Created in COMSOL Multiphysics 6.1

Swirl Flow Around a Rotating Disk

Introduction

This example models a rotating disk in a tank. The model geometry is shown in Figure 1. Because the geometry is rotationally symmetric, it is possible to model it as a 2D cross section. However, the velocities in the angular direction differ from zero, so the model must include all three velocity components, even though the geometry is in 2D.

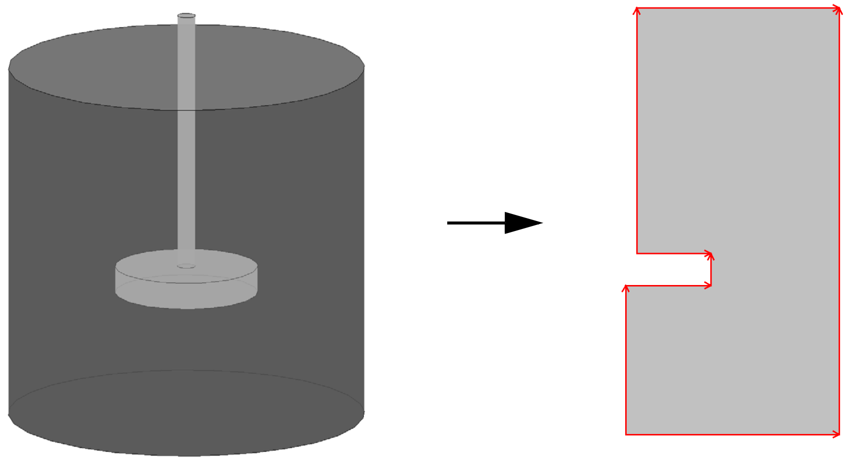

Figure 1: The original 3D geometry can be reduced to 2D because the geometry is rotationally symmetric.

Model Definition

DOMAIN EQUATIONS

The flow is described by the Navier-Stokes equations:

$$\rho\frac{\partial\mathbf{u}}{\partial t}+\rho(\mathbf{u}\cdot\nabla)\mathbf{u} = \nabla\cdot[-p\mathbf{I}+\mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^T)]+\mathbf{F}$$

$$\nabla\cdot\mathbf{u} = 0$$

In these equations, $\mathbf{u}$ denotes the velocity (SI unit: m/s), ρ the density (SI unit: kg/m^3), μ the dynamic viscosity (SI unit: Pa·s), and p the pressure (SI unit: Pa). For a stationary, axisymmetric flow the equations reduce to (Ref. 1):

$$\rho\left(u\frac{\partial u}{\partial r}-\frac{v^2}{r}+w\frac{\partial u}{\partial z}\right)+\frac{\partial p}{\partial r}=\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u}{\partial r}\right)-\frac{u}{r^2}+\frac{\partial^2 u}{\partial z^2}\right]+F_r$$

$$\rho\left(u\frac{\partial v}{\partial r}+\frac{uv}{r}+w\frac{\partial v}{\partial z}\right)=\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v}{\partial r}\right)-\frac{v}{r^2}+\frac{\partial^2 v}{\partial z^2}\right]+F_\varphi$$

$$\rho\left(u\frac{\partial w}{\partial r}+w\frac{\partial w}{\partial z}\right)+\frac{\partial p}{\partial z}=\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial w}{\partial r}\right)+\frac{\partial^2 w}{\partial z^2}\right]+F_z$$

Here u is the radial velocity, v the rotational velocity, and w the axial velocity (SI unit: m/s). In the model you set the volumetric force components F_r, F_φ, and F_z to zero. The swirling flow is 2D even though the model includes all three velocity components.

BOUNDARY CONDITIONS

Figure 2 below shows the boundary conditions.

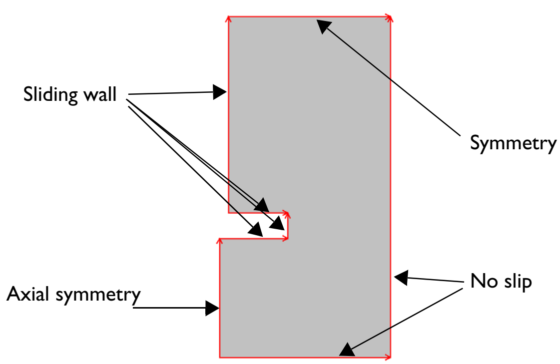

Figure 2: Boundary conditions.

On the stirrer, use the sliding wall boundary condition to specify the velocities. The velocity components in the plane are zero, and that in the angular direction is equal to the angular velocity, ω, times the radius, r:

$$v_{\mathrm{w}} = r\omega$$

At the boundaries representing the cylinder surface a no slip condition applies, stating that all velocity components equal zero:

$$\mathbf{u} = (0, 0, 0)$$

At the boundary corresponding to the rotation axis, use the axial symmetry boundary condition allowing flow in the tangential direction of the boundary but not in the normal direction. This is obtained by setting the radial velocity (and in the case of swirl flow - the azimuthal velocity) to zero:

$$u = 0, \mathrm{v} = 0$$

On the top boundary, which is a free surface, use the Symmetry condition to allow for flow in the axial and rotational directions only. This boundary condition is mathematically similar to the axial symmetry condition.

POINT SETTINGS

In this model you need to lock the pressure to a reference value in a point. The reason for this is that the model does not contain any boundary condition where the pressure is specified (this is often done at outlets). Also the fluid density is constant, which means that the pressure level is not coupled to the density. In this model, set the pressure to zero in the top-right corner.

Results

The parametric solver provides the solution for four different angular velocities. Figure 3 shows the results for the smallest angular velocity, $\omega = 0.25\pi$ rad/s.

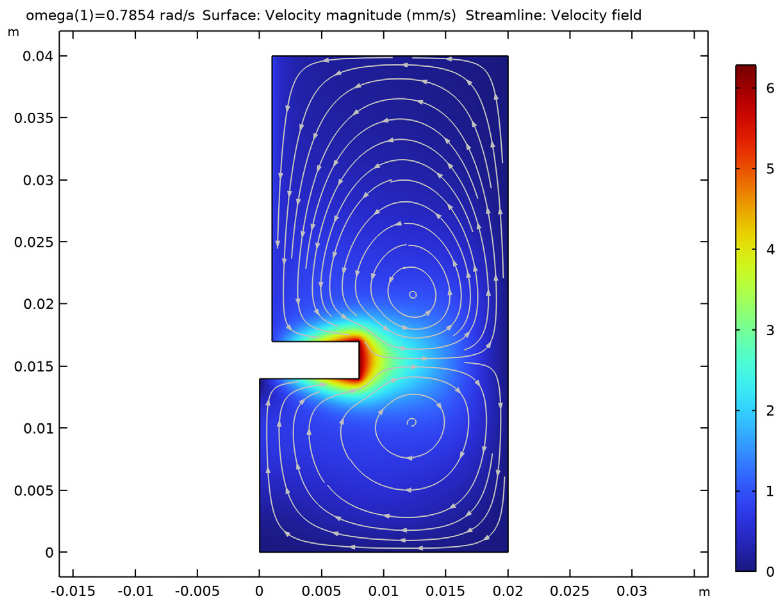

Figure 3: Results for angular velocity ω= *0.25*π *rad/s. The surface plot shows the magnitude of the velocity field and the white lines are streamlines of the velocity field.*

The shape of the two recirculation zones, which are visualized with streamlines, changes as the angular velocity increases. Figure 4 shows the streamlines of the velocity field for higher angular velocities.

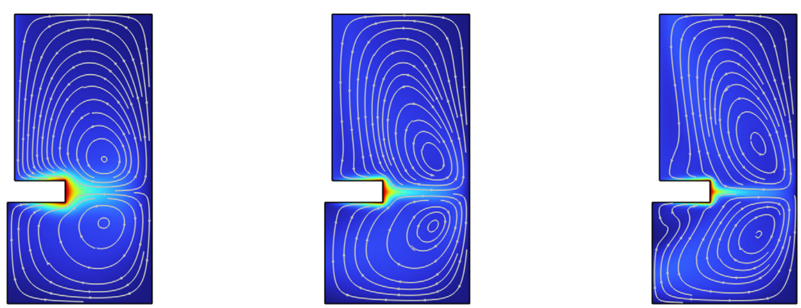

Figure 4: Results for angular velocities ω = *0.5*π*, 2*π*, and 4*π *rad/s. The surface plot shows the magnitude of the velocity and the white lines are streamlines of the velocity field.*

Figure 5 and Figure 6 show isocontours of the rotational velocity component together with surface plots of the velocity magnitude for different angular velocities.

.

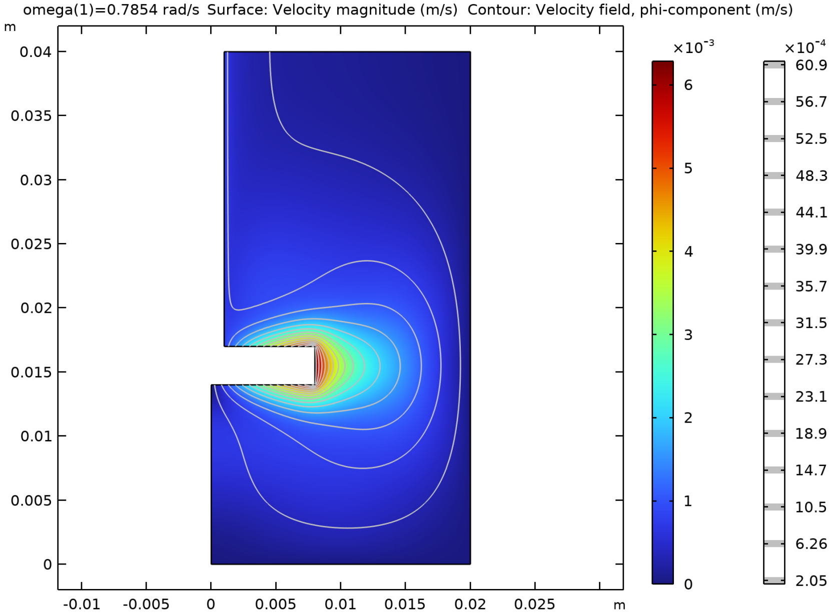

Figure 5: Isocontours for the azimuthal velocity component for angular velocity $\omega = 0.25\pi$ *rad/s. The surface plot shows the magnitude of the velocity.*

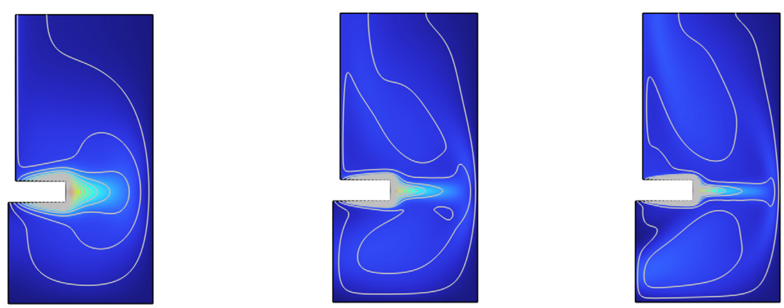

Figure 6: Magnitude of the velocity field (surface) and isocontours for the azimuthal velocity component for angular velocities (left to right) $\omega = 0.5\pi$, 2π, *and* 4π *rad/s.*

Figure 7 shows the turbulent viscosity and flow fields for the angular velocity for $\omega = 500\pi$ rad/s when the flow in the mixer volume is developed.

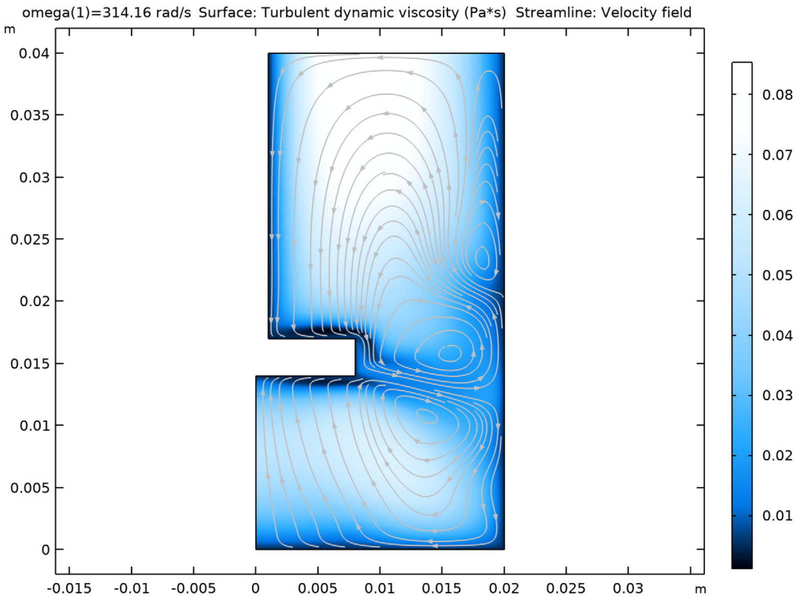

Figure 7: Results for angular velocity $\omega = 500\pi$ *rad/s. The surface plot shows the turbulent viscosity and the white lines are streamlines of the velocity field.*

Reference

1. P.M. Gresho and R.L. Sani, *Incompressible Flow and the Finite Element Method*, vol. 2, p. 469, John Wiley & Sons, 1998.

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

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 Axisymmetric**.
2 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

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
omega	0.25*pi[rad/s]	0.7854 rad/s	Angular velocity

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type 0.02.
4 In the **Height** text field, type 0.04.
5 Click **Build All Objects**.
6 Click the **Zoom Extents** button in the **Graphics** toolbar.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type 0.008.
4 In the **Height** text field, type 0.003.
5 Locate the **Position** section. In the **z** text field, type 0.014.
6 Click **Build All Objects**.

Rectangle 3 (r3)

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

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

3 In the **Width** text field, type `0.001`.

4 In the **Height** text field, type `0.023`.

5 Locate the **Position** section. In the **z** text field, type `0.017`.

6 Click **Build All Objects**.

Difference 1 (dif1)

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

2 Select the object **r1** only.

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

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

5 Select the objects **r2** and **r3** only.

6 Clear the **Keep interior boundaries** check box.

7 Click **Build All Objects**.

8 Click the **Zoom Extents** button in the **Graphics** toolbar.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1e3	kg/m³	Basic
Dynamic viscosity	mu	1e-3	Pa·s	Basic

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 **Physical Model** section.

3 Select the **Swirl flow** check box.

4 Click to expand the **Discretization** section. From the **Discretization of fluids** list, choose **P2+P1**.

This setting gives quadratic elements for the velocity field.

Wall 2

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

2 Select Boundaries 3–5 and 7 only.

3 In the **Settings** window for **Wall**, click to expand the **Wall Movement** section.

4 Select the **Sliding wall** check box.

5 In the v_w text field, type `omega*r`.

Symmetry 1

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

2 Select Boundary 6 only.

Pressure Point Constraint 1

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

2 Select Point 8 only.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
omega (Angular velocity)	`0.25*pi 0.5*pi 2*pi 4*pi`	`rad/s`

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

RESULTS

Velocity (spf)

To create Figure 3 do the following steps:

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

2 From the **Parameter value (omega (rad/s))** list, choose **0.7854**.

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

Surface

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

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

3 From the **Unit** list, choose **mm/s**.

Streamline 1

1 In the **Model Builder** window, 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 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

5 Select the **Number of arrows** check box. In the associated text field, type `100`.

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

7 Locate the **Streamline Positioning** section. In the **Separating distance** text field, type `0.04`.

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

To produce the series of snapshots of the velocity and streamlines of the velocity field shown in Figure 4, proceed with the following steps:

Surface

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

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Clear the **Color legend** check box.

Velocity (spf)

1 In the **Model Builder** window, click **Velocity (spf)**.

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

3 From the **Parameter value (omega (rad/s))** list, choose **1.5708**.

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

5 From the **Parameter value (omega (rad/s))** list, choose **6.2832**.

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

7 From the **Parameter value (omega (rad/s))** list, choose **12.566**.

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

To plot the isocontours for the azimuthal velocity component Figure 5, proceed with the following steps.

Azimuthal Velocity

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Azimuthal Velocity` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (omega (rad/s))** list, choose **0.7854**.

Surface 1

Right-click **Azimuthal Velocity** and choose **Surface**.

Contour 1

1 In the **Model Builder** window, right-click **Azimuthal Velocity** and choose **Contour**.

2 In the **Settings** window for **Contour**, click to expand the **Quality** section.

3 In the **Azimuthal Velocity** toolbar, click **Plot**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Laminar Flow>Velocity and pressure>Velocity field - m/s>v - Velocity field, phi-component**.

5 Locate the **Levels** section. In the **Total levels** text field, type `15`.

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

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

8 Locate the **Quality** section. From the **Resolution** list, choose **Finer**.

9 From the **Recover** list, choose **Within domains**.

To reproduce Figure 6 do the following steps.

Surface 1

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

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Clear the **Color legend** check box.

Azimuthal Velocity

1 In the **Model Builder** window, click **Azimuthal Velocity**.

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

3 From the **Parameter value (omega (rad/s))** list, choose **1.5708**.

4 In the **Azimuthal Velocity** toolbar, click **Plot**.

5 From the **Parameter value (omega (rad/s))** list, choose **6.2832**.

6 In the **Azimuthal Velocity** toolbar, click **Plot**.

7 From the **Parameter value (omega (rad/s))** list, choose **12.566**.

8 In the **Azimuthal Velocity** toolbar, click **Plot**.

Before solving for turbulent flow, store the laminar flow solutions in a separate dataset.

9 In the **Study** toolbar, click **Create Solution Copy**.

STUDY 1

Laminar Swirl Flow

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node, then click **Solution 1 - Copy 1 (sol2)**.

2 In the **Settings** window for **Solution**, type `Laminar Swirl Flow` in the **Label** text field.

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 **Turbulence** section.

3 From the **Turbulence model type** list, choose **RANS**.

Higher order elements are often not cost effective for turbulent flows and can also cause numerical instabilities.

4 Locate the **Discretization** section. From the **Discretization of fluids** list, choose **P1+P1**.

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**.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

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

3 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
omega (Angular velocity)	`range(pi*100,pi*200,pi*500)`	`rad/s`

The continuation solver works best for models with linear dependence on the parameter. A more robust alternative for nonlinear applications is to start from the solution for the previous parameter value.

1 From the **Run continuation for** list, choose **No parameter**.

2 From the **Reuse solution from previous step** list, choose **Yes**.

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

RESULTS

Velocity (spf)

Now create two cut plane datasets in order visualize the velocity on the revolved solution.

Cut Plane 1

1 In the **Results** toolbar, click **Cut Plane**.

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

3 From the **Plane type** list, choose **General**.

4 In row **Point 2**, set **x** to `0`.

5 In row **Point 2**, set **z** to `1`.

6 In row **Point 3**, set **x** to `cos(-90[deg])`.

7 In row **Point 3**, set **y** to `sin(-90[deg])`.

8 Click **Plot**.

Cut Plane 2

1 Right-click **Cut Plane 1** and choose **Duplicate**.

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

3 In row **Point 3**, set **x** to `cos(135[deg])`.

4 In row **Point 3**, set **y** to `sin(135[deg])`.

5 Click **Plot**.

Surface

1 In the **Model Builder** window, expand the **Results>Velocity, 3D (spf)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Linear>GrayScale** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

7 Clear the **Color legend** check box.

8 From the **Color table transformation** list, choose **Reverse**.

Velocity, 3D (spf)

In the **Model Builder** window, click **Velocity, 3D (spf)**.

Streamline Surface 1

1 In the **Velocity, 3D (spf)** toolbar, click **More Plots** and choose **Streamline Surface**.

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

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

4 From the **Parameter value (omega (rad/s))** list, choose **314.16**.

5 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Uniform density**.

6 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

7 Select the **Radius scale factor** check box. In the associated text field, type `1e-4`.

8 Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

9 Select the **Number of arrows** check box. In the associated text field, type `100`.

10 Select the **Scale factor** check box. In the associated text field, type `0.018`.

Color Expression 1

1 In the **Velocity, 3D (spf)** toolbar, click **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Aurora>JupiterAuroraBorealis** in the tree.

5 Click **OK**.

Streamline Surface 1

Use a **Filter** feature to restrict the plot to the cut plane. Note that `rev1y` is the y)-axis of the revolution dataset.

Filter 1

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

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

3 In the **Logical expression for inclusion** text field, type `rev1y<0[m]`.

4 In the **Velocity, 3D (spf)** toolbar, click **Plot**.

Velocity, 3D (spf)

In the **Model Builder** window, under **Results** click **Velocity, 3D (spf)**.

Arrow Surface 1

1 In the **Velocity, 3D (spf)** toolbar, click **Arrow Surface**.

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

3 From the **Dataset** list, choose **Cut Plane 2**.

4 From the **Parameter value (omega (rad/s))** list, choose **314.16**.

5 Locate the **Coloring and Style** section.

6 Select the **Scale factor** check box. In the associated text field, type `0.02`.

7 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `300`.

8 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Streamline Surface 1**.

Color Expression 1

In the **Velocity, 3D (spf)** toolbar, click **Color Expression**.

Filter 1

1 In the **Model Builder** window, right-click **Arrow Surface 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `rev1y>0[m]`.

Velocity (spf)

Next, define a surface plot visualizing the turbulent viscosity and the streamlines of the velocity field (Figure 7).

Turbulent viscosity

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

2 In the **Settings** window for **2D Plot Group**, type `Turbulent viscosity` in the **Label** text field.

Streamline 1

1 In the **Model Builder** window, expand the **Turbulent viscosity** node, then click **Streamline 1**.

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

3 From the **Advanced parameters** list, choose **Manual**.

4 In the **Starting distance factor** text field, type `1.0`.

5 In the **Terminating distance factor** text field, type `0.25`.

6 In the **Turbulent viscosity** toolbar, click **Plot**.

Surface

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Turbulent Flow, k-ε>Turbulence variables>spf.muT - Turbulent dynamic viscosity - Pa·s**.

3 Locate the **Coloring and Style** section. Select the **Color legend** check box.

4 Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Aurora>JupiterAuroraBorealis** in the tree.

6 Click **OK**.