence.

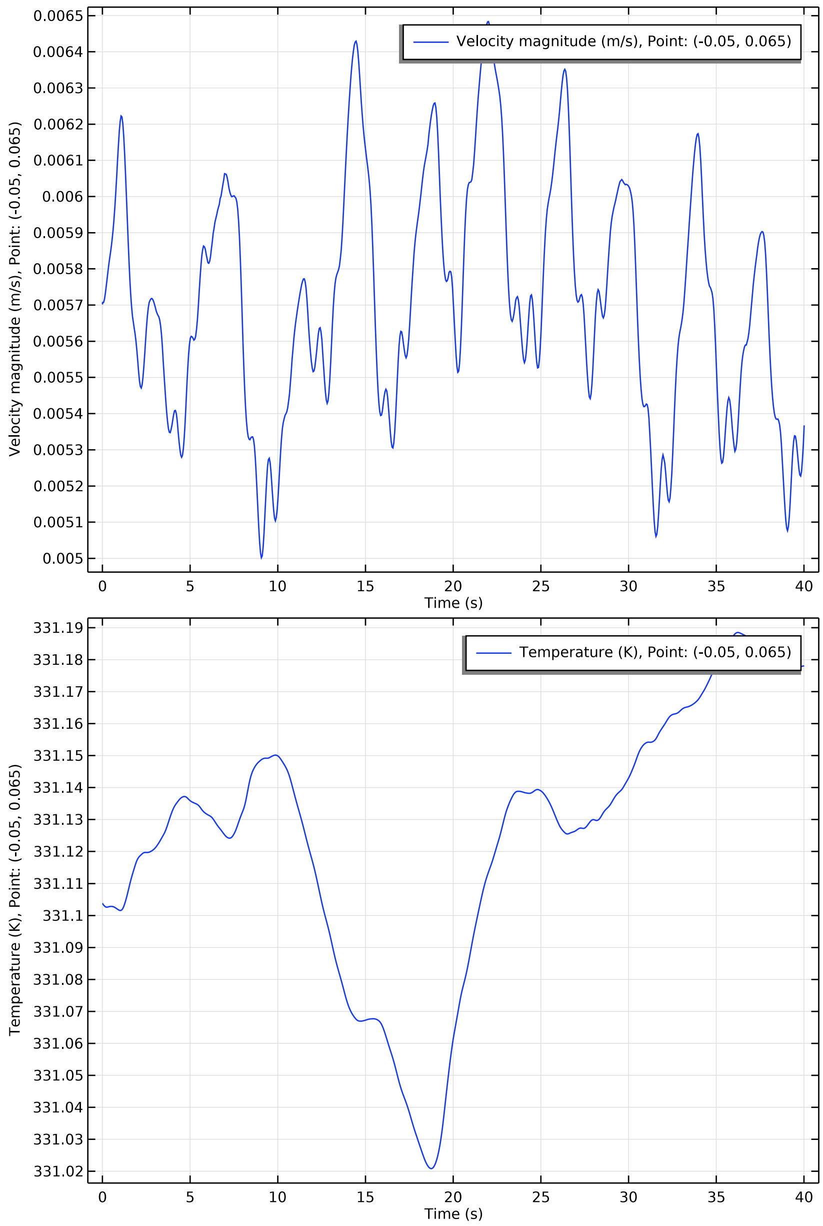

Figure 4: Probe plots of velocity and temperature.

A more complete picture of the progress from the frozen rotor solution toward operating conditions can be obtained through an animation. Figure 5 shows four snapshots from such an animation. The time runs from upper left to lower right. The most notable changes occur in the recirculation zones. The recirculation zone behind the immersion heater shows a periodic shedding pattern typical for flow around a cylinder at moderate Reynolds numbers.

Looking at Figure 3, it can be seen that the recirculation zone along the top wall contains a single, large vortex. As the simulation progresses (from $t = 20$ s to $t = 40$ s), the size and strength of the vortices varies as a result of the interaction between the disturbance, produced by the immersion heater, and the outer wall.

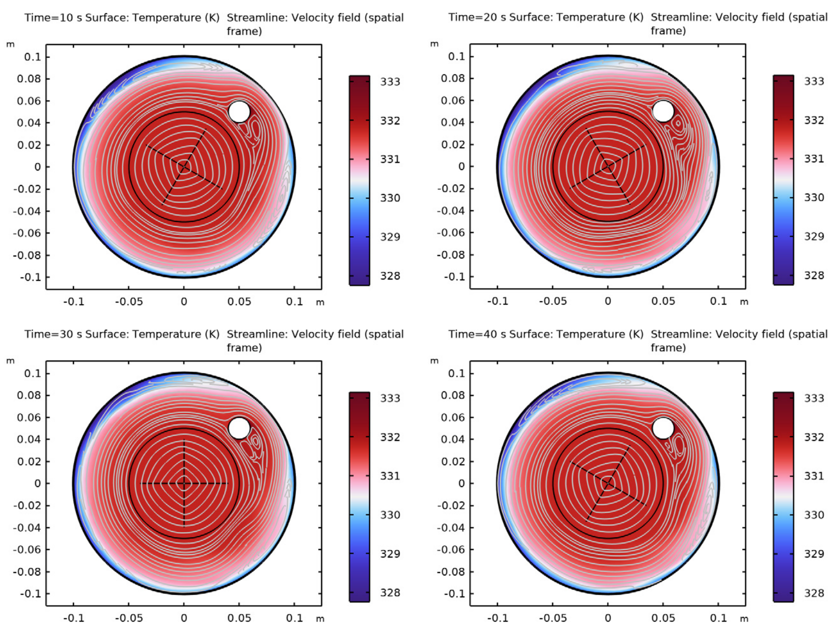

Figure 5: Evolution of the temperature field from frozen rotor solution toward operating conditions.

The results obtained in this model are typical for rotating machinery models: The frozen rotor approach can at a minimal computation effort deliver a decent approximation of the flow and temperature fields. But transient effects can only be captured by a time dependent simulation and these effects can change local temperature and velocity values significantly.

Application Library path: `Mixer_Module/Tutorials/nonisothermal_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 **2D**.

2 In the **Select Physics** tree, select **Fluid Flow > Nonisothermal Flow > Rotating Machinery, Nonisothermal Flow > Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces > Frozen Rotor**.

6 Click **Done**.

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 `0.1`.

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 `0.101`.

Union 1 (uni1)

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

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

Circle 3 (c3)

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 `0.01`.

4 Locate the **Position** section. In the **x** text field, type `0.05`.

5 In the **y** text field, type `0.05`.

Difference 1 (dif1)

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

2 Select the object **uni1** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **c3** only.

Circle 4 (c4)

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 `0.05`.

Difference 2 (dif2)

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

2 Select the object **dif1** 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 **c4** only.

6 Select the **Keep objects to subtract** checkbox.

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 **y** text field, type `-0.04`.

6 Locate the **Endpoint** section. In the **y** text field, type `0.04`.

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 `-0.04`.

6 Locate the **Endpoint** section. In the **x** text field, type `0.04`.

Union 2 (uni2)

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

2 Select the objects **c4**, **ls1**, and **ls2** 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.

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 tree, select **Built-in** > **Steel AISI 4340**.

6 Click the **Add to Component** button in the window toolbar.

7 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Steel AISI 4340 (mat2)

Select Domain 1 only.

MOVING MESH

Rotating Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Moving Mesh** click **Rotating Domain 1**.
2 In the **Settings** window for **Rotating Domain**, locate the **Domain Selection** section.
3 In the list, choose **1** and **2**.
4 Click **Remove from Selection**.
5 Select Domain 3 only.
6 Locate the **Rotation** section. In the *f* text field, type `-20[rpm]`.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
2 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.
3 In the list box, select **1**.
4 Click **Remove from Selection**.
5 Select Domains 2 and 3 only.

Interior Wall 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Interior Wall**.
2 Select Boundaries 17–20 only.

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.
2 Select Point 5 only.

HEAT TRANSFER IN FLUIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.

Solid 1

1 In the **Physics** toolbar, click **Domains** and choose **Solid**.
2 Select Domain 1 only.

Temperature 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2 Select Boundaries 13–16 only.
3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the T_0 text field, type `60[degC]`.

Heat Flux 1

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

2 Select Boundaries 1, 2, 7, and 12 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 From the **Flux type** list, choose **Convective heat flux**.

5 From the **Heat transfer coefficient** list, choose **External natural convection**.

6 In the L text field, type `0.2`.

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**.

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

Size

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

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

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

Free Triangular 1

To avoid unnecessarily small elements in the mixer vessel wall, add a separate size node with reduced resolution in narrow regions.

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 **Domain**.

4 Select Domain 1 only.

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

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

7 Select the **Resolution of narrow regions** checkbox. In the associated text field, type `0.1`.

Free Triangular 1

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

Boundary Layers 1

Use a boundary layer mesh also in the solid domain to increase the resolution there.

1 In the **Model Builder** window, click **Boundary Layers 1**.
2 Click in the **Graphics** window and then press Ctrl+A to select all domains.
3 In the **Settings** window for **Boundary Layers**, click **Build Selected**.

STUDY 1

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

RESULTS

Velocity (spf)

Recreate 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 **Density level** text field, type `8.5`.
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**.
7 Click the **Zoom Extents** button in the **Graphics** toolbar.

Surface 1

Recreate Figure 3 using the following steps.

1 In the **Model Builder** window, expand the **Results** > **Temperature (ht)** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
3 From the **Color table** list, choose **Wave**.

Streamline 1

1 In the **Model Builder** window, right-click **Temperature (ht)** 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 **Density level** text field, type `8.5`.

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

6 In the **Temperature (ht)** toolbar, click **Plot**.

DEFINITIONS

Add a probe to follow the development of the flow during the time-dependent simulation.

Domain Point Probe 1

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

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

3 In row **Coordinates**, set **x** to `-0.05[m]`.

4 In row **Coordinates**, set **y** to `0.065[m]`.

The probe is located at the outer edge of recirculation zone that is positioned along the upper wall.

Point Probe Expression 2 (ppb2)

1 In the **Model Builder** window, expand the **Domain Point Probe 1** node.

2 Right-click **Component 1 (comp1)** > **Definitions** > **Domain Point Probe 1** > **Point Probe Expression 1 (ppb1)** and choose **Duplicate**.

3 In the **Settings** window for **Point Probe Expression**, locate the **Expression** section.

4 In the **Expression** text field, type `T`.

5 Click to expand the **Table and Window Settings** section. From the **Plot window** list, choose **New window**.

ADD STUDY

Add a time-dependent 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(0,0.5,40)`.
3 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**.
4 From the **Method** list, choose **Solution**.
5 From the **Study** list, choose **Study 1, Frozen Rotor**.

For the transient simulation, add a restriction on the time step. This will make sure that the impeller rotation at each time step is bounded, and that a high accuracy is maintained throughout the simulation. First generate the solver sequence.

Solution 2 (sol2)

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

Apply a maximum time step of 0.05 s. This is equivalent to an impeller rotation of 6 degrees.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.
3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
4 From the **Maximum step constraint** list, choose **Constant**.
5 In the **Maximum step** text field, type `0.05`.
6 In the **Study** toolbar, click **Compute**.

RESULTS

Two probe plots will automatically appear when you start the calculation.

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

Temperature (ht) 1

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

1 In the **Model Builder** window, under **Results** click **Temperature (ht) 1**.
2 In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.
3 From the **Frame** list, choose **Spatial (x, y, z)**.

Surface 1

1 In the **Model Builder** window, expand the **Temperature (ht) 1** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **Wave**.

Streamline 1

1 In the **Model Builder** window, right-click **Temperature (ht) 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 **Density level** text field, type `8.5`.

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

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 **Temperature (ht) 1**.

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

6 Locate the **Playing** section. In the **Display each frame for** text field, type `0.25`.

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