

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, and the flow is hence entirely driven 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](#).

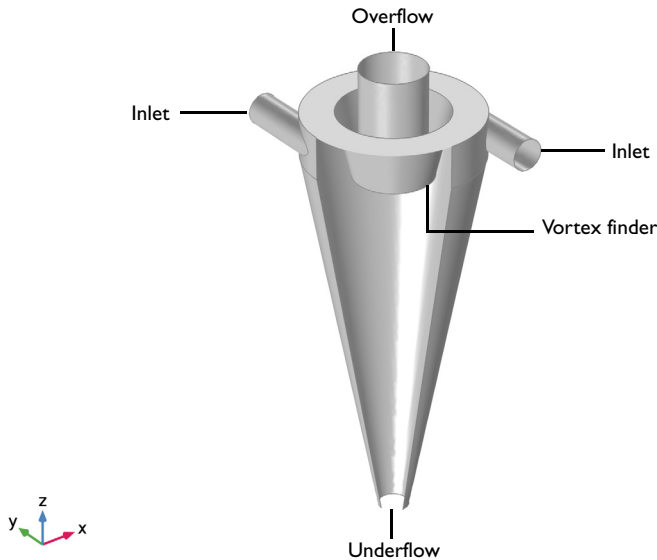


Figure 1: Model geometry showing the inlets, overflow, here the accept outlet, and underflow, reject outlet.

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 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 also happens to be the wall-normal direction in the major part of the hydrocyclone. This makes the v_2 - 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 hence is operating without an air core. Initial values were chosen as zero velocity and zero pressure, and default values for the turbulence variables. No-slip conditions with automatic wall treatment were applied on all walls. At the two inlets the velocity is set to 5 m/s, and the turbulence conditions to isotropic with an intensity of 5 % and a length scale of 0.07 times the inlet diameter. 5 % of the flow exits through the underflow where a uniform velocity profile is specified. A constant pressure condition is applied at the overflow. The outlet conditions can be improved by attaching outlet chambers at both ends.

Results and Discussion

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

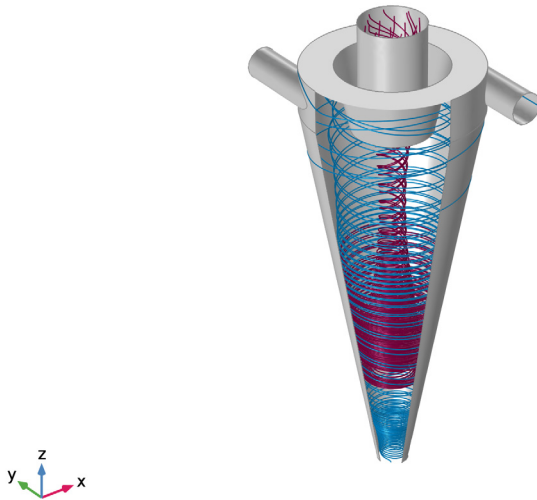


Figure 2: Streamlines for the overflow (burgundy) and underflow (teal).

The streamlines describe the typical flow field encountered in hydrocyclone applications. From the inlet chamber, the flow is diverted toward the underflow. In this case, 95 % of the incoming flow is reversed, and exits through the overflow. This is illustrated by the burgundy streamlines in the core. The remainder (teal) stays closer 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](#).

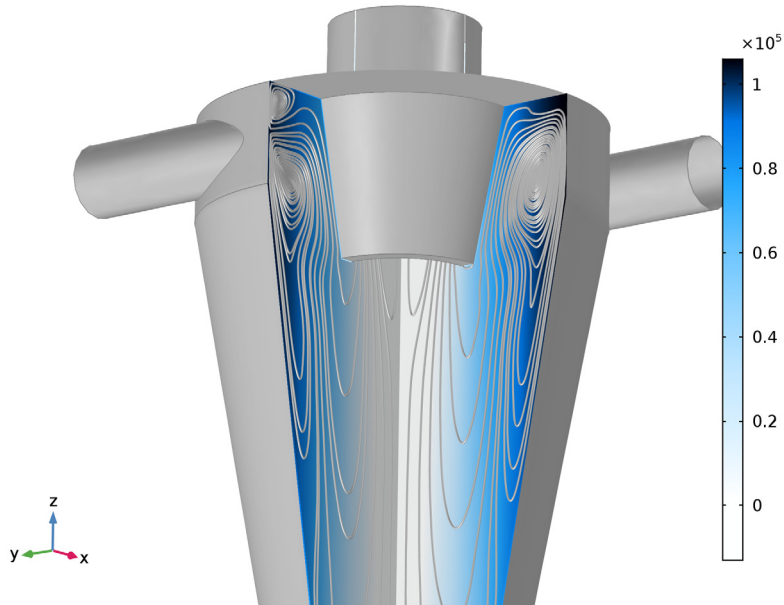


Figure 3: Pressure drop and in-plane streamlines in the xz - and yz -planes.

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, and the design of the inlet chamber, is 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 solid body rotation.

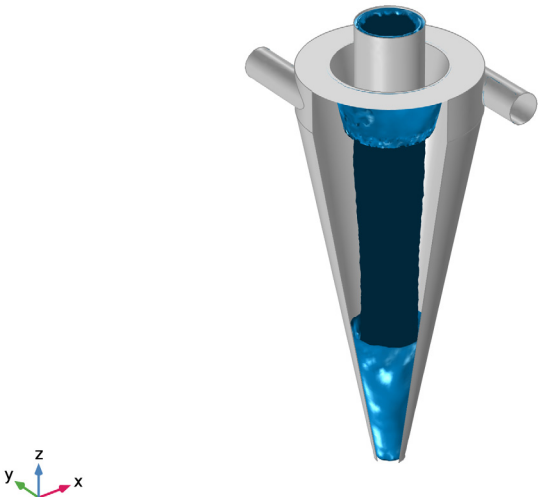


Figure 4: Contour surface of the vortex core.

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. An inner core of solid-body rotation is clearly distinguishable from the outer semi-free vortex.

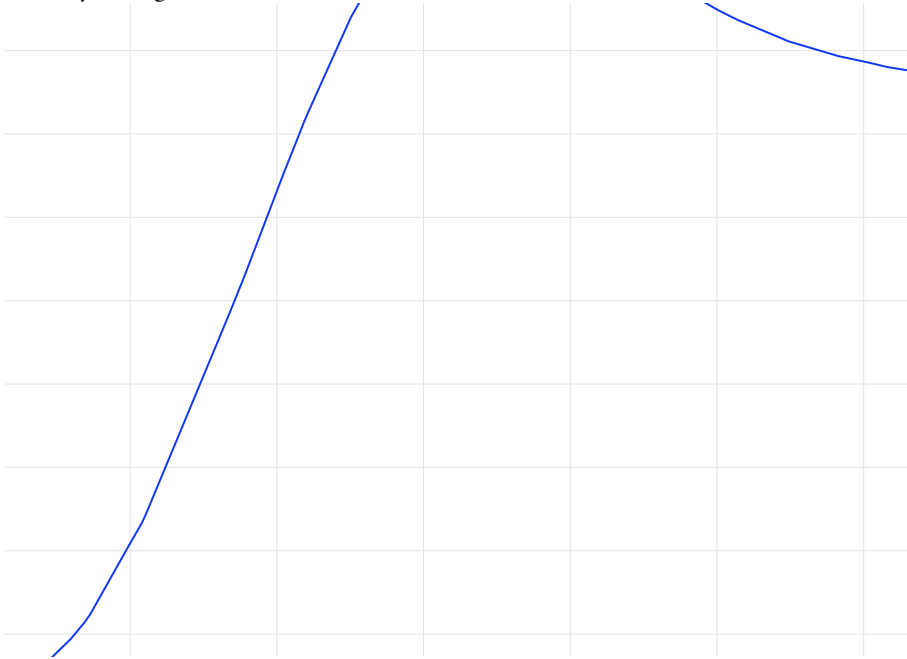


Figure 5: Azimuthal (swirl) velocity versus radius 10 cm below the vortex finder.

Notes About the COMSOL Implementation

The mesh is deliberately made 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.

Reference

1. D. Bradley, “The Hydrocyclone, 1st Edition, International Series of Monographs in Chemical Engineering,” *Pergamon*, 1965.

Application Library path: `CFD_Module/Single-Phase_Tutorials/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>Stationary with Initialization**.
- 6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters**.
- 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

GEOMETRY I

Work Plane 1 (wp1)

- 1 On the **Geometry** toolbar, click **Work Plane**.
- 2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
- 3 From the **Plane** list, choose **xz-plane**.

Plane Geometry

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)** click **Plane Geometry**.

Polygon 1 (poll)

- 1 On the **Work Plane** toolbar, click **Primitives** and choose **Polygon**.
- 2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.
- 3 In the **xw** text field, type 0.4,0.4,r_out,0,0,0.15,0.15,0.18,0.25,0.4.
- 4 In the **yw** text field, type 1,0.8,-1,-1,1.2,1.2,0.7,0.7,1,1.
- 5 Click **Build Selected**.
- 6 In the **Model Builder** window, click **Geometry 1**.

Revolve 1 (rev1)

- 1 On the **Geometry** toolbar, click **Revolve**.
- 2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.
- 3 Clear the **Keep original faces** check box.

Cylinder 1 (cyl1)

- 1 On the **Geometry** toolbar, click **Cylinder**.
- 2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type r_in.
- 4 In the **Height** text field, type 0.5.
- 5 Locate the **Position** section. In the **x** text field, type 0.3215.
- 6 In the **y** text field, type -0.5.
- 7 In the **z** text field, type 0.9.
- 8 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

Cylinder 2 (cyl2)

- 1 On the **Geometry** toolbar, click **Cylinder**.
- 2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type r_in.
- 4 In the **Height** text field, type 0.5.
- 5 Locate the **Position** section. In the **x** text field, type -0.3215.
- 6 In the **z** text field, type 0.9.
- 7 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

8 On the **Geometry** toolbar, click **Build All**.

ADD MATERIAL

1 On 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 **Add to Component** in the window toolbar.

MATERIALS

Water, liquid (mat1)

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

TURBULENT FLOW, V2-F (SPF)

Inlet 1

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

2 Select Boundaries 13 and 46 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} .

5 Locate the **Turbulence Conditions** section. In the I_T text field, type 0.05.

6 In the L_T text field, type $0.07*2*r_{in}$.

Outlet 1

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

2 Select Boundaries 26, 27, 35, and 36 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 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundaries 24, 25, 34, and 37 only.

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

4 Select the **Normal flow** check box.

5 Clear the **Suppress backflow** check box.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **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**, click to expand the **Element size parameters** section.
- 3 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 0.03.
- 4 In the **Minimum element size** text field, type 0.002.

Size 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size 1**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 Click the **Custom** button.
- 4 Click to expand the **Element size parameters** section. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
- 5 In the associated text field, type 0.03.
- 6 Select the **Minimum element size** check box.
- 7 In the associated text field, type 0.001.

Boundary Layer Properties 1

- 1 In the **Model Builder** window, expand the **Component 1 (comp1)>Mesh 1>Boundary Layers 1** node, then click **Boundary Layer Properties 1**.
- 2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Layer Properties** section.
- 3 In the **Number of boundary layers** text field, type 9.
- 4 In the **Boundary layer stretching factor** text field, type 1.1.
- 5 On the **Home** toolbar, click **Build Mesh**.
- 6 Click **Compute**.

RESULTS

Data Sets

First, create data sets needed to produce [Figure 2](#), [Figure 3](#), and [Figure 5](#).

Surface 2

- 1 On the **Results** toolbar, click **More Data Sets** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Selection** section.
- 3 Click **Paste Selection**.
- 4 In the **Paste Selection** dialog box, type 2, 4-6, 10-12, 16-23, 28-33, 38-45, 49, 51-52 in the **Selection** text field.
- 5 Click **OK**.

Edge 3D 1

- 1 On the **Results** toolbar, click **More Data Sets** and choose **Edge 3D**.
- 2 Select Edges 1, 2, 4, 7, 12-15, 21, 23-25, 27, 28, 32, 33, 35-37, 39, 42-45, 47, 49, 51, 53, 55, 57, 63, 66, 67, 69, 71, 73, 74, 80, 81, 83, 84, 89, 91, 93, and 95 only.

Cut Plane 2

- 1 On the **Results** toolbar, click **Cut Plane**.
- 2 On the **Results** toolbar, click **Cut Plane**.
- 3 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.
- 4 From the **Plane** list, choose **xz-planes**.

Cut Line 3D 1

- 1 On 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.

The following steps reproduce [Figure 2](#).

Velocity (spf)

- 1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.
- 2 In the **Settings** window for **3D Plot Group**, type Streamlines in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 4 Locate the **Plot Settings** section. Clear the **Plot data set edges** check box.

Slice

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

2 Right-click **Slice** and choose **Delete**. Click **Yes** to confirm.

Surface 1

1 In the **Model Builder** window, under **Results** right-click **Streamlines** and choose **Surface**.

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

3 From the **Data set** list, choose **Surface 2**.

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

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

Line 1

1 Right-click **Streamlines** and choose **Line**.

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

3 From the **Data set** list, choose **Edge 3D 1**.

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

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

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

7 Click **Define custom colors**.

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

9 Click **Add to custom colors**.

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

Streamline 1

1 Right-click **Streamlines** and choose **Streamline**.

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

3 In the **Number** text field, type 10.

4 Select Boundaries 24, 25, 34, and 37 only.

5 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

6 In the **Tube radius expression** text field, type 0.0025.

7 Select the **Radius scale factor** check box.

8 In the associated text field, type 1.

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 128, 0, and 64, respectively.

13 Click **Add to custom colors**.

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

Streamline 2

1 Right-click **Streamlines** and choose **Streamline**.

2 Select Boundaries 26, 27, 35, and 36 only.

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

4 In the **Number** text field, type 2.

5 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

6 In the **Tube radius expression** text field, type 0.0025.

7 Select the **Radius scale factor** check box.

8 In the associated text field, type 1.

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.

Streamlines

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

2 On the **Streamlines** toolbar, click **Plot**.

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

Continue to reproduce [Figure 3](#).

Pressure (spf)

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

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

3 Clear the **Plot data set edges** check box.

Surface

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

- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Data set** list, choose **Surface 2**.

Line 1

- 1 In the **Model Builder** window, under **Results** right-click **Pressure (spf)** and choose **Line**.
- 2 In the **Settings** window for **Line**, locate the **Data** section.
- 3 From the **Data set** list, choose **Edge 3D 1**.
- 4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 5 From the **Color** list, choose **Custom**.
- 6 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.
- 7 Click **Define custom colors**.
- 8 Set the RGB values to 128, 128, and 128, respectively.
- 9 Click **Add to custom colors**.
- 10 Click **Show color palette only** or **OK** on the cross-platform desktop.

Pressure

In the **Model Builder** window, under **Results>Pressure (spf)** right-click **Pressure** and choose **Delete**. Click **Yes** to confirm.

Surface 2

- 1 In the **Model Builder** window, under **Results** right-click **Pressure (spf)** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Data set** 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** check box.
- 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 **JupiterAuroraBorealis**.
- 9 Select the **Reverse color table** check box.

Surface 3

- 1 Right-click **Pressure (spf)** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Data set** list, choose **Cut Plane 2**.

- 4 Locate the **Expression** section. In the **Expression** text field, type p.
- 5 Locate the **Range** section. Select the **Manual color range** check box.
- 6 In the **Minimum** text field, type -13000.
- 7 In the **Maximum** text field, type 106000.
- 8 Locate the **Coloring and Style** section. Clear the **Color legend** check box.
- 9 From the **Color table** list, choose **JupiterAuroraBorealis**.
- 10 Select the **Reverse color table** check box.

Pressure (spf)

In the **Model Builder** window, under **Results** click **Pressure (spf)**.

Streamline Surface 1

- 1 On the **Pressure (spf)** toolbar, click **More Plots** and choose **Streamline Surface**.
- 2 In the **Settings** window for **Streamline Surface**, locate the **Data** section.
- 3 From the **Data set** 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 **Density** text field, type 16.
- 7 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.
- 8 In the **Tube radius expression** text field, type 0.002.
- 9 Select the **Radius scale factor** check box.
- 10 In the associated text field, type 1.
- 11 From the **Color** list, choose **Custom**.
- 12 From the **Color** list, choose **Gray**.

Pressure (spf)

In the **Model Builder** window, under **Results** click **Pressure (spf)**.

Streamline Surface 2

- 1 On the **Pressure (spf)** toolbar, click **More Plots** and choose **Streamline Surface**.
- 2 In the **Settings** window for **Streamline Surface**, locate the **Data** section.
- 3 From the **Data set** list, choose **Cut Plane 2**.
- 4 Locate the **Expression** section. In the **y component** text field, type 0.

- 5 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Magnitude controlled**.
- 6 In the **Density** text field, type 16.
- 7 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.
- 8 In the **Tube radius expression** text field, type 0.002.
- 9 Select the **Radius scale factor** check box.
- 10 In the associated text field, type 1.
- 11 From the **Color** list, choose **Custom**.
- 12 From the **Color** list, choose **Gray**.

Pressure (spf)

- 1 In the **Model Builder** window, under **Results** click **Pressure (spf)**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Title** section.
- 3 From the **Title type** list, choose **None**.
- 4 On the **Pressure (spf)** toolbar, click **Plot**.
- 5 Click the **Zoom Extents** button on the **Graphics** toolbar.

The following steps reproduce [Figure 4](#).

3D Plot Group 4

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type Vortex core in the **Label** text field.
- 3 Locate the **Title** section. From the **Title type** list, choose **None**.
- 4 Locate the **Plot Settings** section. Clear the **Plot data set edges** check box.

Surface 1

- 1 Right-click **Vortex core** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Data set** list, choose **Surface 2**.
- 4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 5 From the **Color** list, choose **Gray**.

Line 1

- 1 In the **Model Builder** window, under **Results** right-click **Vortex core** and choose **Line**.
- 2 In the **Settings** window for **Line**, locate the **Data** section.
- 3 From the **Data set** list, choose **Edge 3D 1**.

- 4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 5 From the **Color** list, choose **Custom**.
- 6 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.
- 7 Click **Define custom colors**.
- 8 Set the RGB values to 128, 128, and 128, respectively.
- 9 Click **Add to custom colors**.
- 10 Click **Show color palette only** or **OK** on the cross-platform desktop.

Isosurface 1

- 1 Right-click **Vortex core** and choose **Isosurface**.
- 2 In the **Settings** window for **Isosurface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Turbulent Flow, v2-f->Velocity and pressure>Vorticity field>spf.vorticityz - Vorticity field, z component**.
- 3 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.
- 4 In the **Levels** text field, type 90.
- 5 Select the **Interactive** check box.
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](#).
- 6 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
- 7 From the **Color** list, choose **Custom**.
- 8 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.
- 9 Click **Define custom colors**.
- 10 Set the RGB values to 0, 128, and 192, respectively.
- 11 Click **Add to custom colors**.
- 12 Click **Show color palette only** or **OK** on the cross-platform desktop.
- 13 Clear the **Color legend** check box.

Vortex core

- 1 In the **Model Builder** window, under **Results** click **Vortex core**.
- 2 On the **Vortex core** toolbar, click **Plot**.
- 3 Click the **Zoom Extents** button on the **Graphics** toolbar.

The following steps reproduce [Figure 5](#).

ID Plot Group 5

- 1 On the **Home** toolbar, click **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Swirl velocity in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 4 Locate the **Plot Settings** section. Select the **x-axis label** check box.
- 5 In the associated text field, type r (m).
- 6 Select the **y-axis label** check box.
- 7 In the associated text field, type v (m/s).
- 8 On the **Swirl velocity** toolbar, click **Line Graph**.

Swirl velocity

- 1 In the **Model Builder** window, under **Results** click **Swirl velocity**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Data** section.
- 3 From the **Data set** list, choose **Cut Line 3D 1**.

Line Graph 1

- 1 In the **Model Builder** window, under **Results>Swirl velocity** 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 $(x*v - y*u) / \sqrt{x^2 + y^2 + \text{eps}}$.
- 4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 5 In the **Expression** text field, type $-x$.
- 6 On the **Swirl velocity** toolbar, click **Plot**.
- 7 Click **Plot**.
- 8 Click the **Zoom Extents** button on the **Graphics** toolbar.

