Modeling of rotating parts

Introduction

This exercise exemplifies the use of the rotating machinery feature in the Chemical Engineering Module. The rotating machinery application mode makes it possible to model moving rotating parts in for example stirred tanks, mixers and pumps.

The application mode takes care of the formulation of the momentum and mass balances in a rotating coordinate system. Parts that are not rotated are expressed in the fixed coordinate system. The application mode then takes care of the necessary administration to join the parts of the model that uses a rotating coordinate system with the parts that are fixed.

The application mode can be used in cases where the modeled device can be divided into rotational invariant geometries. The desired operation can be for example, to rotate an impeller in a baffled tank. This is exemplified by the figure below, where the impeller rotates from position 1 to 2. The first step is to divide the geometry into two parts which are both rotational invariant, as shown in step 1a. The second step is to specify the parts that should be modeled with a rotating frame and the ones that should be modeled in a fixed frame, step 1b below. The application mode then automatically does the coordinate transformation and the joining of the fixed and moving parts, step 2a.
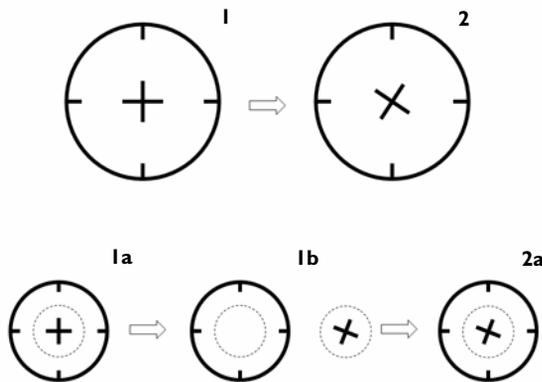

Figure 5-35: The modeling procedure in the rotating machinery application mode in the Chemical Engineering Module.

Model definition

The model treated in this example is that of a pump. Despite the model being somewhat schematic, the example contains all the components needed to describe more realistic geometries. The modeled geometry is shown in the figure below. The impeller rotates clockwise with a speed of 10 RPM.

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

The boundary conditions at the fixed wall of the pump are no-slip conditions. The boundary conditions for the impeller blades are no-slip clock wise rotation conditions. At the pump inlet, pressure is specified, while neutral conditions are implemented at the outlet.

The implementation in the rotating machinery application mode is quite straightforward. First you have to draw the geometry using two separate subdomains for the fixed and rotating parts. Then you have to activate the assembly feature, which makes it possible to treat the two subdomains as two separate parts in an assembly. After this you specify which part that uses a rotating frame. Once you have done this, you can proceed to the usual steps of setting the fluid properties, boundary conditions, and then to the meshing and solving of the problem.
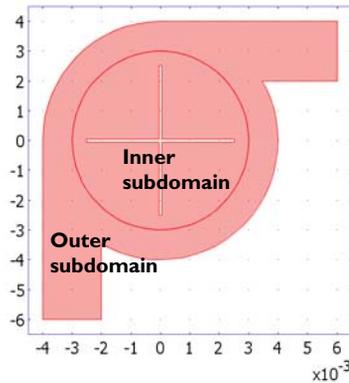

Figure 5-36: Geometry of the schematic pump with the inner subdomain described in a rotating frame and the outer one in a fixed frame.

The figure below shows the flow field as the modulus of the velocity vector and as equi-distance placed streamlines.

Time=20   Surface: Velocity field [m/s]   Max: 2.619e-3
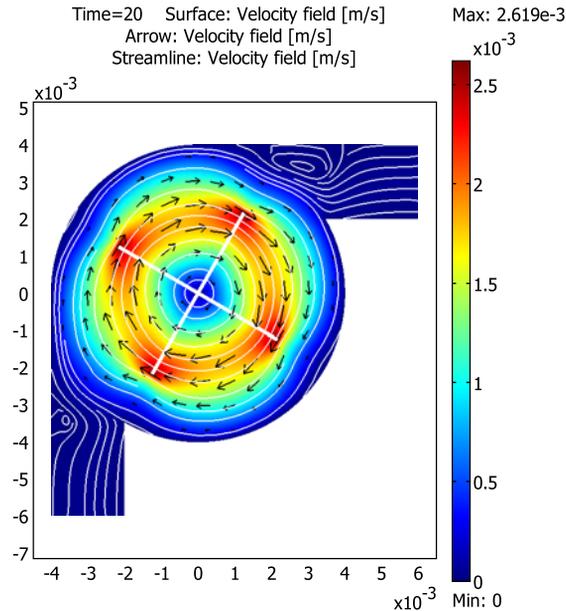Arrow: Velocity field [m/s]
Streamline: Velocity field [m/s]

Figure 5-37: The flow field after 20.5 seconds. The plot shows that two small eddies are formed when the blades enters the inlet and outlet sections.

In the plot above you can clearly see that two eddies are created as the blades of the pump enter the inlet and outlet sections. These eddies decrease the efficiency of the pump. In addition to that, the streamlines show that a large part of the fluid rotates around the blade without actually leaving the pump. By decreasing the cross section of the recirculation area in the pump, it is possible to increase the performance of the pump as a function of the rotating speed of the blades. The figure below shows the velocity field for such a pump geometry. The net flow through the pump compared to

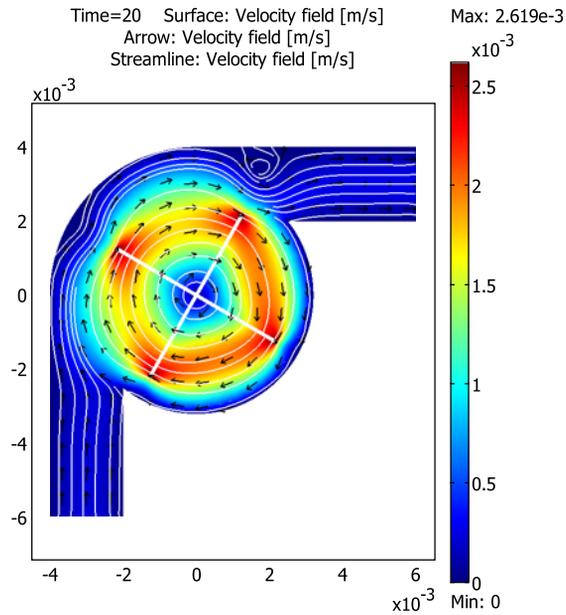the above case is increased by almost a factor of 5 without increasing the rotational speed of the pump.

Figure 5-38: Velocity field for an improved geometry, where parts of the recirculation section has been removed.

Model Library path:

Chemical_Engineering_Module/Momentum__Transport/schematic_pump1

Modeling Using the Graphical User Interface

1 Start COMSOL Multiphysics.

2 In the **Model Navigator** select **2D** in the **Space dimension** list.

3 In the list of application modes select
   Chemical Engineering Module>Rotating Machinery>Rotating Navier Stokes.

4 Click **OK**.

Open the **Options>Constants** dialog box. Enter the following constants; when done, click **OK**:

NAME	EXPRESSION
dens	1e3
visc	1e-2
rpm	10

GEOMETRY MODELING

1 Draw a circle of radius 4e-3 centered at the point (0,0).

2 Select the menu item **Draw>Specify Objects>Rectangle** to create a rectangle with the following settings:

OBJECT	WIDTH	HEIGHT	BASE X	BASE Y
R1	2e-3	6e-3	-4e-3	-6e-3

3 Press Ctrl+A to select both objects.

4 Click the **Create Composite Object** button on the Draw toolbar. Deselect the **Keep interior boundaries** check box. Press **OK**.

5 Create a second rectangle with the following settings:

OBJECT	WIDTH	HEIGHT	BASE X	BASE Y
R1	6e-3	2e-3	0	2e-3

6 Again, press Ctrl+A t select both objects, click the **Create Composite Object**, and press **OK**.

7 Draw a circle of radius 3e-3 centered at the point (0,0).

8 Press Ctrl+A to select both objects.

9 Click the **Create Composite Object** button on the Draw toolbar. Activate the **Keep interior boundaries** check box. Press **OK**.

10 Select the **Split Object** button on the **Draw** toolbar.

11 Shift-click the **Rectangle/Square (Centered)** button on the Draw toolbar. In the **Width** field enter 5e-3, in the **height** field enter 1e-4, and set **Base** to **Center**. Click **OK**.

12 Press Ctrl+C to make a copy of the rectangle.

13 Press Ctrl+V and click **OK**.

14 Click the **Rotate** button on the **Draw** toolbar and set the **Rotation angle** to 90 degrees, click **OK**.

15 Shift-click to select both rectangles. Click the **Create Composite Object** button on the Draw toolbar. Deselect the **Keep interior boundaries** check box and press **OK**.

16 Shift-click again to select the cross and the inner circle, then press the **Difference** button.

17 Press Ctrl+A.

18 Activate the **Use Assembly** option on the **Draw** menu.

19 Choose **Create Pairs** from the **Draw** menu and click **OK**.

PHYSICS SETTINGS

Subdomain Settings

1 From the **Multiphysics** menu, select the **Moving Mesh (ALE)** application mode.

2 Open the **Subdomain Settings** dialog box from the **Physics** menu and select subdomain 2.

3 In the **Subdomain selection** area, set the **Group** selection to rotate_CW.

4 Click **OK**.

5 From the **Multiphysics** menu, select **Incompressible Navier-Stokes**.

6 Open the **Subdomain Settings** dialog box from the **Physics** menu.

7 For both subdomains, set the **Density** to dens and the **Dynamic viscosity** to visc. Click **OK** to close the dialog box.

Boundary Conditions

1 From the **Physics** menu, select **Selection Mode>Boundary Mode**.

2 Shift-click to select the twelve boundaries on the impeller surface. The boundary indices are 15-26.

3 Open the **Boundary Settings** dialog box from the **Physics** menu.

4 In the **Boundary selection** area, set the **Group** selection to no_slip_CW.

5 Select boundary 2 in the list and choose the **Pressure** condition.

6 For boundary 6, use the **Neutral** condition.

7 Click **OK**.

MESH GENERATION

Click the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

COMPUTING THE SOLUTION

1 Open the **Solver Parameters** dialog box from the **Solve** menu.

2 Set **Times** to `0:0.5:60`, **Relative tolerance** to `1e-4`, and **Absolute tolerance** to `1e-5`.

3 Click **OK** to close the dialog box, and then the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

To generate Figure 5-37:

1 From the **Postprocessing** menu open the **Plot Parameters** dialog box and go to the **General** page.

2 Select the **Contour** and **Arrow** check boxes, and deselect the **Geometry edges** check box.

3 Select 20 in the **Solution at time** list.

4 Press **OK**.