Model created in COMSOL Multiphysics 6.4

Flow in a Hydrocyclone

Introduction

Cyclones are used in a variety of applications ranging from the mining industry to vacuum cleaners (Ref. 1). In the pulp and paper industry, hydrocyclones are used for contaminant removal, pulp thickening and fiber fractionation. Most cyclones do not contain any moving parts, hence the flow is driven exclusively by the applied pressure drops between the inlet(s) and the two outlets. The forward stream in the process is referred to as the *accept flow*, whereas the discarded stream is referred to as the *reject flow*. Depending on the application, the accept outlet could either be the overflow located at the base of the cone, near the inlet(s), or the underflow near the apex of the cone. The former configuration is used for removal of heavy (compared to the carrier fluid) contaminants, whereas the latter is used for removal of light contaminants and in thickening processes. In fractionation processes the definition of accept and reject is more or less a matter of convenience since both streams are applied forward in the system.

Model Definition

The model geometry used in this application is shown in Figure 1.

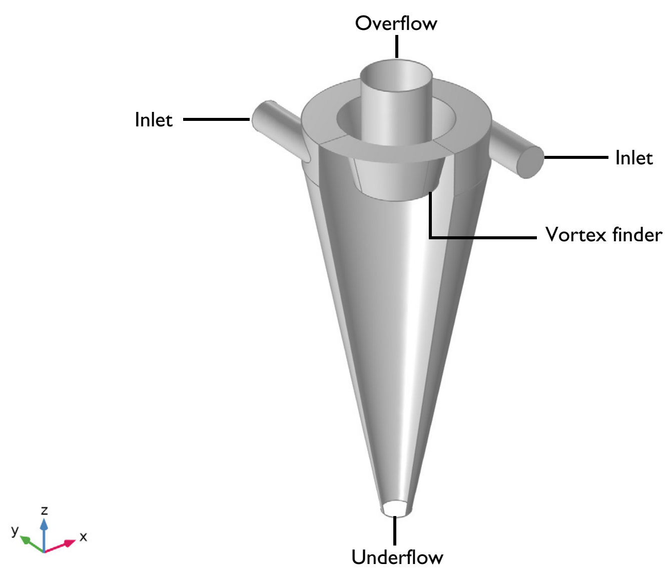

Figure 1: Model geometry showing the inlets, overflow (here the accept outlet), and underflow (the reject outlet).

Two circular inlets are tangentially attached to the annular inlet chamber, which is separated from the overflow by a wall called the “vortex finder”. This design creates a strong swirl in the incoming flow. From the annular inlet chamber, the flow enters a conical chamber where the separation takes place. The conical shape preserves the angular momentum and stabilizes the vortex core — the central region of the swirl motion characterized by nearly solid-body rotation. A portion of the flow is effluxed through the underflow near the apex of the conical separation chamber, and the rest exits through the overflow.

The flow in a hydrocyclone is characterized by a very strong swirl, which makes it difficult to simulate using an isotropic turbulence model. It is imperative that the swirl flow is accurately captured in order to assess the separation efficiency for various particles. The streamlines essentially follow the azimuthal direction whereas mixing of momentum by turbulent fluctuations takes place in the radial direction, which happens to be very close to the wall-normal direction in the major part of the hydrocyclone. This makes the v2-f turbulence model a good candidate for the prevailing flow conditions.

Stationary operating conditions corresponding to those of heavy contaminant removal are studied in this application. The flowing medium is pure water at 20°C. The hydrocyclone is assumed to be pressurized and is hence operating without an air core. Initial values were chosen as zero velocity, zero pressure, and default values for the turbulence variables. No-slip conditions with automatic wall treatment were applied on all the walls. At the two inlets the velocity is set to 5 m/s, and the turbulence conditions to default (medium turbulent intensity and geometry based turbulence length scale). We also prescribe that 5% of the inlet flow exits through the underflow by specifying a uniform velocity profile. A constant pressure condition is applied at the overflow. The outlet conditions can be made more self-consistent by adding outlet chambers, corresponding to the system geometry, at both ends.

Figure 2 shows the streamlines for the swirling flow in the hydrocyclone.

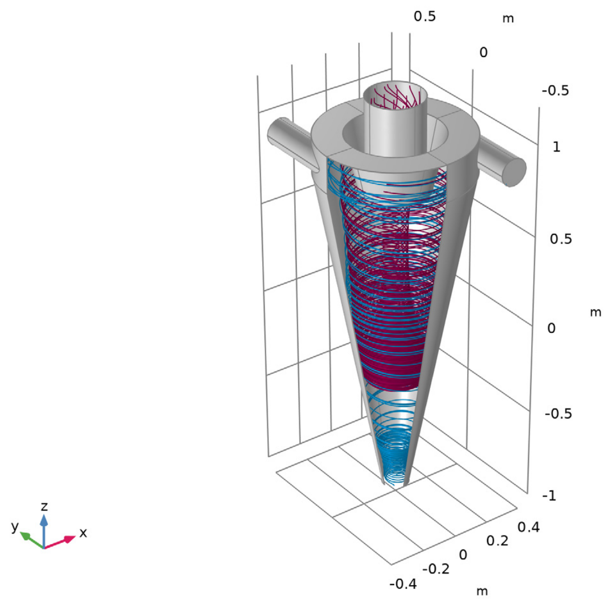

Figure 2: Streamlines for the overflow (burgundy) and underflow (teal).

The streamlines describe the typical flow field encountered in hydrocyclone applications. From the inlet chamber, the flow is diverted toward the underflow. In our case, 95% of the incoming flow should be reversed and exit through the overflow. This is illustrated by the burgundy streamlines in the core. The remainder (teal) almost sticks to the wall and exits through the underflow.

The pressure drop and in-plane streamlines on two orthogonal cut planes through the hydrocyclone are displayed in Figure 3.

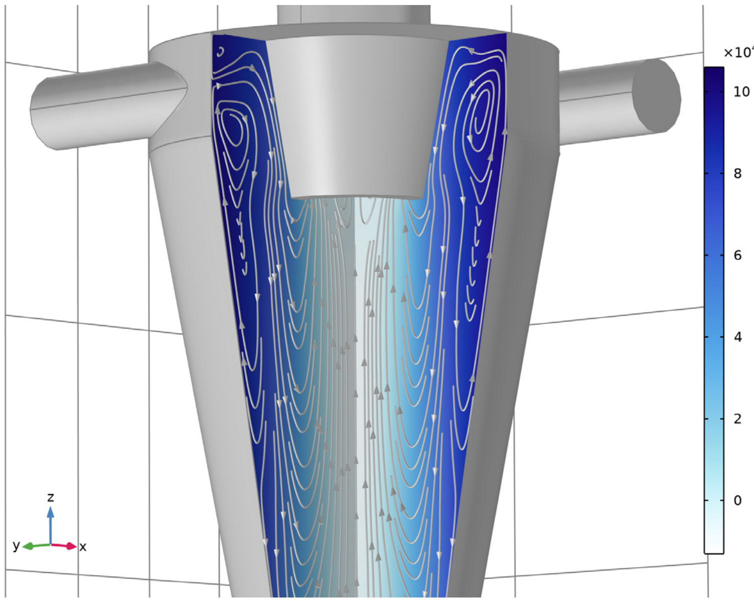

Figure 3: Pressure drop and in-plane streamlines in the xz- and yz-planes.

The two jets mix in the inlet chamber, resulting in azimuthal pressure variations on the vortex core. For certain hydrocyclone designs, this may cause the vortex core to destabilize resulting in poor separation performance. The optimal number and design of the inlet pipes as well as the design of the inlet chamber is still an active research field. The pressure drop between the inlets and outlets is of the order 100 kPa. Figure 4 shows a contour surface displaying a vertical (stable) vortex core. The swirl flow in the hydrocyclone can be divided into an outer region, described by a semi-free vortex, and an inner region of nearly solid body rotation.

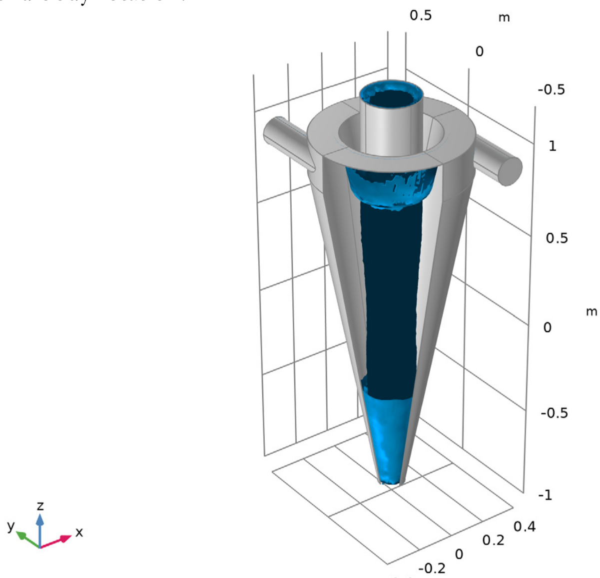

Figure 4: Contour surface of the vortex core.

The graph in Figure 5 shows the azimuthal velocity component as a function of the radius at a vertical position 10 cm below the vortex finder. The inner core of nearly solid-body rotation is clearly distinguishable from the outer semi-free vortex.

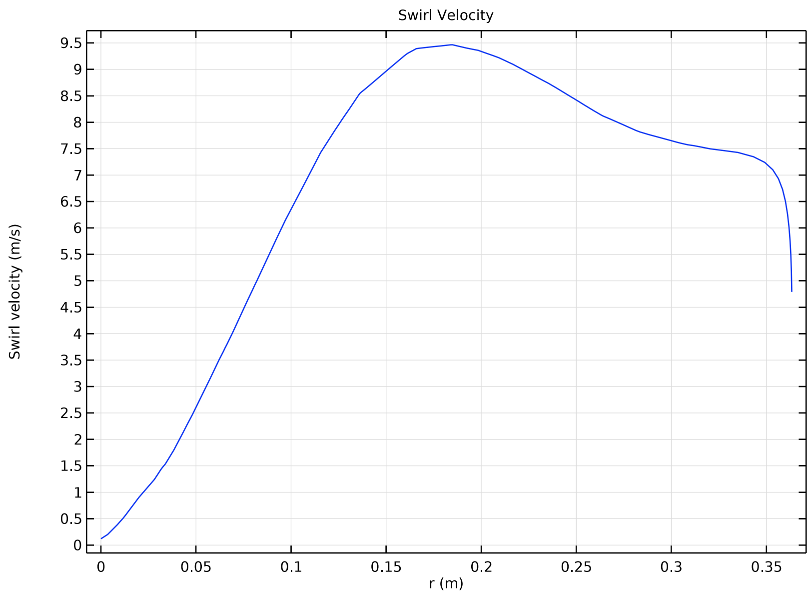

Figure 5: Azimuthal (swirl) velocity versus radius 10 cm below the vortex finder.

Notes About the COMSOL Implementation

The mesh is deliberately made relatively coarse to reduce the computational time for this tutorial model. If the maximum size of the elements is reduced by thirty percent, the maximum swirl velocity in Figure 5 reaches 12 m/s. Particle-tracking can be added to the model to illustrate the separation of heavy and light fractions.

Reference

1. D.Bradley, "The Hydrocyclone, 1^{st} Edition, International Series of Monographs in Chemical Engineering," *Pergamon*, 1965.

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

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 **3D**.

2 In the **Select Physics** tree, select **Fluid Flow** > **Single-Phase Flow** > **Turbulent Flow** > **Turbulent Flow, v2-f (spf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Stationary with Initialization**.

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
u_in	5[m/s]	5 m/s	Inlet velocity
r_in	0.0725[m]	0.0725 m	Inlet radius
r_out	0.07[m]	0.07 m	Reject radius
R_f	0.05	0.05	Reject volume fraction
u_out	R_f*2*(r_in/r_out)^2*u_in	0.53635 m/s	Reject velocity

MESH 1

The mesh will be imported from file and used directly for the simulations.

Import 1

1 In the **Mesh** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `hydrocyclone_mesh.mphbin`.

5 Click **Import**.

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

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

TURBULENT FLOW, V2-F (SPF)

Inlet 1

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

2 Select Boundaries 9 and 40 only, corresponding to the two inlets.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `u_in`.

The fully developed flow condition could be used as well to give a computed flow field for the inlet parameters. However, by doing so, the computing time would increase slightly. Thus, an analytic expression is used for the inlet parameters.

Outlet 1

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

2 Select Boundaries 20, 21, 29, and 30 only, corresponding to the reject outlet.

3 In the **Settings** window for **Outlet**, locate the **Boundary Condition** section.

4 From the list, choose **Velocity**.

5 Locate the **Velocity** section. In the U_0 text field, type `u_out`.

Outlet 2

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

2 Select Boundaries 18, 19, 28, and 31 only.

3 In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.

4 Select the **Normal flow** checkbox.

5 Clear the **Suppress backflow** checkbox.

STUDY 1

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

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

3 Clear the **Generate default plots** checkbox, because the plots are created from scratch.

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

RESULTS

First, create a new surface dataset needed to produce Figure 2, Figure 3, and Figure 5.

Surface 1

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results** > **Datasets** and choose **Surface**.

3 In the **Settings** window for **Surface**, locate the **Selection** section.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `2, 4-17, 22-27, 32-42` in the **Selection** text field.

6 Click **OK**.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Turbulent Flow, v2-f** > **Velocity Streamlines (spf)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Velocity Streamlines (spf)

In the **Model Builder** window, expand the **Velocity Streamlines (spf)** node.

Color Expression 1

1 In the **Model Builder** window, expand the **Results** > **Velocity Streamlines (spf)** > **Streamline 1** node.

2 Right-click **Color Expression 1** and choose **Delete**.

Streamline 1

1. In the **Settings** window for **Streamline**, locate the **Data** section.
2. From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
3. Locate the **Streamline Positioning** section. In the **Number** text field, type `10`.
4. Select Boundaries 18, 19, 28, and 31 only.
5. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.
6. In the **Tube radius expression** text field, type `0.0025`.
7. Select the **Radius scale factor** checkbox.
8. Find the **Point style** subsection. From the **Color** list, choose **Custom**.
9. On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.
10. Click **Define custom colors**.
11. Set the RGB values to 128, 0, and 64, respectively.
12. Click **Add to custom colors**.
13. Click **Show color palette only** or **OK** on the cross-platform desktop.

Streamline 2

1. Right-click **Results** > **Velocity Streamlines (spf)** > **Streamline 1** and choose **Duplicate**.
2. In the **Settings** window for **Streamline**, locate the **Selection** section.
3. Click  **Clear Selection**.
4. Select Boundaries 20, 21, 29, and 30 only.
5. Locate the **Streamline Positioning** section. In the **Number** text field, type `2`.
6. Locate the **Coloring and Style** section. Click **Define custom colors**.
7. Set the RGB values to 0, 128, and 192, respectively.
8. Click **Add to custom colors**.
9. Click **Show color palette only** or **OK** on the cross-platform desktop.

Surface 1

1. In the **Model Builder** window, right-click **Velocity Streamlines (spf)** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `1`.
4. Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
5. From the **Color** list, choose **Gray**.

Use the previously created Surface dataset for the entire plot group to suppress the surfaces and edges that would otherwise obstruct the view into the hydrocyclone.

Velocity Streamlines (spf)

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

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

3 From the **Dataset** list, choose **Surface 1**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Plot Settings** section. From the **Color** list, choose **Custom**.

6 Click **Define custom colors**.

7 Set the RGB values to 128, 128, and 128, respectively.

8 Click **Add to custom colors**.

9 Click **Show color palette only** or **OK** on the cross-platform desktop.

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

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

The following steps reproduce Figure 3.

Cut Plane 1

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

Cut Plane 2

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** list, choose **xz-planes**.

Pressure

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

2 In the **Settings** window for **3D Plot Group**, type `Pressure` in the **Label** text field.

3 In the **Model Builder** window, expand the **Pressure** node.

Streamline 1, Streamline 2

1 In the **Model Builder** window, under **Results** > **Pressure**, Ctrl-click to select **Streamline 1** and **Streamline 2**.

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

Since the exact same setup will be used in a third plot as well, duplicate this one now before proceeding with the pressure plot.

Pressure 1

In the **Model Builder** window, right-click **Pressure** and choose **Duplicate**.

Surface 2

1 Right-click **Pressure** and choose **Surface**.

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

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

4 Locate the **Expression** section. In the **Expression** text field, type `p`.

5 Click to expand the **Range** section. Select the **Manual color range** checkbox.

6 In the **Minimum** text field, type `-13000`.

7 In the **Maximum** text field, type `106000`.

8 Locate the **Coloring and Style** section. From the **Color table** list, choose **Kyanite**.

Surface 3

1 Right-click **Surface 2** and choose **Duplicate**.

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

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

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 2**.

Pressure

In the **Model Builder** window, click **Pressure**.

Streamline Surface 1

1 In the **Pressure** 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 Locate the **Expression** section. In the **x-component** text field, type `0`.

5 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Magnitude controlled**.

6 In the **Minimum density level** text field, type `8.4`.

7 In the **Maximum density level** text field, type `10`.

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

9 In the **Tube radius expression** text field, type `0.002`.

10 Select the **Radius scale factor** checkbox.

11 Find the **Point style** subsection. From the **Color** list, choose **Gray**.

12 From the **Type** list, choose **Arrow**.

Streamline Surface 2

1 Right-click **Streamline Surface 1** and choose **Duplicate**.

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

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

4 Locate the **Expression** section. In the **x-component** text field, type `u`.

5 In the **y-component** text field, type `0`.

Pressure

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

2 In the **Pressure** toolbar, click **Plot**.

Zoom in to get a closer view of the streamlines.

The following steps reproduce Figure 4.

Vortex Core

1 In the **Model Builder** window, under **Results** click **Pressure 1**.

2 In the **Settings** window for **3D Plot Group**, type `Vortex Core` in the **Label** text field.

Isosurface 1

1 Right-click **Vortex Core** and choose **Isosurface**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Turbulent Flow, v2-f** > **Velocity and pressure** > **Vorticity field - 1/s** > **spf.vorticityz - Vorticity field, z-component**.

5 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

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

7 Select the **Interactive** checkbox.

Depending on the boundary conditions, the value may need to be adjusted by sliding the interactive bar. This visualizes the vortex core in Figure 4.

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

9 From the **Color** list, choose **Custom**.

10 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

11 Click **Define custom colors**.

12 Set the RGB values to 0, 128, and 192, respectively.

13 Click **Add to custom colors**.

14 Click **Show color palette only** or **OK** on the cross-platform desktop.

15 Clear the **Color legend** checkbox.

Vortex Core

1 In the **Model Builder** window, click **Vortex Core**.

2 In the **Vortex Core** toolbar, click **Plot**.

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

The following steps reproduce Figure 5. Start with creating a **Cut Line** dataset in radial direction.

Cut Line 3D 1

1 In the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Line Data** section.

3 In row **Point 1**, set **x** to `-0.5`.

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

5 In row **Point 1**, set **z** to `0.6`.

6 In row **Point 2**, set **z** to `0.6`.

Swirl Velocity

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Swirl Velocity` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Swirl Velocity`.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `r (m)`.

7 Select the **y-axis label** checkbox. In the associated text field, type `Swirl velocity (m/s)`.

Line Graph 1

In the **Swirl Velocity** toolbar, click **Line Graph**.

Swirl Velocity

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

2 Locate the **Data** section. From the **Dataset** list, choose **Cut Line 3D 1**.

Line Graph 1

1 In the **Model Builder** window, click **Line Graph 1**.
2 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
3 In the **Expression** text field, type `(x*v-y*u)/sqrt(x^2+y^2+eps)`.
4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
5 In the **Expression** text field, type `-x`.
6 In the **Swirl Velocity** toolbar, click **Plot**.
7 Click the **Zoom Extents** button in the **Graphics** toolbar.