Laminar Flow in a Baffled Stirred Mixer

Introduction

This exercise exemplifies the use of the rotating machinery feature in the CFD Module. The Rotating Machinery user interface allows you to model moving rotating parts in, for example, stirred tanks, mixers, and pumps.

The Rotating Machinery user interface formulates the Navier-Stokes equations in a rotating coordinate system. Parts that are not rotated are expressed in the fixed material coordinate system. The rotating and fixed parts need to be coupled together by an identity pair, where a flux continuity boundary condition is applied.

You can use the rotating machinery predefined multiphysics coupling in cases where it is possible to divide the modeled device into rotationally invariant geometries. The desired operation can be, for example, to rotate an impeller in a baffled tank. This is exemplified in Figure 1, where the impeller rotates from position 1 to 2. The first step is to divide the geometry into two parts that are both rotationally invariant, as shown in Step 1a. The second step is to specify the parts to model using a rotating frame and the ones to model using a fixed frame (Step 1b). The predefined coupling then

©2013 COMSOL

automatically does the coordinate transformation and the joining of the fixed and moving parts (Step 2a).

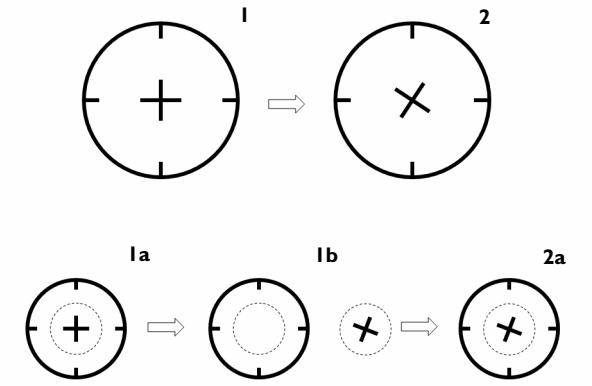


Figure 1: The modeling procedure in the rotating machinery user interface in the CFD Module.

Model Definition

The model you treat in this example is that of a baffled stirred mixer. Figure 2 shows the modeled geometry. The impeller rotates counterclockwise at a speed of 10 RPM, and the fluid in the mixer is water.

The model equations are the Navier-Stokes equations formulated in a rotating frame in the inner domain and in fixed coordinates in the outer one.

At the mixer's fixed walls, no-slip boundary conditions apply. The boundary condition on the rotating impeller will be set to rotate with the same velocity as the no-slip counterclockwise rotation conditions.

The implementation, applying the Rotating Machinery user interface, is quite straightforward. First you draw the geometry using two separate non-overlapping domains for the fixed and rotating parts. The next step is to form an assembly and create an identity pair, which makes it possible to treat the two domains as separate parts. You then specify which part uses a rotating frame. Once you have done this, you

 ©2013 COMSOL

can proceed to the usual steps of setting the fluid properties and the boundary conditions, and finally to meshing and solving the problem.

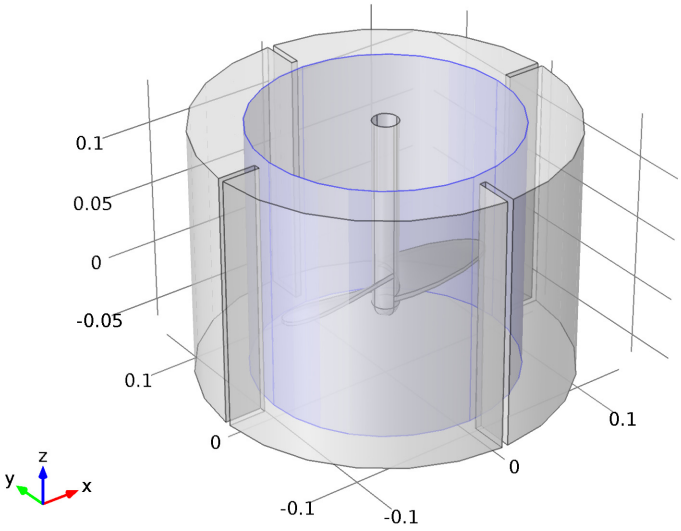


Figure 2: Geometry of the baffled stirred mixer. The inner domain is represented by a rotating (spatial) frame and the outer domain by a fixed (material) frame.

Results and Discussion

Figure 3 below shows the shear rate at the last time step, when the mixer has rotated with full speed for 4 seconds. The shear rate can be important for example when the mixture consists of living cells which are sensitive to too high shear rates.

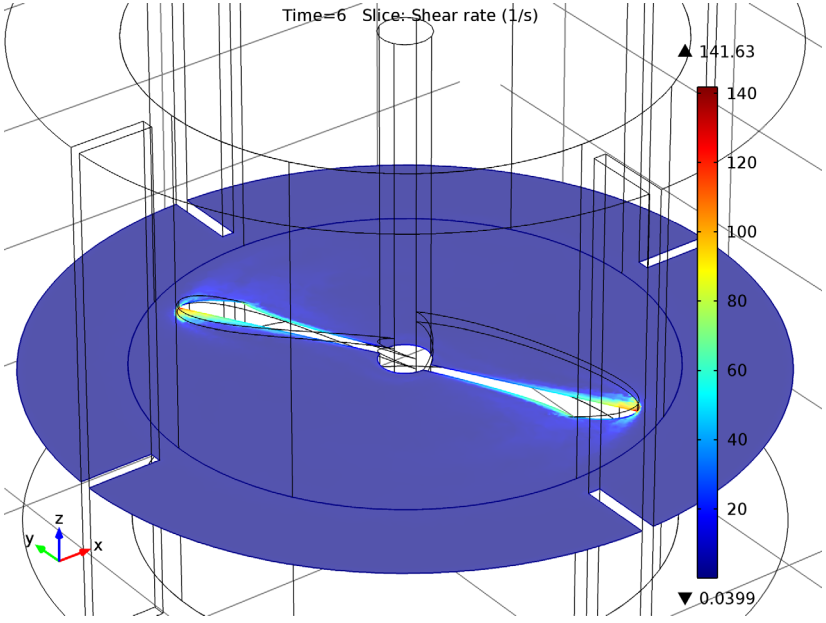


Figure 3: The shear rate after 6 seconds. The plot shows that the shear rate reaches its maximum value at the tip of the impeller blades.

In Figure 3, we can clearly see that the shear rate is highest where the impeller speed is highest, which is expected. If the shear rate is too high, consider to redesign the impeller with sharp leading and trailing edges.

©2013 COMSOL

Figure 4 shows the velocity field in the *yz*-plane at $t = 5.5$ s. It is clear that the impeller is creating a downward flow where the blades pass through the fluid.

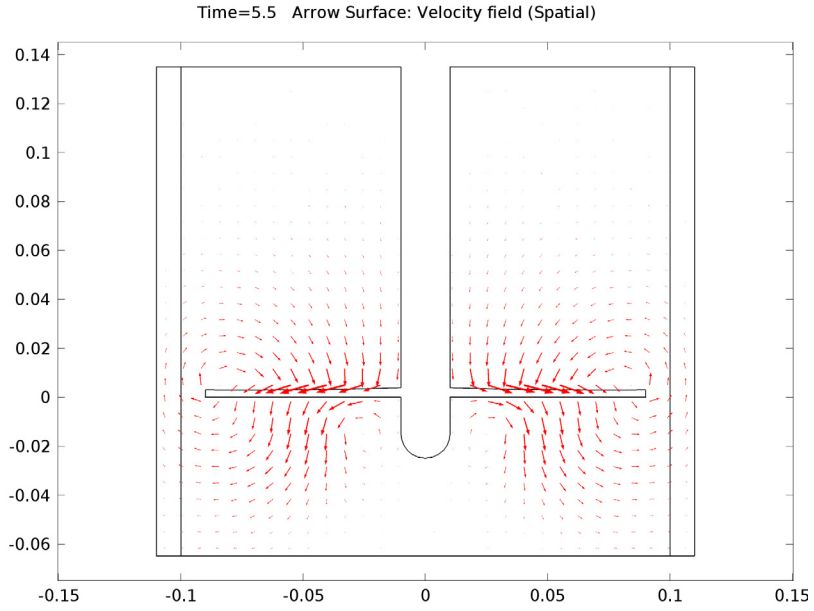


Figure 4: Velocity field in the yz-plane at t = 5.5 s.

Use the Player button on the main toolbar to create an animation.

Model Library path: `CFD_Module/Single-Phase_Tutorials/baffled_mixer`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Fluid Flow>Single-Phase Flow>Rotating Machinery, Fluid Flow>Rotating Machinery, Laminar Flow (rmspf)**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

6 Click **Finish**.

GEOMETRY 1

Import 1

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

2 In the **Import** settings window, locate the **Import** section.

3 Click the **Browse** button.

4 Browse to the model's Model Library folder and double-click the file `baffled_mixer.mphbin`.

5 Click the **Import** button.

You need to use assembly mode to prepare for the creation of the identity pair.

Form Union

1 In the **Model Builder** window, under **Model 1>Geometry 1** click **Form Union**.

2 In the **Finalize** settings window, locate the **Finalize** section.

3 From the **Finalization method** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

5 Click the **Build Selected** button.

DEFINITIONS

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Pairs>Identity Boundary Pair**.

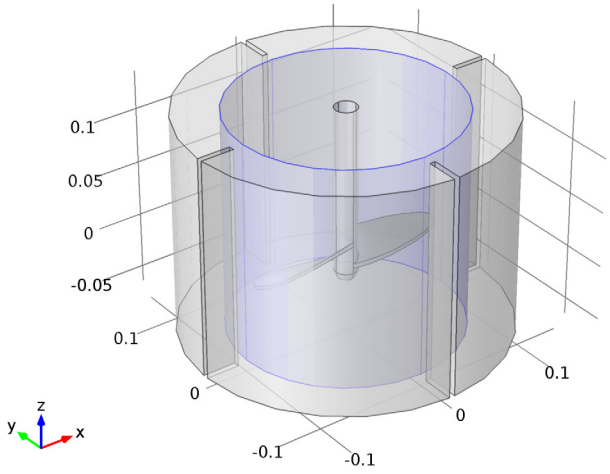
2 Click the **Transparency** button on the Graphics toolbar.

3 Select Boundaries 8, 9, 14, and 15 only.

You can do this by first copying the text '8, 9, 14, 15' and then clicking the **Paste Selection** button next to the **Selection** box or clicking in the box and pressing Ctrl+V.

 ©2013 COMSOL

4 In the **Pair** settings window, click **Activate Selection** in the upper-right corner of the **Destination Boundaries** section. Select Boundaries 23, 24, 37, and 48 only.



Create a **Step** function that you will use to increase the impeller rotation from zero to 10 RPM in a time of 2 seconds.

Step 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Functions>Step**.

2 In the **Step** settings window, locate the **Parameters** section.

3 In the **Location** edit field, type 1.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** edit field, type 2.

Variables 1

1 Right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Description
rpm	10[1/min]*step1(t[1/s])	Revolutions per minute

MATERIALS

Material Browser

1. In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.
2. In the **Material Browser** settings window, In the tree, select **Built-In>Water, liquid**.
3. Click **Add Material to Model**.

ROTATING MACHINERY, LAMINAR FLOW

Rotating Domain 1

1. In the **Model Builder** window, under **Model 1** right-click **Rotating Machinery, Laminar Flow** and choose **Rotating Domain**.
2. Select Domain 2 only.
3. In the **Rotating Domain** settings window, locate the **Rotating Domain** section.
4. In the **Revolutions per time** edit field, type `rpm`.

Next, add the **Flow Continuity** condition on the **Identity Pair** to couple the rotating and fixed domains together.

Flow Continuity 1

1. In the **Model Builder** window, right-click **Rotating Machinery, Laminar Flow** and choose **Pairs>Flow Continuity**.
2. In the **Flow Continuity** settings window, locate the **Pair Selection** section.
3. In the **Pairs** list, select **Identity Pair 1**.

All boundaries that are adjacent to the rotating domain will by default be set to rotating boundaries. The bottom boundary should not rotate, so add an additional no-slip wall boundary condition to enforce this condition.

Wall 2

1. Right-click **Rotating Machinery, Laminar Flow** and choose **Wall**.
2. Select Boundary 25 only.

Symmetry 1

1. Right-click **Rotating Machinery, Laminar Flow** and choose **Symmetry**.
2. Select Boundaries 4 and 26 only.

Finally, add a **Pressure Point Constraint** at one of the top boundaries.

 ©2013 COMSOL

Pressure Point Constraint 1

1 Right-click **Rotating Machinery, Laminar Flow** and choose **Points>Pressure Point Constraint**.

2 Select Point 4 only.

MESH 1

1 In the **Model Builder** window, under **Model 1** click **Mesh 1**.

2 In the **Mesh** settings window, locate the **Mesh Settings** section.

3 From the **Element size** list, choose **Fine**.

Size

1 Right-click **Model 1>Mesh 1** and choose **Edit Physics-Induced Sequence**.

2 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.

3 In the **Size** settings window, locate the **Element Size** section.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Maximum element growth rate** edit field, type `1.2`.

6 In the **Minimum element size** edit field, type `0.002`.

Size 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size 1**.

2 In the **Size** settings window, locate the **Element Size** section.

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

4 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

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

3 In the **Times** edit field, type `range(0,0.25,6)`.

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

Before solving, display the default solver settings to be able to set a maximum time step. The time step is limited by the mesh size and rotational speed with the identity pair in mind.

5 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

Solver 1

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1** node, then click **Time-Dependent Solver 1**.

2 In the **Time-Dependent Solver** settings window, click to expand the **Time Stepping** section.

3 Select the **Maximum step** check box.

4 In the associated edit field, type `0.1`.

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

RESULTS

The default plot groups visualize the velocity field in a slice plot and the pressure contours on the walls. Follow these steps to create a plot of the shear rate.

3D Plot Group 3

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

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

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

4 Right-click **Results>3D Plot Group 3** and choose **Slice**.

5 In the **Slice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Rotating Machinery, Laminar Flow>Shear rate (rmspf.sr)**.

6 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

7 From the **Entry method** list, choose **Coordinates**.

8 Click the **Plot** button.

9 Click the **Transparency** button on the Graphics toolbar.

10 Click the **Go to Default 3D View** button on the Graphics toolbar.

11 Click the **Zoom In** button on the Graphics toolbar.

Finally, create an arrow plot of the velocity field in a 2D cross section through the mixer's axis.

Data Sets

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

2D Plot Group 4

1 Right-click **Results** and choose **2D Plot Group**.

 ©2013 COMSOL

2 In the **2D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Cut Plane 1**.

4 From the **Time** list, choose **5.5**.

5 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

6 Right-click **Results>2D Plot Group 4** and choose **Arrow Surface**.

7 In the **Arrow Surface** settings window, locate the **Expression** section.

8 In the **x component** edit field, type `v`.

9 In the **y component** edit field, type `w`.

10 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** edit field, type `30`.

11 Find the **y grid points** subsection. In the **Points** edit field, type `30`.

12 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Proportional**.

13 Select the **Scale factor** check box.

14 In the associated edit field, type `0.4`.

©2013 COMSOL